International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 17 (2015) pp 37871-37876
© Research India Publications. http://www.ripublication.com

Topology optimization of forming die

Krishnakumar V J
Master student, Department of Mechanical Engineering, Amrita Viswa Vidyapeetham, Coimbatore, India
vj_krishnakumar@yahoo.co.in

Dr Joshi C Haran
Professor, Department of Mechanical Engineering, Amrita Viswa Vidyapeetham, Coimbatore, India
joshi_ch@amrita.edu

Antoni Manekshaw
Project Manager, Altair Engineering, Banglore, India
antoni.manekshaw@altair.com

Abstract

Topology optimization technique is being used in structural engineering to reduce the weight of structures through redistribution of material layout. This technique is used in this study to reduce the weight of a blankholder structure, which is a part of sheet metal forming die. Out of the different methods available, such as Homogenization method, Evolutionary Structural Optimization and Density method, density method is selected. A step bottomed cup is being developed and forming simulation is done using a software called Altair Hyperform. The contact forces from the last step of the simulation is mapped into a blankholder structure, which was already modelled using 3 D elements. Topology optimization is done using the mapped loads and the weight reduction of 18.686% is achieved. Fatigue analysis was done on the optimized and original blankholders and found the results well under acceptable levels.

Keywords— topology; optimization; stamping; sheet metal forming; die; mapping; fatigue; density method; SIMP.

I. INTRODUCTION

Wei Liu et al. [1] introduced two objective functions which account for springback and insufficient stretching. A multi objective generic algorithm (MOGA) which does not involve the conversion of multi-objective functions into single objective function using weighted constraints is used for topology optimization. They used mesh morphing technique for die face redesign, which does not involve alterations in the die structure.

Oguz et al. [2], made use of two methods. In the first method, they used a double binder and found its effect on springback, wrinkling and thickness reduction. In the second method they took positions of upper die, draw bead, draw bead radius, forces applied on the upper die surface and double binder surface as process parameters. Most appropriate values of these parameters are calculated for optimum formability characteristics.

Ji-Hong et al. [3], proposed topology optimization method which is used to design the supporting structures on account of the gravity of the die itself and the surface loads obtained from the numerical stretch forming procedure. Based on the stretch forming simulation (Abaqus) the surface loads on the die are obtained and applied as multiple load cases. Loads can be approximately treated as constants in a certain range, which will favor optimization by iterative calculation. Linear interpolation approach for casting constraints were used by them. Material distribution was constrained in the prescribed casting direction.

Topology optimization is done by Frida et al. [6], for improving stiffness of die and reducing its weight. The loads obtained from forming simulation is being applied to the structure. Some small alterations to the contact force obtained from forming simulations is done to prevent the tearing of the blank. They obtained satisfying result at a volume fraction of 0.15. The mass was reduced by 20% and maximum displacement about 30%.

SIMP (Solid Isotropic Microstructure with Penalty) based topology optimization is proposed by Xu et al. [4]. To validate the proposed method, a step bottomed cup was developed. Sheet metal forming simulation considering interaction between punch and blankholder, and also considering the multiple loading conditions at different forming positions is performed. A local load mapping algorithm was developed to map the loads from forming simulation into a blankholder structure. Topology optimization was performed considering the different loadcases and achieved a weight reduction of about 28.1 %.

So topology optimization method can be effectively utilized to reduce the weight of a die structure. Mesh morphing technique can be used to alter die face without much affecting the die structure. Multi objective generic algorithm can be utilized for optimizing two functions without converting them into a single objective function. Surface loads obtained through forming simulation is used to optimize a die structure and is regarded as the suitable method, because it will consider the operational case. The multiple loads which are obtained during different times of forming can be used for optimizing a structure, which leads to a solution closer to the real case. This paper proposes a complete automation of the topology optimization process and a new technique for load mapping using a software. The load mapping technique [7] is based on meshless mapping technique which uses a polynomial function for interpolation.

