iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 2 **Issue:** VIII **Month of publication:** August 2014

DOI:

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

Title: Design and Optimization of A 30 Ton Hydraulic Forming Press Machine

S. M. Bapat[* 1] and Dessai yusufali [#2]

[*1] (Author for correspondence)

Assistant Professor, Department of Mechanical Engineering, Gogte Institute of Technology,

Belgaum-590008, India;

[#2] post graduate student, Department of Mechanical Engineering, Gogte Institute of

Technology, Belgaum-590008, India;

Abstract: This paper deals with the FEA implementation for analysis and optimization of hydraulic forming press machine. Metal forming is one of the manufacturing processes which are almost chip less. These operations are mainly carried out by the help of presses and press tools. These operations include deformation of metal work pieces to the desired size by applying pressure or force. Press machine always works under impact load condition. Because of continuous impact load, the hydraulic press machine always experience continuous stress. Some parts of the machine experience compressive stresses and some experience tensile stresses. Press machine continuously deals with stress and because of that there are frequent structural failure problems in the machine. Different components of the machine are subjected to different types of loading conditions and are analysed using FEA tool. ANSYS is one of the FEM tool, which is incorporated in the present work. Weight optimization of press frame and upper head is done, which in turn resulted into reduction in thickness of frame structure and material.

Keywords: ANSYS, FEA, Hydraulic forming press, optimization, design

I INTRODUCTION

Fluid Mechanics provides the theoretical foundation of hydraulics and focuses mainly on its engineering applications. The basic law of fluid dynamics that govern the working of any hydraulic system, is the Pascal's law. The development of engineering over the years has been the study of finding ever more efficient and convenient means of pushing and pulling, rotating, thrusting and controlling load, ranging from a few kilograms to thousands of tons. Presses are widely used to achieve this. Presses are pressure exerting machine tools. They can be classified into three principal categories as: hydraulic presses which operate on the principles of hydrostatic pressure, screw presses which use power screws to transmit power and mechanical presses which utilize kinematic linkage of elements to transmit power.

Typical hydraulic press consists of a pump which provides the motive power for the fluid, the fluid itself which is the medium of power transmission through hydraulic pipes and connectors, control devices and the hydraulic motor which converts the hydraulic energy into useful work at the point of load resistance. The performance of a hydraulic press depends, largely, upon the behaviour of its structure during operation. However, these welded structures are becoming complicated and their accurate analysis under given loading conditions is quite important to the structural designer. Hence it is found that optimal design of a hydraulic press in terms of its weight is the need of the hour. The research on machine tool structures was stepped up by the application of the finite element method (FEM). This is a more generalized method in which a continuum is hypothetically divided into a number of elements interconnected at nodal points to calculate the strain, displacement and stress [1] . FEM is preferred because it permits a much closer topological resemblance between the model and the actual machine. It has been only

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

recently employed for press structures. The ANSYS Finite Element software system is used as a tool to establish the theoretically predicted numerical model. This theoretically predicted numerical model is based on different factors, like the boundary condition, the mesh density and the type of the element being used.

The present work is based on the finite element analysis of different parts of the hydraulic press machine and weight optimization of critical components of the hydraulic press machine. Figure no. 1 shows a 3D view of hydraulic press machine whose components are analysed.

II LITERATURE SURVEY

Sinha and Murarka [5] conducted a study on hydraulic presses. It represented a 3-D complex structure. It is found that an exact analytical method of stress and deformation analysis is cumbersome and time-consuming. In order to reduce core memory requirement and the cost of computation, a simplified plane stress (PS) FEM model for a hydraulic press structure (welded frame) has been identified for its analysis. On the basis of this investigation, certain significant guidelines have been obtained for the design of press frames. Such a model has resulted in savings in computational time, core memory requirement and cost of analysis.

