International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE) ISSN: 2349-7947
Volume: 2 Issue: 4 019 – 022

Design and Analysis of a Impact Plate

Aniket Anantrao Deshmukh.
U.G Student, Mechanical Engg. dept.
K.K.Wagh Institute of Engineering Education and Research.
Nashik, India.
deshmukhaniket145@gmail.com

Kartik Balasaheb Datir.
U.G Student, Mechanical Engg. dept.
K.K.Wagh Institute of Engineering Education and Research.
Nashik, India.
kartikbdatir1@gmail.com

Akash Pandit Bhamare.
U.G Student, Mechanical Engg. Dept.
K.K.Wagh Institute of Engineering Education and Research.
Nashik, India.
akash.bhamare@gmail.com

Chandrakant Prakash Chaudhari.
U.G Student, Mechanical Engg. Dept.
K.K.Wagh Institute of Engineering Education and Research.
Nashik, India.
cpc1993@gmail.com

Ramesh B. Bhandare.
Assistant Professor, Mechanical Engg. Dept.
K.K.Wagh Institute of Engineering Education and Research.
Nashik, India.

*****

Abstract— The plate in the rubber molding press plays an important role in molding operation. The press is a four column compression rubber molding press. When the impact load is been applied on the plate, the deflection of the plate takes place and stresses are induced in the plate. In this paper the various parameters of plate like thickness, deformation are calculated. The plate model is drawn using the modeling software solid works and using ANSYS workbench15 software, the deflection and stress is been found out by using ANSYS and we also do the thermal analysis of that plate. The results obtained by ANSYS are compared with theoretical results which are very helpful to avoid the stresses and deflection in the plate.

Keywords-*rubber moulding press,impact load,solid work,ANSYS workbench 15.*

*****

I. INTRODUCTION

Impact plate, base plate which is important component as far as compression rubber molding is concern, compression molding is process which ensures, compound to flow , fill the cavity and spilling out into overflow according to heat and pressure which is provided in volume production system or those involving machining of delicate material.

So design of impact plate which handle direct load from hydraulic cylinder must given prime concern particularly in molding process. Thickness of impact plate plays important role in handling the impact force in order to limit deflection and stresses of plate, but basic stress equation for compressive, tensile loading are developed with prime assumption that, there should not be any discontinuities along cross section and also no abrupt change in cross section of plate, which fails when applied on actual operation. Practical operation are always concern with discontinuities like holes, threads for special functional requirement , such discontinuities alter stress concentration in vicinity of space along plate so parameter which reflects the mechanical thermal properties must taken in consideration which includes thickness and other mounting such as guiding plate for accumulation of flexible die and punch according to requirement which eliminates the heat loss problem dangers to operator, downtime when mold's brought out after each operating cycle, overflow, blow holes.

Mounting accessories on impact plate given strength to plate and gives innovative features for best solution for critical and precision molding with fast production rate, improved quality product, save manpower, reduction in rejection time, consistency dimensional stability, surface finish product, with high automation capability that makes it ideal for today's molding industries.

Finding out dimension of impact plate, thickness, stress and deflection are major consideration, when plate is subjected to loading it deflect from its original position, when it is sufficient strong to resist force and bending moment then design based on strength while, if it has adequate stiffness criteria Macaulay's is one who gave basic formula which provides basic equation for bending moment at any section expressed in systematic order, Which is similar to double integration technique.

Whenever plate subjected to transverse loading it bent. Flexure is one which gives formula for pure bending which ultimately given satisfactory results for designing thickness of plate.

