Structural Analysis of Industrial H-Type Hydraulic Press by Using Finite Element Method

Haşmet Çağrı Sezgen, Mustafa Tınkır

Abstract

In this study, structural analysis of an industrial 300 tons H-type hydraulic press is investigated for geometric optimization using finite element method. For this purpose, linear static analysis of press body parts is realized and maximum Von Misses stress locations, safety coefficients, maximum deformation results and required optimization locations are determined via ANSYS Workbench software. The obtained results are given in the form of the graphics.

Keywords: Structural analysis, hydraulic press, finite element method, stress, deformation, safety coefficients, optimization.

1 Introduction

Metal forming machines and presses are one of the classic applications of hydraulic science and they are used in many branches of the industry for high quality and series production. New materials, products and new manufacturing process are new areas of application for press technology. Major power is provided by using hydraulic in presses for effective and high-volume production. Today, hydraulic presses which are the most important part of the industrial hydraulic are used in iron and steel industry such as plastering, twisting, extrusion and forging process [1].

However, some structural problems occur in the hydraulic press manufacture and use. These critical problems are investigated and listed below:

1) Time to time cracking, plastic deformation and fracture problems are seen in press body and components. This situation prevents the

performance of press, it leads to the deformation and breakage of the molds and decreases the efficiency and capacity of hydraulic press.

2) Some values are given about power, capacities and fatigue behavior of the press but these values cannot be validated exactly.

3) Press types are increased because of every manufacturer produces different type and size of press for customer desire. But increased production rate disrupt standard production of this kind of machines. Also manufacturers use lots of raw materials to produce without using engineering calculation.

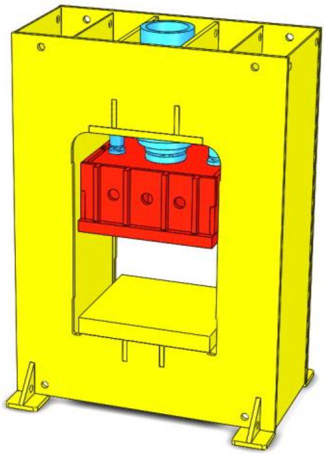

Figure 1. Computer aided design model of H-type hydraulic press.

In the light of all, these problems are considered and a large market, literature review and relationship with manufacturers are realized to help solving problems. As a result of investigations it can be said that engineering knowledge and realistic methods or formulas have not been used in press industry exactly. In literature review, different studies have been made about the hydraulic presses such as design and structural analysis. But the most important and similar studies are considered and given in this study. Arslan [2] studied the structural analysis of the body of an eccentric press using ANSYS software. Yağbasan [3] realized finite element analysis of C-type hydraulic press body. Raz et al. [4] analyzed stress-strain of hydraulic press components using finite element method. Zahalka [5] studied the modal analysis of a hydraulic press. Zhang et al

[6] have implemented structural optimization of the hydraulic press. Again, Zhang et al. [7] investigated mechanical analysis of the cylinder block of a hydraulic press.

In this paper, an industrial 300 tons H-type hydraulic press of manufacturer company given in Figure 1 is chosen and structural analysis of press is realized for geometric optimization. For this aim, linear static analysis of press body parts is realized and maximum Von Misses stress locations, safety factors, maximum deformation results and required optimization locations are determined via ANSYS Workbench finite element software. The obtained results are useful and realistic for chosen press manufacturer company to decrease using raw material for press production.

2 Structural Analysis

The relationship between external forces and displacements can be described as linear equations to solve static problems using finite element method. Spring can be used for elastic problem. Spring force is the product of displacement and spring constant. In the solutions of the finite element method, according to deformation and external forces the stiffness matrix can be written as:

$$f = k.x \tag{1}$$

where, k is the global stiffness matrix and x is the displacement vector.

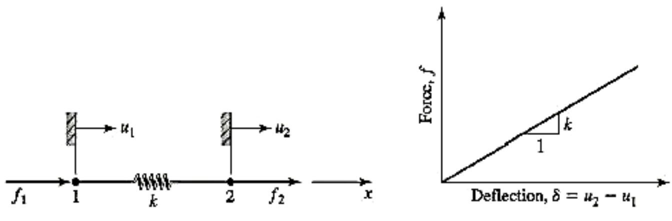

Figure 2. Relation between spring force, deflection and stiffness.