Mohamad M. Saleh [6] has given a complete thesis on design study of a heavy duty hydraulic machine using finite element techniques. The machine is designed by ENERPAC without any measurement or variable hydraulic system. The investigation dealt the theoretical and experimental model of the machine to establish the accurately optimal design analysis and further development of the present machine at minimum time and lower cost. The applicability of the existing PC based FE package as a computer aided design tool is also investigated. A comparison has been made between the experimental and theoretically predicted results. Both the results are found to be in good agreement with each other.

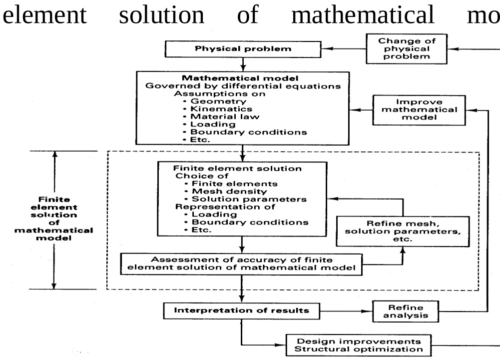
Work carried out by Muni Prabaharan and V.Amarnath [7] shows that, topology optimization has been applied on various components of scrap baling press and 5 Ton hydraulic press using ANSYS WORKBENCH software. It is inferred that topology optimization results in a better and innovative product design. A.G. Naik and N. K. Mandavgade [8] attempted for FEA implementation for analysis and optimization of top and bottom frame for hydraulic cotton lint bailing press. It is observed that selection of good shape provides strength to the system as the system is only undergoing through bending. According to the FEA analysis the best solution is obtained by changing the shape and design of the Top and Bottom frame structure.

Thus from the literature it is found that very less or little attention is given towards the analysis of frame structure in terms of its material, geometry and stresses induced in it. Hence the objectives of the present work are as defined below:

- Design and analyze the tie rods, support plates and frame of hydraulic press machine.
- Reduction of bending stresses causing bending of frame and other parts.
- Reduction of cost and Improve safety
- Changing the geometric structure and material of the frame -Design Optimization.

III. ANALYSIS PROCEDURE

The analysis procedure involves manual calculations of stresses and deflections using conventional design data hand book [4], FE analysis using ANSYS software (version 5.4), followed by weight optimization. Figure 1 shows a line diagram of the analysis. Once the physical problem is identified, a mathematical model is prepared which is governed by differential equations with the assumptions on geometry, kinematics, loading, boundary conditions, etc.
Then a finite element solution is obtained for the mathematical model. This solution includes choice of different types of finite elements, mesh density and solution parameters. The finite element solution represents the type of loading, boundary condition etc. Finally the results are interpreted after a proper assessment of accuracy of finite element solution of mathematical model is done.



INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

Figure 1: Line diagram of analysis

A) CALCULATIONS USING STRESSES AND DEFLECTIONS:

The tie rod acts under compression. Hence the stresses acting on each tie rod are compressive stresses and also design has to be done for buckling load.

L/K=880/12.5 = 70.4

As the L/K ratio is less than 120 we use Johnson's straight line formula [4].

$$F_{cr} = AS_y\left[1-\left(\frac{2}{3\pi}\right)\left(\sqrt{\frac{S_y}{3En}}\right)\left(\frac{L}{k}\right)\right] = 402534.42 \text{ N} = 41.03 \text{ Tonne}$$

Rankine's formula has been used in order to find the induced stress in the tie rod [4].

$$\sigma_i = \frac{p}{A}\left[1+\frac{a}{n}\left(\frac{L}{k}\right)^2\right] = 38.30 \text{ N/mm}^2$$

The lower plate supports all four tie rods which are fixed at its four edges and the load of 30 Tonne acts at the centre of the plate. Deflection at the centre is given by [3]

$$\delta = \alpha\frac{Pa^2}{Eh^3} = 0.07729$$

Maximum stress is given by [10]

$$\sigma_x = \beta\frac{p}{h^2} = 114 \text{ N/mm}^2$$

Since b = 450 mm (breadth of the plate) and a= 300mm (width of the plate). Hence b/a ratio is 1.5. The values of α and β are taken from the tables [9] for b/a value of 1.5. (refer table I).

