International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878 (Online), Volume-8 Issue-2, July 2019

Design and Analysis of Structural Elements of H-Frame Press

Dani Prafullkrishna Kiran, S.Sivarajan

Abstract*: *Automobile manufacturers nowadays release new models or new variants of their vehicles in a very short span of time. As a result, for every new product, the manufacturing equipment has to be modified or updated to suit the production requirements. This leads to development of new customized manufacturing equipment for production. Thus along with the new variants, there is an increase in the fixed cost related to production due to investments in the development of the machines. To reduce fixed cost involved due to development and fabrication of the machine, it is necessary to minimize the material utilized during fabrication of the machine. Hence it is important not to overdesign the equipment with respect to strength and deformation. General practices include procedures like modifications in design of the previously used equipment to suit the new equipment followed by fabrication of the new equipment. This can lead to increase in material consumption and thus overdesign of the equipment with respect to the same. This study deals with the design and analysis of the structural elements of a press. This press is used to assemble two thrust bearing races in the motorcycles steering tubes connected to the frame of the motorcycle. The design of the machine is customized to withstand loads for the specific application. The approach of the design process is concurrent instead of iterative design. The process tackles each structural element in a way to determine the best suitable dimensions to maximize strength and minimize material, thereby optimizing the design

Index Terms*: *About four key words or phrases in alphabetical order, separated by commas.

I. INTRODUCTION

Presses are used in various industrial processes during mass production. These presses can be of various types. The common ones are Mechanical Press, Pneumatic Press and Hydraulic Press. Out of these three, the most commonly used is the hydraulic press. Hydraulic presses 2 are known for their efficient design and dependability. The other major reason which has made hydraulic presses popular is the amount of force that can be generated by them which cannot be easily matched by the other two types. 1.2.2. Types of Frames The two types of common frames are C Frame and H Frame [1]. A C type frame basically behaves as a cantilever beam. Whereas on the other hand, an H type frame as an additional member which makes the structure stiffer thereby leading to smaller deflections s as compared to C type frame. However, C type frames generally take up less space as compared to the H type frame. When compared to H type frame, C type frames have much higher accessibility and hence are more preferred in prototyping shops where the machine needs to accommodate different types of parts and components of various sizes and dimensions. However, for mass production, H type frames can be more suitable due to the uniformity on components. For the application addressed in the project, the H Type frame has been selected for its suitability due to the above mentioned reasons, The trend in providing product variety of products is increasing. On one hand, increasing product variety leads to differentiation in market place, however on the other it also leads to increase in product development, process development and production cost. Hence it is necessary to focus on maximum possible variety offered as well as the cost it adds to the development of the variants [1]. In this paper, efforts have been taken on one segment of the latter point. A case of motorcycle steering bearing assembly process has been analyzed. The project involves with design of structural elements of H-type Bearing Pressing machine of the above mentioned applications. Direct and quantified cost analysis has not been carried out due to the very high number of variable parameters involved in the costing model such as location, material grades, local vendors and supplier chains, which do not lie in the scope of the project. The project deals with primary variable such as amount of material required, number of parts involved in assembly etc. which will definitely reduce the cost of the product irrespective of the above-mentioned variables in cost. Detailed analysis on the structural elements of the pressing machine has been carried out by solving for analytical solution for the simplified geometries and correlating the acquired result for the actual geometry through finite elements methods. Analysis of the assembly elements has also been carried out as they form a very crucial part of the design topology and can have considerable effect on development cost of the product. The focus of this paper is to design and analyze the structural elements such as frame [2], stand and assembly elements such as bolted and welded joints of an H frame press for stresses and deformations using analytical and finite elements methods.

II. METHODOLOGY

The paper aims at designing a press machine for the bearing fitting processes carried out to assemble the bearing races in the steering column or tube by interference fit. Such machines are often manufactured newly for new products or different variants of the same product.

Revised Manuscript Received on 30 July 2019.

* Correspondence Author

Dani Prafullkrishna Kiran*, School of Mechanical & Building Sciences, Vellore Institute of Technology, Chennai, India.

S. Sivarajan, School of Mechanical & Building Sciences, Vellore Institute of Technology, Chennai, India.

As a result, they contribute to a portion of the fixed cost in production of the equipment.