II. THEORIES INVOLVED

The general optimization problem can be written as [6]
The objective function min $f(x)$
Constraints $g_i(x) \leq 0$ for $i=1,\ldots\ldots, m$
$h_i(x)=0$ for $j=1,\ldots\ldots,n$
$x \in R^n$ (the real coordinate space)
The solution of the problem is x^* where $f(x^*) \leq f(x)$

A. Toplogy optimization- Mathematical derivation [5]

Minimum compliance design

Assume Ω as the reference domain
Optimal design problem is defined as the selection of optimal stiffness tensor $E_{ijkl}(x)$.
$a(u,v)=\int_\Omega E_{ijkl}(x)\varepsilon_{ij}(u)\varepsilon_{kl}(v)d\Omega$
a(u,v)-Internal virtual work done at u(equilibrium) and for v(arbitrary virtual displacement)
ε -Linearized strains.
$\varepsilon_{ij}(u)=1/2(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})$
Work done by the external load which acts on the structure
$l(u)=\int_\Omega fud\Omega+\int_{\Gamma T} tuds$
f-body force
t-boundary traction
$\Gamma T c\ \Gamma \equiv \partial\ \Omega$
So minimum compliance problem is now,
$\min_{u \in U,E}\ l(u)$
S.T. $a_E(u,v)=l(v)$ for all $v \in U$
$E \in E_{ad}$
$E_{ijkl}(x)=\rho(x)\square E\square_{ijkl}$;$p>1$
$\int_\Omega \rho(x)d\Omega \leq V;0<\rho_{min} \leq \rho \leq 1$
u-Space of kinematically admissible displacement fields
Ead-Set of admissible stiffness tensors
p-Penalty factor which makes solution discrete
$E\square_{ijkl}$-Elastic modulus of a given isotropic material
ρ(x)-Density
Topology optimization with a volume constraint is expressed as
Min : $C=F\square\ U=U\square\ KU=\sum_{e=1}^{N}((\rho_e)\square u_e\ \square\ k_e\ \square\ u$
S.T. :$V=f_vV_0=\sum_{e=1}^{N}\rho_eV_e$
$F=KU$
$0 \leq \rho_e \leq 1$
C-Compliance of the structure
K-Global stiffness matrix
ρe- Relative density of the eth element
k e □-Stiffness matrix of the eth element
F, U -the vectors of the force and nodal displacement
u e -Element displacement vector
N-Total number of elements
V-Material volume after optimization
fv- Volume fraction defined to constraint the structure volume
V0-Initial design volume
Ve-Volume of the eth element
The density interpolates the stiffness between the material properties 0 and $E\square_{ijkl}$ .So density function can be minimized for topology optimization.