Table I: Values for α and β for b/a ratio of 1.5 [9].

b/a	1.0	1.2	1.4	1.6	1.8	2.0
α	0.0611	0.0706	0.0755	0.0777	0.0782	0.0788
β	0.754	0.894	0.962	0.991	1.000	1.004

For upper head it is calculated as,

$$I = \frac{bh^3}{12} = 106875000 \text{ mm}^4 \text{ [2,4]}$$

The deflection is given by [2,4]

$$\delta = \frac{PL^3}{48EI} = \frac{30x1000x9.81x390^3}{48x2.1x10^5x1.06x10^3} = 0.0148mm$$

The section modulus is given by [2,4]

$$Z = \frac{bh^2}{6} = 1425000 \text{ mm}^3$$

The bending moment is given by [2,4]

$$M = \frac{WL}{4} = 27958500 \text{ N-mm}$$

Bending stress is given by [2,4]

$$\sigma_b = \frac{M}{Z} = 19.62 \text{ N/mm}^2$$

The analysis for frame is done only for one column considering the loading pattern and geometry of both columns of frame is same. Since the frame acts as a column analysis for buckling has to be done.

Area is calculated as [4]

A= (B x H) – (b x h) = (150 x 150) – (139.2 x 139.2) = 3123.36 mm^2

Moment of inertia is given by [4]

$$I = \frac{(BH^3 - bh^3)}{12} = 10899651.86 \text{ mm}^4$$

L/K= 1745/59.07 = 29.54

As the L/K ratio is less than 120 we use Johnson's straight line formula [4],

$$F_{cr} = AS_y\left[1-\left(\frac{2}{3\pi}\right)\left(\sqrt{\frac{S_y}{3En}}\right)\left(\frac{L}{k}\right)\right] = 71.73 \text{ Tonne}$$

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

Rankine's formula has been used in order to find the induced stress in the tie rod [4],

$$\sigma_i = \frac{p}{A}\left[1 + \frac{a}{n}\left(\frac{L}{k}\right)^2\right] = 2.35 \text{ N/mm}^2$$

B) FINITE ELEMENT ANALYSIS USING ANSYS

The Figure 2 shows the 3D model of the hydraulic press machine prepared using ANSYS. Analysis is also done using ANSYS 5.4. The meshing is done using element solid 45 node 8 element. Figure no. 3 shows the maximum stress distribution in the tie rod which is in compression.

Figure 2: 3D model of hydraulic press machine

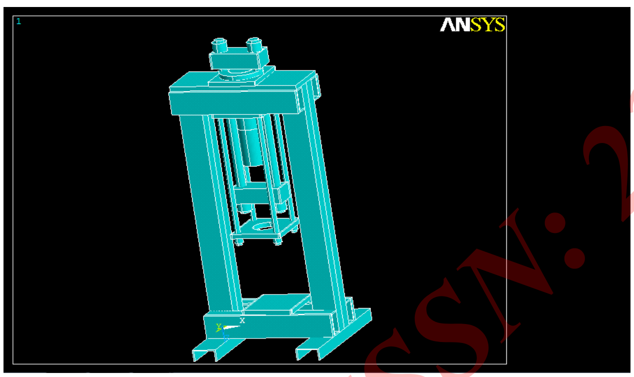
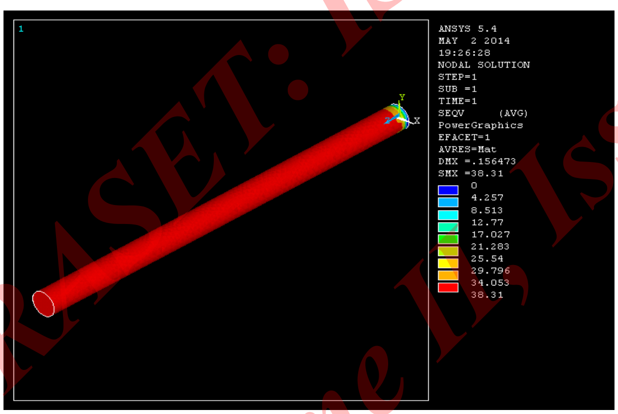
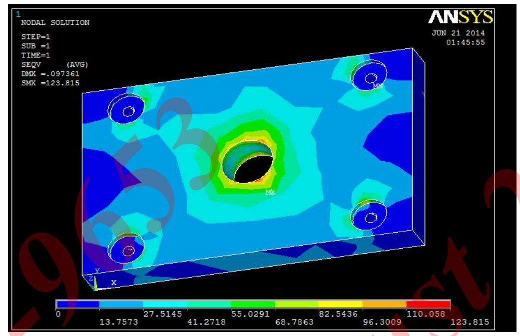