According to Figure 2, u_1 and u_2 are displacements that occur in the spring forces f_1 and f_2 applied to spring and k represents the spring constant.

Accordingly, net displacement formed in the spring is written as:

$$\delta = u_2 - u_1 \,. \tag{2}$$

External force applied to spring is described as:

$$f = k.\delta = k.(u_2 - u_1). \tag{3}$$

Applied forces can be written separately as:

$$f_1 = -k.(u_2 - u_1). \tag{4}$$
$$f_2 = -k.(u_2 - u_1).$$

These equations can be written as matrix form:

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix}. \tag{5}$$

When the initial equation is considered:

$$\begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} = [k_e] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}.$$

$$\downarrow \qquad \downarrow \qquad \downarrow$$

$$\{F\} = [K]\{X\}. \tag{6}$$

the stiffness matrix can be written as follow :

$$[k_e] = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}. \tag{7}$$

Assumptions for linear static finite element analysis are as follows:

- [K] is the global stiffness matrix, is constant.
- linear elastic material behavior is assumed.
- small deflection theory is used.
- {F} is the global load vector, is statically applied.
- No time-varying forces are considered.
- No damping effect is considered.

- Young's modulus and Poisson's ratio are always necessary for the linear static analysis.
- if there are the forces related inertia, mean density is needed.
- if a thermal load is applied, thermal expansion coefficient is required.
- stress-strain boundaries are required for safety coefficients

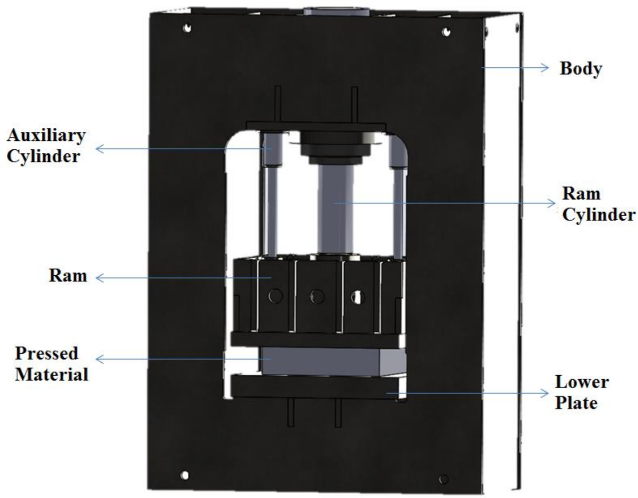

Figure 3. Hydraulic press main components.

The structural analysis of hydraulic press is realized, weak and more strength areas of the press body parts are determined using finite element method according to these assumptions. Also ANSYS Workbench finite element software is used in analysis. Computer aided design (CAD) model of the proposed hydraulic press is created in SolidWorks CAD program before the analysis and required shape optimizations of cad model are done for analysis. The connection holes, teeths and hydraulic adapter of press are removed from cad model. Some radius disturbed mesh geometry are removed. Moreover separate surfaces are drawn on touching parts of assembly components to superpose mesh geometry. Also ¼ symmetry is utilized to facilitate the solution. The mathematical model of hydraulic press given in Figure 2 is obtained by meshing cad model.

So cad model is divided into two identical or equivalent to close to geometric shapes. In this study, the mathematical model is achieved by using quadrilateral and triangular elements in ANSYS Workbench software and structural analysis is performed according to this model. The details of the mesh structure used in modeling are as follows:

For press body;

- ✓ maximum element size 36 mm.
- ✓ maximum face size 18 mm.
- ✓ minimum element size 2 mm.
- ✓ the growth rate is 1.85
- ✓ normal inclination angle is 45 degrees.
- ✓ mesh method is Sweep and Tetrahedrons.

