Design and Analysis of Hydraulic Press Machine Used for Gasket Cutting Application

Govind Thakar[1] Sanket Sirsat[2] Munna Sontakke[3] Dipak Sirsat[4] Ashwin Josh[5] Kirankumar Nuti[6]
[1,2,3,4,5]Zeal College of Engineering & Research, Savitribai Phule Pune University, India

Abstract— The paper consist Hydraulic press having 20 Ton load and gasket cutting application. The 3D model of Hydraulic Press machine mainly includes component are cylinder, four pillar ,four pillar base, ram, upper plate for whole simulation of static structure the method used is finite element analysis. For 3D solid design of project software used is catia. This will results in position of sress and strain distribution, deformation in pillar, analysis of the strength and stiffness of structure. The result obtained from the modal, structural analysis used for the eliminating the resonance occur in working

Key words: Hydraulic Press, Stress Strain, Four Pillar, Finite Element Analysis

I. INTRODUCTION

Hydraulic power systems and the actuators are used where the large force is applied. Hydraulic system serves the weight and price benefit over the electomechanical system producing same effect. Cylinder piston positions and the main chamber pressure are control by the two optical scale or analog pressure sensor. For the mechanical calculation of the hydraulic press factors used are theory of; "material mechanics" and "elastic mechanics".[1]. Hydraulic press are used for process where we need the mechanical power such as punch, press, extruding, stamping, forming. Among this processes various process are perform on Hydraulic press. There are two type press first is manually control and other is plc operated

For a long time, the researches of hydraulic press structure design are mainly focus on the static analysis. Since the operation speed of equipment has been improved greatly, the dynamic characteristic of each major part has an equal importance with the structural static characteristic. In the equipment running process, the equipment vibration is caused by frequent loading and unloading action. The resonance which is generated by operating frequency and equipment natural frequency could influence the service life of equipment. This paper, the equipment natural frequency could be obtained by modal analysis, and the result from calculation is used to avoid the resonance[1].

This paper include the design analysis of hydraulic press which is conroled by the control unit which performs the application of pressing (e.g. gasket)as well as compression test. To achive the flexibility and accuracy in the operation the position and pressure sensors are used which are programmed on matlab though the computer control system.

II. MODAL ANALYSIS OF HYDRAULIC PRESS FRAME

A. Pre-Processing:

The first parts of equipment frame include four pillars, bottom plate. Since the symmetry of equipment frame, the 1/4 equipment is chosen as the simulation model. At first the 1/4 equipment is drawn in the catia as shown in Fig.1. Then the model is imported in the Ansys. The connections be-tween lower cross beam(bottom plate), column and lock nuts are set as contact connection, and the connections be-tween other parts are set as rigid connection..

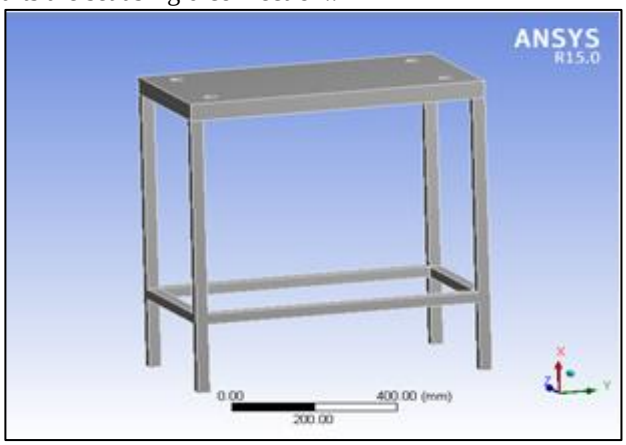

Fig. 1: The overall structure of the three-dimensional model of the hydraulic Machine table