D. Ravi (2014), analyse the power press for reducing its frame thickness,web thickness and bed thickness[1].Amit kalekar, S.B.Tuljapure (2015), do analysis of a frame of a bush poressing machine and they investigate the stress and deflection on lower and upper region of C frame[2].Ankit H. Parmar, Kinnarag P. Zala,Ankit R. Patel,(June 2014) , investigate for the foremost element of hydraulic press machine by using FEM tool wich helps to innovative design,reduced weight and cost effective foremost element of press[3].WANG Yi-xiang, SUN Hui-xue (2010), studied

the influence of the initial deformation on the working performance for MLB hydralic press machine[4].H. N. Chauhan,M .P. Bambhania (2013),studied the amount of fillet provided by press machine which is depend on the load condition and analyse the frame by FEM tool,which is also helpful to reduce the plate size of frame structure so that material saving and cost benefit will be considerable[5].S. M. Bapat,Dessai Yusufali (2014),investgated the implementation of FEM tool for analysis and optimisation of hydraulic forming press machine[6].

This work focused on generation of modeling the functional plate along with molding and its analysis, and by providing the heater on plate as it works on high temperature we do thermal analysis also, to check whether the plate is sustain the high temperature on particular load and thickness.

From above references it is clear that analysis limited to c frame was done. There is a scope to work on the impact plate. So, we take impact plate of rubber press for analysis and for which supporting frame is H type. Plate in this paper has optimum surface to impact force which simply required for effective working. Main steps involved in this work as follows

1. Modeling of plate in solid work software.
2. Calculating maximum stress, deflection and its thickness.
3. Analyzing same parameter by using ANSYS Workbench15.
4. Comparing the theoretical with analytical results.

II. PLATE MODELLING

Along with holes were done equidistances from all corner for functional requirement considering stress concentration constraints, nature of contacting surfaces other parameters regarding to plate were design it is used for molding guide for moulting flexible punch and die.

Impacting force from hydraulic cylinder is transmitted towards impact plate .thus it is very important to analyze factor on plate. Various components for specified operation shear the total load impacting on plate so separate analysis is to be done for each component.

Both plate been modeled on solid work software. The material of plate is mild steel, machining is done taken into consideration of all parameters like width, shearing area, which were calculated from standard empirical relations. Plate and mounting of guide ways were assembled by using mate commands with various relationships, constraints of parallism, surface coincides.

TABLE .1 PLATE PARAMETER

Parameter	Unit
Length	600 mm
Width	400 mm
Thickness	50 mm
Load	340 kN (point load)

TABLE .2 SPECIFICATION OF PLATE

Material Name	Mild Steel
Mass Density	7833 kg/m^3
Young's Modulus	2.08*e5 N/mm^2
Poisson's Ratio	0.27
Thermal Expansion Coefficient	0.00/C
Thermal Conductivity	0.032kW/m-C
Yield Strength	208.508 N/mm^2
Tensile Strength	580 N/mm^2

Thickness calculation –

Plate is designed to withstand maximum bending stresses. There is problem of bending of plate, stress concentration at particular point, abrupt change in cross section of plate. This can be minimized or avoided by applying various methods, parameter to be considered. In this, we are focus on maximum bending phenomenon. Flexures formulae play important role for calculating thickness of plate,

$$\therefore \quad \frac{M}{I} = \frac{\sigma}{y}$$

Consideration- self weight of plate and point load from cylinder which act opposite towards plate

When force is imparted on plate it gives rise to changing in dimensions, its deformation is linearly changes with applied force. Such deformation is called as deflection of plate

Deflection play major role during working conditions as it changes the dimensions of auxiliary which ultimately results in operation condition in order to avoid this various equation are tried out. Macaulay's given basic formulae for finding out maximum deformation.

Macaulay's equation

$$\therefore EI\frac{d^2y}{dx^2} = M_b$$

This bending generally observed along shorted bent dimension.

III. ANALYSIS OFIMPACT PLATE

Failure of plate due to the bending stress and deformation are the two important considerations while designing impact plate. In order to predict failure due to the maximum tensile and compressive stress on the plate must be given prime importance and must require specifying. In the past research, the bending stress sensitivity of the impact plate has been calculated using photo elasticity or relatively coarse FEM meshes. But by using coarse FEM meshes the analytical results does not closely resembles to actual working condition. However present advancement in the technology has lead down various added function, so as to withstand global advance in market.

