International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

Thermal Analysis of Complex Die for Lug Preparation Subjected To Impact Loads

Sravan Kumar, Dr.S.ChakradharGoud

Research Scholar , S.J.J.T.University, Rajastan

Prof & Principal Sana Engg college, Kodad

Abstract: Forging dies are usually made of high-alloy or tool steel. Dies must be impact resistant, wear resistant maintain strength at high temperatures, and have the ability to withstand cycles of rapid heating and cooling. By considering the thermal heat flux generation at critical areas of complex component production analysis optimization is needed to estimate life cycle analysis. Present paper describes modeling and thermal analysis of bottom die of lug component for 1 ton hammer load to check the thermal contour regions. The modeling done in UNI-graphics NX 8.0 and the analysis carried out in ANSYS work bench. This will compared with practical results to get optimum accurate coincidence of life cycle.

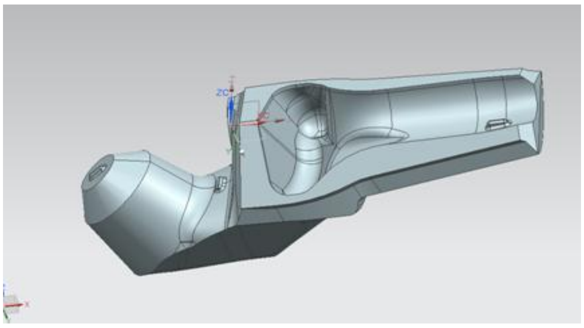
Key words: thermal load analysis, heat flux, forging die.

Introduction: In warm forging, the billet is heated to temperatures about the re-crystallization temperature, for example, for steels the range of 800-1000^{o}C, which is lower than the conventional hot forging range, is considered as warm forging temperature range. In warm forging, flow stress and the forging pressures are reduced compared to cold forging. A greater degree of deformation can be achieved in a single operation compared to cold or warm forging. Main disadvantages are extensive scale formation, low dimensional accuracy, and necessity of larger tolerances for further machining and requirement of heating equipment. At present drilling is the basic need In mining. Hammers are used to make digging the ground .In the present project we undertaking a product called lug which is a partial assembly of hammer. The lug is a component which is used to get hold on digging bit. Three of the lug components act as assembly of cullet to hold the bit.

