Journal of Mechanical and Mechanics Engineering
Volume 1 Issue 2

Design and Analysis of Hydraulic Operated Press Brake

R. K. Kanakaraddi, Mahesh H. Halli

Department of Mechanical Engineering, BLDEA's Dr. P. G. Halakatti College of Engineering and Technology, Vijaypur, Karnataka, India

E-mail: rkanakaraddi@gmail.com, mhalli09@gmail.com

Abstract

Hydraulic press brake is a machine tool and it is used for bending the sheet and plate material, most commonly used for sheet metal. The present work includes the simulation (stress analysis) of hydraulic operated press brake subjected to load. The stress analysis is carried out using advanced fem tool ANSYS. The static structural analysis was done for hydraulic operated press brake that analysis may be static or dynamic analysis. A number of analysis techniques like analytical and experimental are available for the structural analysis of hydraulic press brake. Same as number of optimization techniques are available for optimization of it like Taguchi method. Determination of the different structural analysis and optimization through the various methods lie FEA in a hydraulic press brake has been reported in literature.

Keywords: *FEA, ANSYS, structural analysis, hydraulic press brake, solid work*

INTRODUCTION

HYDRAULIC PRESS BRAKE

The hydraulic press brake is also called as brake press. Hydraulic press brake is a machine tool and it is used for bending the sheet and plate material, most commonly used for sheet metal. The one of the most commonly used industrial forming process is linear bending. We can see them in electric home appliances, such as washing machine, ovens and freezers and it is also used in computers frames, wind power mills, ships, trucks and airplanes etc. The main advantage of bending in components is the additional stiffness and rigidity to the parts. The bending operations are carried out most commonly in special long press called press brakes. The hydraulic press brakes are machines generally used to bend sheet metal. To carry out bending of sheet metal a bottom tool is mounted on lower stationary plate (beam) and the top tool is mounted on a moving upper plate (beam).The sheet metal is placed between

the two tools and top tool is pressed down. There will be a force exerted between the two beams is transferred through a frame. The beams and most other parts of the frames are generally made up of steel plates and that parts are either welded or bolted together. The conventional press brakes are driven using one hydraulic cylinder at each end of moving upper beam connected by link. Because the work piece is made between these actuators, both beams are exposed to three point bending deflection of the beams lead to unwanted variations in the bend angle throughout the length of work piece [1–4].

LITERATURE REVIEW

Pedro G. Coelhoa *et al.* (2005) conducted a study on structural analysis and optimization of hydraulic press brakes [5]. It has been found that the source of deflection parallelism errors and minimize them through a structural optimization methodology. Based on the model of the bending process in press brake defined by Timoshenko theory of beams, it has been not possible to design a machine that achieved uniform Bending angles for every bending length because no optimal shapes or dimensions of the bed and ram that leads to parallel deflections for all bending lengths. To achieve uniform bending angles for every bending length work piece bending errors have been derived by considering for all the influence of shape, dimensions and initial deformation of the machine structural components, shape optimization, dimensional optimization has been performed and to decrease the bending error optimal initial deformation was performed.

Work carried out by H. N. Chauhan and M. P. Bambhania (2013) shows that, FE analysis has been applied on a frame of 63 ton power press machine by using Finite Element Method. Analysis of this frame was done by using simulation software [2]. In which they concluded that instead of sharp corner of the c-frame, fillet is provided to reduce failure in the structure. Amount of fillet depends on load condition experienced by frame and it is analyzed by using Finite Element Method Tool. It is concluded that simulation software is the powerful tool for prediction of plate required for a given load. This analysis given the result of reduction in thickness of plate of frame structure, material saving and as well as cost benefits.

S. M. Bapat and Dessai Yusufali (2014) have investigated on the design and optimization of a 30 ton hydraulic forming

press machine. The work done in this paper shows that analysis of the frame structure in terms of its material, geometry and stressed induced in it. Metal forming is one of the manufacturing processes which are almost chip less [6]. In this paper they focused on the causes of structural failure problem in the machine because hydraulic press continuously deals with the stress that may be compressive or tensile for that press machine always works under impact load condition and because of impact load the hydraulic press always experienced continuous stress .it is studied that different components of the machine are subjected to different types of loading condition and are analyzed by using FEM tool ANSYS. Weight optimization of press frame and upper head is done, which in turn reduces in thickness of the frame structure and material.

Work carried out by Muni Prabharan and V. Amaranath (2011) shows that; topology optimization has been applied on various components of scrap baling press and 5 ton hydraulic press using ANSYS work bench software. Shape optimization is a promising tool to explore optimal solution to engineering products. The benefits are numerous, including load path visualization, weight savings, systems design space, ballistic protection and improved fatigue resistance it is inferred that topology optimization results in a better and innovative product design.