III. METHODOLOGY

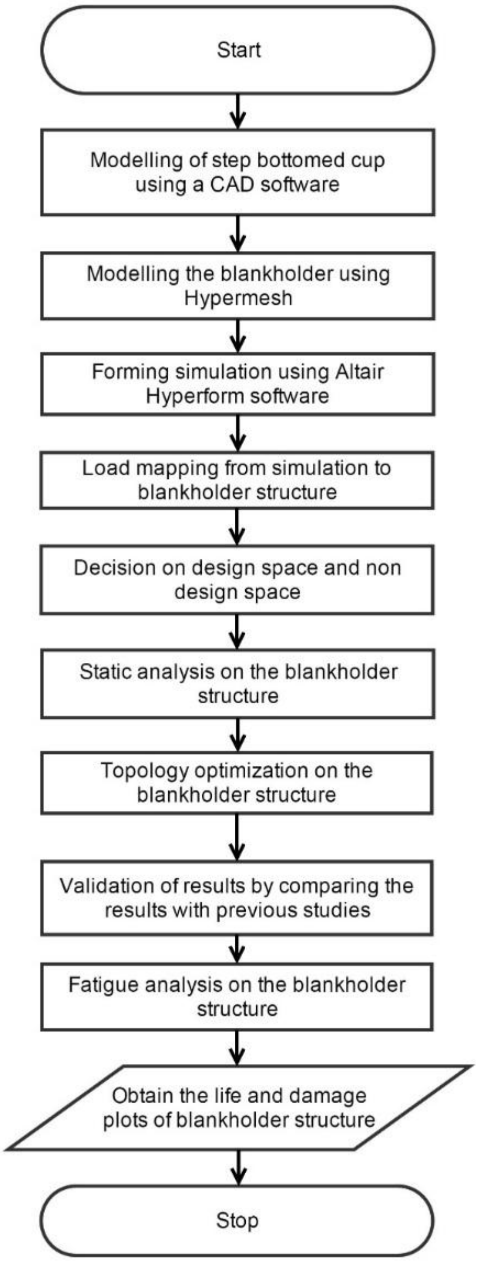

Fig 1.Methodology overview

This study is an extension of the work done by Xu et al. [4], so the same dimensions of the cup as given in their work is taken for modelling the step bottomed cup.
5 trias and 5149 quad elements were used for modelling the blankholder.
The blankholder surface was meshed with 2D elements, first. The average size of the mesh was kept at 5 mm. Then the mesh is being converted into 3D hexa mesh. 39680 hexa elements were formed.

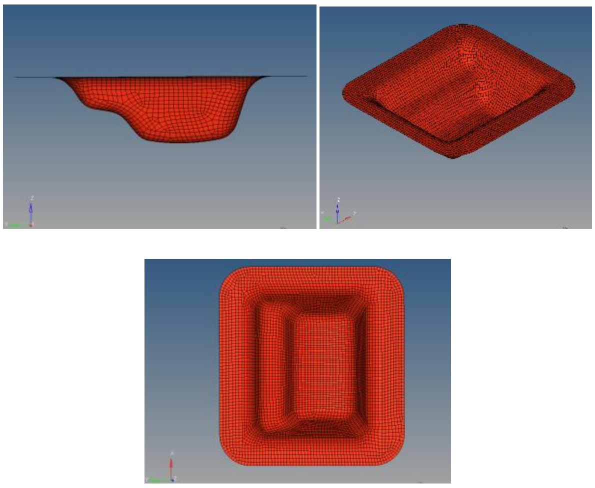

Figure 2-(Clockwise from the bottom)-Plan, elevation, ortho view

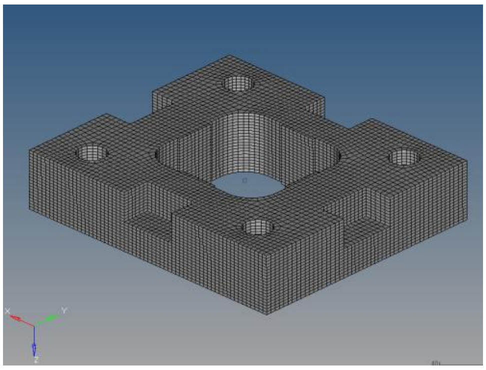

Figure 3- The blankholder mesh

Table 1- Forming characteristics

Drawing Depth(mm)	Coefficient of Friction	Blank holding force (KN)
43	0.14	120

Table 2- Material properties

Material	Thickness (mm)	E (MPa)	υ	σs (MPa)	K (MPa)	n	R00	R45	R90
DP600	1.4	207000	0.28	421	1110	0.212	0.877	0.886	1.128