Thus, higher the cost to develop and to manufacture the equipment leads to delay in achieving break even. As the equipment does not form a very critical component in the product development phase, general practices lead to over design of the equipment which involves increased material consumption and thus leads to increase in cost. Other than that, increased material consumption leading to over design means inefficient design due to wastage of raw material. Therefore it also has a small impact on environment as well. The goal of this study therefore involves designing and analyzing the structural elements for the required components. While doing so, it also involves in developing a procedure which can help in design similar products or machines ensuring minimum material consumption. It can therefore be called as a lean designing procedure. Figure 1 show the methodology followed in design analysis. The materials for tube and bearing races and their specification and properties are taken from [3]. The tolerances were selected for the press fit according to ISO standards [4].All the analytical calculations are carried with reference to [5],[6].

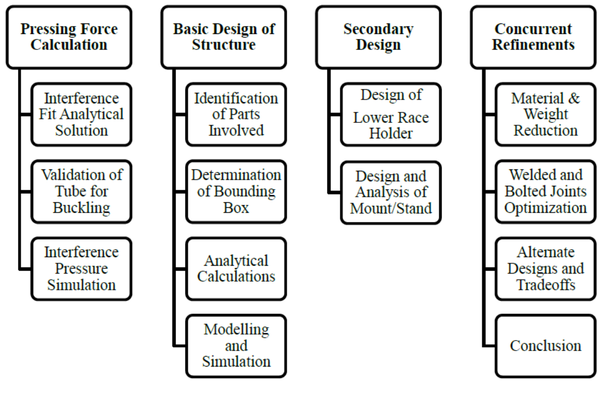


Figure 1 Methodology

III. FINITE ELEMENT ANALYSIS OF STRUCTURAL ELEMENTS OF H FRAME PRESS

Simulation of Interference fit

Analysis of Simplified Geometry

The first analysis was performed on a simplified geometry to validate the boundary conditions [7]. A parametric model was prepared with varying diameters of the bearing race and tube. A half cut model was created to reduce calculation time with a frictionless support at the faces of symmetry. The half cut model also helps to generate a swept mesh of high quality. An interference offset was defined at the contact of the two surfaces (viz. bearing outer surface and tube inner surface) of 0.035mm (=0.070/2). Analysis of Actual Geometry – Upper Assembly A quarter-segmented model was generated for the actual geometry. Similar to the previous case, frictionless supports were provided at the faces of symmetry. The segmented model helps to generate a swept mesh in a highly controlled manner. Figure 3.shows how the two meshed bodies are meshed conformably. Thus meshes quality and simulation results are reliable.

