International Journal of Scientific Engineering and Research (IJSER)
ISSN (Online): 2347-3878
Index Copernicus Value (2015): 62.86 | Impact Factor (2015): 3.791

Simulation and Analysis of Hot Forging Process for Automotive Coupling Flange

Manoj Kumar[1], Nippu Tayeng[2]

[1]Asso. Professor, Dept of Forge Technology, National Institute of Foundry and Forge Technology, Ranchi, India
[2]M.Tech Student, National Institute of Foundry and Forge Technology Ranchi, India

Abstract: ***In this paper hot forging process for industrial automotive coupling flange is simulated and analyzed. An increase in demand of industrial coupling flange (mass produced component) with better quality and lower price are produced by hot forging process. Production of industrial coupling flange by means of hot forging process is affected by many parameters such as billet temperature, geometry of die and geometry of pre-formed billet. In this study the influences of billet temperature on effective plastic strain, radius of die corners on internal stress of billet and thickness of flash on required force of press are investigated by means of computer simulation. Three-dimensional modeling of initial material and die are performed by SolidWorks 2014, while simulation and analysis of forging are performed by Deform 3D. Based on the computer simulation the required dies are designed.***

Keywords: Coupling Flange, Finite Element method-Computer simulation, Forging, Die design

.

1. Introduction

Forging process is one of the metal forming processes where desired size and shape is obtained by using compressive loads with the help of hammers or presses. Many of the industrial parts are produced by this method because of its high strength and production rate of parts. Production cost and increasing part quality are affected by many forging parameters like temperature, geometry of raw material and die design parameters such as fillet and corner radii, flash design etc[1]. If the design and manufacturing of parts are on basis of experience that results in time waste and high cost. The prediction of material flow can be achieved completely by computer simulation. Main parameters in computer simulation are filling the die completely without leaving any defect, reducing material loss and stress in die and increasing die life. In order to reduce the cost of forging process and make it competent with other production methods, it is essential to optimize the design of part and die. Moreover, considering required forces, energy and time in such a way to attain an accurate analysis and design could be highly important. Therefore, using of the accurate computer simulation can be an effective method for reaching these goals. Forging of complex parts is usually performed in several steps. Design of step numbers, step types and intermediate steps (pre-forms) along with the design of final die, forging parameters and process conditions are the most complex tasks in forging simulation. Generally, application of computer to design the required steps based on analytical and experimental data has been considered excessively by other researches previously. There is much simulation software which is used in forging process. Some are based on finite element methods while other are based on finite volume method. Using these tools, one can build a virtual model of dies and defines different materials characteristics in order to simulate the forging process precisely and to get valuable information about how the material is being formed in the die. Thus it can be guaranteed that the parts have desired shapes before making the real die. In addition, using the results of performed numerical analysis many of the important design parameters such as forming forces, stresses on the part and die and also plastic energy are determined. Because of the functions and capabilities of this method in simulating the process, the manufactures know this method as the so-called virtual production. Most of computer aided research studies about the forging process were based on finite element method. Thus, today this method is being replaced with the finite volume method (FVM) [1]. Several researches have been performed on design of forging dies. Behrens and colleagues have applied Super Forge to design the vehicle's propeller shaft [2]. To analyze the bending deformations of thick and thin plates, Wheel and his colleagues have used FVM [3]. Rranatunga et al have applied FVM method to design the furrow of die in a closed die forging [4]. Eivazi made a software package called Auto Forge in order to automate the hot forging process of round parts [5]. Sanaie and Jafari compared closed die forging for two parts based on three methods: analytical method (Qachy method), numerical method (ANSYS), and experimental method. Some analytical and numerical methods predict more or less load compared to experimental method, respectively [6]. In this paper, the modeling and analysis of hot forging process for a coupling flange has been investigated by Deform 3D and the deformation mechanism is investigated in detail.