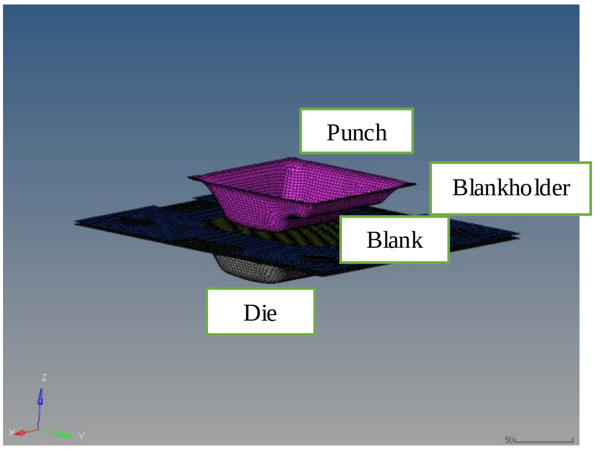

Figure 4-The stamping simulation model Hyperform

The punch, blankholder and die are considered as rigid bodies. So no displacement occurs inside the structure. Therefore the surfaces of these parts are used for simulation and computation time is saved. Double action draw technique is used for forming simulation where the punch moves and deforms the blank.

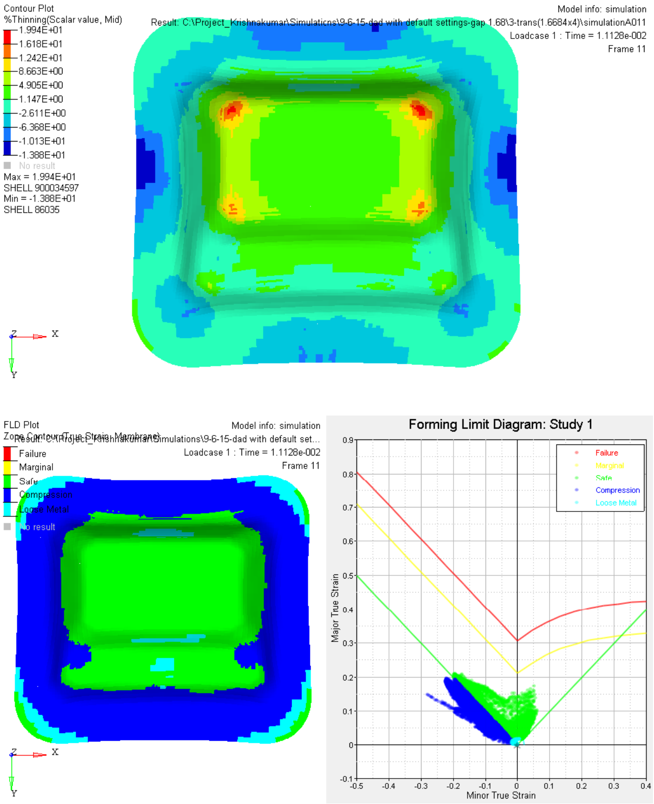

Figure 5- Plots of percentage thinning and FLD.

Percentage thinning option allows to view the thinning in percentage after the blank is being formed. Forming limit diagram is used to detect forming failures such as failure by tearing or wrinkling. It is formed from major and minor strains found out after the deformation of the circles imprinted in the initial blank. If an element is coming inside FLD, it is highly formable.

The contact force is found to be very large at one side because at that point, the blank is drawn very deep. Contact force massively depends upon the final blank shape. It is not uniformly distributed around the blankholder. The pattern is somewhat similar to that which is in the study [4]. Same conditions are used here. The loads were mapped using the die stress analysis template which is available with Hyperworks. The summation of loads was found to be 120.3kN which is exactly equal to that of the final simulation time step.