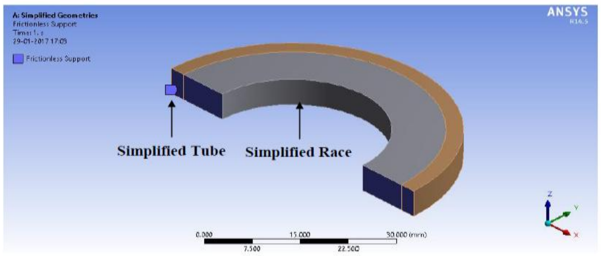


Figure 2. Half-cut parametric model with boundary conditions

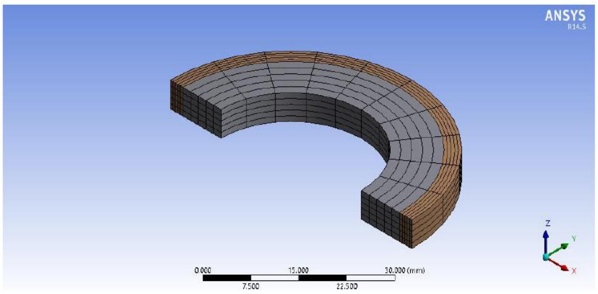


Figure 3. Conformal swept mesh of hexahedral elements

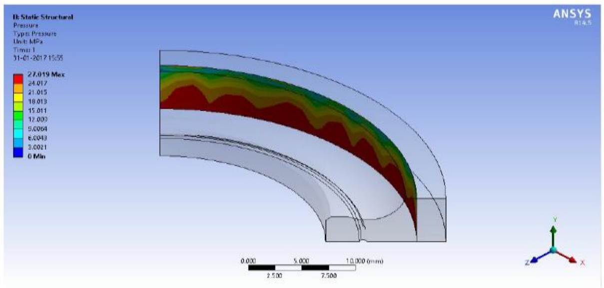


Figure 4. Results for Upper Assembly Contact Pressure - Max = 26.125MPa

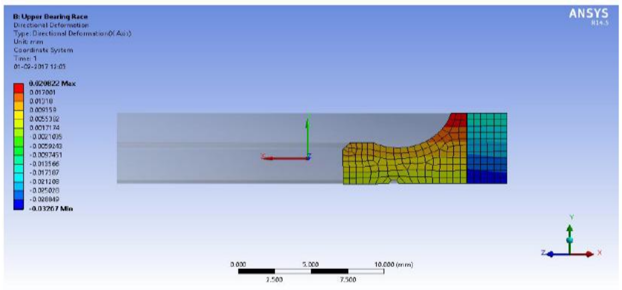


Figure 5. Results for Upper Assembly Deformation

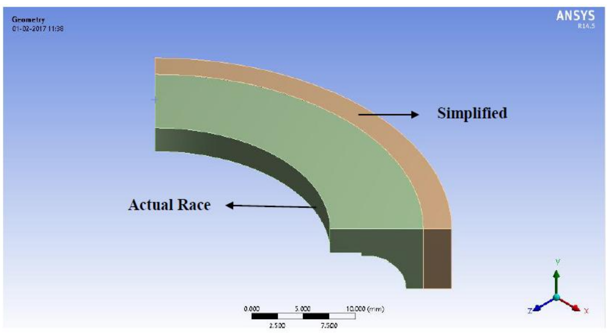


Figure6. Quarter Segment Model of Lower Assembly with Boundary Condition

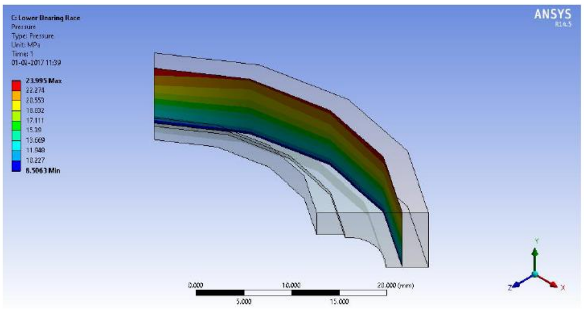


Figure 7. Results for Lower Assembly Contact Pressure - Max = 23.905 MPa

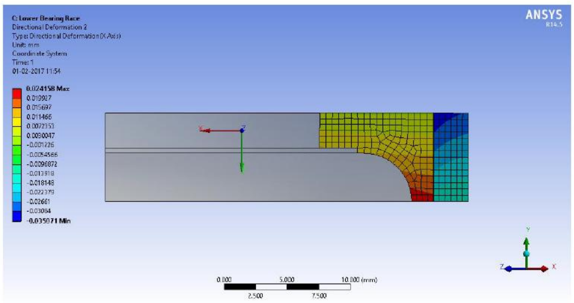


Figure 8. Results for Lower Assembly Deformation

Table 1. Analytical result summary for interference fit

	Analytical Simplified	Simulation Simplified	Simulation Actual
Upper Assembly Contact Pressure (MPa)	26.110	26.125	27.019
Lower Assembly Contact (Pressure (MPa)	22.587	22.601	23.905

The finite element analysis results of upper and lower assembly are presented in figure 4, 5, 7, 8.Table .1 tabulates the contact pressure comparison of upper and lower assembly analytically and by simulation

Table 2. Pressing force requirement

Upper Assemby Pressing Force (F_1) (tonnes)	0.3311	**Total**	**Design**
Lower Assembly Pressing Force (F_2) (tonnes)	0.3949	**0.7260**	**7500 N**

Column Frame Analytical Calculations

After careful consideration, it can be seen that the column frame is subjected to eccentric tension. For calculating the analytical solution, a single columnar frame member was considered as a cantilever being subjected to half of the total reaction load. Also, it would be subjected to dead load of the weight of the upper frame and cylinder on it. This member is as well subjected to cyclic loading condition. To minimize variety in procured material, the same cross section as for the upper frame was considered for the first design iteration. Hence, the dimensions of the cross section for the first iteration were considered as 200mm x 16mm.

