Stress Analysis of a Frame of a Bush Pressing Machine for Pumps

Amith Kalekar [1], S. B. Tuljapure [2]

P.G. Student, Department of Mechanical Engineering, Walchand Institute of Technology, Solapur,India[1]

Assistant Professor, Department of Mechanical Engineering, Walchand Institute of Technology, Solapur, India[2]

ABSTRACT: A bush pressing machine is designed for pressing bronze bushes into a pump frame or casing. Finite element method (FEM) based software ANSYS is used for modeling & analysis work. As the machine is symmetrical about a vertical plane, only half model is prepared using the modeling facilities available in ANSYS. Meshing is done using tetrahedral (Solid-45) elements. A force of 17.78 KN is applied on the job mounting plate & cylinder mounting plate in the form of pressure. Deflections and stresses are observed after the analysis. Both deflection & stresses are the allowable within limit. The stress values obtained by FEM are in well agreement with the analytical values.

KEYWORDS: Finite Element Analysis, Press, Bush, Von Mises stress.

I. INTRODUCTION

Due to the uneven cooling of the bronze bushes or due to stringent tolerances and some other reasons, the bush is not getting fitted easily in the pump frame in few cases. Manual insertion is leading to inclined entry of bushes. Thus due to these problems, a bush pressing machine is designed to assemble the bush & frame of the pump.

The machine consists of two 'C' plates at sides. These plates are held together by frame strengthening plates. One hydraulic cylinder is mounted on the upper side. One job mounting plate is provided on the lower side to keep the jobs. Analysis of the machine is done to observe the stresses & deformation.

II. LITERATURE REVIEW

A. G. Naik, N. K. Mandavgade (2012) incorporated design process successfully into a structural shape optimization problem. The aim of their work was reduction of bending stresses causing bending of frame of a hydraulic cotton lint bailing press by optimizing the Top & Bottom frame. Reduction of cost and Improvement in safety was another aim of their work. Software ANSYS was used for this work. Due to new design a reduction in weight of frame was 13%.

H.N.Chauhan & M.P.Bambhania (2013) designed & analyzed frame of a 63 tonne power press machine using Finite Element method. Due to the impact loading at the end of the bolster plate there was development of a crack at the corner and stress generated was more due to continuous loading and stress concentration. Modifications were done by introducing the fillets of proper size. Also plate thickness was reduced which saved material.

Ankit H Parmar and et al (2014) performed optimization of a hydraulic press structure to decrease total mass of structure while assuring adequate stiffness. A method of structure optimization for hydraulic press was proposed in order to reduce mass while assuring adequate stiffness. Key geometric parameters of plates which have relatively larger impacts on mass and stiffness were extracted as design variables. In order to research relationship between stiffness, mass and design variables, common batch file was built by CREO and analysis was done in ANSYS. Top plate, movable plate and column design and analysis were done. A reduction of weight was 42 % keeping the stresses & deflection within limit.

Bhavesh Khichadia, Dipeshkumar Chauhan (2014) took review on design and analysis of mechanical press frame. They dealt with the analytical method and corresponding design and analysis of mechanical press frame. Modeling of press frame has created by the CAD software and according to the modeling structure failure analysis done by FEA tool. Stress distributions in press frame have been found out by the analytical and simulation methods.

Reference data were used for the new design for modification of a new press structure. With regard to design specification, stress distribution, deflection, optimization, ergonomics, stiffness and rigidity was focused on recent design and development in press frame obtained from structural components of press machine frame.

B. Parthiban and et al (2014) designed a 'C' type hydraulic press structure and cylinder. They analyzed press frame and cylinder to improve its performance and quality of press working operation. The frame and cylinder were modeled by using software CATIA. Structural analysis was done using analysis software ANSYS. An integrated approach was developed to verify the structural performance and stress strain distributions were plotted by using ANSYS software. According to the structural values the dimensions of the frame and cylinder were modified to perform the functions satisfactory.

D. Ravi (2014) analyzed a power press of 10 tonne capacity under static condition. The modeling of the C- frame power press was done using Pro/E software. The 3D model of the power press was analyzed in static condition to find the stresses and deflections in the structure. Later part involved the reduction in weight of the power press by varying or reducing the thickness of frame and bed and the press was analyzed in static condition to find the results. The result obtained from analysis package is within the limit.

Santoshkumar S. Malipatil and et al (2014) made an attempt with the objective to reduce the volume of material of a press. They considered an industrial application project consisting of mass minimization of H-frame type hydraulic press. ANSYS software was used for this analysis the main aim was to reduce the cost of the Hydraulic presses without compromising on the quality of the output. With regarding to design specification, stress distribution, deflection, and cost, were focused on optimized design. The methodology followed in this work was comparison of stresses induced in machine for different thickness used for construction of frame and column of the H-frame type hydraulic press.

III. DESIGN OF BUSH PRESSING MACHINE

Following are the figures of frame & bush. The tolerances provided for hole in the frame & bush outer diameter are studied in detail.
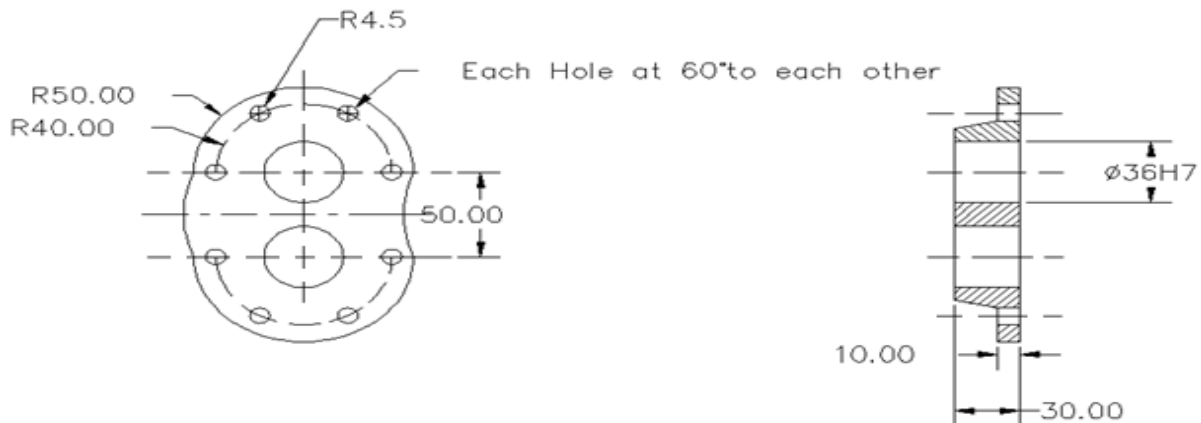

Fig. 1 Details of frame

The material of the frame shown in the above figure is Cast iron. Bush is to be fitted in the hole of diameter 36 mm. Only important dimensions & tolerances are shown in the figure to reduce the complications.
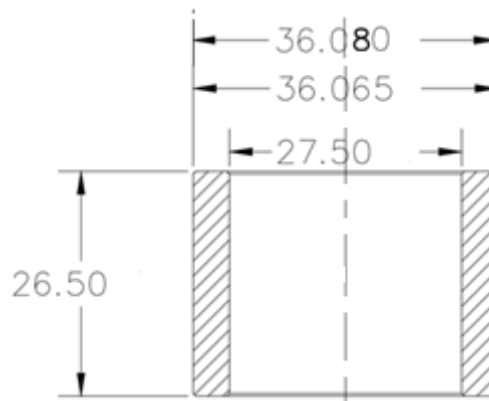
Fig. 2 Details of bush

The material of the bush used is Phospher Bronze. Here also only important dimensions with tolerances are shown. The diameter of the bush is larger than 36 mm to get the require fit.

Design of Press Body/Frame

The permissible tensile stress for the selected 'C' plates is given by

$\sigma_t = \frac{\text{Yield Stress}}{\text{Factor of Safety}} = \frac{304}{2.5} = 121.6$ N/mm^2

For design of the bush pressing machine, the maximum interference between bush & hole in pump frame after nitrogen cooling of bushes is determined. But the machine is designed for maximum interference which can be obtained by normal press fitting operation for the present diameter of bush. This interference is slightly higher than the one which is obtained due to nitrogen cooling of bushes. The force required for this operation is 17.78 KN.

Since two identical plates (C shaped) are taken, the force on each plate is (17.78/2) = 8.89 KN

These 'C' plates are subjected to direct tensile stress & bending stresses. The stresses are maximum at the inner fiber. At the inner fibre,

$\sigma_t = \frac{P}{A} \times \frac{M_b \times y}{I}$ …………[Ref. 1]

σ_t = Permissible tensile stress
P   = Force acting
A   = Resisting area of C plate = Width of C plate x Thickness of C plate
M_b = Bending moment
y   = Distance of neutral fiber from the point of application of force
   I = Moment of inertia of resisting cross sectional area of plate

Let us take, Width of 'C' plate = 150 mm
Hence,

$121.6 = \frac{8890}{150 \, x \, t} \times \frac{8890 \, x \, (150+75) \, x \, 75}{\frac{1}{12} \, x \, t \, x \, 150^3}$

t = 4.87 mm
Let us take a standard 6 mm thick plate for C plates of the body.
Considering the height of the normal person, the space required for mounting the casting frame & bushes, & the free space for the hand movements, mounting of hydraulic cylinder etc. other dimensions of the C plate are finalized and are as shown below in the figure.

Design of job mounting plate
The maximum force acting on the plate is 17.78 KN.
Let us take 250 mm x 300 mm (width) as area of mounting plates of job.
Now, load acts at the centre of the plate.
So, this is a case of a simply supported beam of length 300 mm with a central load of 17.78 KN.
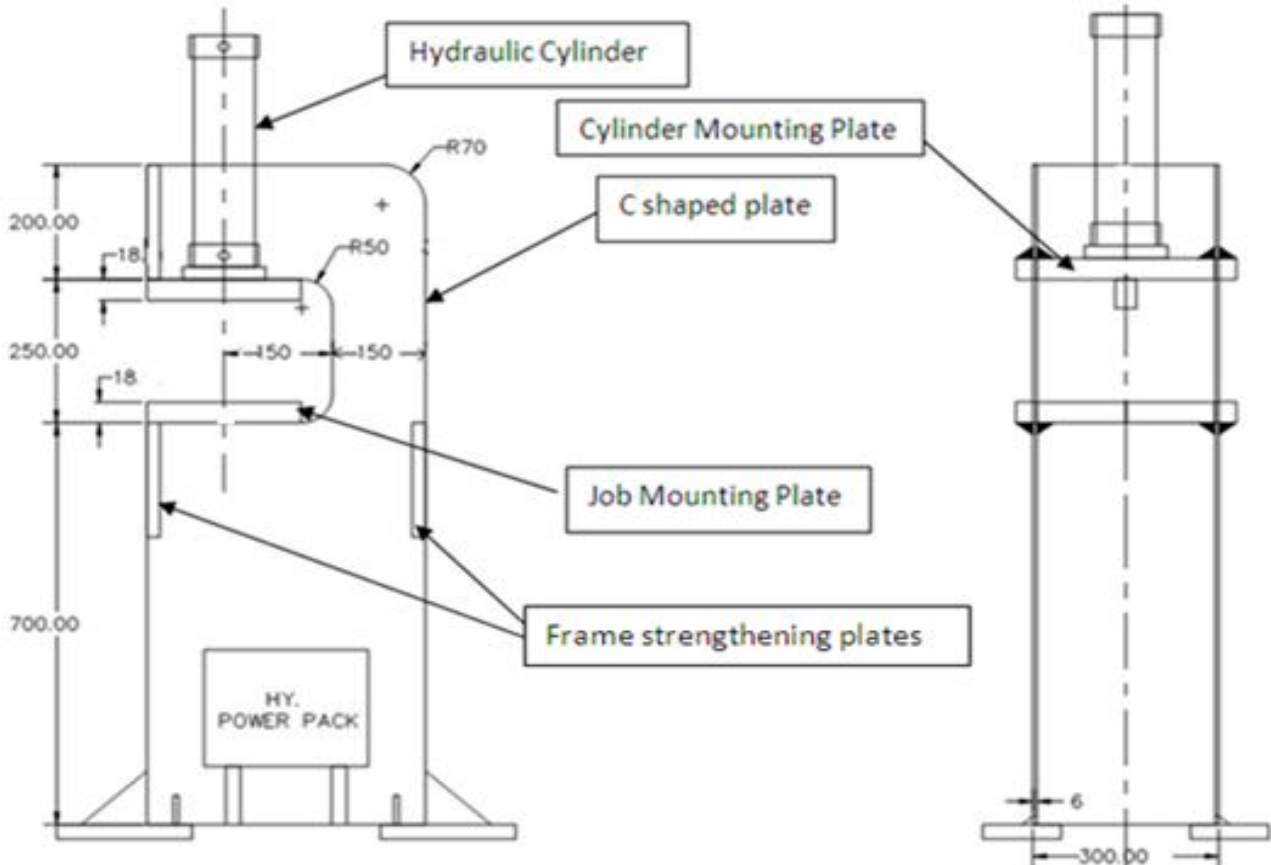Thus, the plate will be only under bending load.

Fig. 3 Bush Pressing Machine

The above figure shows details of the different parts of the complete machine.
Using,

$$\sigma_t = \frac{M_b x\, y}{I}$$

For simply supported beam,

Max. Bending Moment = M_b = (Force x Length)/ 4 = (17780 x 300)/4 = 1333.5 KN-mm

$$121.6 = \frac{1333500\, x \frac{t}{2}}{\frac{250t^3}{12}}$$

t = 16.22 mm = **18 mm (Say)**

Now the dimensions of the job mounting plate are 250 mm x 300 mm x 18 mm

The calculations done for job mounting plate are also applicable to hydraulic cylinder mounting plate. This plate will have same dimensions as job mounting plate.

Frame strengthening plates: Four plates are used for keeping two 'C' plates together & strengthening the body of the machine. The dimensions of these plates are 300 mm (width) x 200 mm (height) x 10 mm (thickness). Two plates are mounted in the front side, one at the top side & one at the bottom side. One plate is mounted at the back side of the machine. Leveling and installation plates are provided at the four corners of the base.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 2, February 2015

IV. FINITE ELEMENT ANALYSIS

Finite element analysis is a powerful tool in the field of engineering. Initially, finite element analysis was used in aerospace structural engineering. The technique has since been applied to nearly every engineering discipline from fluid dynamics to electro-magnetics.

While finite element analysis offers another way to analyze structures, it requires an understanding of the program and subject being modeled. If the operator does not use the correct model, time is wasted and more importantly the data is useless.

Finite element analysis is one solution to the problem. In finite element analysis the structure is broken up into small pieces that are easier to analyze. In the case of the cylinder it might be broken up into small cubes or plates called elements. All of the elements make up the mesh. Each of these elements can be easily solved for by using simple equations for stress and strain. As the number of elements increases (increasing mesh size), their size decreases and the solution will grow more accurate.

Analysis of C plate:
Initially 'C' plate of the machine is modeled & analyzed to check the suitability of element type & mesh density. The meshed model along with the load condition & boundary condition is shown in the following figure.
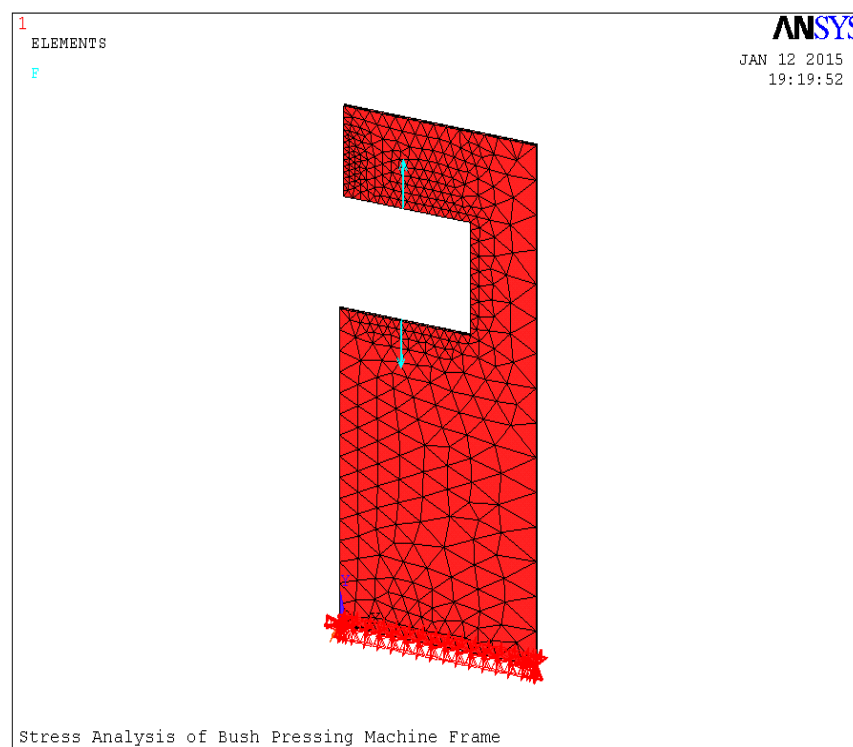

Fig. 4 Meshed model of C plate along with Load condition & Boundary condition

Meshing: Model is meshed using 8 Node Brick 45 (SOLID45) elements. The geometry of this element is as shown in the figure 5.

The tetrahedral option is used for the present problem. Solid 45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. At the inner fibre,
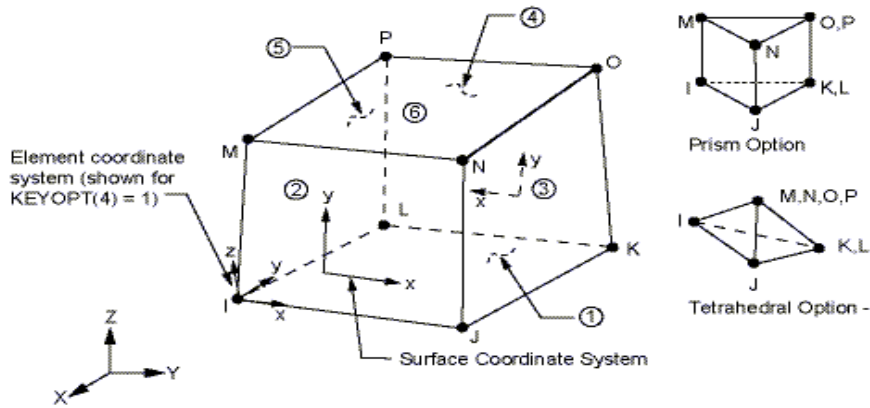
Fig. 5 Geometry of SOLID45 element [13]

$$\sigma_t = \frac{P}{A} \times \frac{M_b x \, y}{I} \qquad\qquad \text{.......[Ref. 8]}$$

with symbols indicating same parameter as earlier,
Hence,

$$\sigma_t = \frac{8890}{150 \, x \, 6} \times \frac{8890 \, x \, (150+75) \, x \, 75}{\frac{1}{12} \, x \, 6 \, x \, 150^3}$$
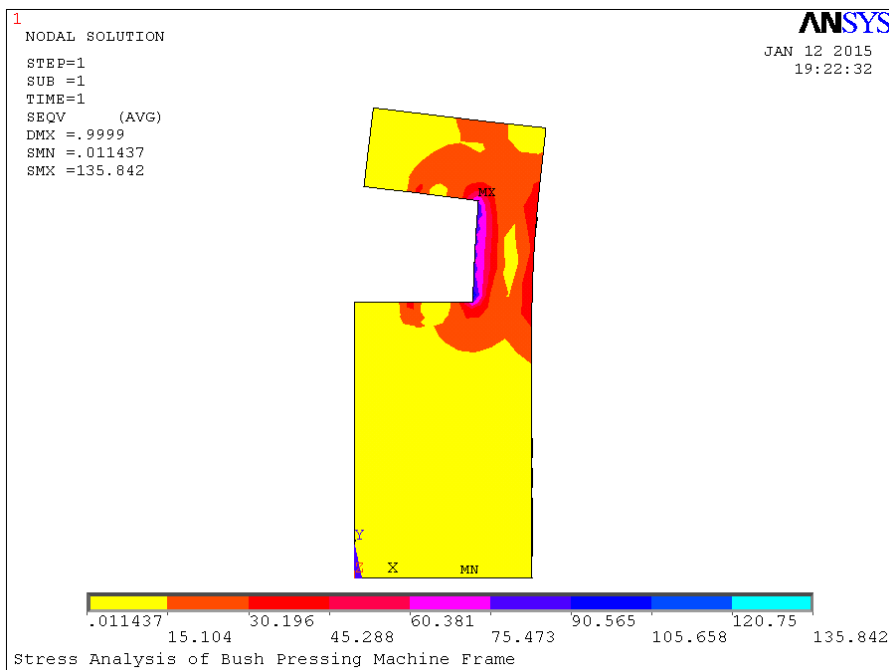
$\sigma_t = 98.698$ MPa

Fig. 6 Von Mises stresses of C plate

At the inner fiber, the stress values should be near to 98.698 MPa on the inner side of the C plates.
Figure 6 shows the stresses in the C plate. The stresses at the inner fiber are as per the calculations.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 4, Issue 2, February 2015

As the stress values obtained on the 'C' plate are as per the analytical values, it can be said that the type of element chosen & mesh density for the present structure are correct. Hence the analysis of complete frame is can be done taking same element & mesh density.

Modeling of the bush pressing machine: The modeling facilities available in ANSYS were used for the present work & found to be sufficient. As the frame or body of the machine is symmetrical along a vertical plane, only half model is prepared & symmetry plane is defined. This will reduce the computational time.

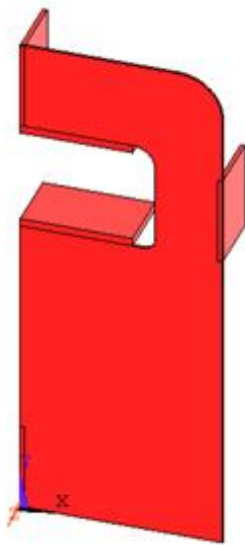Figure 7 shows the model prepared for finite element analysis.

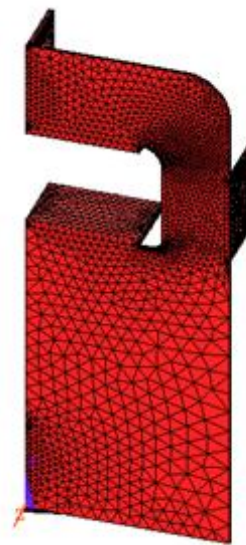Fig. 7 Model of press machine frame

Fig. 8 Meshed Model of press machine

Figure 8 is a view of the meshed model. Total 16583 elements & 5828 nodes are used for meshing the model.

Boundary Conditions: The lower surfaces of the frame are constrained for movement in all directions. Here areas of symmetry are also defined.

Load Conditions: The force in the form of pressure is applied on the job mounting plate & cylinder mounting plate.
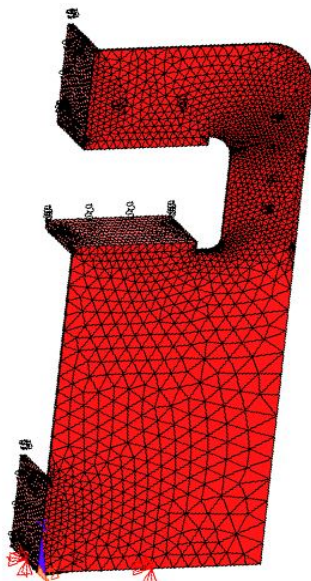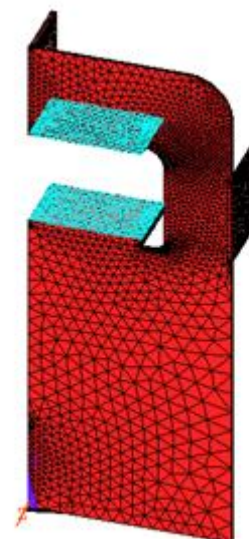

Fig. 9 Boundary conditions

Fig. 10 Load conditions

V. RESULTS & DISCUSSIONS

During post-processing, deformation, stresses etc are observed.

Deformed shape: Figure 11 shows the deformed shape of the machine frame along with the un-deformed shape. The maximum deformation is 1.053   mm. This maximum deformation will occur at the highest interference fit in press fit category. This situation will not come as due to nitrogen cooling of the bushes interference is going to reduce much. In those cases, actual deformation will be far less than the above value. This deformation is due to the unsupported upper part. In case of lower region, it is resting on ground. So deformation will be lesser in lower region. Thus the present dimensions of different parts of press machine are found to be satisfactory.
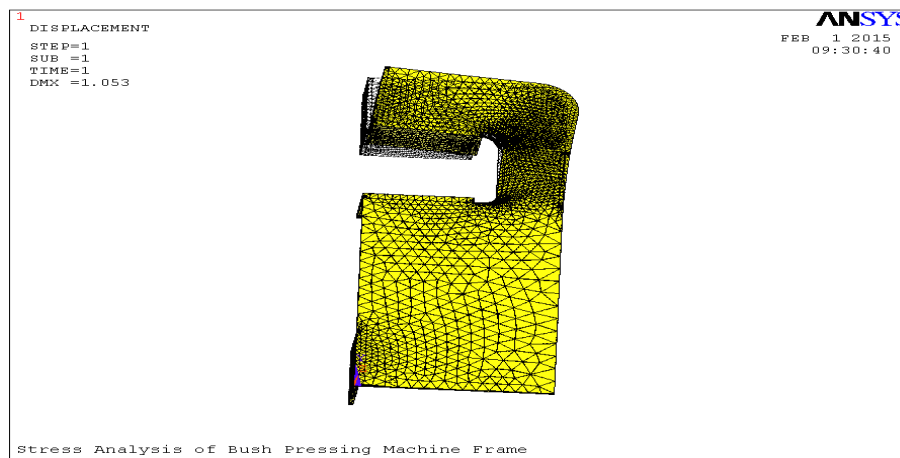

Fig. 11 Deformed shape

Von Mises stresses: The side plates of the machine frame are under combination of tensile & bending stresses. Here on the inner side due to the less deformation at the lower side of the side plate near to the job mounting plate, the stresses are more.
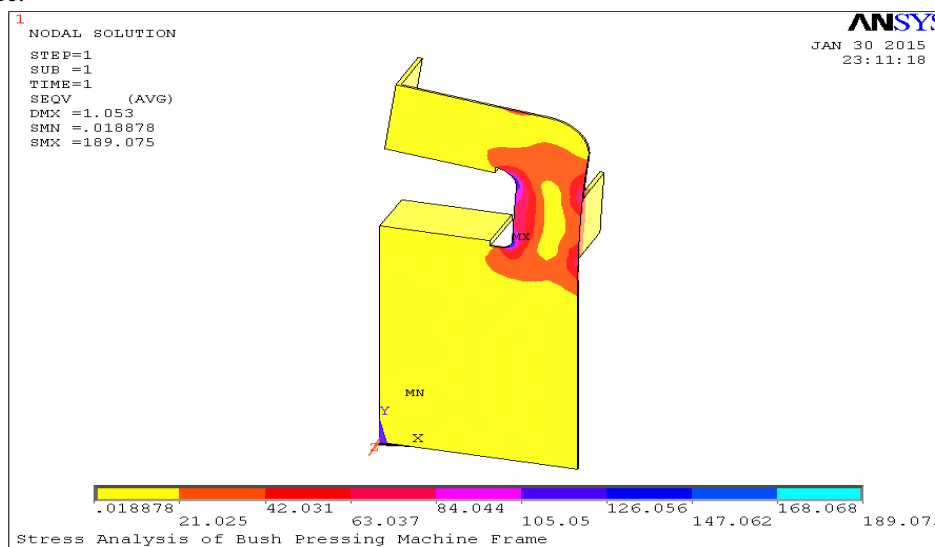

Fig. 12 Von Mises Stresses

Figure 12 shows Von Mises stresses for machine frame. The maximum stress is 189.075 MPa. This is occurring at the lower fillet region.