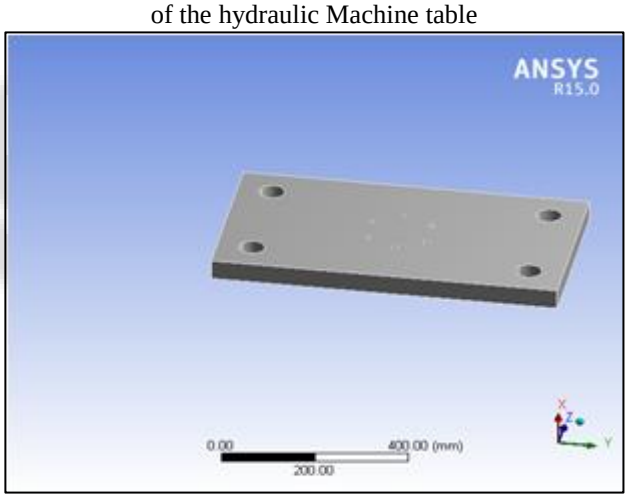

Fig. 2: The overall structure of the three-dimensional model of the hy-draulic ram(movable plate)

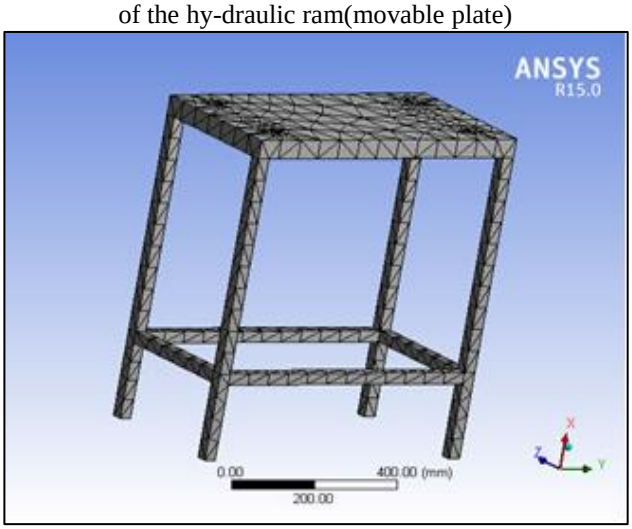

Fig. 3: Overall structure of model of hydraulic press table

The column material uses the 45 steel normalizing treatment, modulus of elasticity E=200GPa, poisson's ratioμ=0.3, yield strengthσs=270 MPa, allowable stress [σ]=200 MPa; beam material is gray cast iron HT300, modulus of elasticity E=143GPa, poisson's ratioμ=0.3, tensile strength σb=200MPa, allowable stress [σ]=250 MPa.[2] .

Assume that the hydraulic press column and bottom plate are rigid connection, at the same time do not consider the stress of installation and temperature; the bottom plate which supported above the frame, its role is primarily to limit the movement of hydraulic press in the horizontal direction, the ground to prevent the movement of vertical direction.

III. RESULTS ANALYSIS

A. Stress Analysis:

As the structure is subjected to force action, the Mises stress and the first principal stress overall distribution are respectively shown as Fig. 3 and Fig. 4. It is obvious that the maximum Mises stress values is 173MPa. at lower beam, and the first principal stress is 173MPa., The maximum Mises stress on the movable beam is 66.31MPa , which connected to piston rod, as shown in Fig. 4.strength does not exceed the allowable stress of their material.

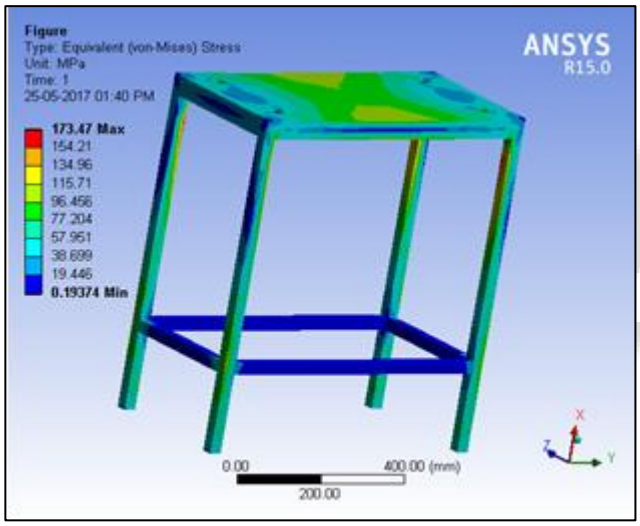

Fig. 3: Mises stress distribution of the frame (table).

