Available online at www.sciencedirect.com

ScienceDirect

Materials Today: Proceedings 5 (2018) 7320–7328

materialstoday: PROCEEDINGS

www.materialstoday.com/proceedings

IMME17

Simulation and Analysis of hot forging dies for Pan Head bolt and insert component.

R. Rajiev*[a] Dr. P. sadagopan[b]

[a]*Asst Professor, Dept. of Mechanical Engineering, Bannari Amman Institute of Technology ,Sathyamangalam,638401,Tamilnadu ,*
Professor,Department of Production Engineering , PSG College of Technology, Coimbatore,641004, Tamil Nadu

Abstract

Hot forging is a metal-forming process in which hot metal, in the form of bar stock or cut pieces, is shaped by pressing between impression dies. In high-temperature metal forming processes, die life has an important role on the productivity and the quality of the finished products. In this study wear analysis was carried out in a pan head bolt die in local industry. The simulation of the forging process on the die and the work piece was carried out by using commercially available software (ANSYS). In this paper discussed about to determine the stress distribution during hot forging operation of an asymmetric cross-sectioned in forging dies. The finite element simulation is used to find out the amount of stress occurred in dies that influence of friction on the die wear and the material flow is discussed for this forging component. This paper focuses on the modeling, simulation and analyses of the hot forging die for Pan head bolt and insert component die. The final results are in the form of different stresses distribution occurs in die cavity.

Keywords: Hot forging, Finite Element analysis, Contact Algorthim, Strees distribution, Die wear.

1. Introduction:

Hot metal forming including forging is one of the most important production processes [1]. Forging is a metal-forming process in which hot metal, in the form of bar stock or cut pieces, is shaped by forging between impression

* Corresponding author.
E-mail address: cheryuvaraj@gmail.com

2214-7853

dies. As a process, forging can be characterized by good mechanical properties of the work piece, optimal material utilization, near net shape a short production time, high productivity. These advantages are achieved normally for rather large production quantities because of high costs of tooling as well as long set-up times for production lines [2]. During the forging process, the life of the tools is affected by a complex combination of high mechanical and thermal stresses [3] The main factor influencing the life of tools/dies is the damage caused by wear, fatigue and plastic deformation, in which wear is the predominant factor [4-5]. Increasing the die life can be achieved by a combination of optimal process conditions as well as materials improvement [6-7].This is partly due to the benefits of the process in which mass-produced complex parts can be achieved. The production cost to a large extent depends on the tool cost. The former originates from repeated impact and high-pressure flow of metal. It is reported that while forging brass work piece, thermal stresses are generated in dies at a temperature generally between 200ºC and 300º C due to cyclic contact between die and work piece [8]. High work piece temperatures and high contact pressures during the forging process lead to large mechanical, thermal softening, wear and plastic deformation of dies [9]. Proper selection of the die material and of the die manufacturing technique determines, to a large extent, the useful life of forming dies. Dies may have to be replaced for a number of reasons, such as changes in dimensions due to wear or plastic deformation, deterioration of the surface finish, breakdown of lubrication, and cracking or breakage. Many researchers have investigated the influences of process conditions on die service life during metal forming process [10-11]. By the 1990s with staggering enhancement of computing power, finite element plasticity based methods had gained sufficient ground to result in the emergence of several forging-specific simulation packages. The user interface of these packages continues to improve as does the take-up by the forging industry. In this paper, the analyses of die wear are focused and the results obtained from the computer simulation using finite-element-analysis (FEA) is an effective tool to predict die fill, residual stresses and forming forces. In addition, the FEA plays a decisive role in construction and optimization of forging tools. In order to analyze different stresses distribution occurs in die cavity in the forging tools the considered forging process was represented by a finite-element (FE) model.

2. Methodology

2.1. Case Study 1:

Wear in a die depends on properties like surface hardness, surface finish, friction coefficient, lubrication, temperature etc. Forging wear is a complex phenomenon and takes place during forging as well as during ejection of the component from the die. Forging wear is a complex phenomenon and takes place during forging as well as during ejection of the component from the die In this paper, the analysis has been done on the hot forging die used for manufacturing Pan head bolt billet (SS316) used in door locker. Before starting the pan head bolt forging process, the dies are preheated to 150°C initially to prevent die failure due to thermal stress. Billets and dies are similar to the one shown in Fig. 2.1 are heated to temperature of 1100°C. Using screw forging press of 100 Ton capacity, the pan head bolt (SS316) is forged as shown in Fig.2.1.a and Fig 2.1.

2.2. Case Study 2:

In this paper, the analysis has been done on the hot forging die used for manufacturing insert shown in Fig.2.2 used in railways components .Before starting the insert forging process, the dies are preheated to 150°C initially to prevent die failure due to thermal stress are shown in Fig. 2.3 (a) and Fig. 2.3 (b) Billets are heated to temperature of 900°C. Using screw forging press of 100 Ton capacity, the insert component is forged as shown in Fig. 2.4.a and Fig. 2.4. b.

3. FEA and contact Algorthim:

In order to establish the factor that contributes for die failure, modeling of the components were done to find the component stress distribution, temperature distribution and velocity of metal flow. Simulation and analysis were carried out using the commercially available software (ANSYS). Nonlinearity due to contact conditions arises