Figure 3: ANSYS result for maximum stress in tie rod



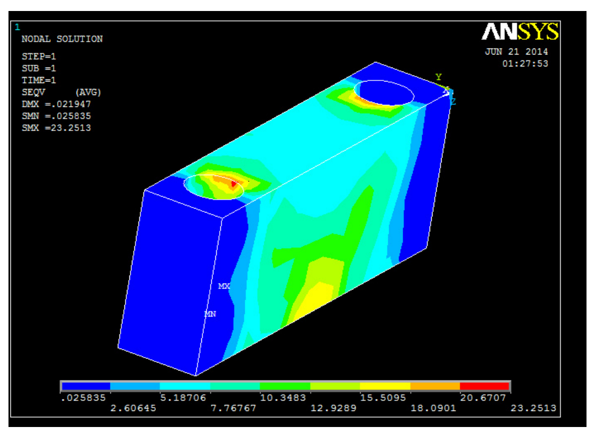
The tie rod is fixed at both ends and a compressive load is applied axially. From the figure it is clear that the maximum compressive stress is uniform throughout the length of the tie rod. Figure no. 4 shows the stress results for the supporting plate. The plate is fixed at all corner holes near the edges and load is applied at the centre of the plate. The maximum stress is shown at the centre region.

Figure 4: ANSYS result for maximum stress in lower support plate



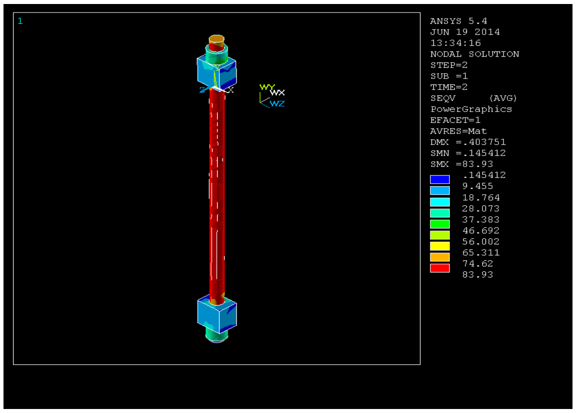
The stress distribution for the upper head is shown in figure no. 5. Here the boundary conditions are such that the head is fixed at both the ends near the hole where the tie rods are placed, and load is applied at the centre. The result shows clearly that the maximum stress is at the centre region. Figure no. 6 shows the stress distribution in the tie rod in tension and we can see the stress is uniformly distributed. The stress distribution of frame is shown in figure no. 7. The vertical members of the frame are in compression since the base is fixed and the load is applied axially from the top. The results interprets that the stress in the vertical members is uniformly distributed, but the maximum stress is in the region of contact between the vertical member and the base.