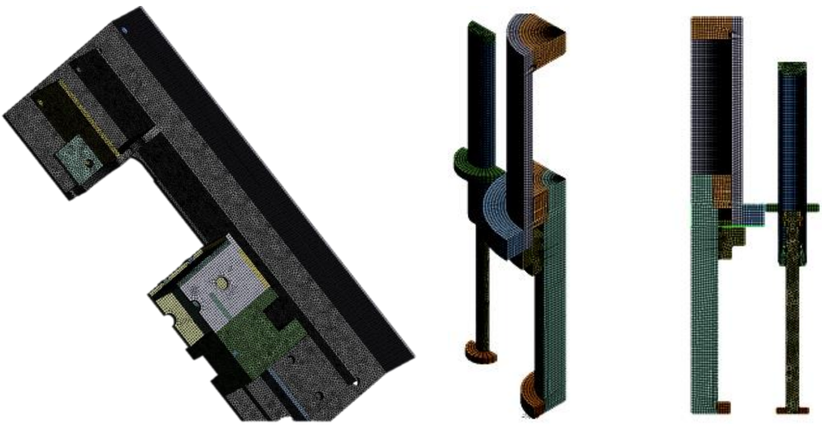

Figure 4. Mesh geometry of auxiliary and ram cylinders of the press.

For cylinders;

- ✓ maximum element size 12 mm.
- ✓ maximum face size 6 mm.
- ✓ minimum element size 2 mm.

- ✓ the growth rate is 1.85
- ✓ normal inclination angle is 45 degrees.
- ✓ mesh method is Sweep and Tethrahedrons

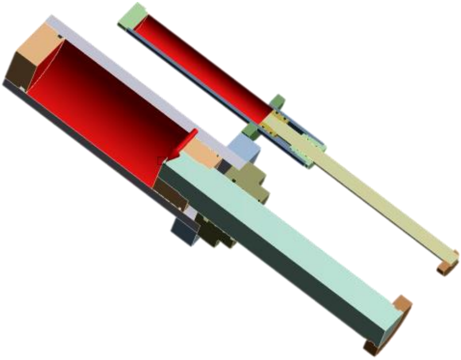

Figure 5. The applied pressure of the cylinder parts.

It is shown in Figure 4, appropriate mathematical model is obtained which contains overlapping part of close dimensional face. 250 bar hydraulic oil pressure is applied to drive the cylinder rod after mathematical modeling. In Figure 5 the pressure, formed in the piston top face, the upper face of the rod shaft, the cylinder inner face of the back cover and the inner face of the sleeve, are shown. Also the analysis model is simplified by carrying reaction forces instead of modeling material.

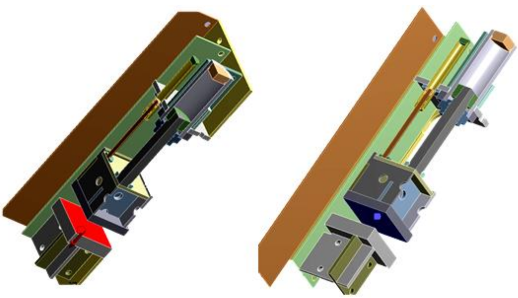

Figure 6. Assumption of pressing material.

Defined frictionless support to ram head and defined per load to bottom plate are shown in Figure 6. After loading the model the boundary conditions are determined and large deflections which are one of our boundary conditions for structural analysis is prevented. A screw connection is defined in the lower leg of press and all axes other than the direction of the normal are released (Frictionless support). The frictional connection is defined between barrel and piston, rod shaft and throat because the friction small displacements are perpendicular to surface normal. Fixed faces are shown in Figure 7.

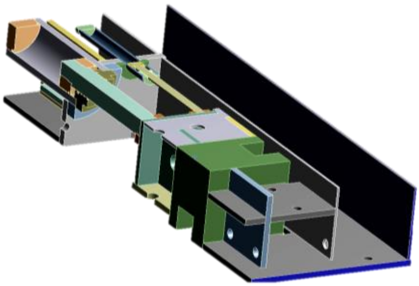

Figure 7. Fixed faces.