The following equations along with the previous equations were solved to calculate the required values of the beam [5].

$$Bending\ Moment\ for\ loaded\ condition\ (M) = \frac{P}{2} \times \frac{L}{2}\ (for\ fixed\ ends)$$

$$Dead\ weight\ (P') = \ Weight\ of\ Frame + Weight\ of\ Cylinder$$

Factor of Safety (n) according to Goodman Equation

$$\frac{1}{n} = \frac{\sigma_{mean}}{\sigma_{tenslie}} + \frac{\sigma_{amp}}{\sigma_{endurane}}$$

n	2.05
W	5.4931 kg

The system of the column frame is complex due to the upper frame connecting the two members. This leads to significant increase in stiffness of the system. Hence the values for deformation and stress have been calculated using finite element analysis alone

Finite Element Analysis of Frame Structure

A 3D model was generated in Ansys Design Modeller according to the dimensions obtained from the analytical solution. The model was segmented into various blocks to achieve a highly structured hexahedral mesh of high resolution. The boundary conditions were defined according to the practical scenario and not according to the analytical approach. For the first iteration, a simple bounding box and dimensions for the press frame were determined graphically from the geometry of the tube. It was made sure to enclose the entire tube within the frame. For the total height, of the frame, considerations were made to accommodate the bearing race holder/post and easy movement of the tube during operations (viz. loading in and unloading from the machine). These dimensions only form the bounding box of the frame geometry. Actual dimensions would be calculated after analysis of each member for the loads acting on them and their boundary condition. The frame was modelled to consist of 4 elements. These include the bottom plate or the bolster plate, two columnar frames and an upper frame on which the hydraulic cylinder mount will be attached. Free body diagram for each element was drawn to observe the type of load acting on each member of the frame. Figure 9 shows the 3D model of the basic frame.

All members were studied for strength and displacement analytically and will be modelled and studied by Finite Elements Methods A 3D model was generated in Ansys Design Modeller according to the dimensions obtained from the analytical solution. The model was segmented into various blocks to achieve a highly structured hexahedral mesh of high resolution. The boundary conditions were defined according to the practical scenario and not according to the analytical approach.