Figure no. 5 ANSYS result for maximum stress in the upper head



ISSN: 2321-9653

Figure 6: ANSYS result for maximum stress in tie rod in tension



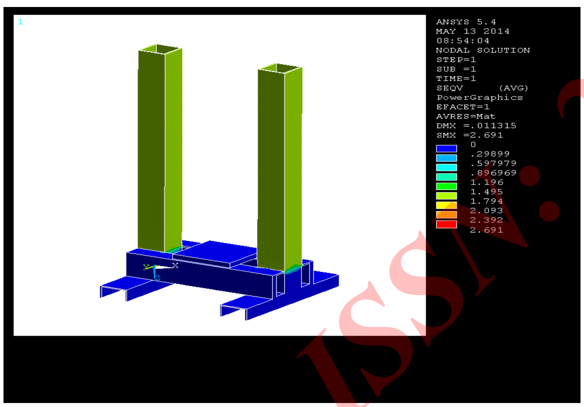


Figure 7: ANSYS result for maximum stress in the frame

C) OPTIMIZATION

Different cases are presented here with the modification in the thickness of the frame material and the results are also observed. Modification in the geometry of upper head is also done to save material of the same.

The thickness of the frame is reduced to 4 mm and the width and breadth is reduced to half [10] i.e. 75*75 mm. Then L/K ratio is found to be 60.80.Checking for buckling [4],

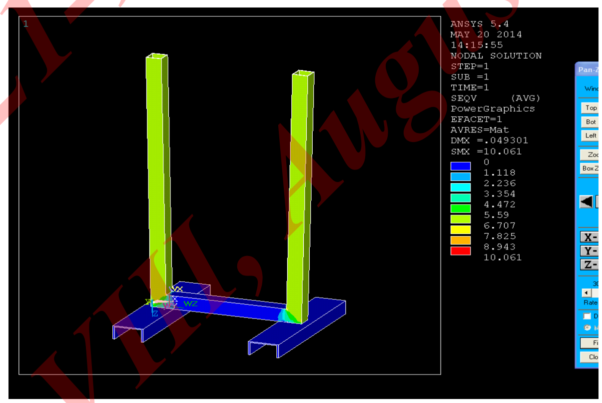
$$F_{cr} = AS_y\left[1-\left(\frac{2}{3\pi}\right)\left(\sqrt{\frac{S_y}{3En}}\right)\left(\frac{L}{k}\right)\right] = 23.41 \text{ Tonne}$$

Rankine's formula has been used in order to find the induced stress in the tie rod [4],

$$\sigma_i = \frac{p}{A}\left[1+\frac{a}{n}\left(\frac{L}{k}\right)^2\right] = 6.73/\text{mm}^2$$

The ANSYS solution is shown in figure no. 8. The model is analysed in the same way the actual frame is analysed with the same application of elements, boundary conditions and loads. The difference is in the geometry of the model.

Figure 8: ANSYS result for maximum stress in optimized frame



The actual upper head in the machine used is of rectangular section. This geometry is modified as shown in the figure 9.

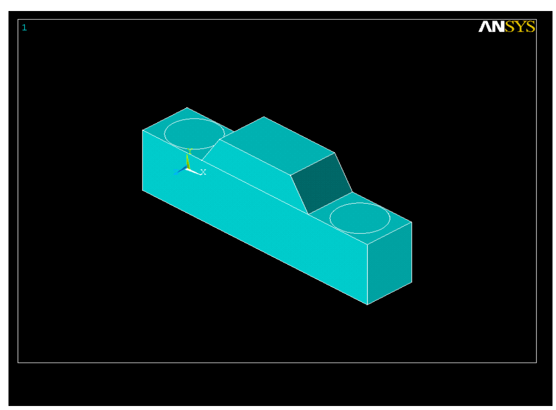


Figure 9: 3D model of the optimized upper head

The maximum stress from calculation is found out to be

ISSN: 2321-9653

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

Total section modulus $Z_{XX} = Z_A + Z_B = 691145.83$ mm^3

Bending moment [4] M=wL/4= =27958500 N-mm

Bending stress [4] $\sigma_b = \dfrac{M}{Z}$ = 40.45 N/mm^2

The ANSYS stress distribution result is shown in fig. 10.