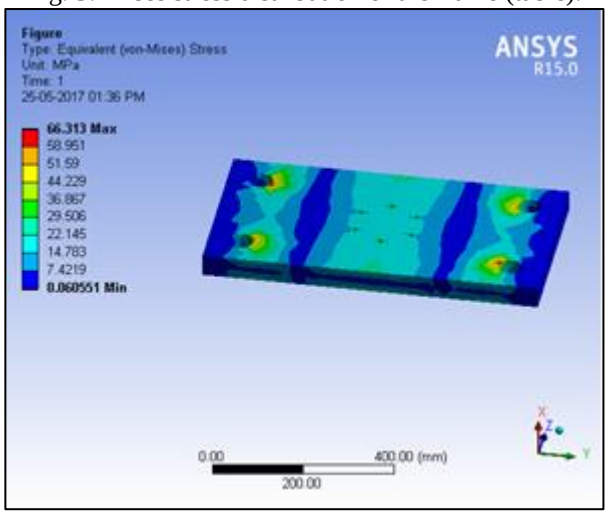

Fig. 4: Mises stress distribution of the ram (moving plate).

B. The Deformation Analysis:

The maximum deformation occurs in the movable beam, is 0.107mm Fig. 5 shows the deformation of movable beam. And also the maximum deformation occurs in the frame i.e bottom plate is 2.64mm. Fig.6 shows the deformation in the frame (bottom plate)

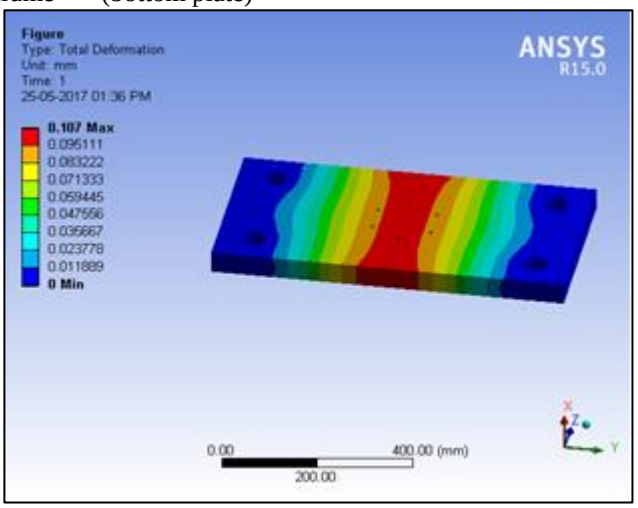

Fig. 5: Total deformation of the ram (moving plate).