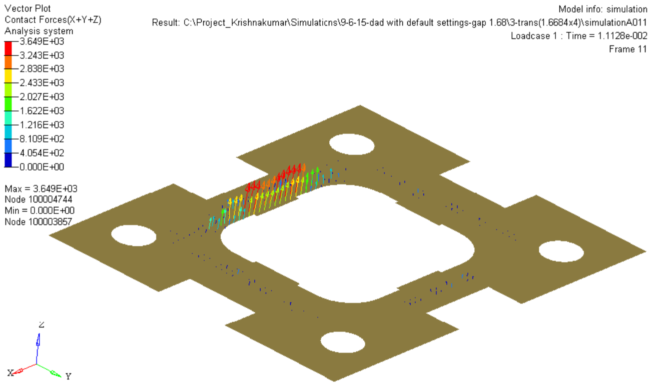

Figure 6-Contact forces at the final time step of simulation

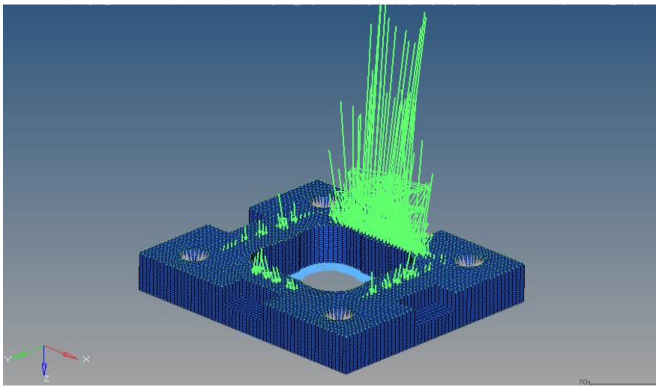

Figure 7-Mapped loads on the structure

IV. RESULTS

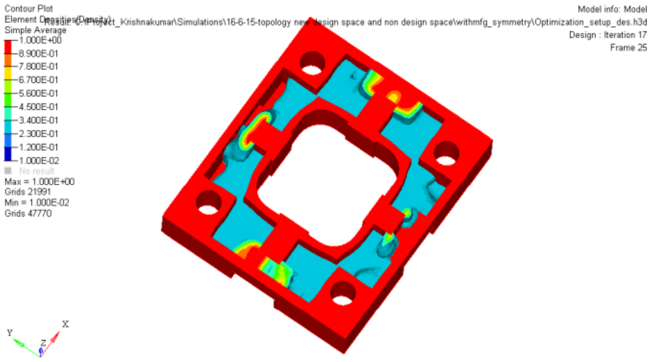

Figure 8- Topology optimization result

A. *Comparison of structural performance*

The default results of static analysis in Optistruct are displacement and Von Mises stress. Both the displacement and stress levels of original and optimized blankholder are compared. Figure 9 shows that the displacement of the structure (a) increased from 0.0487 to 0.05147. The difference between them is 0.00277 which is well under acceptable level (≤5%). As compliance is minimized, Stiffness is increased. So the displacement is increased to account for stiffness increase as the force is constant.Structural performance (b) shows that the stress levels in the structure are not increased by the removal of material. This is because at the iso value of 0.3, the material is not removed from the critical parts of the structure. So the maximum stress remains exactly the same.

Validation of structural performance is done using the study [4]. This study was done by taking the same model from [4]. Instead of taking load distributions at three different positions and doing optimization, only one load case involving the contact forces from the final forming step, that is when the blank was completely formed was taken and performed optimization. The interaction between punch and blankholder is also neglected as it requires significant alterations of the default model in the sheet metal forming software Hyperform. The displacement results obtained from study [4] is in the range of 0.04mm-0.6mm. They obtained an increase in displacement of 18.8 % when the structure is optimized. The displacement results in this study are very similar to the obtained values. The maximum stress values found in [4] is near 80 MPa. The maximum stress value obtained in this study is in the range of 150 MPa which is very high. Because in this study, the punch force is applied vertically on the blankholder whereas in [4] some punch force is gone horizontally as interaction between punch and blankholder is considered by the authors.