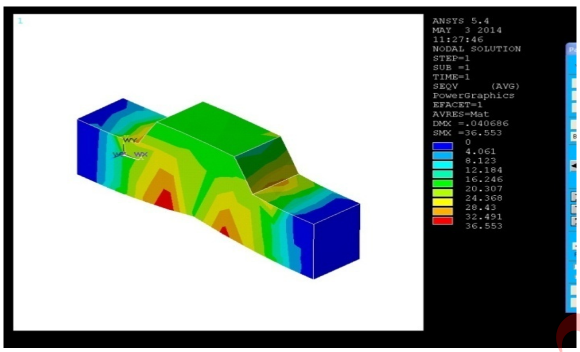


Figure 10: ANSYS result for maximum stress in the optimized upper head

The figure shows clearly that the maximum stress is acting at the centre bottom region and there is negligible stress acting near the ends where the heads is fixed.

C1) Weight optimization: It is a very important phenomenon in terms of cost and load considerations. We know that the density of the mild steel is 7801 kg/m^3 [4] and volume is 1910775 x 10^{-3} mm^3. But Weight = volume x density. Hence,

Weight of the actual frame = volume x density = 42.5 kg
Weight of both frame columns = 42.5 x 2 = 85 kg
Weight of the optimized frame = 14.9 kg
The amount of material saved is given by = actual frame weight – optimized frame weight= 85 kg – 29.8 kg = 55.2 kg.
Weight of the actual head = volume x density = 33.34 kg
Weight of both the upper and lower head = 66.68 kg.
Weight of the optimized head = 26.26 kg.
Weight of both the optimized head =26.26 x 2 = 52.52 kg.
The amount of material saved is given by = actual head weight – optimized head weight = 66.68 kg - 52.52 kg = 12.14 kg.

IV RESULTS AND DISCUSSION

The following results have been obtained by ANSYS and theoretical calculations for different parts and the results are compared. Table II compares the results for stress analysis.

Table II: Comparison of results of stress.

Component	ANSYS results		Theoretical results	
	Maximum stress in N/mm^2	Maximum deflection in mm	Maximum stress in N/mm^2	Maximum deflection in mm
Four tie rods	38.31	*	38.30	*
Support plate	123.875	0.0973	114	0.07729
Upper head	23.25	0.021	19.62	0.0148
Two tie rods	83.93	*	75	*
Frame	2.69	*	2.35	*

The ansys results for maximum stress for four tie rods, support plate, upper head, two tie rods and frame are 38.31, 123.875, 23.25, 83.93 and 2.69 N/mm^2 respectively. Whereas theoretical results of maximum stress for four tie rods, support plate, upper head, two tie rods and frame are 38.30, 114, 19.62, 75 and 2.35 N/mm^2 respectively. A comparison of both results shows that both of them are in close agreement with each other. A close observation of table II also shows that the values of maximum deflection for support plate and upper head are also in good agreement.
Table III shows that the load acting on the components are well below the critical load for buckling.

Table III: Buckling results for tie rod and frame.

Component	Critical buckling load (tons)	Load acting on the component (tons)
Tie rods in compression	41.03	7.5
Frame	71.73	0.75

As the applied loads are not exceeding the critical loads the components will not buckle. The difference between the

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

critical buckling load and the applied load for the frame is very high which demands the design of even thinner frame. But taking aesthetics of the machine in consideration an optimum thickness is selected for the design of the frame.

Table IV shows the % error in the results of Ansys and theoretical analysis.

It is found that maximum % error exists for upper head/frame and lowest for the four tie rods. Table V shows the results for the optimized parts.

A careful observation reveals that % error for ansys results and theoretical results is more in terms of stress (N/mm^2) as compared to deflection (in mm).

Table IV: Percentage error in the analysis of different machine parts.

Component	ANSYS results	Theoretical results	% error
	Maximum stress in N/mm^2	Maximum stress in N/mm^2	
Four tie rods	38.31	38.30	0.01 %
Support plate	123.875	114	8 %
Upper head	23.25	19.62	15 %
Two tie rods	83.93	75	10%
Frame	2.69	2.35	12 %

TABLE V: RESULTS OBTAINED FOR THE OPTIMIZED PARTS

Optimized components	ANSYS results		Theoretical results	
	Stress in N/mm^2	Deflection in mm	Stress in N/mm^2	Deflection in mm
Frame	10	*	6.75	*
Upper head	36.553	0.0406	40.45	0.0449

Table VI: Percentage reduction of weight

Components	Actual weight in kg	Optimized weight in kg	Material saved in kg	% reduction
Frame	85	29.8	55.2	64
Both heads	66.68	52.52	14.16	21

Table VI shows the values of weight reduction. It is found that a major reduction in weight of 64% is found for frame, whereas it is 21% for both the heads i.e. upper and lower head.