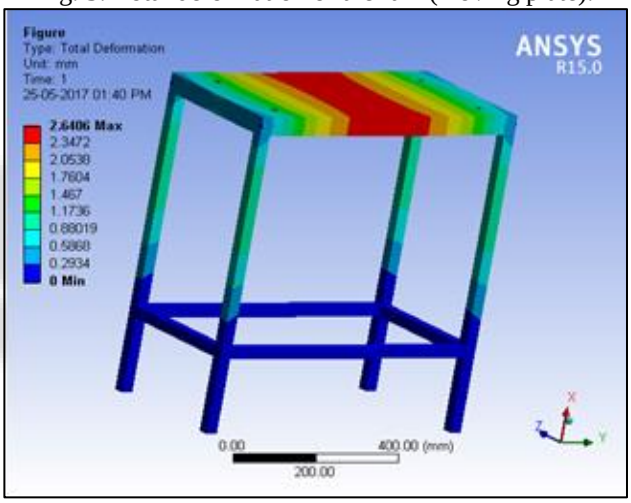

Fig. 6: Total deformation of the table

C. Buckling Analysis:

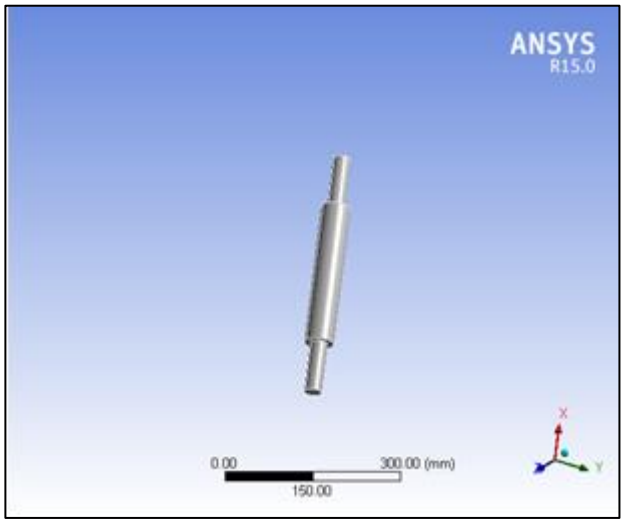

Fig. 7: The overall structure of the three-dimensional model of the hy-draulic machine column

In the hydraulic press machine shown in the fig on the column some weight is going to act act so that's why it going under diflecion so the maximum deformation in the column is about 1.0004mm as shown in the Fig.7 due to short length of the column there is no larger deflection in the column.

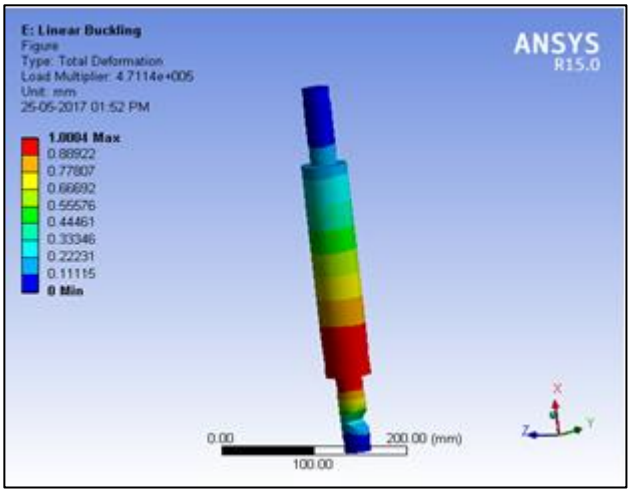

Fig.8: bukling of column

IV. CONCLUSION

The present paper concludes the designe and stess strain analysis of the hydraulic press machine to performe the given task (gasket cutting).for the analysis of the whole system use the ansys software and for 3D model drawing use the catia software. Through the ansys software we are able to find out the structral strength of hydraulic press is enough or not. By the strain diagram of the stress, can understand that: the structure strength of the hydraulic press is enough, it does not belong to the difficulty of design, here the strength of bottom plate, movable plate(ram) is enogh so the design is safe.

ACKNOWLEDGMENT

We want to wish special thanks to Dr. Sunil G Dambhare, Head of Mechanical Department, Zeal College of Engineering & Re-search, for their guidance and attention. We have to appreciate the guidance given by other Professors as well as the panels especially in our project presentation that has improved our presentation skills thanks to their comment and advices.

REFERENCES

[1] Hu Guoliang, Ren Jiwen.ANSYS11.0 finite element analysis introduction and enhancement [M].Beijing: National Defence Industry Press,2009,1.
[2] Yuefeng Li , Tieqiang Wang The Structural Static Analysis of Four-Column Hydraulic Press
[3] ''Fluid power with Application''-Pearson Education Inc. by Anthony Espisito
[4] '' Strengh of Materials '' Laxmi publication ltd. By Dr.R.K. Bansal
[5] X. Shang, F. Qiu, The senior analysis method of ANSYS struc-ture finite element and example applications [M]. Beijing: China water conservancy and hydropower press, 2006.
[6] S. Gong, Y. Huang, The finite element analysis and ANSYS APDL programming and advanced application[M]. Beijing: Mechanical industry press, 2009.3.