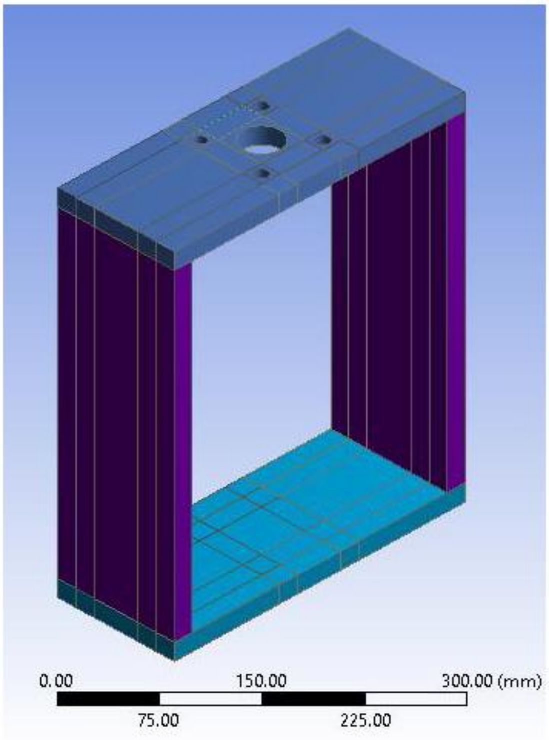


Figure 9. 3D Modelled Geometry of Frame

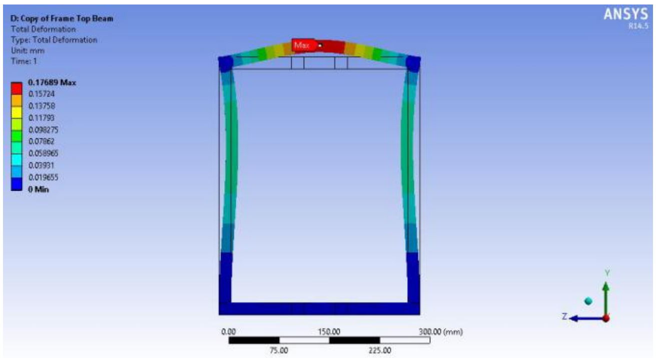


Figure 10. Total Deformation of Frame – Max 0.17689mm

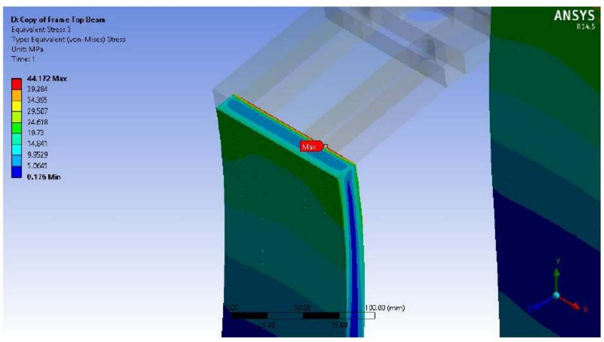


Figure 11. Stress Induced at Joint - Max 45 MPa

Stresses induced in the FEA Model(figure 11.) appear to be lower at the joint however seem to be higher at the holes provided for the cylinder actuator. However the values do not exceed the maximum permissible value of the material (Hot Rolled Steel) The variation in values of maximum stress is due to the assumption of the upper beam being completely fixed. However, the stiffness of the actual system is less than the system assumed in the analytical solution. This also leads to variation in the values of deformation. Also, the presence of holes in the upper frame for fastening the hydraulic cylinder as well as the center hole for the actuator leads to higher values of deformation in the finite elements model(Figure 10.).

Table 3. Stress induced in Frame

	Analytical Results	Simulation Results	% Deviation
Maximum Induced Stress (MPa)	54.913	44.172	19.7
Maximum Deformation (mm)	0.12874	0.17689	27.2

The comparison of stress induced in the frame from analytical and simulation and the deviation are provided in Table 3.

Stand Design

A conventional structure of stand was selected for the first iteration. This was done to ensure minimum material consumption. The structure consists of the 4 horizontal members and 4 legs or vertical member. Addition of reinforcing elements was avoided in the first step to ensure minimum material consumption. IS Standard was followed for selecting cross section of the frame members. All members were designed for same cross sections dimensions to simplify material procurement. IS 4923 was adopted for square tube section. Initially a parametric geometry of the frame was modelled in Ansys Design Modeller (figure 12.). For initial approximation and identification of range of stresses, a 3D model was created using beam elements. Using beam elements instead of 3D elements helps in good solution initialization and reduces computation time of the solution. The cross section dimensions and thickness were defined as parameters. Overall dimensions for the frame were calculated from the main press structure dimensions. The above mentioned values were selected as defined variables in the parametric model. External force of 7500 N was defined using remote force feature. A remote geometry point was created for the 4 horizontal members at the center of the geometry. The four legs of the stand were defined with fixed boundary condition at the lower ends.

The values of deformation, Maximum combined stress and mass of the stand for different cross sections are taken according to IS 4923 Standard. From this analysis the dimensions of the square tube were selected to be 25 x 25 mm of thickness 2mm as it provides a sufficient factor of safety of 3.39. Also it requires minimum material requirement of 3.813 kg due to the least cross sectional area of the geometry.FEA results are given in figure 13 and 14.