V CONCLUSIONS

The following conclusions are drawn from the present study:

- An attempt was made to analyse and optimize the 30 tonne hydraulic press machine using ANSYS software.
- The values of stresses obtained by ANSYS software conform with the values obtained theoretically within 15 % of error. The buckling analysis is done for the tie rods in compression and the frame. The critical load in both the cases is much less than the actual load acting on these components, hence buckling will not occur.
- Weight optimization is done for frame and upper head. From results we can say that as the thickness is reduced the maximum stress in the frame is increased but still it is well below the yield stress of the mild steel.
- The geometry of the pull up head i.e the upper head is modified to save material. Analysis shows that the stresses are well below the yield stress and hence it is safe.
- The total percentage reduction of weight in frame is 64 % and of both heads is 21 %. The weight of the frame is drastically reduced.

ACKNOWLEDGEMENT

The authors would like to express sincere thanks to Mr. M.S. Mishrikoti, Head of Design Department MSM HYDROPNEUMATIC ENGINEERS Belgaum, India, for his

INTERNATIONAL JOURNAL FOR RESEARCH IN APPLIED SCIENCE AND ENGINEERING TECHNOLOGY (IJRASET)

valuable guidance and sustained encouragement throughout the tenure of this work.

Nomenclature:

A - Area of the cross section.

E - Young's modulus of the material.

$\mathbf{F_{cr}}$- Critical load for buckling.

I - Moment of inertia.

K – Radius of gyration.

L - Length of the tie rod.

M - Bending moment.

P - Load acting on the machine.

S_y - Yield stress of the material.

Z - Section modulus.

n - Constant for end condition.

σ - Nominal stress.

σ_i - Rankine's induced stress.

σ_b - Bending stress.

δ - Deflection.

REFERENCES

[1] Bathe K. J. "*Finite Element procedures"*, Englewood Clis, NJ: Prentice hall. 1996.

[2] Spotts M. F. "*Design of machine elements"*, sixth edition. Prentice Hall of India pvt Ltd. 1988.

[3] Spotts M. F., Shoup T. E., Hornberger L. E., Jayaram J. R., C.V.
Venkatesh C.V. "*Design of Machine Elements"*, eighth edition.
Pearson Education Inc. 2006.

[4] Patil H. G., "*Machine Design data hand book"*, fourth edition,
IK International publishing house New Delhi. 2007.

[5] Sinha S. P. and Murarka P. D., "Computer-aided design of hydraulic press structures". *Mathematical Computing and Modelling*, Vol 10, pp 637-645,1988.

