IN THE NAME OF GOD

Iran Science & Technology University

Stress Analysis and Fatigue Life Estimation of a Cold Forging Die

By

Valiallah Rahmani

Supervisor: Dr. F. Javanroodi

February 2006

Acknowledgements

I am grateful to Dr. Javanroodi, my academic advisor, for putting forth invaluable information about this thesis and Dr. Ayatollahi and Dr. Abrinia and Dr. Aleiha for their practical support.

I wish to express my sincere thanks to my family for their dedicated support and encouragement to my life.

Contents

1. Introduction

2. Forging process

3. Simulating the cold forging process

4. Fatigue life estimation models

5. Life estimation of the tooling

6. Conclusions

Appendix

Chapter 1

INTRODUCTION

1.1 Introduction

Metal forming is one of the most widely used fabrication processes in industry. Metal forming has experienced scientific progresses that caused production of complex parts. Metal forming processes can be classified into two groups: sheet metal forming processes and bulk metal forming processes. Unlike sheet metal forming, where the enormity of stresses is not a dominant factor to cause failure, in bulk forming dies, since the die is subject to the high stress levels, the greatness of stresses can be a significant factor to die failure. Forging is one of the most common bulk forming processes used to fabricate components with different shapes. Safe life of a die and how long a die can be employed safely and reliably is a significant issue that should be considered and many studies are dedicated to determine the safe life of the die and its failure. Die failure due to fatigue is one of the most important reasons that have been deliberated in many researches. Among forging die failure modes, about 46% of die failure is fatigue fracture [1]. When a die is developed, it’s not clear how many parts the die can produce and when the die fails.

1.2 Abstract

This thesis concentrates on the fatigue failure of a cold forging tooling (die and punch) using ABAQUS package based on finite element analysis. Fatigue failure plays a dominant role in stoppage time of the forging die due to high cyclic stresses. Life analysis was investigated for two states. At first, the tooling life or the number of cycles was calculated until a crack initiates in the critical point standing the highest cyclic stresses using S-N curve of the tooling material. Due to axisymmetry, a two-dimensional axisymmetric analysis was carried out to calculate the stress distribution in the tooling. The effect of mean stress was considered in multiaxial fatigue life and validated models such as Goodman, Gerber and Smith-Watson-Topper and etc were used and compared to each other. To simulate the forging process and study the strain rate sensitivity of the material, the true stress-strain curve of the forged material at the different strain rates was needed. Therefore, five tensile tests were carried out according to ASTM standard E8 at different strain rates. Consequently, fatigue crack propagation was studied and the linear elastic fracture mechanics approach (LEFM) and Paris law were utilized to compute the number of cycles before die failure during fatigue crack growth.

1.3 Literature survey

In recent years many studies have been concentrated on the simulation of metal forming processes using numerical methods especially finite element technique. Also many researches have been focused on the stress and strain analysis of metal forming dies and also some works are devoted for investigation of die failure modes and estimation of die life. Some studies that have been conducted earlier are summarized:

2D and 3D finite element analysis of a three stage forging sequence used to form a spline shape on the head of an aerospace fastener were conducted by C. Cormack and J. Monaghan. The workpiece material was Ti–6AL–4V having a yield stress of 1300 MPa at room temperature. The complicated process was a combination of extrusion (forward and backward) and forging of a very complex spline geometry. The first two stages of the forging operation were modeled using 2D finite element analysis because of the symmetry in the workpiece geometry. Between stages two and three, the stresses within the 2D workpiece were imported into a 3D model to enable the simulation of the spline formation. The authors used the DEFORM software to simulate the process which has the capability to convert a 2D model into a 3D one by using a few simple Dos command. One of the fundamental points about modeling this process was that the residual stresses within the workpiece after each station were conserved as they will affect the flow characteristics of the material due to strain hardening. The automatic remeshing technique in DEFORM package used to improve the quality of meshes that enables continues simulation of a forming process without any interface by the user, even if several remeshes were required. To consider damage of the workpiece, the damage distribution that is the critical damage value “C” in Cockfoft and Latham’s ductile fracture criterion was obtained and the critical damage value ‘C’ gave an indication as to areas within the workpiece of possible fracture problems. The authors finally concluded that using the powerful 2D to 3D conversion within DEFORM enabled the first two axisymmetric stages of the forming process to be modeled in 2D, which reduced computational time, allowing the final and more complex stage 3 to be modeled in 3D. [2]