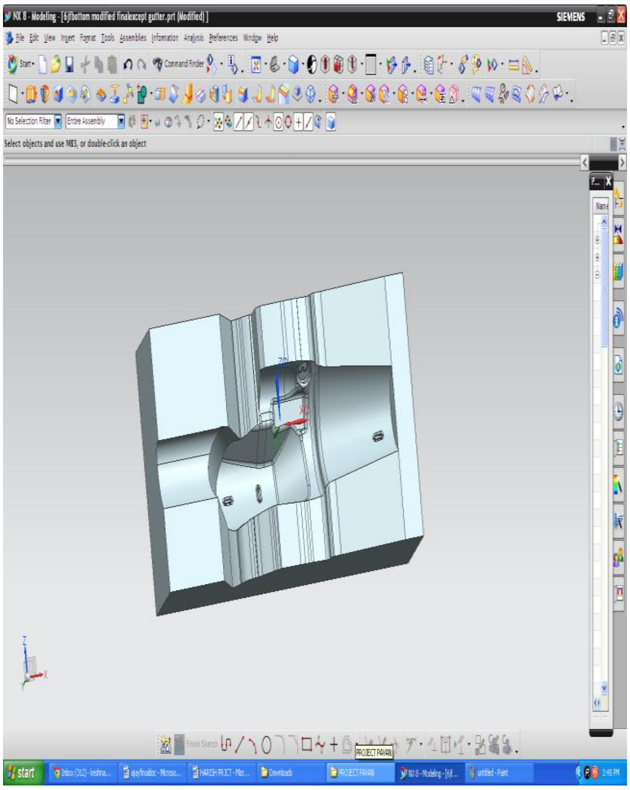
Literature review

In the study of Ceran et al.., [1], hot forging process was simulated coupled with thermal analysis, in order to determine effects of the process on the header die for the taper perform stages in upset forging process. The effects of wall thickness and base thickness of the header die on the stress distribution in the header die were examined. For different cases, the stress distribution in the header die had been presented. The obtained stress distributions were also used for the fatigue analysis of the die.In the study of Garat, Bernhart and Hervy et al.., [2], hot forging die for a nut was investigated in order to increase its life time. In the paper, brittle failure phenomenon of tools is addressed and the influence of process parameters like billet length, billet initial temperature or tool design are studied. Service life prediction considering fatigue is made using the universal slope method proposed by Manson.Kim, Lee, Kim, Kim et al..,[3] studied the estimation method of die service life based on wear and the plastic deformation of dies in hot forging processes. Two methods are suggested for estimating the service life of hot forging dies by plastic deformation and abrasive wear, and these are applied to predict the product quantity according to two main process variables, forming velocity and initial die temperature for a spindle component. Through the applications of the suggested methods, the thermal softening of dies due to the local temperature rise led to the reduction of the service life of hot forging dies by plastic deformation more than by abrasive wear.Brucelle and Bernhart et al.., [4] described the methodology that has been applied to gain understanding of the thermo mechanical stress field in a cemented carbide punch used for the manufacture of airbag container type parts. The stresses are the result of a combination of purely mechanical stresses due to the forging process, and thermo mechanical stresses induced by the thermal cycling of the punch surface during successive hot forging and waiting periods. In the paper, the importance of a simultaneous use of numerical simulation (process simulation and thermo mechanical stress calculation) and experimental testing (laboratory and industrial tests) is highlighted.In the study of Jeong, Kim, Kim, Kim and Dean et al.., [5], experiments and numerical analyses were performed under various conditions, two kinds of surface treatment, two lubricants, different initial billet temperatures and different loads, to investigate the effects on thermal softening and the amount of heat transfer. Carbon Nitride (CNx), ion-nitride and no surface treatment for the dies were used and oil-based and water-based graphite were used as lubricants.In the study of Lee and Jou et al.., [6], the experimental techniques, wear model and numerical simulation method was combined to predict the wear of warm forging die. The non-isothermal ring compression test was adopted to estimate the friction coefficient in different temperatures and the on-line temperature recording system was setup to correct the heat transfer coefficient of the interface. The wear coefficients in different temperatures were acquired from high temperature wear experiment. Additionally, the Archard wear theory and a FEM code, were used to analyze the warm forging of automotive transmission outer-race and predict the die wear condition.In the study of Kim, Yagi, Yamanaka et al.., [7], a history of practical use of FE simulations in forging area was briefly reviewed. Then, practical use and benefits of FE simulations in forging area were discussed with examples. Finally, key points for successful and effective use of FE simulations were explained followed by current issues for better use of the FE simulation as a must tool in the forging industry.

Modeling: 3D modeling has been done by using NX 8.0, component as well as die design of bottom also modeled and exported to Ansys work bench.



Above figure represents the component model of lug for cullet application assembly. The component developed in surface modeling module. The complex die also developed by using surface trimming methods to avoid correlation at the corners



Figureshows the bottom die cutting for analysis representation for checking machining possibilities to avoid destortions

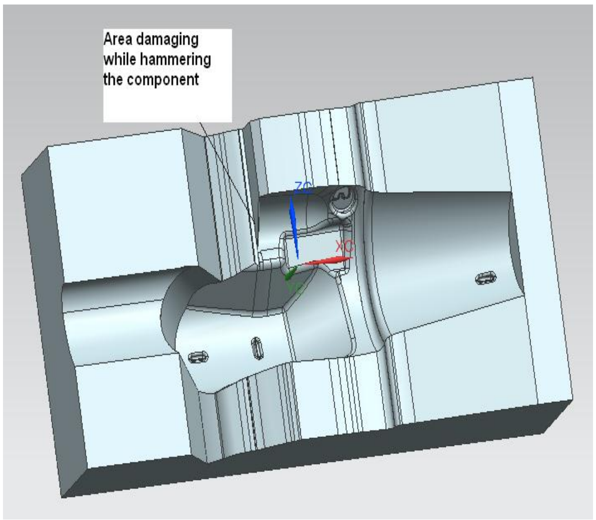
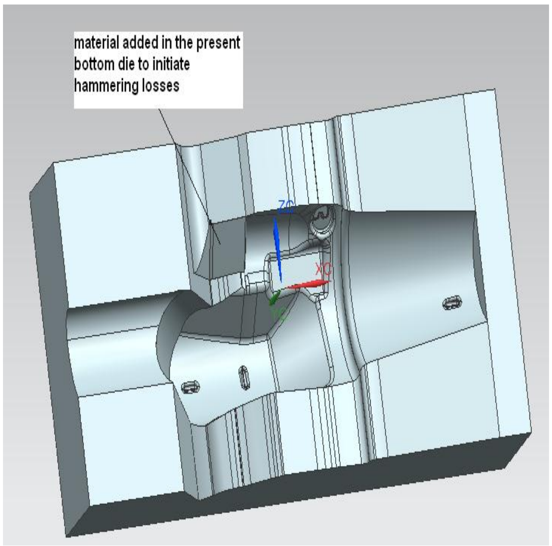


Figure represents the area of surface weakness in the die for the probability of damage section in present die. Material addition is need for wall strengthening procedures.



Figureshows the design modification without changing component impression

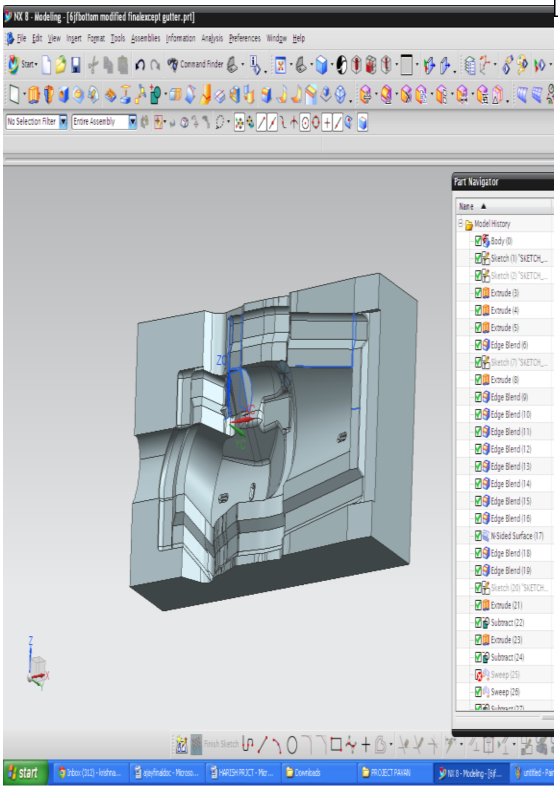


Figure shows the design of rectified bottom die with metal flow gutter for manufacturing with model tree.

Meshing and analysis

Material properties:

Material- D6

Chemical Composition

The chemical composition of D6 tool steels is outlined in the following table.

Element	Content (%)
Iron, Fe	83.05
Chromium, Cr	12.5
Carbon, C	2.05
Tungsten, W	1.3
Manganese, Mn	0.8
Silicon, Si	0.3

Physical Properties

The following table shows the physical properties of D6 tool steels.

Properties	Metric
Density	7.67 g/cm^3

Mechanical Properties

The mechanical properties of D6 tool steels are displayed in the following table.

Properties	Metric
Compressive strength	1320 MPa
Modulus of elasticity	194 GPa
Hardness, Rockwell C	46

Thermal Properties

The thermal properties of D6 tool steels are given in the following table

Properties	Metric
Thermal expansion co-efficient (@21-400°C/68-	10.8 µm/m°

752°F)	
Thermal conductivity (@20°C/68°F)	20

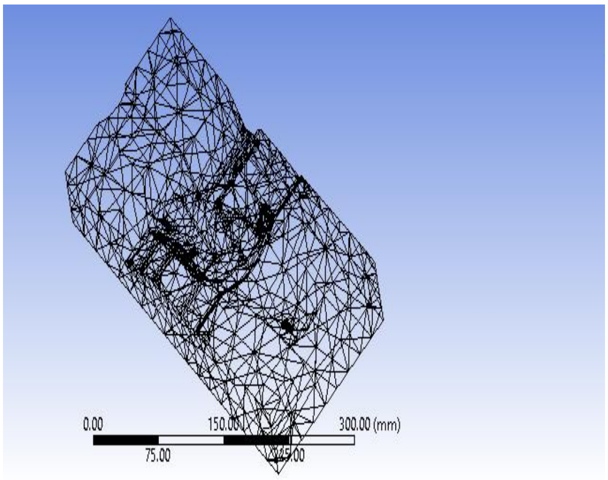
Fabrication and Heat treatment

Forging

D6 tool steel is heated slowly and uniformly to 700°C (1292°F) and then more rapidly to 900-1050°C (1652-1922°F).

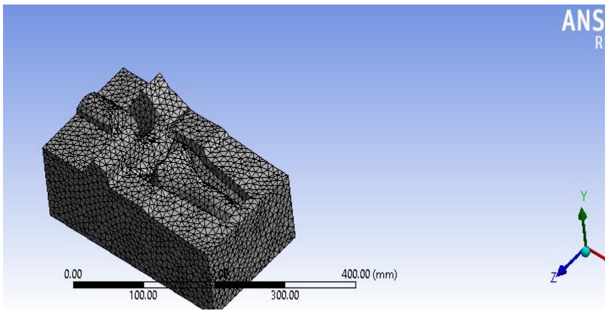
Annealing

D6 tool steel is then annealed at 800-840°C (1472-1544°F) and cooled. Hardness of the steel after annealing will be approximately 225 Brinell.



Hexagonal meshing of bottom die

Figure below shows the meshed component with 9506 Hexagonal elements and 15831 nodes of top die



Hexagonal meshing of Top die for top loading force area by assembling with 1 ton hammer

Input parameters used in the numerical simulation.

Initial temperature-100^0c

Temperature on the die cavity -600^0c

Convection on the surfaces with convective coefficient of air=25w/m^2

Figure shows the boundary conditions used for thermal analyses, convection and temperature was given as shown.

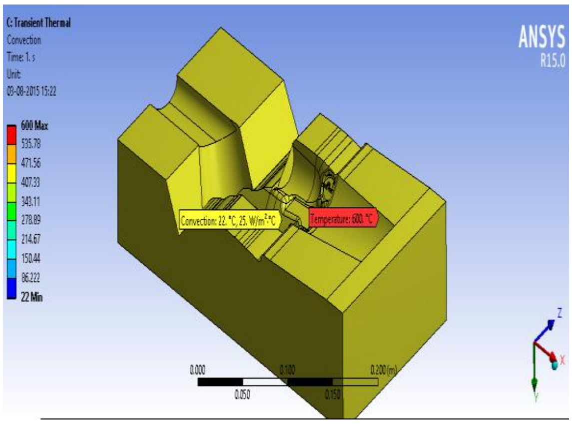


Figure shows the thermal boundary conditions of bottom die

THERMAL ANALYSIS

Figure shows the temperature distribution on the die, minimum of 390^0c was found.

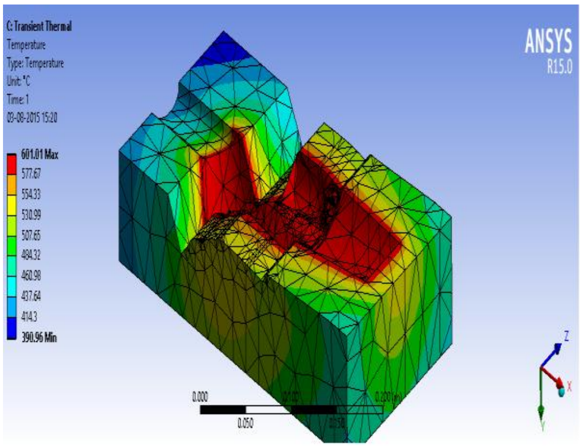
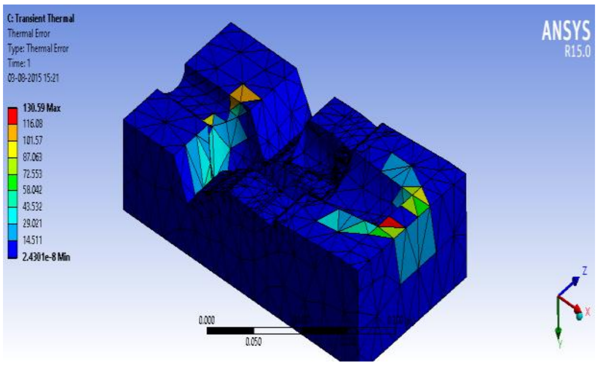


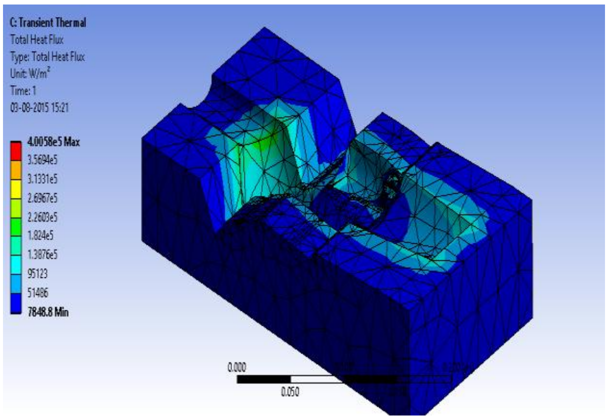
Figure shows Temperature distribution over the die

Figure below shows the thermal error of the die, maximum of 130 was observed



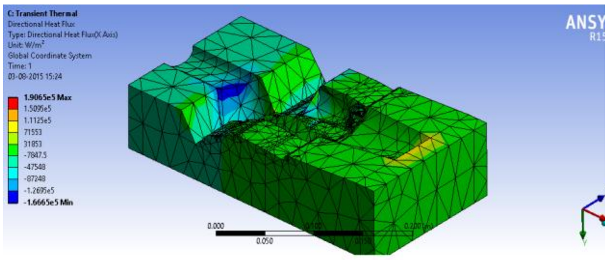
Temperature gradient

Figure shows the Heat flux variation across the die in W/m^2



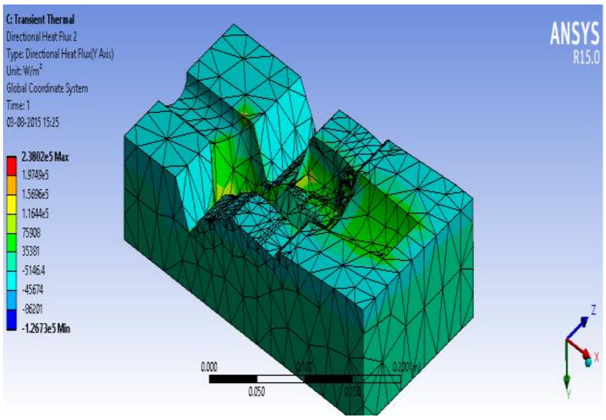
Total Heat flux

The figure shows the variation of heat flux in X direction along the die.