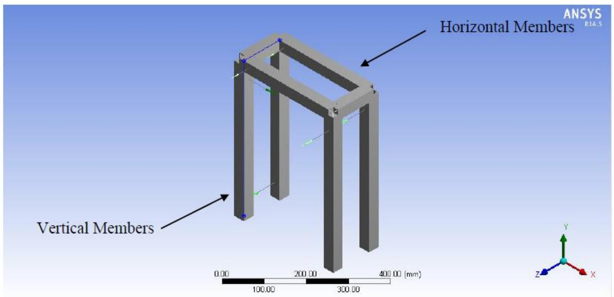


Figure 28 Basic Model of Stand

Figure 12. Basic Model of Stand

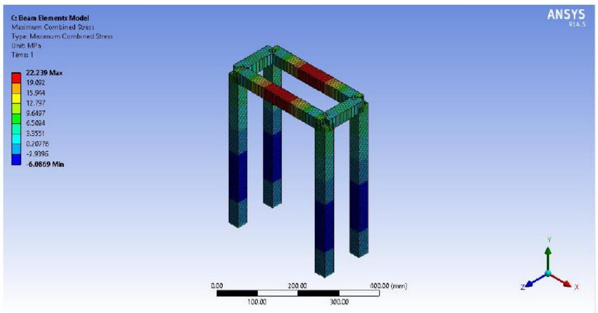


Figure 13. Maximum Combined Stress = 22.239 MPa

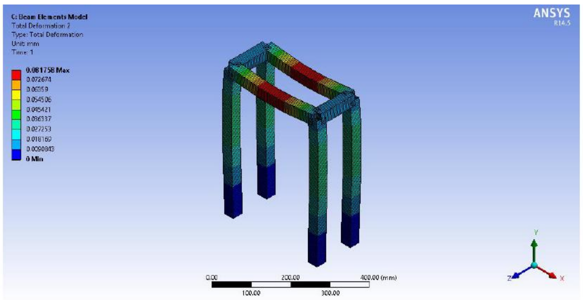


Figure 31 Total Deformation at 180x Scale

Figure 14. Total Deformation at 180x Scale

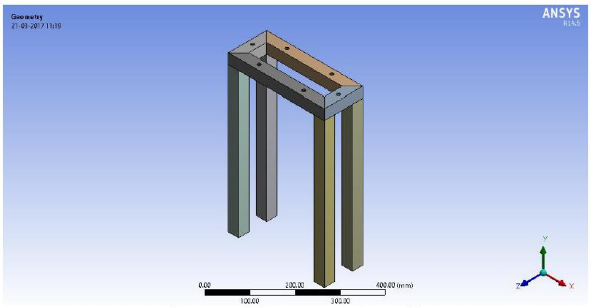


Figure 15. Detailed 3D weld Geometry (Volume Body

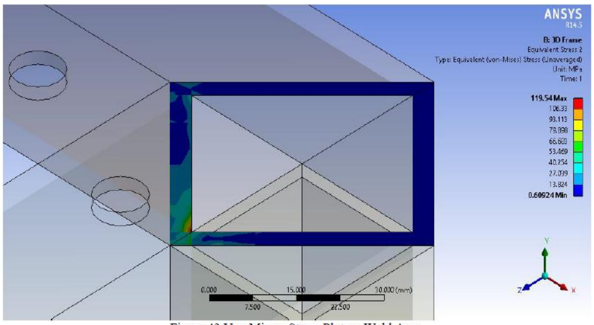


Figure 42 Von Misses Stress Plot on Weld Area

Figure 16. Von Misses Stress Plot on Weld Area

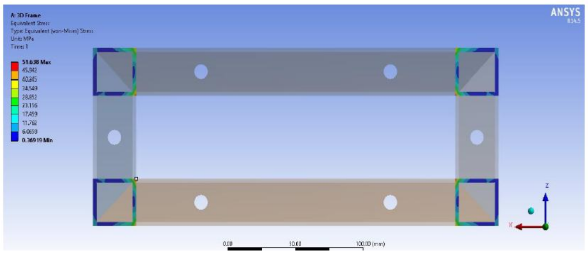
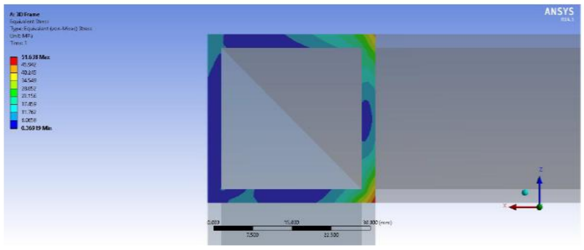


Figure 17. FEA Results for Vertical to Horizontal Beam Welded Joint



Figue 18.FEA Results for Vertical to Horizontal Beam Welded Joint

Design of Weld joints of stand

Welded joints of the stand were grouped into two categories viz. horizontal member to horizontal member welds and horizontal member to vertical member welds. The main differences between the two joints are the geometries and the type of loading each one is subjected to. Horizontal members were designed for welds at 45 degrees for maximum weld length.

Finite Elements Methods Analysis of Welded Joints

A higher-level 3D model (Figure 15.) of volume elements was created according to the new dimensions from the Excel programme. Welded surfaces were defined using contact regions and were set to bounded contacts. The entire stand geometry was meshed using Multi-zone and Swept mesh controls. Hence a high quality controlled mesh was created which would provide accurate results. The maximum value of stress induced in the welded region can be noted to be 119.54 MPa. Thus there is good co-relation between the finite elements solution and the analytical solution (114.68 MPa). The maximum value of stress induced in the welded region can be noted to be 55.961 MPa.

Thus there is good co-relation between the finite elements solution and the analytical solution (51.638 MPa).The FEA results are shown in Figure 16, 17, 18 and comparison with analytical solution is shown in table 4.

Table 4. Result summary for welded joints of stand

	Analytical Solution	Simulation Solution
Maximum Stress in H-H Weld Joint (MPa)	114.68	119.54
Maximum Stress in H-V Weld Joint (MPa)	51.638	55.961

The design can be modified according to the requirements of the function and adjusted according to the part. Following are a few modifications suggested (figure 19):
The stand of the press can be made stiffer by adding additional support members to minimize deformation of the part. Rubber padding can be used to increase the grip of the machine with the floor in case bolting the machine is not an option. In case the parts center of gravity is at a considerable height and the machine is susceptible to instability, additional members can be attached to the bottom of the stand legs to prevent roll of the machine by increasing the distance of the point of rotation. All these modifications lead to increase in material consumption. However, for proper functioning of the machine and safety of the operator, it is recommended that these modifications are made, if necessary.

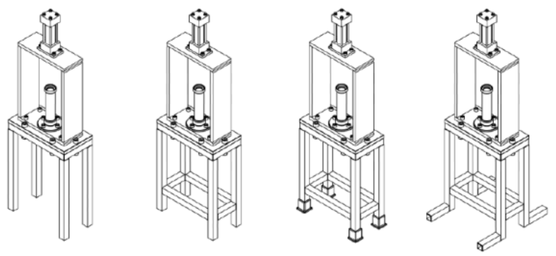


Figure 19. Design Alternatives Suggested

IV. CONCLUSION

The structural elements of an H Frame Press, viz. press frame, mounting stand and assembly elements such as welded joints for different parts of the machine were designed from clean sheet for the required components. The results from the analytical solution and from simulation showed good co-relation especially for values of interference fit force and welded joint stress . Results for part such as frame member showed some amount of deviation (27%) for deformation values due to the difference in the actual and analytically simplified boundary conditions. It was found from the results of stress values for the stand that although the member with minimum area (25mm x 25mm - 2mm thickness) and thus minimum mass did not fail under loading, the welded joints did fail due to the small area of cross section. Hence the stand members were re-designed with larger cross section dimensions (32mm x 32mm - 2.6mm thickness) so that the area of the welded joint would increase thereby reducing the stress values. Instead of following the traditional approach of refining previous models or the following iterative method, a concurrent and non-iterative approach was selected in the project. This method makes the best use of simple equations from theoretical background, to generate results for parametric equations of solid mechanics. Further work can be carried out by developing a program which only requires the input of the part dimensions in the assembly (bearing races and tube) and generates the machine dimension details as output.

REFERENCES

1. L. F. Scavarda, J. Schaffer, H. Schleich, A. d. C. Reis and T. C. Fernandes, “Handling Product Variety and its Effects in Automotive Production,” POMS 19th Annual Conference, Vols. 008-0447 , p. 2,
2. I. Suchy, “Chapter 5-2: Parts of a Press,” in Handbook of Die Design, pp. 5-5.
3. V. Agarwal, “Japanese Standard Specifications,” in Steel Handbook - Bible to Steel “
4. C. M. T. Division, “ISO System of Tolerances,” in Machine Tool Design Handbook, Bangalore
5. V . B. Bhandari, Design of Machine Elements, Third Edition ed., Tata McGraw Hill Education Private Limited, 2010
6. R. S. Khurmi and J. K. Gupta, A Textbook of Machine Design (S.I. Units), Eureka Publishing House (PVT.) LTD.,
7. Mechanics Development Group - ANSYS, “Surface - Surface Contact 171 - 174,” in Nonlinear Contact AnalysisTechniques using ANSYS, pp. 21, 75, 88.

AUTHORS PROFILE

Dani Prafullkrishna Kiran obtained his B.Tech degree in Mechanical Engineering from Vellore Institute of Technology, Chennai, India.

S. Sivarajan obtained his Bachelor of Engineering degree In Mechanical Engineering from Coimbatore Institute of Technology, Coimbatore. He obtained his Master degree in Manufacturing Technology from Indian Institute of Technology, Madras, Chennai. He has twenty two years of teaching experience. Currently he is working in Vellore Institute of Technology, Chennai, India. He has published around fifteen reputed journals. He has also written fifteen book chapters published by reputed publishers. His research interests include Sheet metal forming, Metal cutting, Finite element analysis, Surface engineering and Tribology