3 Results

To be able to comment on the modeling results, firstly stress results are obtained. The critical regions and safety factors are determined using Von Misses stress results. Also displacement results are considered to verify analysis according to given boundary conditions. In Figure 8, Von Misses stress and the displacement results of the press are given. Von Misses stress results in critical areas of the body are given in Figures 9 to 13. According to these results, the minimum safety coefficient for the outer wall layer is 12. For the front wall plate the minimum safety coefficient is obtained as 2.12. The minimum safety coefficients are for upper and rib platinum, inner wall and cylinder platinum 2.08, 2.89, 2.69 and 2.66 respectively. Moreover 1.98 and 4.44 safety coefficients are obtained for first and second lower platinum. Stress of the outer wall of press body is lower and no loads occur in this region. Thinner sheet metal can be used in this area but this time the buckling effect makes noise due to displacements while press running.

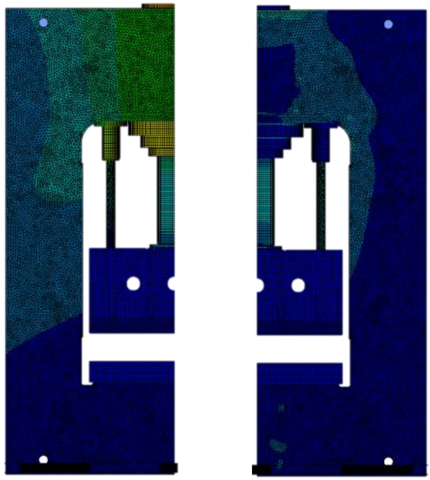

Figure 8. Von Misses stress and the displacement results of the press.

The stress as 90MPa is determined on the window radius of the front wall platinum of the body. This value can be decreased by radius optimization or rib assembly for this region. In addition, high stress rises due to the harsh design transition in connection ports.

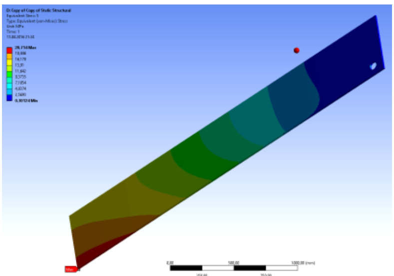

Figure 9. Critical region Von Misses stress results of the outer wall sheet metal of the body.

In connection with the upper platinum of body, the stress is obtained as 115 MPa. By smoothing, the transition stress can be reduced. Also 95 MPa. stress is found in the hole for transferring press. By changing the location of this hole stress can be reduced. Body rib platinum has the same stress with the upper platinum. Stress is determined as 80MPa in the inner walls of the body. This stress can be decreased with parametric optimization in this area. Then part of thickness can be reduced via topological optimization. The stress as 90MPa of connection port of the cylinder can be decreased by using a radius transition.

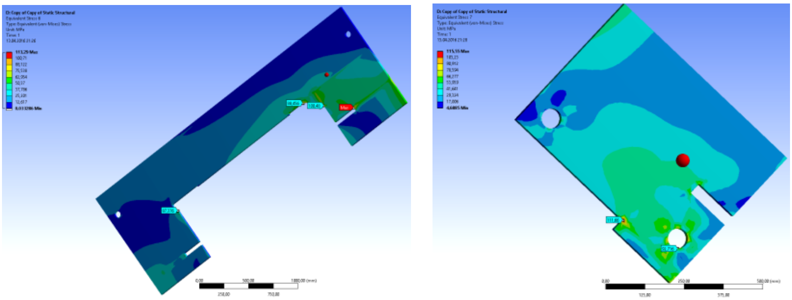

Figure 10. Von Misses stress results of the front wall and upper platinum of the body.

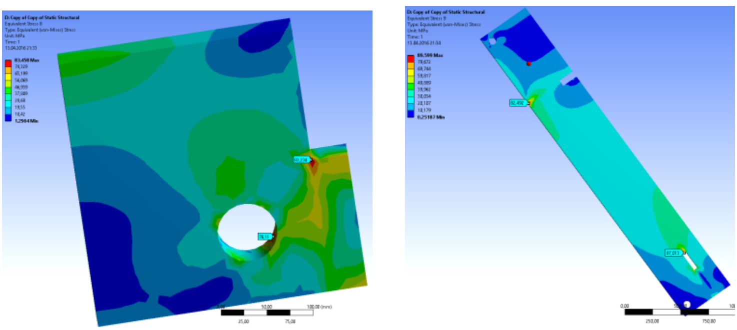

Figure 11. Von Misses stress results of the rib platinum and inner wall sheet metal of the body.