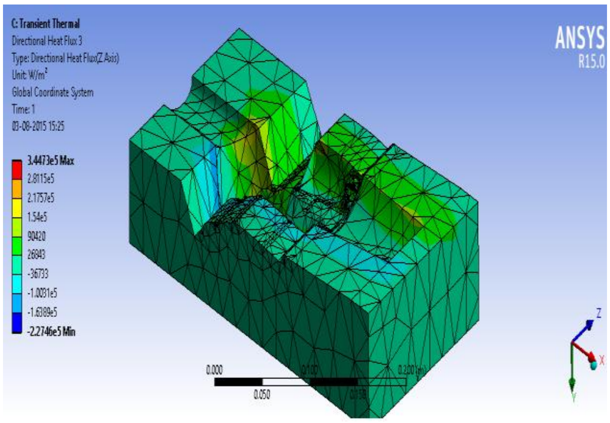
Heat flux along x Axis

The figure shows the variation of heat flux in Y direction along the die.



Heat flux along y Axis

The figure shows the variation of heat flux in Z direction along the die.



Heat flux along z Axis

Discussions: The maximum temperature observed obtained in the die is 601.1^{0}c and the minimum value obtained is 390.9^{0}c at the centre portion. The maximum heat flux obtain will be at tip of the die which is obtained was 130.59 W/m^2 and it is varied at tip and tail where minimum value obtained is 2.430e^{-8}W/m^2. The variation of heat flux in x-direction is at center portion of die is maximum of 4.058e^5 W/m^{3}and is differentially varied all along the section where the minimum value obtained is

78.48 W/m^3.The variation of heat flux in Y-direction is obtained at tip of the die which is maximum value of 1.906e^{5}W/m^{3}and the minimum value is at corner portion which is 1.2673e^5 W/m^3. The variation of heat flux in Z-direction is obtained at tip of the die which is maximum value of 3.477e^5 W/m^3.

CONCLUSIONS:

1.The knowledge of these values is the key to optimizing the existing processes and developing new procedures and materials. This is, however, impossible without verified models, required for a reliable analysis of the process.

2. The first application of the numerical simulation involved constructing a temperature model. It described the temperature changes in closed-die forgings during the heating to the austenitizing temperature before the quenching. Using this model, the heating and soaking times of the forgings in the furnace can be shortened, the optimum layout of the forgings in the furnace can be found and various types of problems solved.

Future scope

It was found that thermo-mechanical treatment can produce practically identical properties of a work piece as conventional hardening. However, such results should be interpreted with caution and this finding should be supported by a larger body of statistical data. In future efforts, the FEM simulation of forming processes will be refined, e.g., the plastic behavior of a forged part instead of the curve plots employed so far.

REFERENCES:

1.0 Ceran et al..,nvestigation of Defect Formation in a 3- Station Closed-Die Forging Operation, CIRP Ann., Vol 381, 1989, p 243
2.0 Garat, Bernhart and Hervy et al.., Thermal Resistances of Pressed Contacts, Appl. Energy, Vol 22, 1986, p 31–84
3.0 Manson.Kim, Lee, Kim, Kim et al..,“Multistage Forging Simulations of Aircraft Components,” Engineering Research Center for Net Shape Manufacturing, ERC/NSM- 02-R-84, 2002
4.0 Brucelle and Bernhart et al.., Metal Forming Fundamentals and Applications, American Society for Metals, 1983
5.0 Dean et al.., “Preforming in Forging and Preparation of Reducer Rolling,” Ph.D. dissertation, University of Hannover, 1959
6.0 Lee and Jouet al.., New Concepts in Die Design—Physical and Computer Modeling Applications, J. Mater. Process. Technol., Vol 98, 2000, p 212–223
7.0 Kim, Yagi, Yamanaka et al..,Forging Equipment, Materials and Practices Vol 8, 2000, p 22–23