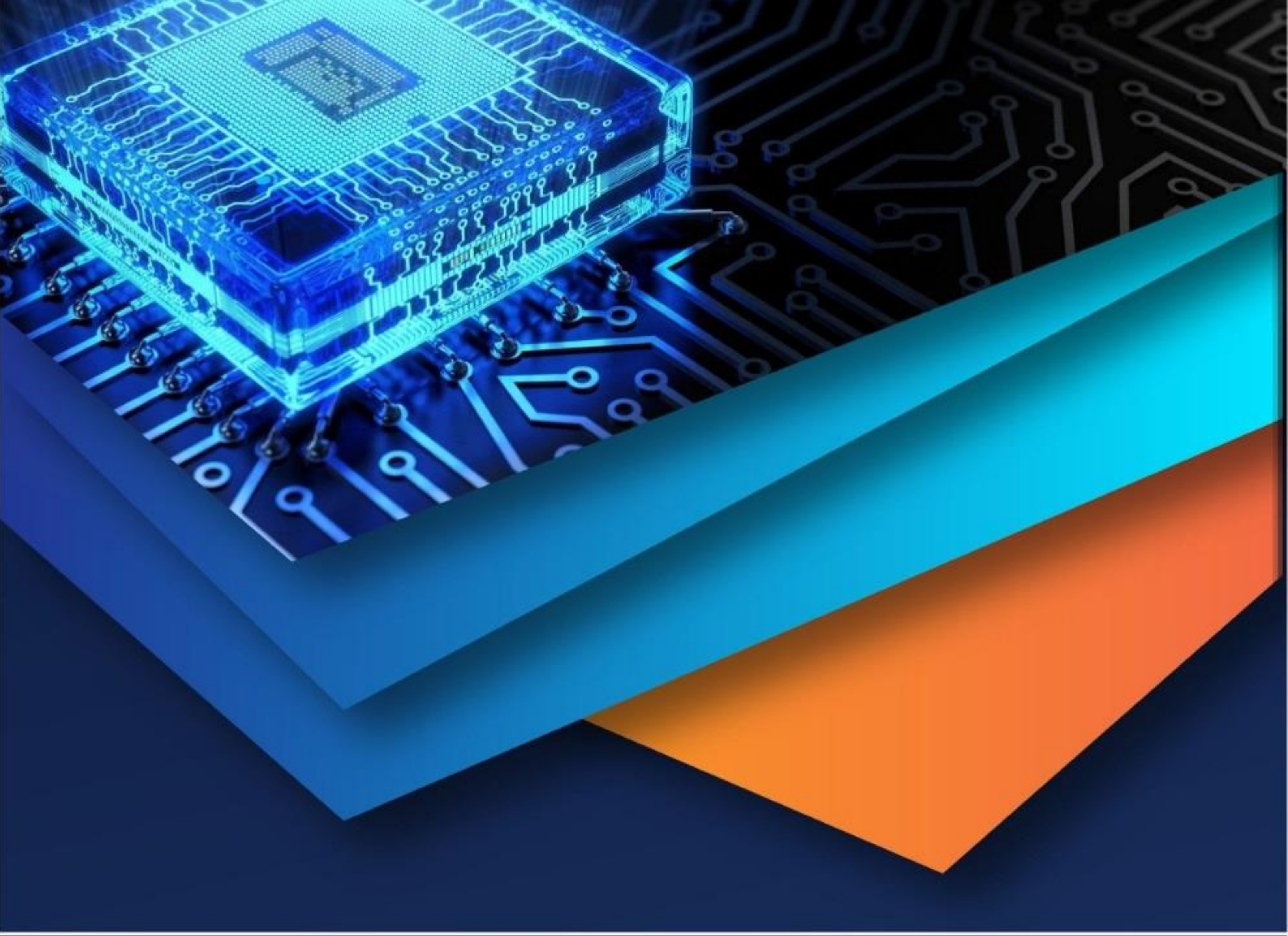
[6] Mohamad M. Saleh, *" Design study of a heavy duty hydraulic machine using finite element techniques"*, Doctoral Thesis, Dublin city university, 1992.

[7] Muni Prabaharan and V.Amarnath "Structural Optimization of 5Ton Hydraulic Press and Scrap Baling Press for Cost Reduction by Topology", *International Journal of Modelling and Optimization*, Vol 1, pp 185-190, 2011.

[8] A. G. Naik and N. K. Mandavgade, "FEA implemention in analysis and optimization of top and bottom frame for hydraulic cotton lint bailing press", *International journal of scientific & engineering research*, Vol 3, ,pp 1-8,2012.

[9] Vishwas Aggrawal, *" Steel Hand Book*" , Vishwas Techno Publications,2002.

[10] H.N.Chauhan and M.P.Bambhania , "Design & Analysis of frame of 63 ton power press machine by using finite element method", *Indian Journal of Applied Research*, Vol 3, pp 285-288,2013.



10.22214/IJRASET

INDEX COPERNICUS

45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY