DESIGN & ANALYSIS OF C-FRAME OF 40 TON PNEUMATIC POWER PRESS USING FEA

[1]Rajdipsinh G. Vaghela, [2]Ravi C. Patel, [3]Kanaksinh Gohil

[1]Researcher, [2]Assistant Professor, [3]Managing Director
[1]Mechanical Engineering Department,
[1]Atmiya Institute of Technology and Science, Rajkot, India

***Abstract*—The objective of this research work is to design and analyze the C-frame of pneumatic power press for TARAK MACHINE TOOLS. The design of the frame is made using Creo-2.0 software. Design parameters are evaluated by analytical as well as software simulation using Ansys software. The obtained results are within the permissible limits which includes the stresses and deformation of the frame. Further efforts have been made to optimize the obtained results by using hyper mesh optistruct solver. It optimizes the stress value and gives a modified thickness of the frame, which thereby reduce the material required to manufacture it given the stress of the frame remaining under permissible limit. The results are validated by comparing the theoretical values with the obtained software values.**

***Index Terms*— C- frame, Creo-2.0, Ansys 14.5, Hypermesh 11, stress concentration, topology optimization numerical and experimental analysis**

I. INTRODUCTION

Power press are used for producing large quantities of articles quickly, accurately and economically from the cold working of mild steel and other ductile materials. The components produced range over an extremely wide field and are used throughout industry. Sometimes the pressings may be complicated and more than one pressing operation may be required. Now-a-days practice is to produce most of the sheet parts of any shape by using specially designed press tools and other combination of operations. For economical production of quantities of pressings, consideration has to be given to the rate of production, the cost of the press tools to be employed and the expenditure involved in setting them. It is also necessary to plan the operations to reduce scrap material to a minimum and to use waste material for other smaller pressings. For any operations to be performed on press, the selection of the proper press and the design of the tool or die to be mounted on it are very important.

II. DESCRIPTION OF THE PROBLEM

Power presses are used to produce large quantities of articles economically, quickly and accurately. Here a 'C' framed power press is taken into consideration, for punching operation. There are many operations performed in a power press, but in this project we consider punching operation. While punching, the Ram has to travel from the punch holder to the die block. So in this operation, due to the ram speed and force, some deflections are undergone in the structure.

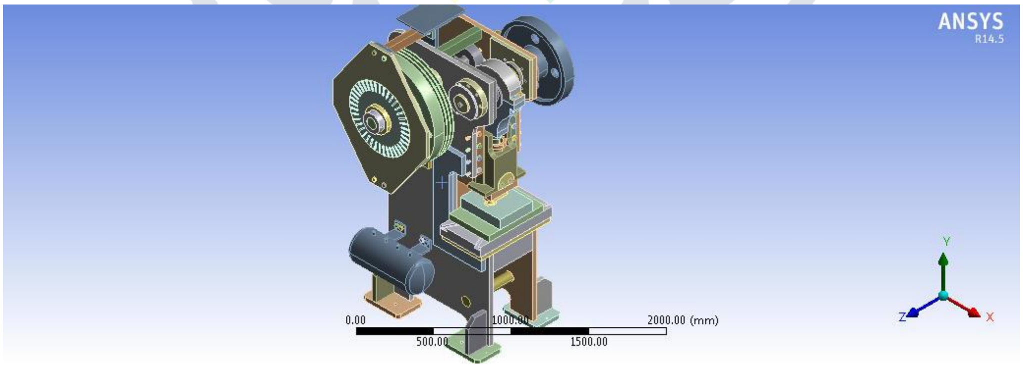


Fig. 1 C-type Power press

The drawback of the open frame design is the fact that such presses are generally limited in practice to the use of single dies. This is a result of several factors including the lack of stiffness and the typically small force capacity and die area of open frame presses. Here we are going to consider the deflections and stresses in the structure and minimize the material wherever there is no deflection and stress. Hence this reduces the weight of the power press and minimizes the cost of the production.

III. SOURCE OF THE PROJECT

The source of the project is from the company, TARAK MACHINE TOOLS, Pvt Ltd, Rajkot. The dimensions and specifications of a 40 ton capacity press are obtained from this company.

Technical specification

Table 1 Technical specification of press machine

Technical specification	
Description	**Dimension**
Tonnage	40
Diameter of Crankshaft	89
Adjustment of Stroke	10×88
Adjustment of Ram	51
Punch Hole of Ram	45
Hole in Bed (Bolster)	89
Length and Width of Bed	660×330
Thickness of Bolster	70
Distance between Bed to Ram	350
Flywheel Diameter	610
Stroke per minute in double geared	40
H.P / R.P.M (Geared)	3/1440
H.P / R.P.M (Ungeared)	4/960
Weight Approximate (kg.)	1900
Power supply	400/440 volts, 3 phase, 50 cycles

Table 2 Technical specification of Pneumatic Clutch/Brake unite

Capacity of CCB 600 Model			
Torque capacity (N.M)			
At 6.0 BAR		At 5.5 BAR	
Clutch torque	Brake torque	Clutch torque	Brake torque
7500	4500	6300	4500
Inertia kgm2			
EXT.		INT.	
2.35		4.0	
Stroke Volume Ltr.			
Worn out lining		New lining	
1.37		1.0	
Max. Speed (RPMS)			
1000			
Approx. Weight			
157			
All Dimensions are in mm			

IV. DESIGN PROCEDURE OF C-FRAME PRESS

The frame is the base machine element in press [16]. It is designed by the following steps.

1. Function

The main function of the frame is to withstand the force developed by the RAM. Frame is used for mounting and housing the press accessories like ram, die block, motor, flywheel, gears etc.

2. Determination of forces

The capacity of the press determines the major forces acting on the frame structure. The capacity of frame here is 40 TON.

3. Selection of materials

The “St 42 W” from WESTERMANN TABLE is selected for the frame because it is soft, ductile and they can be easily welded and machined.

Specification of Material

Designation: St 42 W

Tensile strength: 420 to 540 MPa

Density: 7850 kgf/m^3

Young’s Modulus: 2.1 x 105 N/mm^2

Poisons Ratio: 0.3.

Factor of Safety: 4.

For impact loading

Stress Concentration Factor: 1.25
Maximum Allowable Stress [σmax]: 460/4 = 115 N/mm^2

Theoretical design of C-frame

Assumptions considered.

1. As the frame is having the symmetrical cross section area, one side is to be consider for the purpose of analysis.
2. Maximum deflection may occur at the points A and B as shown in figure 6.2.
3. "C" web has been considered as integral with the frame.
4. The material of the frame is perfectly homogeneous and isotropic (i.e., it is same material throughout and of equal elastic properties in all the direction).
5. The material of frame obeys Hook's law.
6. The Young's modulus E is same in tension and compression.

Calculations

As the section is symmetrical, one side of the frame has been considered for the analysis. (Refer fig.2)

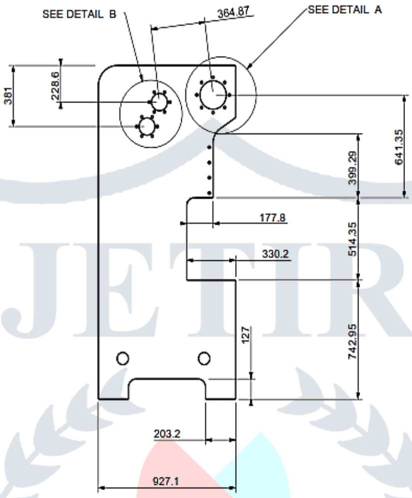


Fig.2 Drawing of C-Frame of Press

The permissible tensile stress for the plates is given by,

$$[\sigma_{max}] = \frac{\sigma_{yt}}{fs} = \frac{250}{4} = 62.5\ N/mm^2$$

Since the plates are identical, the force acting on each plate is (392400/2) or 196200 N [4]. The plates are subjected to direct tensile stress and bending stresses. The stresses are maximum at the inner fibre. At the inner fibre,

$$\sigma_{max} = \sigma_{tensile} + \sigma_{bending}$$

$$\sigma_{max} = \frac{P}{A} + \frac{M_b \times y}{I}$$

Where,
σ = Permissible stress in N/mm2
P = Applied load/ Force in N
A = Area of the plate section in mm2
Mb = Bending moment in N.mm
I = Moment of inertia in mm4
y = Distance from the neutral surface to the extreme fiber in mm
e = Perpendicular Distance in mm
Now we know that,

$$\sigma_{max} = \frac{P}{A} + \frac{M_b \times y}{I}$$

$$= \frac{P}{b \times t} + \frac{(P \times e) \times y}{\frac{1}{12} \times t \times b^3}$$

$$= \frac{196200}{70.1040 \times 35} + \frac{(196200 \times 774.7) \times 298.45}{\frac{1}{12} \times 35 \times 596.9^3}$$

$$\sigma_{max} = 79.96 + 73.13\ N/mm^2$$

$$\sigma_{max} = 153.09\ N/mm^2$$

$$\sigma_{max} = \frac{153.09}{4} = 38.27\ N/mm^2$$

Now as σmax < [σmax], so the press frame is safe.

Checking for the deflection

Deflection along the axis of application of force at A and at the extreme end B.
The deflection at the load axis A.
R = Radius of gyration.

H = R φ $\varphi = \frac{H}{R}$ … (1)

H + δ = R1 φ $\varphi = \frac{H + \delta}{R1}$… (2)

Where, R1 = R + a + L

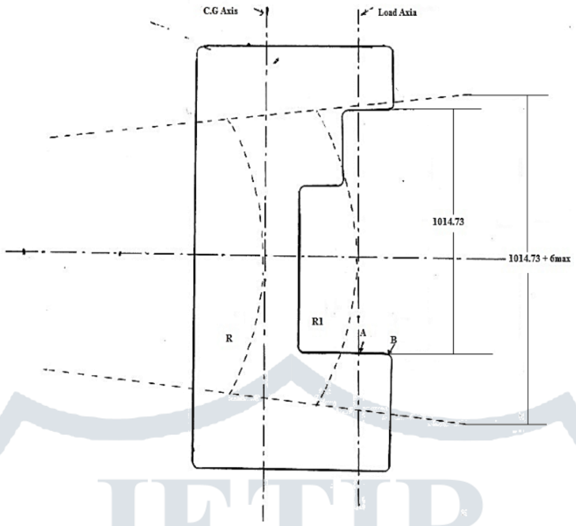


Fig.3 Effect of Radius of Gyration

Equating both equation,

$$\frac{H}{R} = \frac{H + \delta}{R + a + L}$$

So,

$$\delta = \frac{H(L+a)}{R}$$

In order to determine the increase in throat height i.e. δ, it is considered that the neutral axis bends under the load P to consider arc of radius R. In unloaded condition the neutral axis doesn't bend and in loaded condition the angle of inclination become φ.

Now,

$$\frac{M}{I} = \frac{\sigma}{Y} = \frac{E}{R}$$

$$R = \frac{E \times I}{M}$$

Where, Mb = 151.996 x 106 N.mm

I = 620.285 x 106 mm4

E = 2.1 x 105 N/mm2

$$R = \frac{2.1 \text{ x } 10^6 \text{ X } 620.258 \text{ x } 10^6}{151.996 \text{ x } 10^6}$$

R = 856.99 x 103 mm

Now we know that,

$$\delta = \frac{H(L+a)}{R}$$

Where, H = 1014.73 mm.

a = 177.4 mm.

L= 596.9 mm

R= 3.22 x 106 mm

So,

$$\delta = \frac{1014.73\ (596.9+177.7)}{856.99\ X\ 10^3}$$

δ = 0.9172 mm.

The deflection at the extreme end B.

$$\frac{\delta_{max}}{0.9172} = \frac{2(596.9) + 177.7}{596.9 + 177.7}$$

$$\frac{\delta_{max}}{\delta} = \frac{(2L + a)}{(L + a)}$$

δmax = 1.6237 mm

Checking for the maximum stress

For the maximum stress we use the principle stresses.

$$\frac{M}{I} = \frac{\sigma}{Y}$$

Where, Mb = 151.996×106 N-mm

I = 620.285 x 106 mm4

E = 2.1 x 105 N/mm2

Now we know that,

$$Z = \frac{I}{y}$$

$$Z = \frac{620.285\times 10^6}{298.5}$$

$Z = 2.0780 \times 10^6 mm^3$, and

$$\sigma_b = \frac{Mb}{Z}$$

$$\sigma_{b=} \frac{151.996 \text{ X } 10^6}{2.0780 \ X\ 10^6}$$

$\sigma_{b=}73.145\ N/mm^2$

Now we know that for transverse /direct shear stress, τ

$$\tau = \frac{P}{A}$$

$$\tau = \frac{196200}{2453.64}$$

$\tau = 79.962\ N/mm^2$

The maximum principal stress is

$$\sigma_{\text{max=}} \frac{\sigma_b}{2} + \sqrt{\frac{\sigma_b^2}{2} + \tau^2}$$

$$\sigma_{\text{max=}} \frac{73.145}{2} + \sqrt{\frac{73.145^2}{2} + 79.962^2}$$

$$\sigma_{max} = 36.572 + 95.23$$

$$\sigma_{max} = 131.80\ N/mm^2$$

$$\sigma_{max} = \frac{131.80}{fs}\ N/mm^2$$

$$\sigma_{max} = \frac{131.80}{4}\ N/mm^2$$

$$\sigma_{max} = 32.95\ N/mm^2$$

Now the maximum stress obtained is 32.95 N/mm2 and allowable stress is 62.5 N/mm2. As the maximum stress is less then allowable stress, the design is safe.

V. ANALYSIS OF C-FRAME POWER PRESS

There are six basic steps for doing the simulation in press machine to predict the defects by applying impact load.

Step 1- create CAD model of press frame structure.

Step 2- Import the CAD model in simulation software

Step 3- Pre processing

Step 4- Apply boundary conditions

Step 5- Post processing

Step 6 – Result and analysis

After creating the CAD model, import the CAD file in simulation software in .IGES or .STL format.

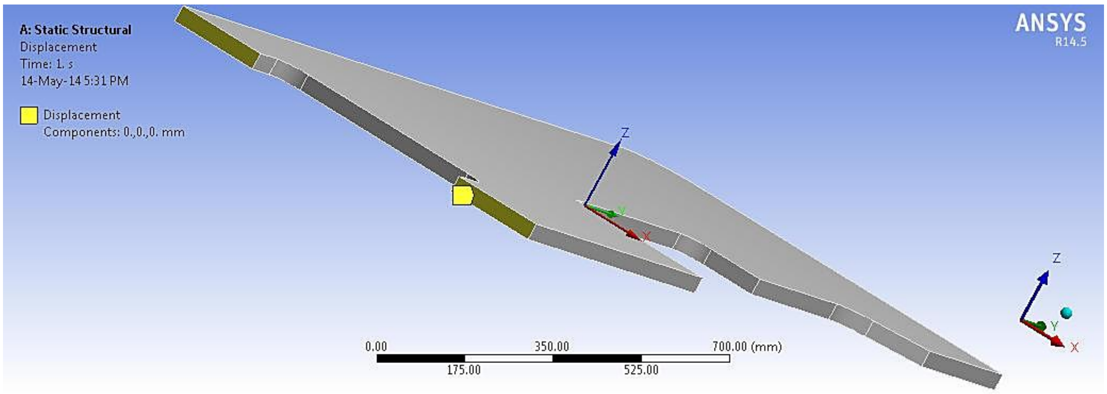


Fig.4 Appling boundary condition

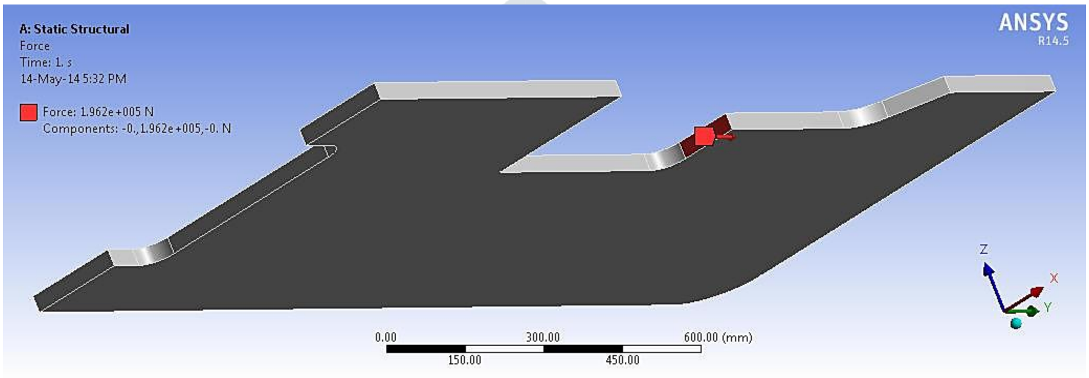


Fig. 5 Application of **l**oad at Ram line of action

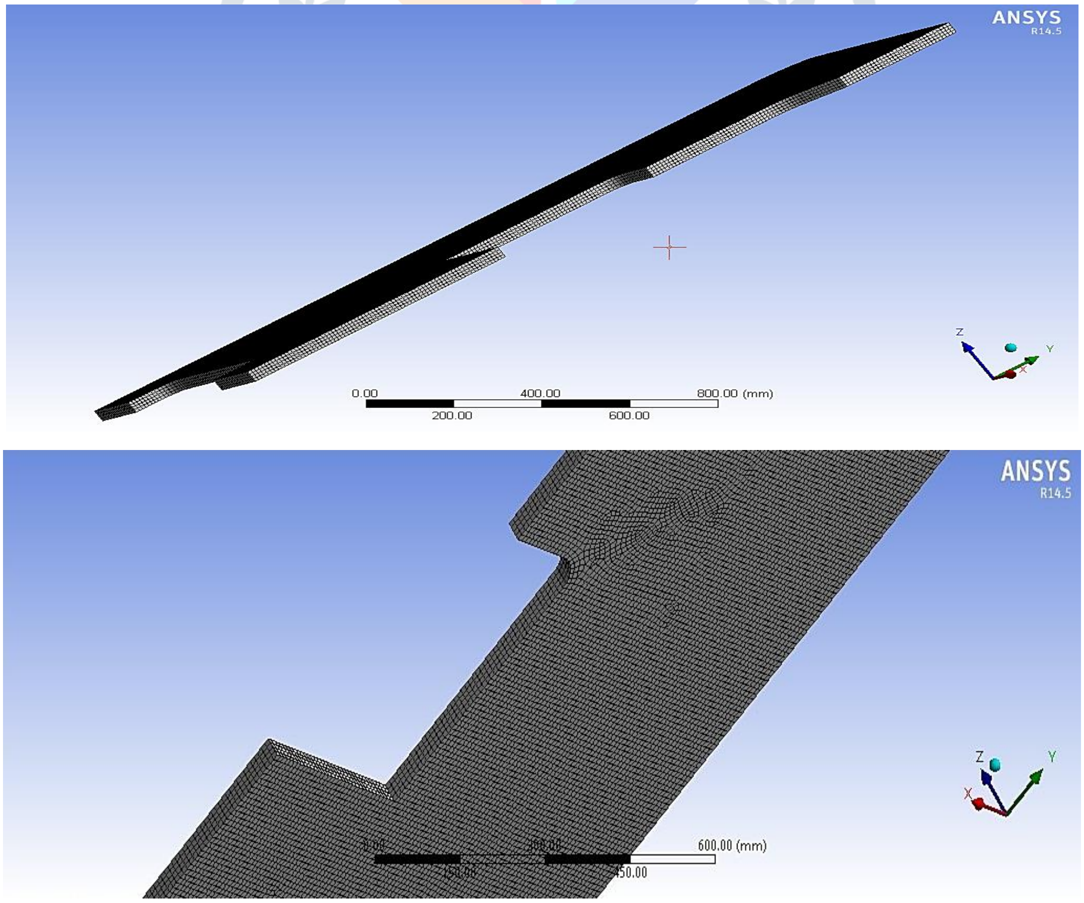


Fig.6 Meshing of C-frame model

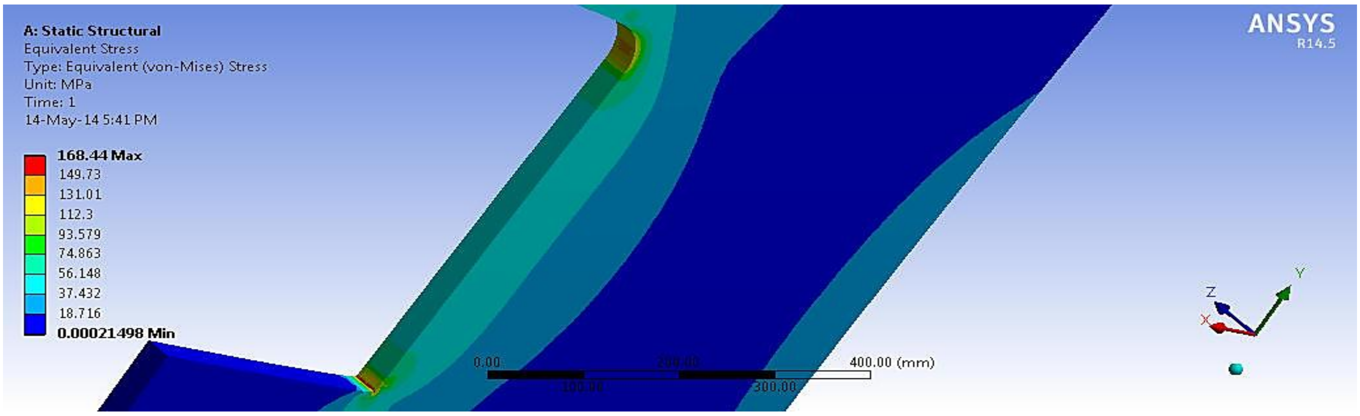


Fig.7 Stress affected area

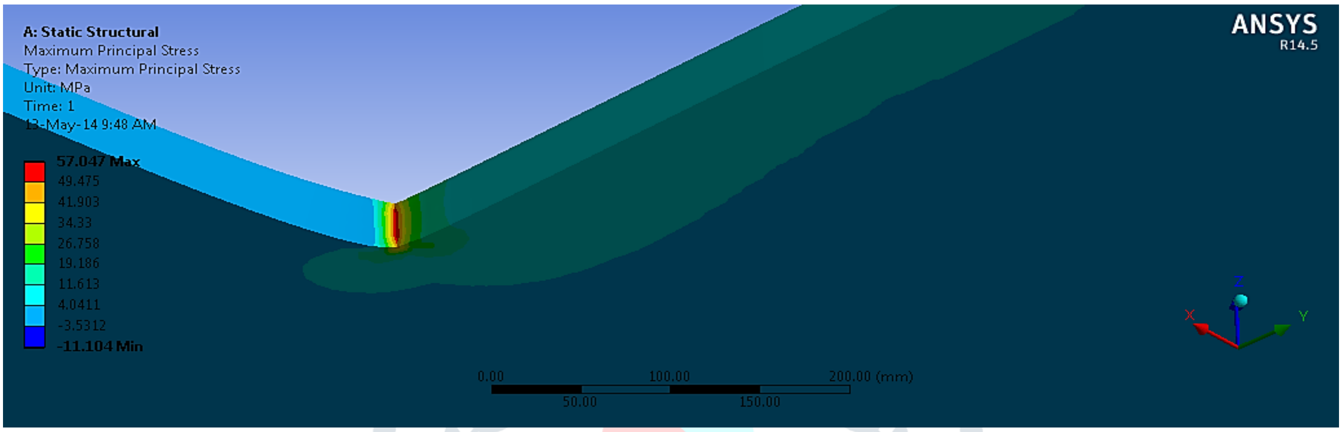


Fig.8 Maximum stress generated

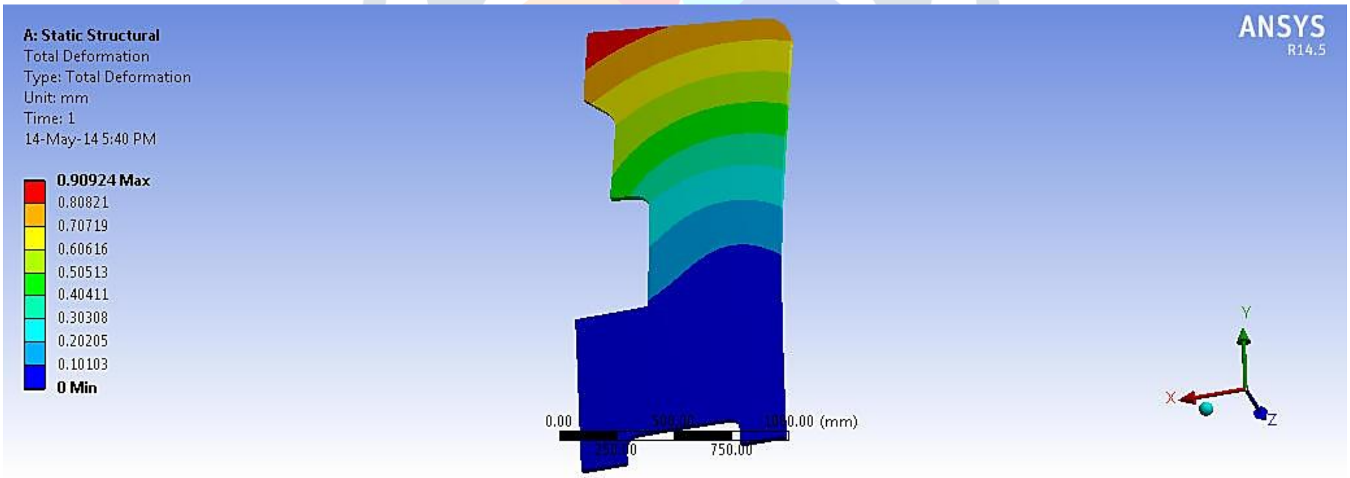


Fig.9 Deflection of frame

VI. OPTIMIZATION

The FE method is a well-known tool for structure analysis. Mathematically, it can be seen as a numerical tool to analyze problems governed by partial differential equations describing the behavior of the system. Finite elements can be applied equally well to commercial airplanes, automobiles, and to micro electromechanical devices to determine the deformations and stresses in the structures given the load and boundary conditions. Structural optimization has been categorized into two types in the literature: Topology optimization and Geometry optimization. Topology optimization deals with determining the layout or load distribution path in the design space for specified loading conditions subject to behavioral constraints and in some cases geometric constraints. For geometry optimization, the topology of structure is predefined. Thus, the result of geometry optimization is highly dependent on the initial topology. Geometry optimization deals with optimizing the shape and size of the layout obtained from topology optimization. Finite element-based optimization assumes that a discretized FE model exists; whereas, the classical structural optimization approaches directly work on the differential equations governing equilibrium. FE-based optimization is embodied in a computer program for the optimization of mechanical structures. The program typically consists of an analysis module and an optimization module. The analysis module is used to analyze the structural responses (e.g., deflections and stresses) and to perform sensitivity analysis. Sensitivity analysis calculates the change in the structural response for a small change in the design. Based on the sensitivity analysis, the optimization module calculates a change in the structural design that improves the structural response and updates the design. Typically, the optimal design is not achieved after only one optimization step. Instead, the procedure, consisting of the FE analysis, the sensitivity analysis, and updating response, is repeated several times. After a number of iterations, the design cannot be further improved and an optimal structure has been reached .A flowchart of finite element-based optimization is shown in Figure 10. First, the design variables for the optimization problem are initialized. Second, FE-based optimization tools use finite elements to analyze the structural response followed by the next step, sensitivity analysis for the gradient information. Next, the design variables are updated based on the sensitivity analysis information. The above process is iterated until a convergence criterion is met. After few iterations, the optimization problem converges and the results of the optimization are checked in post-processing stage.

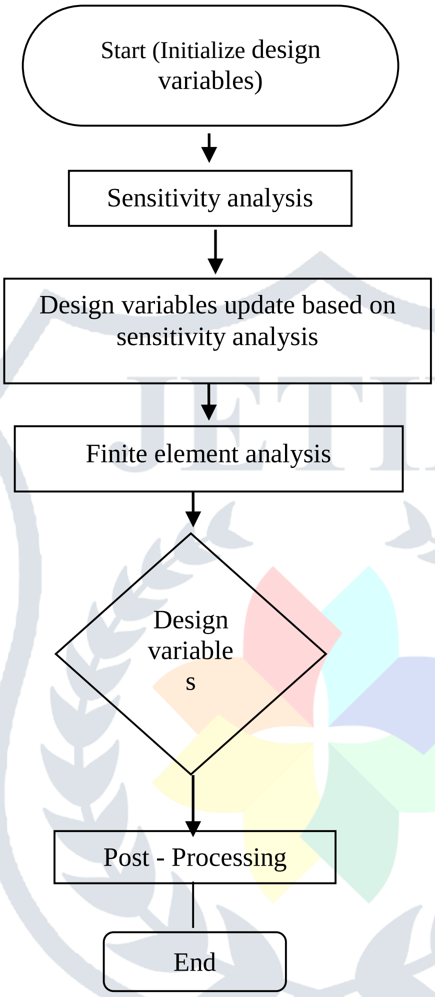


Fig.10 FEA Based Optimization

The calculation of the design sensitivity analysis can be performed by two different approaches: the variational approach and discretized approach. The variational approach applies the variational principle to differentiate the governing differential equations before they are discretized, and then solves the resulting equations by a structural analysis program. The discretized approach uses finite element models directly. The sensitivity calculations are performed by determining the derivatives of a given function. The discretized approach for sensitivity analysis has three different computational schemes: the analytical method, the semi-analytical method, and the finite differencing approximation.

Types of optimization

There are mainly three types of optimization.

1. Size Optimization

Size optimization is the simplest structural optimization that can be performed because the number of design variables are less in comparison to shape and topology optimization. Size optimization relates to determining the wall thicknesses, dimensions of the cross sections, etc.

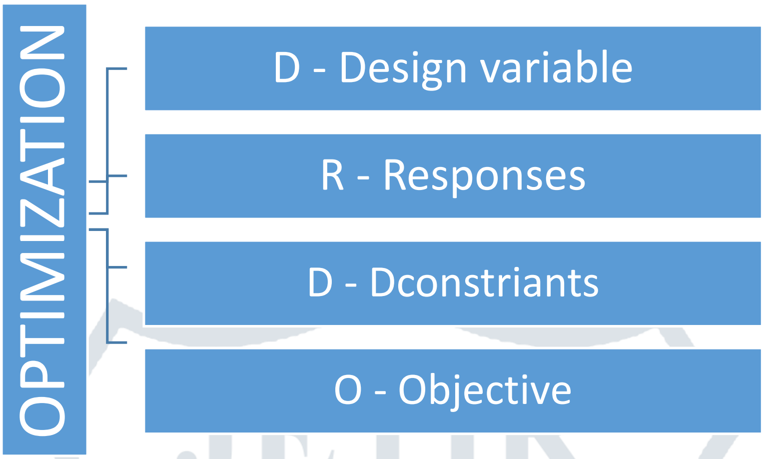
2. Shape Optimization

In this method, the x, y, z-coordinates of the nodes or grid points in the finite element model are changed iteratively. In other words, the coordinates of the nodes on the surface, which needs to be shape-optimized, are regarded as design variables, which will be modified during the optimization. Mesh parameterization is the way the x, y, z coordinates of the nodes in the FE model are related to the design variables. Special care must be taken to avoid element distortion during optimization. This type of shape optimization usually leads to a large number of design variables, which might cause considerable mathematical difficulties. Shape optimization is very effective, when p-adaptive finite elements are used. Typically, a coarser mesh of the p-elements can be used to replace the usual h-elements. Mesh distortion is no longer a problem with p-elements because they allow large changes in shape to occur.

3. Topology Optimization

Topology optimization is often referred to as layout optimization in literature. It is the most difficult form of structural optimization. Topology optimization is most useful during the early design stages. If used early in the design stages, topology optimization has a great impact on strength, reliability, durability, and maintainability of structures and machines. Moreover, usual sizing and shape optimization cannot change the structural topology during the solution process. Therefore, topology optimization is, therefore, most valuable as a preprocessing tools for sizing and shape optimization. The topology optimization of a structure can be classified into two classes based on the distribution of material: the material or micro-approaches and the geometrical or macro-approaches.

Basic procedure of topology optimization



Topology optimization

After the analysis, the C-frame is imported in Hypermesh for its topology optimization. The file is modified into .fem format and loaded into Optistruct user profile. It filters the material properties and loading conditions of Optistruct template. Again analysis is done by defining the material, properties, load and load steps in different collectors. The analysis is run and the result files are stored in the specified directory in .fem format. Now for optimization the basic procedure is followed as shown in figure 8.2 and the optistruct solver solves it to get the topology optimization files in .h3d and .s1 format. The .h3d file stores the modified shape and .s1 file stores the load steps information. It shows the result of the desired objective before and after optimization.

Mesh generation & boundary condition

The c-frame is meshed by using the *Quad* elements. The elements generated are solid 3D elements. The base of the frame is kept fixed (refer figure 8.3 (c)) and load is applied on the c-frame as shown in figure 8.3 (a) and (b). As both the frames are identical one single frame is taken for analysis and optimization. The value of load taken is equal to the capacity of the press frame, i.e. 40 TON which equals 392400 N.

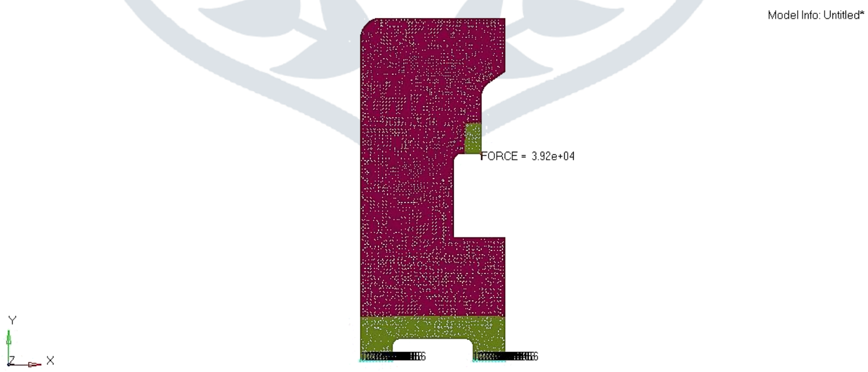


Fig.11 Meshed Model of C-Frame for Optimization

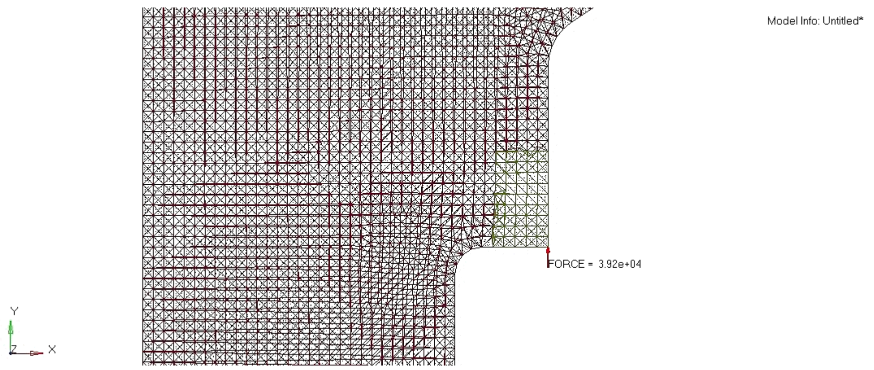


Fig.12 Force Applied on the Nodes

Fig.13 Boundary Condition – Frame Base Fixed

Results – element stresses

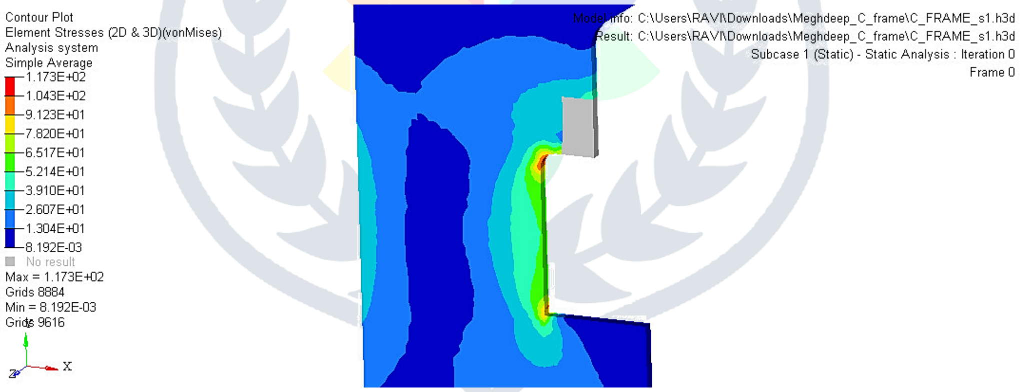


Fig.14 (a) Stresses at Iteration 0

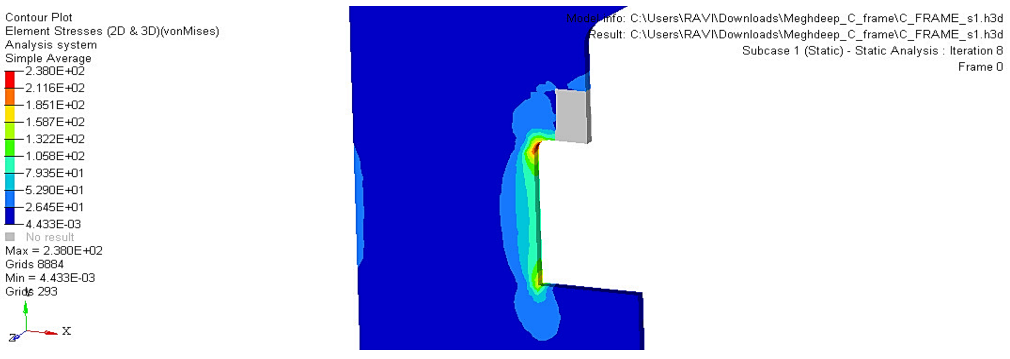


Fig.14 (b) Stresses at Iteration 8

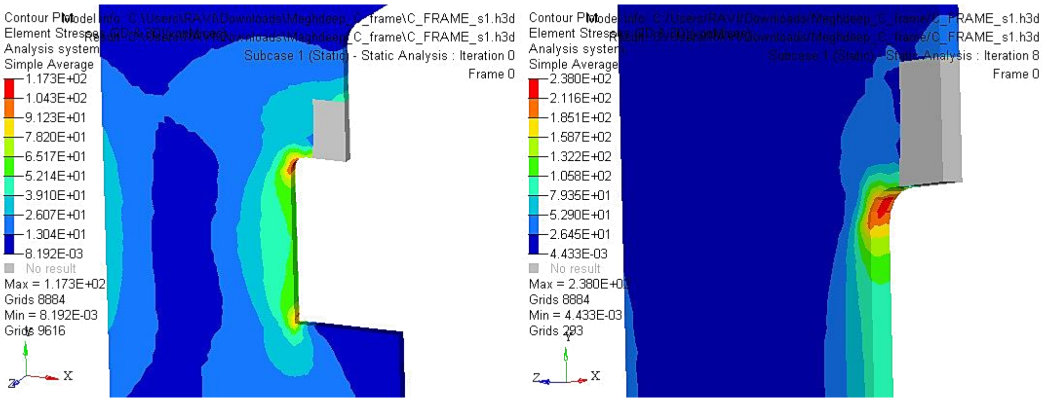


Fig.14(c) Comparison of Stresses at Iteration 0 and 8

The values of stresses obtained before and after optimization are shown in figure 14 (c). Maximum value of stress is 1.179E+02 MPa in iteration 0 and the value of stress in iteration 8 is 2.990E+02 MPa.

Results-Displacements

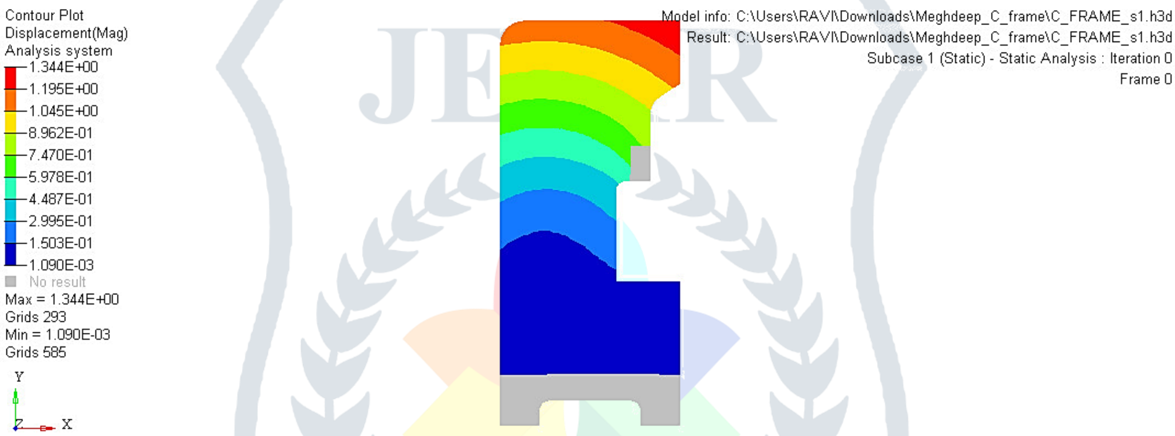


Fig.15 (a) Displacement at Iteration 0

Figure 15 (a) shows the maximum displacement value of the frame to be 1.344 mm.

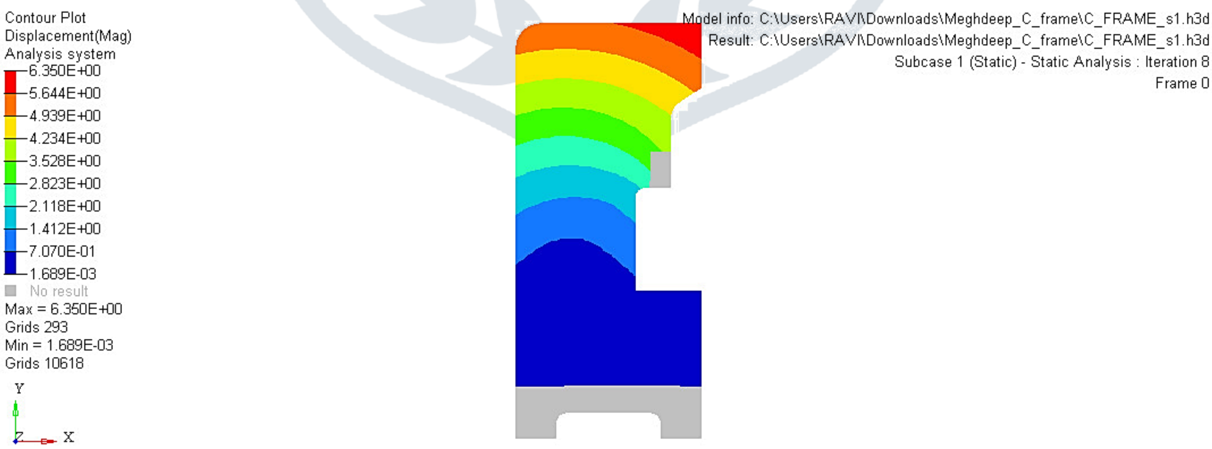


Fig.15 (b) Displacement at Iteration 8

Figure 15 (b) shows the maximum displacement value of the frame to be 6.350 mm. The difference in the values is due to the reduction in the thickness of the frame which is obtained after optimization.

Results - element densities

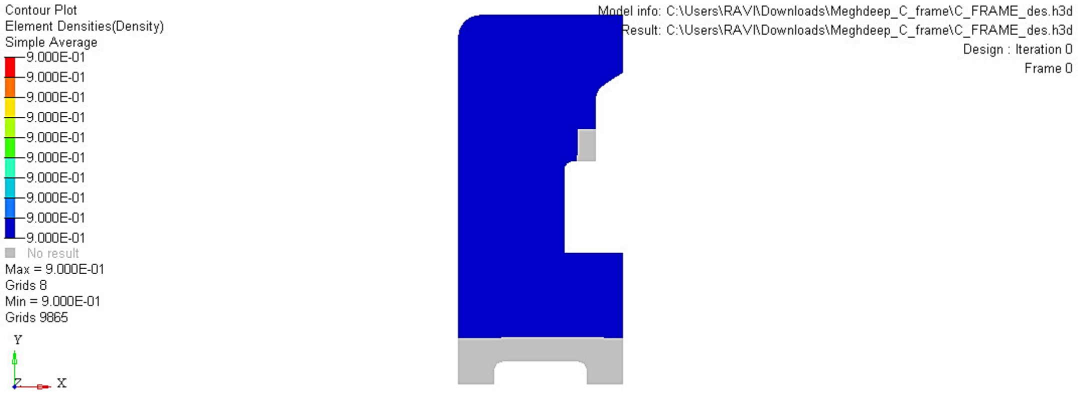


Fig.16 (a) Density at Iteration 0

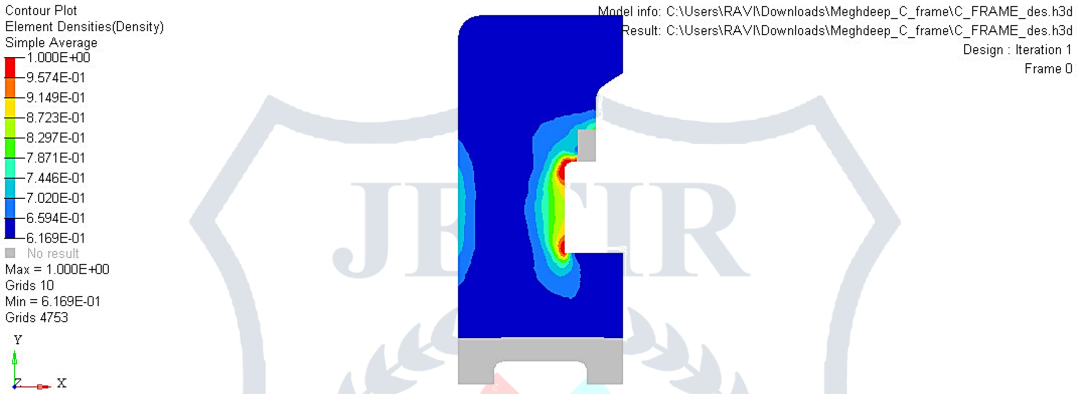


Fig.16 (b) Density at Iteration 1

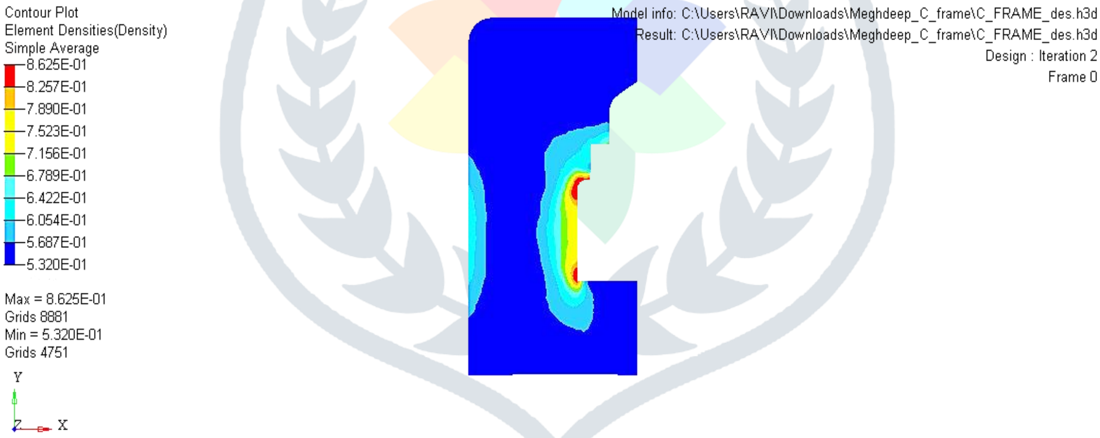


Fig.16 (c) Density at Iteration 2

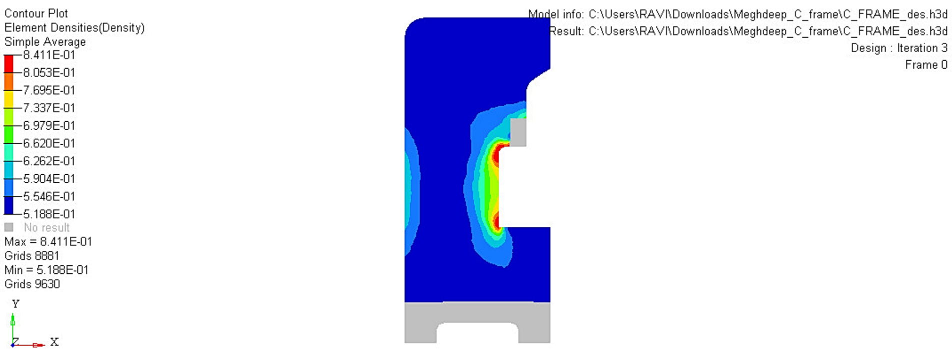


Fig.16 (d) Density at Iteration 3

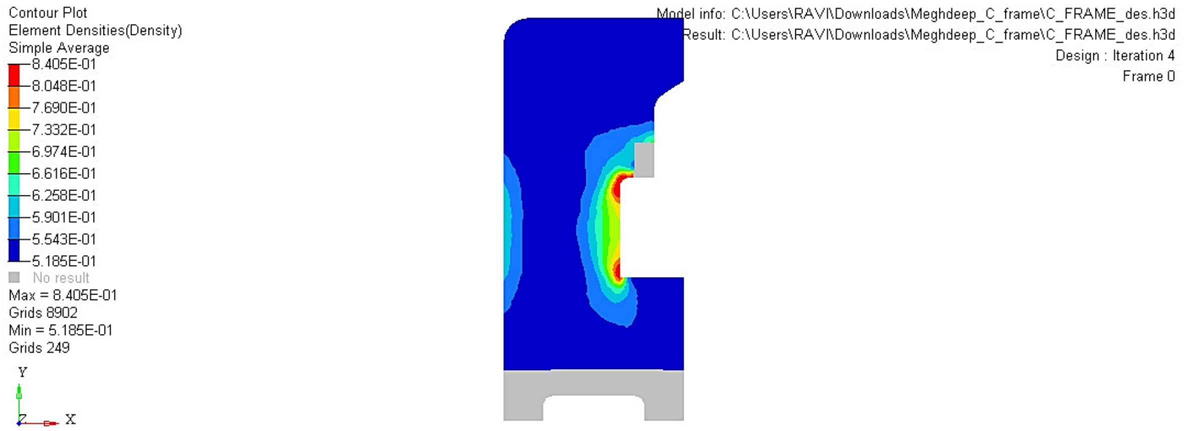


Fig.16 (e) Density at Iteration 4

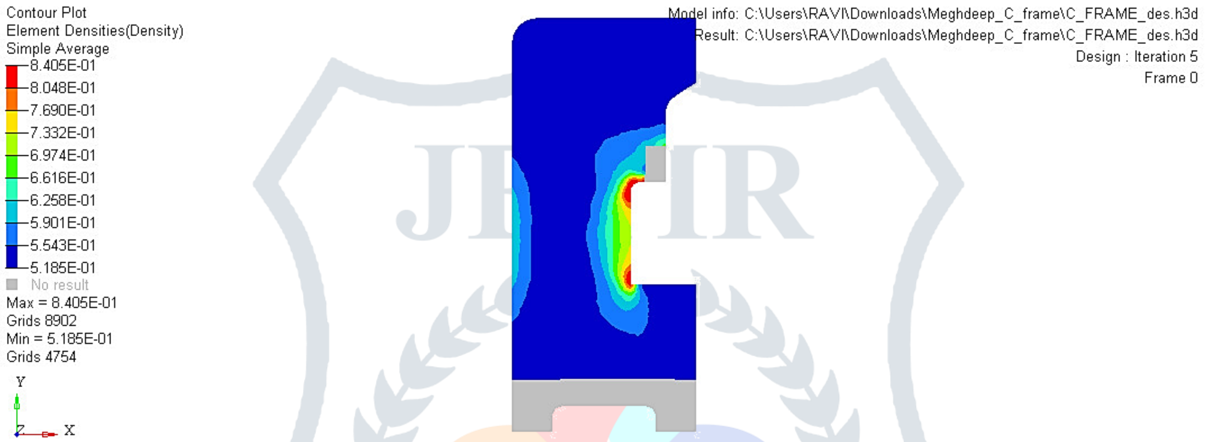


Fig.16 (f) Density at Iteration 5

Figure 16 (a) to (f) shows the number of iterations carried out in reduction of the element densities. With minimum member size control the elements are reduced to achieve the optimum thickness of the frame.

VII. CONCLUSION

Main findings of Research work

This dissertation work was carried out to design and analyze the C-frame of 40 Ton pneumatic power press using FEA tools. Design of the frame was made in Creo-2.0 and analyzed using Ansys 14.5. At later stage Topology Optimization was also carried out with objective of minimization of thickness. Table 9.1 shows the values of stresses and displacement for both analytical and simulation results.

Simulation Results				
Sr. No.	C-Frame Iterations	Element Stresses (MPa)	Maximum Displacement (mm)	Element Densities
1	0	117.3	1.34	0.9
2	8	238.0	6.35	0.84
Analytical Results				
1	N/A	153.09	0.91	N/A

The major conclusions of the work is summarized as follows:

- ✓ Analytical and simulation results show that the stress values lies well within the permissible limits, so the design of C-frame is safe and there was a further scope of optimizing the existing design.
- ✓ The outcome of the optimization analysis gives an indication that the existing thickness can be reduced up to 20% without compromising on functionality.
- ✓ Topology Optimization provides a way of minimizing or maximizing our objective function depending upon the design variable parameters, responses and constraints.

VIII. SCOPE OF FUTURE WORK

- ✓ Topography and Shape optimization can be done which can give more strength and modified shape to the existing frame and also there can be further reduction in frame thickness

✓ Same procedure can also be adopted for optimization of other types of presses.

IX. REFERENCES

Papers

[01] D. Ravi-''Computer Aided Design and Analysis of Power Press'' Middle-East Journal of Scientific Research 20 (10): 1239-1246, 2014 ISSN 1990-9233© IDOSI Publications, 2014
[02] Romeo CIOARA & Ioan DAN ''Increased Rigidity Solutions for C-Frame Mechanical Presses''- July 2011
[03] Sandip Godse-''Static Load Analysis of Tata Ace Ex Chassis and Stress Optimization Using Reinforcement Technique''-2013
[04] Frantisek Trebuna "Analysis of crack initiation in the press frame and innovation of the frame to ensure its further operation"2011, vol. 45, pg.244-255
[05] A. G. Naik, N. K. Mandavgade "FEA Implementation in Analysis and Optimization of Top and Bottom Frame For Hydraulic Cotton Lint Bailing Press" ISSN 2229-5518 Volume 3, Issue 7, July-2012
[06] P. D.Murarka "Computer- Aided Design of Hydraulic Press Structures"1988, Vol. 10, No. 9, pg. 637-645.
[07] V. Amaranth- "Structural Optimization of 5 Ton Hydraulic Press and Scrap Baling Press for Cost Reduction by Topology" Vol. 1, No. 3, August 2011.
[08] Rakesh. A. Oza- "Analysis and Optimization of Drive Shaft in Eccentric Mechanical Press" Vol. 1, Issue 2, pp.055-058
[09] M. Fulland- "Analysis of fatigue crack propagation in the frame of a hydraulic press" Engineering Fracture Mechanics 75 (2008) 892–900
[10] N.K.Mandavgade –"Review on Analysis and Optimization of Hydraulic Cotton Lint Bailing Press"-2012
[11] Mihai TALU, Stefan TALU- "Static Analysis of a Press Ram-Linear Hydraulic Motor piston Assembly from Horizontal Hydraulic Press – 2 Mn"-2010
[12] K. Chodnikiewicz -"The Calibration of Metal-Forming Presses"-2000
[13] Y.Y.Lin-Che -"A Software Tool Development for Pneumatic Actuator System Simulation and Design"- 2003.
[14] Catalin Iancu -"Comparison between Analytical Calculus and Fem for a Mechanical Press Bed"- 2013.
[15] Chen Yan hong & Zhu Feng –"The Finite Element Analysis and the Optimization Design of the Yj3128-Type Dump Truck's Sub-Frames Based on Ansys"-2011.
[16] B.Parthiban, P.Eazhumali – "Design and Analysis of C Type Hydraulic Press Structure and Cylinder"- March-2014

Books

[01] Ibrahim Zeid and R. Siva Subramanian, "CAD/CAM Theory and Practice", By, McGraw Hill, 2010
[02] A. Cowley and S.Hinduja, "The finite element method for machine tool structural analysis." Ann. CIRP 9, 1971, pp. 171.
[03] Westermann Tables for metal trade revised IS SKPI: 1976, New Age international (P) Limited, Publisher -1996, pg-5
[04] V.B. Bhandari, "Design of Machine Element" by, Tata McGraw Hill Company Limited, New Delhi, pg-117
[05] K. K. Pujara & B.L. Juneja "A Course in Machine Design" by Dhanpat Rai & Sons Publication, New Delhi. pg. 557-585.
[06] Machine Design Exercises by S.N. Trikha, Khanna Publication, July 1997 edition, pg. 226-241.
[07] David Alkire Smith "Mechanical Press Types and Nomenclature" by, © 1993-2005 C02.doc C02.PDF, 530 Hollywood Drive, Monroe, Michigan 48162-2943, August 19, 2005

Web site

[1] Tarakpowerpress ''about, cframe, pnepress''
www.tarakpowerpress.com/cframe.htm
www.tarakpowerpress.com/aboutus.htm
www.tarakpowerpress.com/pnepress.htm
[2] Airtorq ''pneumatic-industrial-combination-clutch-brakes''
www.airtorq.com/pneumatic-industrial combination-clutch-brakes.html
www.airtorq.com/download.html
[3] Fancort industries, "tonnage calculation"
www.fncort.com/Documents/tonnage_calc.html