The authors also analyzed a cold forging process using finite element analysis to produce a hexagonal shape on the head of a bolt. The process was a combination of cutting and forging. The main focus of the analysis was to predict the level of the stresses within the die material. The analysis consisted of creating a different die geometries called “standard trim die “ using AutoCAD/Mechanical desktop and importing them into the finite element analysis package DEFORM. The trimming of the bolt head was considered as an axisymmetric problem. This assumption was not completely correct due to the hexagonal hole within the trim die. The reason of considering the problem as axisymmetrical was a necessary first step in predicting stresses. The finite element analysis indicated that the highest stress concentration occurred within the body of the tool and not along the contact surfaces. The authors investigated the effect of some parameters such as land width, petal angle, rake angle and friction on the process and finally suggested values for die trim parameters to reduce the stress levels within the tooling during the forging process. [3]

Y. Park and J. S. Colton investigated the failure of V-bending dies fabricated from an easy-to-machine, polyurethane-based, composite stock. Several failure criteria were proposed to predict die failure mode and the corresponding die life. Both computational and experimental methods were employed to assess the accuracy of the criteria and to identify the dominant process parameters in V-die bending. Based on the low-ductility nature of the tooling material they found that the local stress-based fatigue approach was appropriate for die failure mode and life predictions. The damage analysis of V-bending dies involved the stress–strain analysis of the forming process using the FEA and applying the damage parameters to the materials data to predict die failure mode and life. The experimental results showed that fracture due to overload and fatigue is the competing die failure modes in V-die bending. The normal stress on the maximum shear plane and the Smith–Watson–Topper stress provided the most accurate fatigue life prediction, the latter being more conservative; the maximum tensile principal stress provided suitable fracture prediction. A parametric study revealed that over traveling of the punch was the most significant parameter, followed by bend radius, sheet thickness and strength. The authors used the results as guidelines for V-bending die design for low-ductility, powder-filled, polymer composite tooling materials. The authors utilized maximum principle stress, equivalent Von Mises stress and SWT approaches to estimate the die life and finally concluded that the SWT parameter provides the most accurate fatigue life prediction. [4]

The authors also considered the failure modes of a cylindrical cup drawing tooling machined from a polymer composite rapid tooling using finite element method and experiments. The authors stated that the stress states in the die, which govern the failure, were not intuitive, and thus require computational simulations. The simulations were performed by constructing a FE model and obtaining the stress–strain responses. They obtained that possible failure modes in cylindrical cup drawing dies are fracture, wear, and plastic deformation.

The damage parameters used were the maximum tensile principal stress for fracture and the maximum normal stress on the die surface for wear. The authors showed that plastic deformation occurs primarily due to the wrinkles in the sheet metal when no blank holder is used and they concluded that the drawing die fails when the maximum principal stress reaches the flexural strength of the die material. Another important observation was that the peak in the drawing force curve did not correspond to the maximum stress, which necessitates the FEA of the process prior to selecting process parameters. A statistical analysis of the two-level fractional factorial design showed that sheet strength and thickness were the most dominant parameters, followed by draw ratio, punch–die clearance, and run-off. The experimental study showed that the punch corner radius must be selected carefully to prevent premature plastic deformation failure of the die. Finally, the authors concluded that this study provided

- A method to predict the failure mode of a cylindrical cup drawing die

- The die design guidelines that can be employed in the preliminary design stage.
- The valid damage parameters for various die failure modes based on the underlying failure mechanisms. [5]

M. W. Fu and K. K. Tong presented the S-N approach for die life estimation by employing the FEM and metal fatigue theory. They stated that fatigue failure starts from crack initiation, propagation to finally not being able to produce quality products. In fatigue failure, brittle fatigue is the most common fatigue mode in cold forging die. In forming process, the preheating of die and billet, high forming pressure in the die and the unbalance force between the die and billet can lead to the deformation or deflection of die. For the simulation of the deforming body the authors used the varitional approach which is the basis of the plastic FEM to formulating functions and getting the velocity or displacement solution. To linearize the functions, the Newton-Raphon approach or direct iteration method can be used. In Newton-Raphon approach, the initial value for all the node velocity is pregiven. In iteration process, if the velocity and force meet certain criterion, the iteration of the specific loading step is considered as convergence. For analysis of die stress, the authors used the elastic FEM approach. The Haigh diagram of the tooling was obtained using Goodman equation to compute the die life when non-zero mean stress conditions occur. In Haigh diagram, X-axis is the mean stress, while Y-axis is the amplitude stress as shown in Fig.1.1. To generate a Haigh diagram, there are two critical lines. The former can be constructed by connecting the ultimate strength in X-axis and the endurance limit in Y-axis. The latter can be generated by connecting the ultimate strength in X-axis and the amplitude stress of 10^3 cycles in Y-axis. The 10^6 line is usually considered as the infinite line, while the 10^3 line is the line between the low cycle and high-cycle fatigue regions.

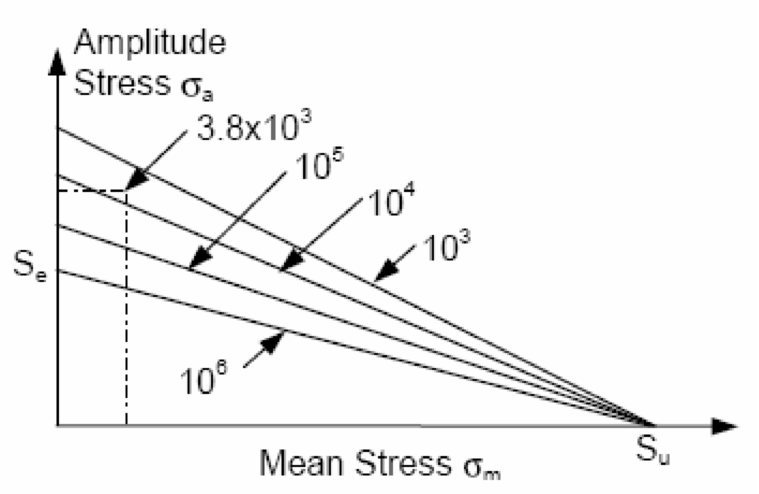

Fig.1.1 Haigh diagram

The authors used the maximum principle stress and Goodman equation to calculate the alternating and mean stresses. The simulation showed that the critical regions are located at the teeth of the die insert. The die life using simulation and Haigh diagram was obtained 4500 cycles. [1]

The authors also analyzed a cold forging circular die insert using finite element based ANSYS/ LSDYNA and DEFORM simulation softwares. Since the die insert life was important they attempted to determine the stress distribution in the die insert and provide a systematic study to reduce the stress levels and thereby increasing the tool life. The fracture was predominantly caused by low cycle fatigue, and they intended to recommend various die design combinations to improve the fatigue life of the die. The authors suggested the following procedures outline the steps in the verification process and to finding out the stress distribution and high stress regions in the die insert, and recommending die life improvement methods using FEA.

Verification of FEA results using forward and backward extrusions and a die ring:

To verify the 2D simulation results, the working loads and pressures required for forward and backward extrusions were determined using the ANSYS/ LSDYNA software. The results obtained from FEM were then compared with experimental results from the literature to ensure that the former was accurate.

To verify the 3D simulation results, a die ring being subjected to side wall and reinforcement pressure was used. The resultant hoop stresses at the inner and outer surfaces of the die ring due to the side wall and reinforcement pressures were obtained using analytical equations. Hoop stresses calculated using FEM were then compared to the theoretical results from calculations to ensure that discrepancy is minimal. The material properties were used to define the power law behavior ($\sigma = K\varepsilon^n$) of the simulated die insert. Prestressing of the die was carried out by applying a pressure on the outer surface of the die insert to generate a compressive hoop stress in the die insert. This pressure, which was 300 MPa, was due to the interference mismatch between the mating surfaces of the die insert and the die, and this pressure will result in a compressive hoop stress in the die insert.

The entire forging operation including the punch, die and workpiece was simulated in the DEFORM simulation software. The minimum principal stress in the die insert after reinforcement prestressing and the maximum principal stress after application of the working load were evaluated. The prestress changes were suggested to improve the die insert life by lowering the maximum principal stress values. The reduction of detrimental principal stress in the die insert led to the improvement in fatigue tool life. The authors concluded that the FEA results were very similar to those from the experiments and the slight discrepancy might be due to factors such as different state of the same material (annealed, cold worked etc.), and different friction coefficient. They proved that imposing a high reinforcement prestresses will result in a high compressive minimum principal stress before the working load was applied. This high compressive principal stress will counteract the tensile stress due to the working load and result in a comparatively lower maximum tensile principal stress. [6]

V. Vazquez, D. Hannan and T. Altan summarized an investigation of different alternatives to improve the life of a tungsten carbide insert used in a cold forming operation performed on an automatic cold header. To improve the life of the insert, the metal flow and stress analysis of the carbide insert were conducted using the commercial FEM software DEFORM. The resistance of carbide to fail by fatigue is greatly reduced if there are tensile stresses present in any portion of the tool during the forging cycle. Therefore, it is important to free the carbide tooling of tensile stresses or at least lower them to the minimum value. The authors showed that the cause of failure was due to the tensile maximum principal stresses near the transition radius in the tooling. The conclusion of the work was:

-Use FEM software to prevent any flow related defects that may affect the performance of either the forging operation or the product.

- Once flow related problems were solved, perform a stress analysis on the tool to determine possible failure locations.
- Change the design of the tooling based on the stress analysis to minimize the possibility of failure.
- Perform stress analysis of the new design and compare with original design.
- If the execution of the new design is not economically possible select a more ductile tool material. This may increase the tool life. [7]

Chapter 2

Forging process

2.1 Introduction

Forging, is a forming process in which a part is deformed plastically between two tools (or dies) to achieve the desired shape. A simple forging process has been shown in Fig. 2.1. In forging processes the production is fabricated in a very short time. Therefore, forging causes potential savings in energy and material, especially in medium and large production quantities. Besides, for a given weight the parts manufactured by forging reveal better mechanical and metallurgical properties and reliability than do those produced by casting or machining.

Forging is an experience-oriented technology and throughout the years, good experiences have been accumulated in forging process, mostly by trial-and-error techniques. Nevertheless, the forging industry has supported products that are manufactured from developed, difficult-to-form alloys. The physical phenomena describing a forging operation are difficult to express with quantitative relationships.

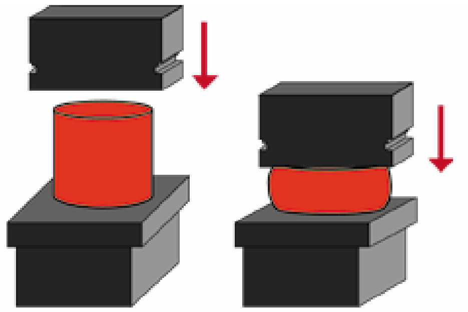

Fig. 2.1 A simple forging process

The metal flow, the friction at the tool/material interface, the heat generation and transfer during plastic flow, and the relationship between microstructure properties and process conditions are difficult to predict and analyze. There are many different parameters influencing the forging process and final shape.

2.2 Types of forging processes

There are a large number of forging processes that can be summarized as following:

Closed/impression die forging with flash- Closed/impression die forging without flash- Radial forging – Hobbing- Isothermal forging - Open-die forging - Powder metal (P/M) forging- Upsetting - Nosing – Coining and etc.[8]

Types of forging based on temperature:

- Cold forging
- Warm forging
- Hot forging

In cold working the recrystallisation rate is so low that recrystallisation essentially does not occur during forging. Recrystallisation is subsequently achieved statically by annealing.

Warm forging has a number of cost-saving advantages which emphasize its increasing use as a manufacturing method. The temperature range for the warm forging of steel is from above room temperature to below the recrystallization temperature.

Hot working refers to the high temperature plastic deformation of metal in which recrystallisation occurs simultaneously with deformation, thus avoiding strain hardening. [9, 10]

2.3 Variables in forging process

In forging, the main objective of the process is to construct a good production with considered properties. Many variables control the metal flow and ultimate shape of the production including:

-Billet: Flow stress - thermal and physical properties- Initial conditions- Plastic anisotropy- Billet size and thickness and etc.

-Tooling/Dies: Tool geometry- mechanical and thermal properties under conditions of use- hardness- temperature and etc.

-Deformation zone:

The mechanics of deformation- model used for analysis- metal flow-velocities-strain- strain rate- stresses (variation during deformation) and etc.