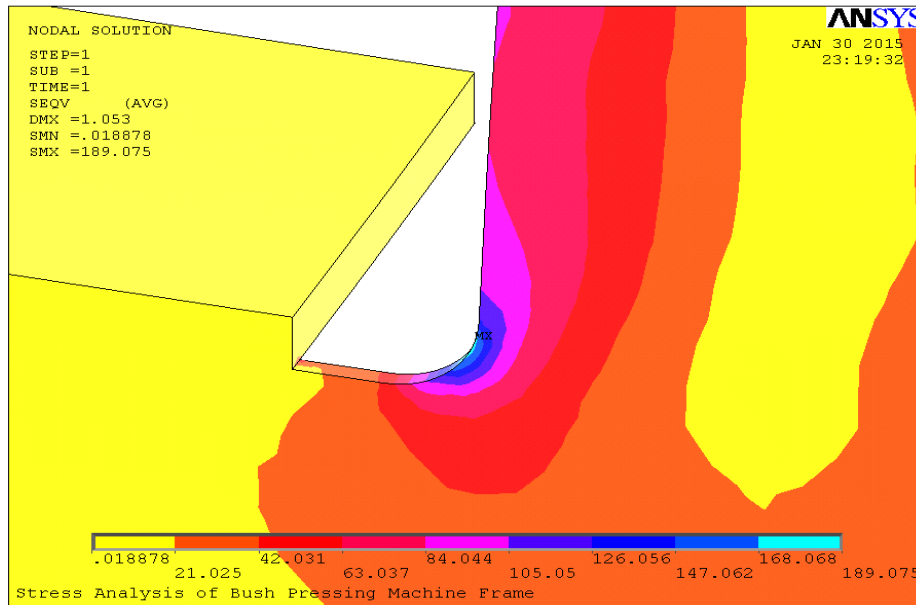Figure 13 is the exaggerated view of the frame.

Fig. 13 Exaggerated view showing region of highest stress

It is clearly showing maximum stresses at the lower fillet. Still the stresses are within limit. So the tructure is safe.

VI. CONCLUSION

Following are the conclusions.
The deformation & stresses for the designed bush pressing machine are within limit.
The deflection in the upper region is more due to unsupported part.
The stress is more in the lower region as there is less deformation.
Maximum stress occurs at the fillet in the lower region.

REFERENCES

1. V. B. Bhandari, "Design of Machine Elements", Tata McGraw-Hill Publishing Co. Ltd., New Delhi, 2008
2. Bernard Hamrock, "Fundamentals of Machine Elements", McGraw-Hill Publication, 1999
3. R. S. Khurmi, J. K. Gupta "A Text Book of Machine Design", S. Chand Publications, New Delhi 2012
4. S. R. Majumdar "Oil Hydraulic Systems-Principles and Maintenance", Tata Mc-Graw Hill Publishing Co. Ltd. New Delhi, 2008
5. PSG Design Data Handbook, 2008
6. A. G. Naik, N. K. Mandavgade, "FEA Implementation In Analysis And Optimization Of Top And Bottom Frame For Hydraulic Cotton Lint Bailing Press", International Journal of Scientific & Engineering Research Volume 3, Issue 7, 2012
7. H.N.Chauhan & 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, Issue:7, pp.285-288, 2013
8. Ankit H Parmar and et al, "Design and Modification of Foremost Element of Hydraulic Press Machine", International Journal of Advanced Scientific and Technical Research, Issue 4, Vol. 3, 2014
9. Bhavesh Khichadia, Dipeshkumar Chauhan, "A Review on Design And Analysis of Mechanical Press Frame", International Journal of Advance Engineering and Research Development, Volume 1, Issue 6, 2014
10. B. Parthiban and et al, "Design and Analysis of C type hydraulic press structure and cylinder", International Journal of Research in Aeronautical and Mechanical Engineering, Vol.2 Issue.3, pp. 47-56, 2014.
11. D. Ravi, "Computer Aided Design and Analysis of Power Press", Middle-East Journal of Scientific Research 20 (10): pp.1239-1246, 2014
12. Santoshkumar S. Malipatil and et al, "Analysis and Structural Optimization of 5 Ton H-Frame Hydraulic Press", International Journal of Innovative Science, Engineering & Technology, Vol. 1, Issue 5, pp.356-360, 2014
13. ANSYS User Manual, 2013