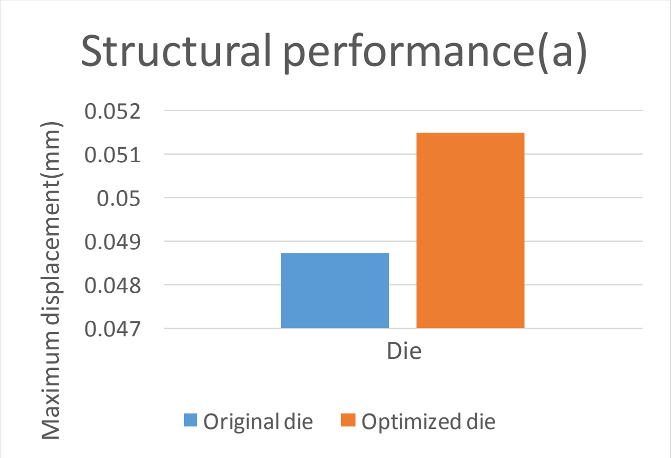

Figure 9- Comparison of structural performances

B. Punch load vs punch displacement

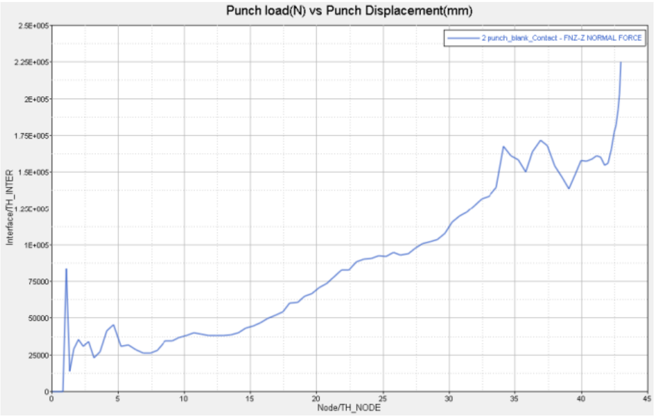

Figure 10- Punch load vs punch displacement

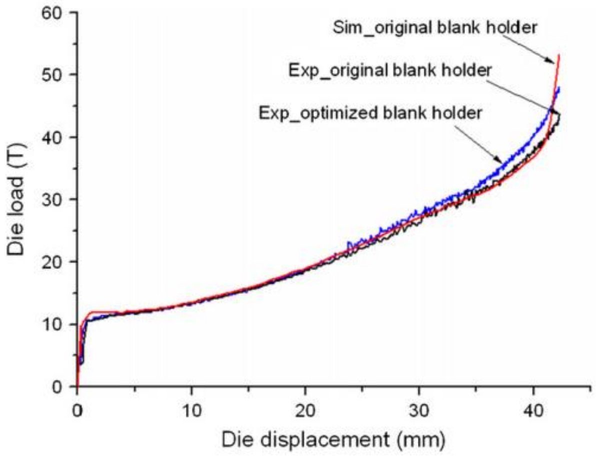

Figure 11- Die load vs die displacement

$$1 \text{ ton force}=\frac{Force\ in\ N}{9800}$$

The forming simulation can be validated by taking figure number 14 from paper [4]. There die load in ton force is plotted against die displacement in mm. The characteristics of experimental original and optimized blankholder is compared with simulated original blankholder. They follow the same pattern initially and shows some deviations at end. The maximum die loads of experimental original and optimized blankholder and simulated original blankholder are about 40, 45, 53 ton forces respectively. The difference is caused by the reduction of material flow for optimized blankholder. In this project, double action draw has been adopted, so punch will be moving. So punch load vs punch displacement has been plotted. The forming simulation shown in figure 4 also follows same pattern but with a maximum punch force of 225000 N which is 22.9436 ton forces. Punch load is less compared to [4] because interaction between punch and blankholder is not considered in this project.

C. Fatigue analysis

Stress-life approach was taken into consideration because stress levels in the structure is considered to be in elastic range. The loading-unloading curve can be created only after knowing the characteristics of die machinery. So an arbitrary loading history table, where the interval of application of load is 2s, is used. The loading history helps to convert static load into varying load.