So this advancement focused to use FEM which closely resembles with actual working condition and gives result which are more précised and accurate.

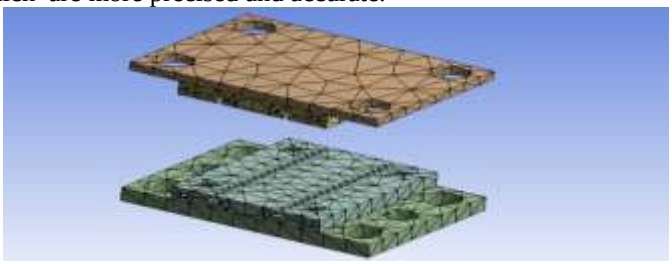

Figure1. Mesh Model.

A. Analysis:

In the present impact plate model problem, we export the modelled drawing from SOLIDWORKS software with the .igs as extension and do its analysis in the ANSYS WORKBENCH 15 software. Initially geometry and parts are defined. The boundary condition and loading condition should be specified. At the starting condition the load is applied on the plate and the amount of load transfer from plate to mounting parts were observed.

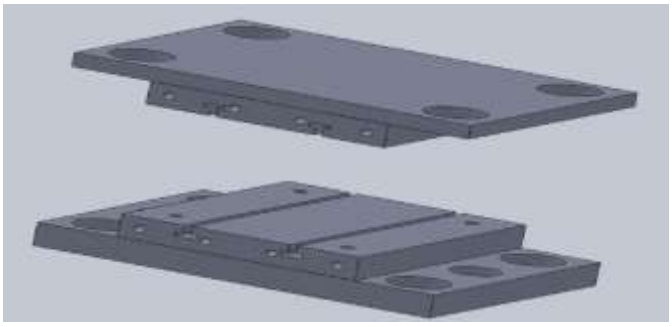

Figure2. Solidworks model of plate.

The area defining the contact region of the plate holes which are made for functional requirement and other component made frictionless. At initial condition as the load is applied stress starts building up gradually and as the time passes it reaches towards maximum value.

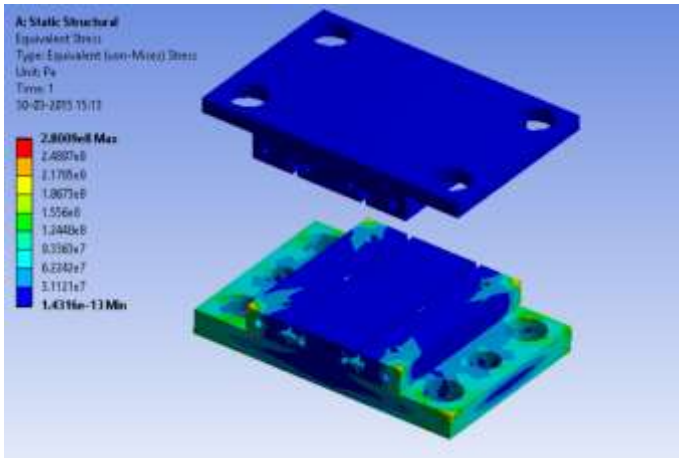

Figure3. Stress analysis of plate.

From, the figure it is clear that the minimum stress on the plate surface is 1.431e^-7mpa and the maximum stress goes to 280mpa. The maximum stress obtained from software is within permissible limit.

The deformation analysis is also done on the plate to know the maximum deformation.