2. Materials and Method

First step in design of die for the coupling flange is to define a three dimensional model of the workpiece in the SolidWorks 2014, which is a modeling software. Then, the initial model will be transformed to the final shape of the workpiece. Materials of workpiece and die are steel AISI-1045 and steel AISI-H-13 respectively. Forging process for producing coupling flange is performed in two stages. In the first stage (pre-form) the workpiece will achieve its suitable diameter by reducing its height. While, in the second stage the workpiece will obtain its final shape after trimming.. The input data used here is listed in table 1 including type of press, materials of workpiece and die.

Table 1: Used Input data

Billet Mat	Die Mat	Die Temp	Ambient Temp	Press
AISI 1045	AISI-H-13	200^0C	20^0C	Crank type

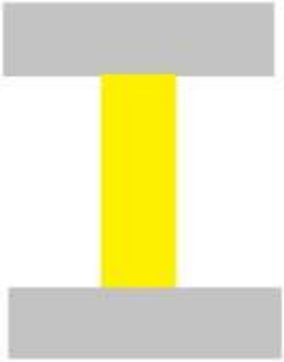


Figure 1: Locations the billet inside the die to reduce the height

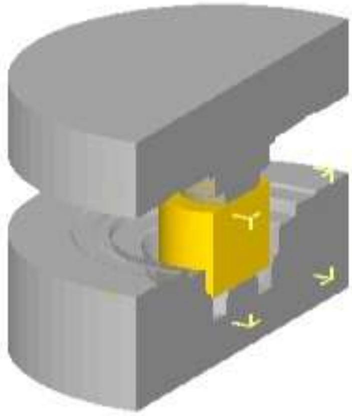
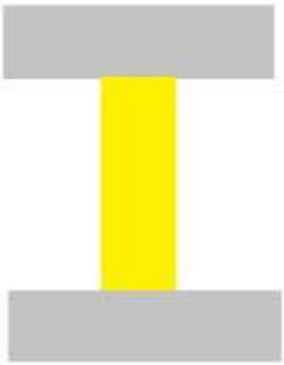


Figure 2: Workpiece in the die.



3. Forging Simulation by Using Pre-Form Die

After converting the machine drawing of coupling flange into forging drawing and designing flash with the help of empirical relations a two stages forging process was designed. The first stage is a pre-form including height reduction. The original cut billet is upset and then in the second stage this upset billet is placed in the final die to form its final shape. Figure 1 illustrates the position of the workpiece within the die Figure 2 shows the position of the upset workpiece within the die, while Figure 3 illustrates the forging stages. The simulation results indicate that metal flows as intended and completely fill the die cavities. Finally, after trimming the forging will be ready for machining.

Addition of a pre-form stage to forging process will ease the forging of billet in the final die. In the pre-form stage the workpiece will achieve the suitable diameter based on the diameter of final die. Using pre-form stage following advantages are obtained [7]

1. Obtaining the desired diameter.
2. Scale removal of initial billet.
3. Proper material flow in the final die

The height and diameter of initial billet are taken as 87mm and 68mm, respectively. After pre-form stage, the height and diameter is reduced and increased to 82mm and 70mm, respectively. As it is mentioned, the scale removal is one of the pre-form benefits. It will take away the oxide layer from the surface of initial materials which will improve the surface quality of workpiece. In fact, the height reduction stage will cause the billet fiber to approach toward the final fiber structure of workpiece which facilitates the flow of materials in the final die.

In the simulation stage, the initial billet will be the input of height reduction stage and its output, in turn, will be the input of the second stage (final forming). Figure 1 illustrates the position of initial billet in the stage of height reduction.

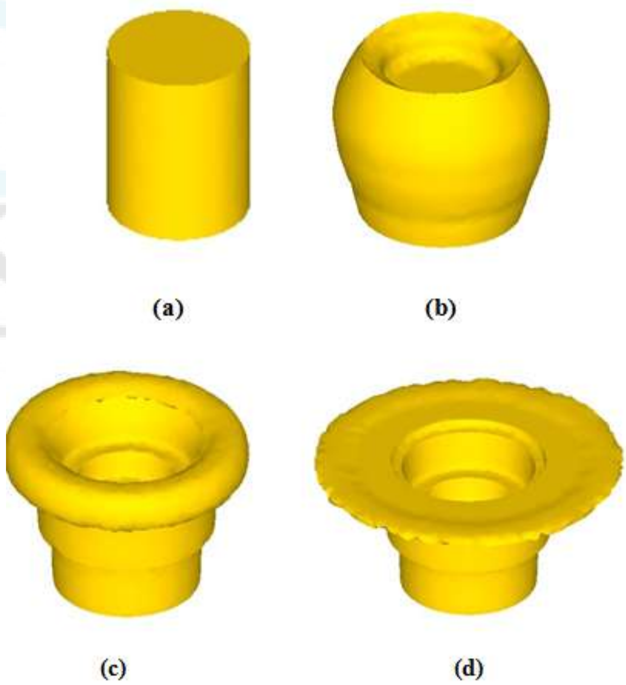


Figure 3: Workpiece geometry in process simulation (a) billet after upset, (b) after 40% change in the substance, (c) after 80% change in the substance, (d) after 100% change in the substance.

4. Simulation of Final Forming Stage

In the final stage, the obtained workpiece from the height reduction stage is placed in the final die, Figure 2. Because the diameter of pre-formed workpiece is fit for the final die, it can be easily placed inside it. Figure 3 depicts the forging process stages for a coupling flange. After initial height reduction process, the work piece will be placed in the final die for final forming process. Then and based on the suitable width of the flash, the trimming process can be performed easily.

5. Results and Discussion

In this part the basic variables which affect the forging process such as flash thickness, temperature and die corner radius are investigated. Then, based on the most appropriate values of temperature, die radius and flash thickness was chosen for the purpose of die production.

5.1 Effect of temperature

The purpose of this part is to monitor the effect of temperature on the effective plastic strain during the final forging stage. In this test, a pre-formed workpieces with temperatures of 900, 1000, 1100 and 1200 ° C are placed inside the final die and the maximum effective plastic strains at various temperatures are obtained, Figure 4. The distribution of effective plastic strain at different temperatures is illustrated in Figure 5. When the temperature goes beyond a specific value, the surface quality of the workpiece decreases seriously because of the formation of oxide layers [8]. In this study, the temperature value upto 1200 ° C, was selected for simulation.

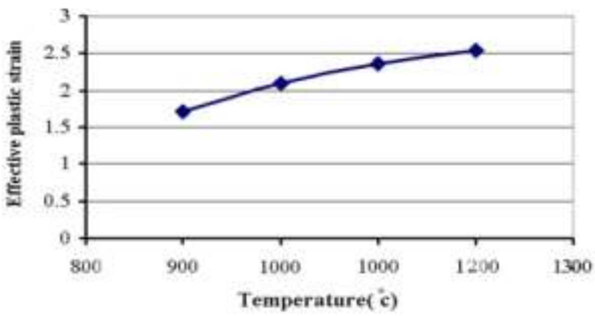


Figure 4: Effect of temperature on the maximum effective plastic strain

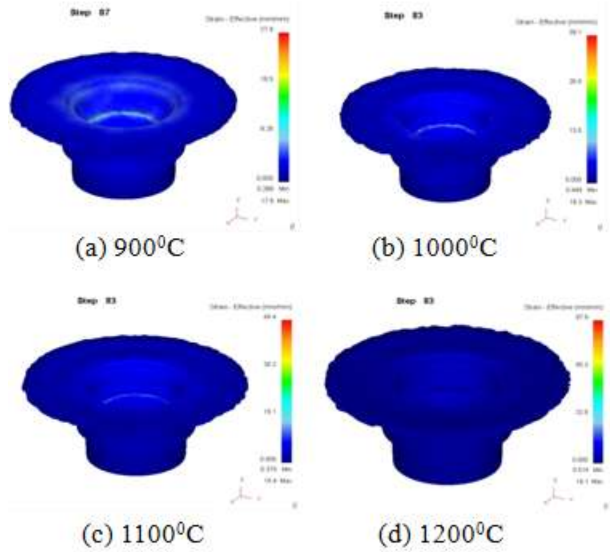


Figure 5: Effect of temperature on effective plastic strain

5.2 Effect of die corner radius

The effect of die corner radius on the effective stress in the work piece is investigated here. For the selected radiuses (2, 2.5 and 3 mm), the maximum effective stresses are obtained, which is shown in Figure 6. 3D models of dies with different radiuses were designed in Solidworks 2014. Then the obtained results are transferred to the simulation software. The minimum and maximum amounts of effective stresses are achieved at die corner radiuses of 3 and 2 mm, respectively. The main reason for this observation is the easier flow of metal inside the die. Therefore, in this study radius of 3 mm was selected for the design of final die. Von mises stress distribution at different radiuses of die is shown in Figure 7.

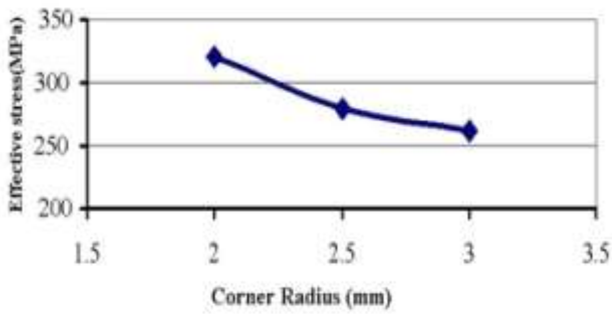
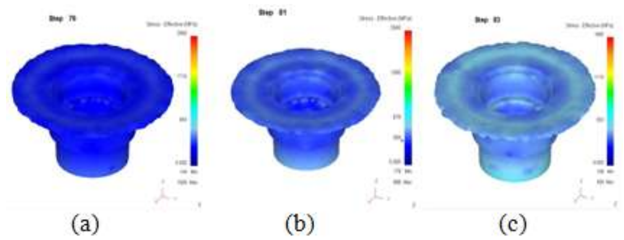


Figure 6: Variation of the effective stress with die corners radius



(a) Effective stress r=3(mm)
(b) Effective stress r=2.5(mm)
(c) Effective stress r=2(mm)

Figure 7: The effect of die corner radius on the workpiece stress.

5.3 Thickness of flash

The purpose of this test is to evaluate the effect of flash thickness on the press force. For the selected flash thicknesses (1.8, 2, 2.4 and 3 mm), the required press forces are obtained which is shown in Figure 8. It has to be pointed out that the flash thickness is directly related to the amount of waste material. Thus, an optimal flash thickness should be chosen based on the capacity of press, life time of die and the number of products. According to Figure 9, the press force increases significantly during the formation of flash. Hence, the smaller flash thickness will exert more pressure to the die and may cause break. On the other hand, the bigger flash thickness will cause more waste material which in turn increases the production cost. Hence, the designer has to find the most appropriate thickness of flash by considering the factors such as workpiece and die materials and the required number of work pieces [9]. The obtained results show that the maximum and minimum press loads belong to the flash thickness of 1.8 mm and 3 mm, respectively. In this study, considering the number of produced workpieces, their materials, die and press capacity, the flash thickness was taken as 3 mm.

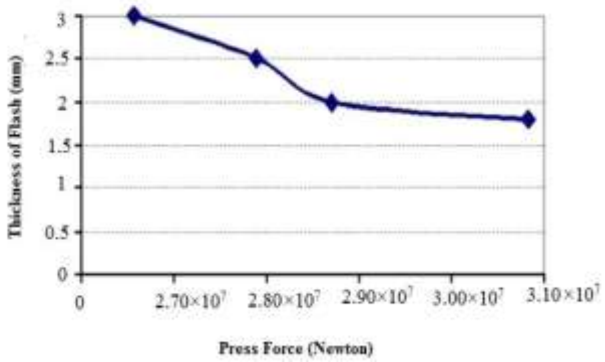


Figure 8: Variation of flash thickness with press force

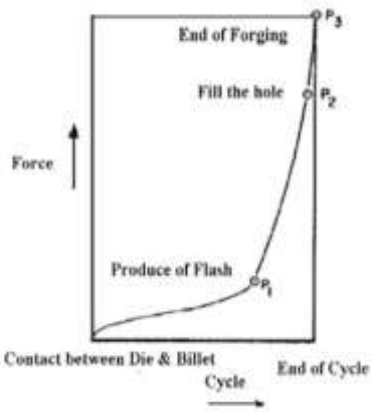


Figure 9: The Curve of force-cycle in forging

6. Conclusion

Hot forging process for industrial coupling flange has been simulated and analyzed by Deform 3D software. The simulation of forging process was performed in two stages, in the first stage the height reduction process was applied to the workpiece and in the second stage the final forming was performed. Using a computer simulation the optimal design parameters such as flash thickness, temperature and die corner radiuses were selected. The obtained results show the great potential of Deform 3D methods for simulation of complicated industrial forging process. This will reduce the cost of die manufacturing as well as by selecting suitable temperature forging defects can be minimised and load on the die is also reduced which will result in higher die life. The following results have been achieved during the current study.

- The effective plastic strain will increase with rising of temperature. However, if the temperature exceeds a certain limit, the work piece will failure, and the final product will be defective..
- The effective stress decreases as radius of corners increases.
- With an increase in the flash thickness, the required press force will decrease. The flash thickness is directly related to the amount of waste material. Thus the most favourable flash thickness should be designed based on the capacity of press, life time of die and number of products.
- For manufactuing a die based on simulation result above values will be used and accordingly die will be manufactured and subsequently forging will be produced. Validation of simulation result will be made with experimental results.

References

[1] Product Design Guide for Hot Forging. Forging Industry Association (F.I.A.).www.forging.org, 2002.

[2] A. Behrens, H. Schastall, “2D and 3D Simulation of Complex Multistage Process by Use of Adaptive Friction Coefficient”, Journal of Materials Processing Technology, Volumes **80-81**, 1 August 1998, pp. 298-303

[3] M.A. Wheel, “A Finite Volume Method for Analyzing the Bending Deformation of Thick and Thin Plates”, *Computer Method in Applied Mechanics and Engineering*, Volume **147**, Issues 1-2, 30 July 1997, pp. 199-208.

[4] V. Ranatunga, J.S. Gunasecra, W.G Fraizer, K.D. Hur, “Use of UBET for design of flash gap in closed-die forging”, *Journal of Materials Processing Technology*, Volume **111**, Issues 1-3, 25 April 2001, pp. 107-112.

[5] P. Eivazi, “Automatically Formats Designed to Help Round Forging Computer Parts”, *The Second Iranian Annual Conference of Mechanical Engineering*, May 1994, pp. 153-162.

[6] F.F. Saniee, M. Jaafari, “Analytical, numerical and experimental analyses of the closed-die forging”, *Journal of Materials Processing Technology*, Volumes **125-126**, 9 September 2002, pp. 334-340.

[7] H. Parikh, M. Shaltout, B.V. Mehta and J.S. Gunasekera, “Forging Process Analysis and Perform Design”, *Current Advances in Mechanical Design and Production*, Volume **3**, Cairo, Egypt, Jan. 2004, pp. 117-123.

[8] M. Marumo, S. Sonoi, “Effect of the surface structure on the resistance to plastic deformation of a hot forging tool”, *Journal of Materials Processing Technology*, Volume **113**, Issues 1-3, 15 June 2001, pp. 22-27

[9] M. Maarefdoust and M. Kadkhodayan, “Simulation and analysis of hot forging process for industrial locking gear elevators”, AIP Conference Proceedings **1252**, 903 (2010)