As FG300 material is used for blankholder structure, the following properties were used

Ultimate tensile stress=300 MPa
Yield strength=124 MPa
First fatigue strength exponent=-0.15
Fatigue strength coefficient=300 MPa
Point at endurance limit occurs=10^6 cycles
Mean stress correction= Goodman method

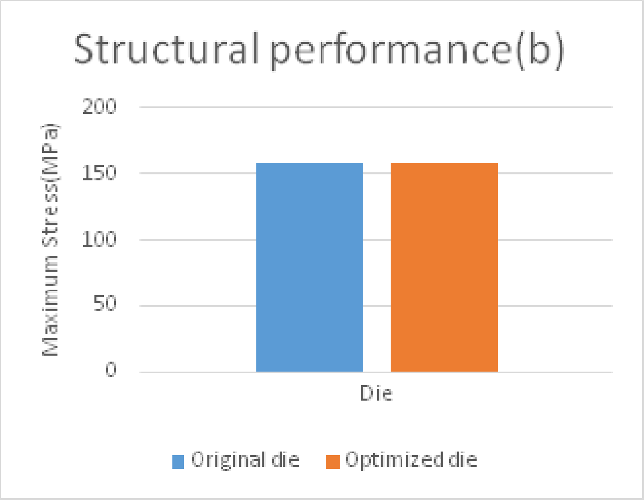

Figure 12-Original blankholder life plot

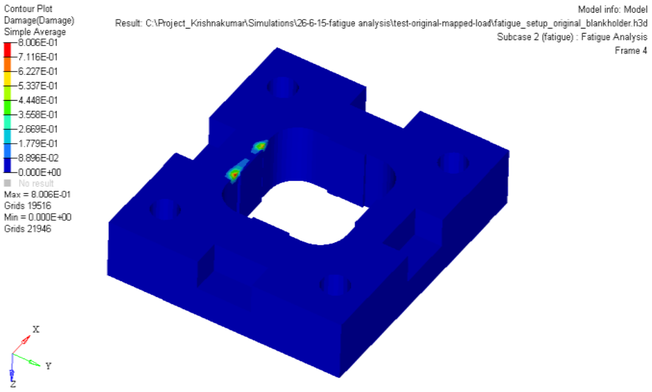

Figure 13-Original blankholder damage plot

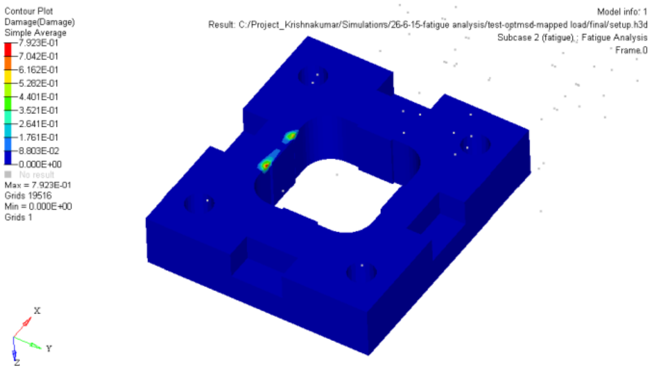

Figure 14- Optimized blankholder life plot

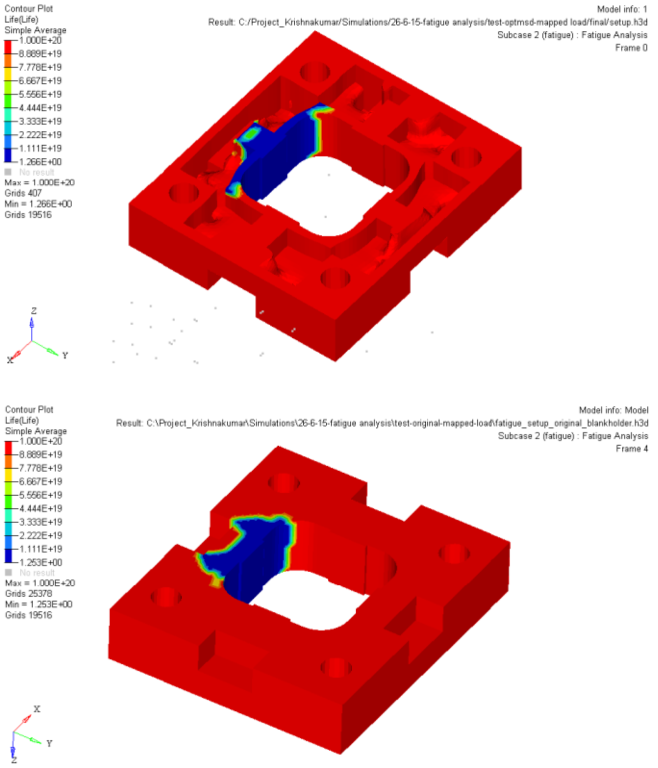

Figure 15- Optimized blankholder damage plot

Life is the number of loading cycles that cause fatigue at each location. At one node, for both original and optimized blankholder life is near about 1.2 cycles which is very less. At all other points, the life is above accepted levels.

The damage is calculated as, Damage = n/N = no. of cycles applied/Total life

It is considered as Damage < 1 = safe design and if Damage > 1 =design failure. So for original blankholder, the maximum life is 0.8 and for optimized blankholder, the life is 0.7923. Both are less than 1 and the design is safe.

V. CONCLUSIONS

The proposed methodology of topology optimization depending on density method with SIMP interpolation works well for decreasing the weight of any die structure. The study resulted in a weight reduction of 18.686%.The previous problem of manual local load mapping can be automated using the template "die stress analysis" available with Hyperworks suite. So a complete automation of the optimization considering the operational case is being developed.The simulation of topology optimization was validated using [4]. The variations in punch load vs punch displacement (die load vs die displacement) and equivalent stress vs die displacement can be attributed to the non-consideration of the interaction between punch and blankholder.High life of optimized blankholder structure similar to the original one and damage which is less than 1 for the optimized blankholder structure points the fact that the optimized design is very safe.