The lower platinum of the body has a stress as 120 MPa. With the design optimization in this area stress can be lowered. The stress as 50MPa occurred in the other sub platinum connection port of the body and the transfer hole can be decreased by design changes and further thickness can be reduced via topological optimization.

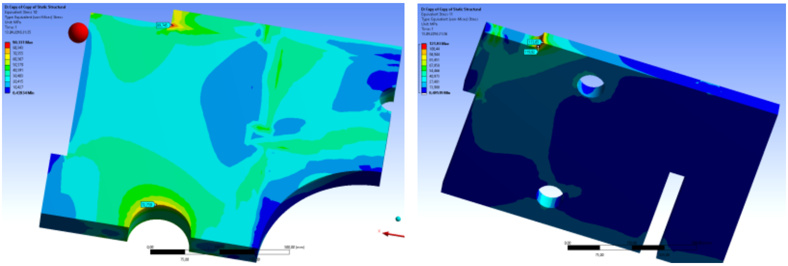

Figure 12. Von Misses stress results of cylinder port platinum and lower platinum 1 of the body.

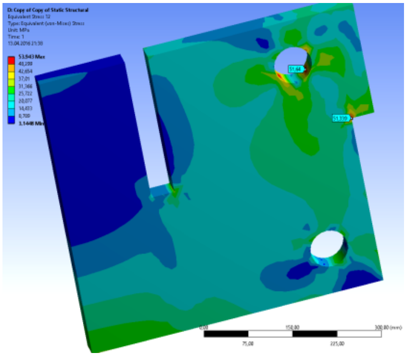

Figure 13. Von Misses stress results of lower platinum 2 of the body.

4 Conclusion

In this paper, an industrial 300 tons H-type hydraulic press is chosen and structural analysis of press body parts is realized for geometric optimization. For this aim, linear static analysis is performed and maximum Von Misses stress locations, safety coefficients, maximum displacement results and required optimization locations are determined via ANSYS Workbench finite element software. The obtained results are useful and realistic for chosen press manufacturer company to decrease using raw material for press production. The main contribution of the paper is that press company verified analysis results with their experiences, changed design parameters and sheet metal thickness of same type hydraulic press for lower cost production. The obtained results can be improved according to fatigue analysis in the future works by topological optimization.

Acknowledgments. This work has supported by the Coordinatorships of Necmettin Erbakan University Scientific Research Projects. Also many thanks to Hidroliksan Halim Usta Press Company.

References

[1] Taş, B., "Hydraulic press design", Master thesis of Institute of Science of Gazi University, (2008), 90 pages.
[2] Arslan, O., "Analysis of an eccentric press body via ANSYS", Thesis of Mechanical Engineering Dept. of Dokuz Eylül University, (2009), 35 pages.
[3] Yağbasan, O., "Analysis of a C-type press body using finite element method", Master thesis of Mechanical Eng. Dept. of Eskişehir Osmangazi University, (2010), 77 pages.
[4] Raz, K. and Vaclav, K., 2014. Using of a Hydraulic Press in Production and Manufacturing of Large Rings. Procedia Engineering, 69, 1064–1069.
[5] Zahalka, M., 2014. Modal Analysis of Hydraulic Press Frames for Open Die Forging. Procedia Engineering, 69, 1070–1075.
[6] Zhang, W., Wang, X., Wang, Z. and Yuan, S., 2014. Structural optimization of cylinder-crown integrated hydraulic press with hemispherical hydraulic cylinder. Procedia Engineering, 81, 1663-1668.
[7] Zhang, W.W., Wang X.S., Wang, Z.R., Yuan, S.J., He, Z.B., Liu G. and Dai, K., 2015. Mechanical analysis on the cylinder-crown integrated hydraulic press with a hemispherical cylinder. Journal of Mechanical Engineering Science, 299(3), 407-416.

Haşmet Çağrı Sezgen [1], Mustafa Tınkır [2]

[1]Mechanical Engineering Department of Institute of Science of Necmettin Erbakan University, Konya-TURKEY
E-mail: cagrisezgen@gmail.com

[2]Mechanical Engineering Department of Necmettin Erbakan University, Konya-TURKEY
E-mail: mtinkir@konya.edu.tr