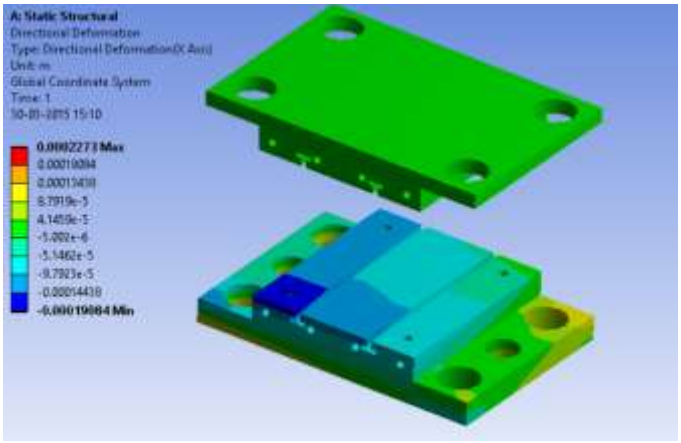

Figure4. Deformation Model.

It has been found that deformation is maximum at the corner of the plate. This indicates that corner is under high temperature and pressure due to which deformation is maximum. Here the deformation ranges from -0.19m to 0.27mwe can say that Analytical deformation is in permissible limits.

IV. COMPARING THE RESULTS

The results obtained from the ANSYS 15 software are compared with the theoretical results. The first comparison is done between the stresses obtained theoretically and by using ANSYS. The graph is plotted between stress v/s the no. of tooth. The following graph shows the stress distribution for the larger gear.

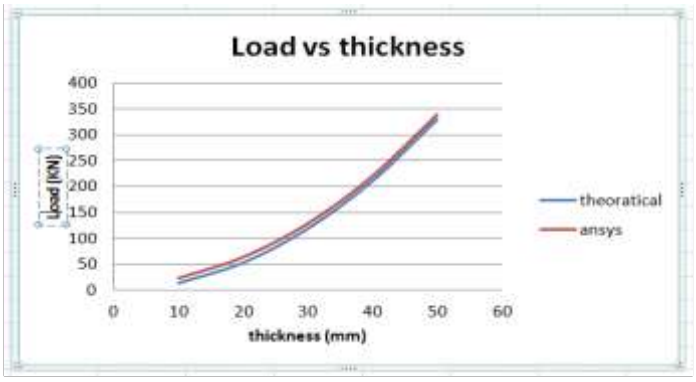

Figure5. Load varies with respect to thickness.

From the graph it seen that as the load varies the corresponding thickness for sustaining the load varies linearly.

Figure6. Deformation varies with respect to load.

From above graph it is seen that analytical and theoretical value differs marginally .This is due to fact that in analytical condition is closely resembles with actual working condition by giving proper constraints. While in theoretical assumption makes results slightly deviated from actual results due to assumption made in theoretical.

CONCLUSION

An attempt was made to analyzed and optimizes the impact plate of rubber molding press machine by using ANSYS Workbench 15.From results the deformation and stress for the designed plate are within limit. From results it is concluded that as load increases the deformation increases linearly. As the load increases thickness required to withstand corresponding load also increases.

REFERENCES

[1] D. Ravi "Computer aided design and analysis of power press" IPOSI publications,2014, DOI:10:5829/ idosi.mejsr.2014.20.10.114100.

[2] Amit Kalekar, S.B.Taljapure "Stress analysis of a frame of a bush pressing machine for pump" IJIRSET ISSN 2319-08753 vol.4,Issues 2,February 2015.

[3] Ankit H Parmar, Kinnarraj P. Zala, Ankit R. Patel,"Design and modification of foremost element of hydraulic press machine ",issue 4,vol. 3,may-june 2014 ISSN 2249-9954.

[4] WANG Yi-xiango and SUN Hui-xae ,"Influence of the initial deformation on working performance for MLB hydraulic press",unpublished.

[5] H.N. Chauhan And M.P. Bambhania,"design and analysis of frame Of 63 tonn power press machine by using finite element method",vol. 3,issue 7,july 201,ISSN 2249-555X.

[6] S.M. Bapat And Dessai Yusufali,"Design and optimisation of a 30 tonn hydraulic forming press machine",vol.2,issue 8,august 2014,ISSN 2321-9653.