ACKNOWLWDGMENT

The softwares for this work, Hypermesh, Optistruct, Radioss, Hyperform, were provided by Altair Engineering. The work was done at Altair Engineering, India.

REFERENCES

[1] Wei Liu, Yuying Yang, "*Multi-objective optimization of an auto panel drawing die face design by mesh morphing", *Computer-Aided Design, pp 863–869, 2007.

[2] Oguz Kayabasi, Bulent Ekici, "*Automated design methodology for automobile side panel die using an effective optimization approach",* Materials and Design, pp 2665–2672, 2007.

[3] Ji-Hong Zhua, Xiao-Jun Gua, Wei-Hong Zhang, Pierre Beckers, "*Structural design of aircraft skin stretch-forming die using topology optimization",* Journal of Computational and Applied Mathematics, pp 278–288, 2013.

[4] Dongkai Xu, Jun Chen, Yucheng Tang, Jian Cao, "*Topology optimization of die weight reduction for high-strength sheet metal stamping",*International Journal of Mechanical Sciences, pp73–82, 2012.

[5] Bendsøe MP, Sigmund O, "*Topology optimization—theory, methods and applications*" New York: Springer; 2003

[6] Frida Birath, Anna Nilsson, "*Topology optimization of a stamping die",* Master's dissertation, Division of Solid Mechanics, Lund University, Box 118, SE-221 00 Lund, Sweden, 2006.

[7] User's guide- Optistruct 13. Obtained through the internet: *http://www.altairhyperworks.com/*, [Accessed 10/2/2015]

[8] Nitin S. Gokhale, Sanjay S. Deshpande, Sanjeev V. Bedekar, "*Practical finite element analysis",* Pune: Finite to infinite, 2008.

[9] Topology optimization. Obtained through the internet: *http://training.altairuniversity.com/optimization/topology/*, [Accessed10/2/2015]