Sutasn Thipprakmas *et al.* (2011) conducted a study of the process parameters design of spring back and spring go into V-bending process using Taguchi technique. They show that the process parameter of bending angle, material thickness and punch radius are very important to achieve a high precision of bent parts of complex shapes. The combination of the FEM simulation, the TAGUCHI method, and the anova technique was an effective tool to predict the degree of importance of the process parameters in the v-bending process, in an addition to aiding in the improvement of the quality of the required bending angle by optimization of the process parameters relating to the spring-back and spring-go.

PROBLEM DEFINITION AND PRESENT WORK

The present work includes the simulation (stress analysis) of hydraulic operated press brake subjected to load. The stress analysis is carried out using advanced fem tool ANSYS. The static structural analysis was done for hydraulic operated press brake

PROJECT DETAILS

SPECIFICATIONS

Material used for manufacturing the hydraulic press brake: Mild steel.

Carbon content of the mild steel: 0.40

Density: 7801Kg/m^3.

Ultimate tensile strength of the mild steel: 650 N/mm^2.

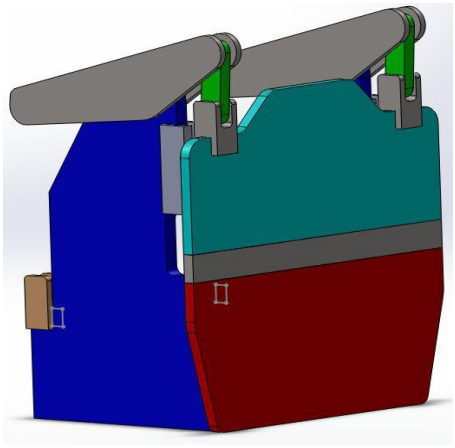

Fig. 1: *Shows 3D View of Hydraulic Press Brake.*

Young's modulus: 2X10^5. N/mm^2

Poisson's ratio: 0.3.

Coefficient of thermal expansion: 12 x$10^{-6}/^0$C. Thermal conductivity: 43.3 W/m-K

Operating temperature: 30–80^0C.

Load acting on the hydraulic press brake: 30tons Yield strength: 250 N/mm^2

For analysis purpose the model is recreated with separate parts as frame, cylinder and ram by using ANSYS software.

Modeling: The Machine assembly consists of top plate bottom plate, side plate and guide. The whole assembly is made up of mild steel

ANALYSIS OF HYDRAULIC OPERATED PRESS BRAKE

The hydraulic operated press brake assembly consists of link, top plate bottom plate, clamp and guide, side plate etc. The whole assembly is made up of mild steel in solid works software

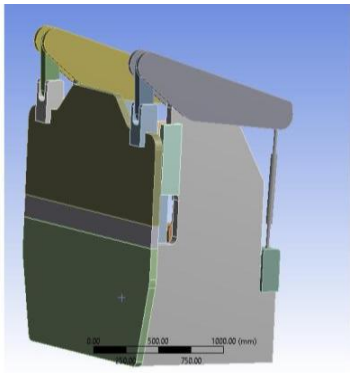

Fig. 2: *Modeling View of Hydraulic Operated Press Brake.*

MESHING

Elements used SOLID185 element (8 nodes with 3DOF) is used to mesh the hydraulic operated press brake. The whole press brake is free meshed. The Figure below shows the meshed view

BOUNDARY CONDITIONS

The hydraulic operated press brake is constrained in all DOF at the base and force of 30 ton is applied on each cylinder position.

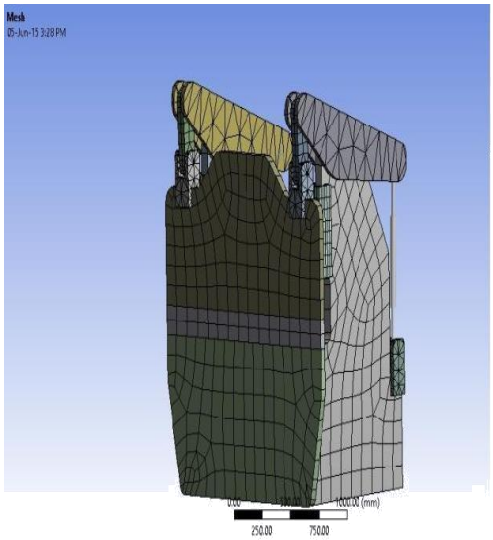

Fig. 3: *Meshed View of Hydraulic Operated Press Brake.*

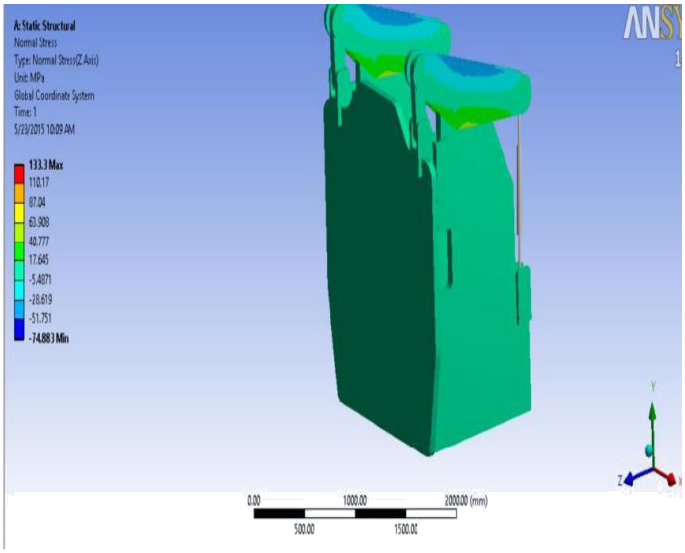

Fig. 4: *Application of 30 ton Load on the Frame and Stress Results Obtained.*

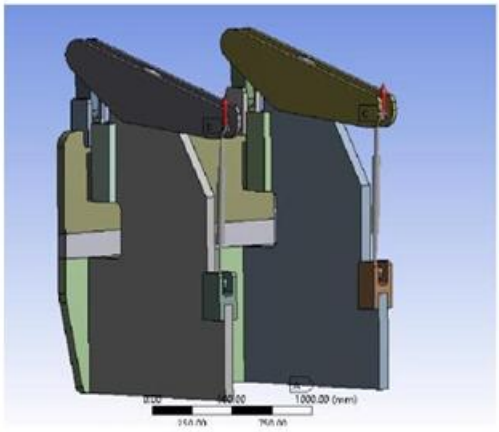

Fig. 5: *Shows Application of 30 ton Load and BC's on Hydraulic Operated Press Brake.*

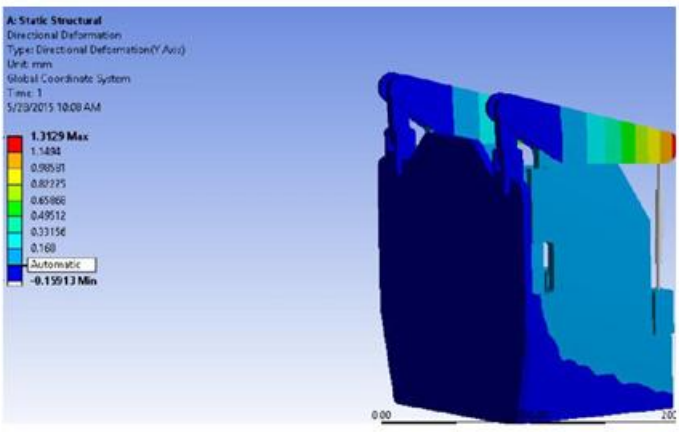

Fig. 6: *Application of 30 Ton Load on the Frame and Deformation Obtained.*

RESULTS OBTAINED

A force of 30 ton is applied on frame and results obtained are shown in below Figure

RESULTS OBTAINED FROM ANALYSIS

The results obtained from analysis

1. The maximum stress induced in hydraulic operated press brake 133.3 Mpa.
2. The maximum displacement for this load 1.3 mm.
3. The maximum stress induced in hydraulic operated press brake from ANSYS is 133.3 Mpa and from theoretical calculations is 145.3 Mpa.
4. Factor of safety from above results

will be 250/145.3=1.720.

5. Therefore, now we reduce the side plate thickness of hydraulic press brake size from 75 mm to 60 mm and we go for further analysis and material optimization.

CONCLUSION

Finally, we concluded that by using ANSYS14 workbench software an attempt was made to analyze the 30 ton hydraulic operated press brake the static structural analysis carried out for parts of hydraulic operated press brake and the project work of hydraulic operated press brake is carried out successfully and designed to meet the requirements as per constraints. The hydraulic operated press brake is carefully designed and cross checked where it does meet the requirement. With reference to the assumptions made above the maximum stress induced in the hydraulic operated press brake is 133.3 Mpa which is less than allowable stress of the material given.

REFFERENCES

1. R. Lown. Hydraulic Presses in the 80's. Based on SME Technical Paper, MF82-918. The Society of Manufacturing Engineers, Dearborn Michigan; 1982.
2. Bambhania M. P, Chauhan H. N. Design & analysis of frame of 63 ton power press machine by using finite element method. 2013; 3: 285–288p.
3. B. Parthiban, P. Eazhumali, S. Karthi, et al. Design and analysis of c type hydraulic press structure and cylinder. *International Journal of Research in Aeronautical and Mechanical Engineering*. 2014; 2(3): 47–56p.
4. Mohammad M. Saleh. Design study of a heavy duty hydraulic Machine using Finite element techniques.
5. Pedro G. Coelhoa, Luis O. Fariab, Joao B. Cardosoa. Structural Analysis and Optimization of Press Brakes. *International Journal of Machine Tools & Manufacture*. 2005; 45: 1451–1460p.
6. Jahan Taghizadeh. Design and optimal calculate of a hydraulic press with one hundred tons capacity. *International Journal of Current Life Sciences*. 2014; 4(2): 704–708p