Optimization of Hot Forging Process by Smart Design and Numerical Analysis Method

Turki Diwan Hussein [1, a)], Rasha Abed Alkhider Abed Allah [2,b)], Kamil Jawad[3,c)]

[1,2] (*Southern Technical University/ IRAQ*), [3] *(Al- Furat Al-Awsat Technical University/ IRAQ)..*

[a)] *turki.hussein@stu.edu.iq*
[b)] *suns1980@yahoo.com*
[c)] inm.kml@atu.edu.iq

Abstract. In this time being, the trend in manufacturing process is toward fast and mass productions. Forging process is classified as one of the most important methods to produce a semi and final products with suitable durability. In deformation proses like forging, the problems of high sticking and frictional contact between the workpeice and tools during and after the process need to be solved. Also it's very important to develop analysis and simulation method for optimizing the products and increase the tools life by avoiding and minimize the stresses concentration in some weak points. However; by using F.EM analysis before empirical steps, the design process can be optimize, minimizing the manufacturing errors and tools life can be predicted with time saving. In this research paper; the commercial finite element software ABAQUS/ CAE have been conducted for analysis and simulation this deformation process. This design and manufacturing study including a forging process for a symmetric and cylindrical product with constant wall thickness. The methodology of optimization in this work was included developing the design and manufacturing method to minimize the generation stresses in hot forging products through controlling some parameters and variables. An axis-symmetric model of finite element (FE) was established for was established for modelling and simulation this forging process. As a results from this work; it's found that due to high deflections of tools, dies and machines which caused by the forging forces; the generation stresses and stiffness of tools are the important variables on the overall deformation process. It's concluded from this study that; it's very important to link between the manufacturing considerations and design aspects in any effective optimization. Analysis and simulations results were verified and the optimization presented was carried out.

Keywords: *Forging process; Optimization; ABAQUS; Analysis; Die design; and Metal deformation.*

INTRODUCTION

Nowadays; the tendency towards high quality and accuracy demands of products which produced by forming process make the manufacturing and research centres focus on increasing the stiffness of employed machines and tools [1]. Hot and warm forging with extra cold finishing are used up to date in manufacturing to extract the exact geometry for many precision parts like bevel gears and connecting rods. In order to match these goals; flux of the forces, stress behaviour and high strained areas should be optimize and improved in all components of forging tools components [2].

Many manufacturing process in classical approach is done based on experience which depends on trial and error methods. The revolution in computational technology make the classical approach cannot match with the rapid increasing and development in modern era. For any manufacturing process with highly cost requirements, the trial and error will result in decreasing the efficiency and lost in time [3].

In process planning; the physical and numerical modelling has a very important role regarding to saving of material, energy and reducing of flawless process. Also physical modelling can reduce the necessary costs. It's a powerful tool which can allow investigating the process whatever its complicated they are [4]

Properties and microstructures for many formed products can be predicted by using numerical process simulation. FEM can used for reduce the number of necessary experiments and operations in forging process. By simulate this process it's possible to minimize the error and reduce the cost throughout decreasing the necessary time prior to experiment [5].

Forging process can simulated as two dimensional process, plane-strain, three dimensional or axisymmetric process depending on the product complexit. If the process involving many stages, then each stage should be define separately with different boundary conditions depending the type of interaction in each step [6,7].

Simulation technique can helps the technical designers to visualize the metal flow and provide more insights on manufacturing process. The most attractive branch in F.E simulation technique is that called (auto optimization). There are a lot of aspects in forging manufacturing process need to be optimized like products quality improvement and production costs reduction [8].

The service life of forging tools and the quality of forging parts can be predicted by using finite element analysis. It's an optimal solution and forecasting tool for stresses concentration problems which caused the tools faller. Improving and optimizing the forging quality is require that the deformation should be homogeneous for the material under forging [9].

To optimize any deformation process like forging; the designer can control some parameters and variables used in design and manufacturing process. Optimization includes product quality improvement, cost reduction and tools life [10, 11].

Geometric Properties and Modelling Process

The starting point in this forming process is to define product and workpiece geometry with the other tools for modelling procedure. In closed-die forging; it's very important to define all the starting geometrical properties like dimensions and material specifications. "Figure 1" illustrates the billet dimensions before forging process.

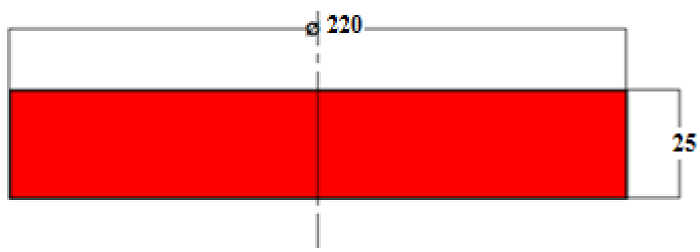

FIGURE 1.. Billet dimensions before forging process

"Figure 2" and "Figure 3" illustrates the product with all dimension after completion the forging process and front sectional view in three dimensions respectively.

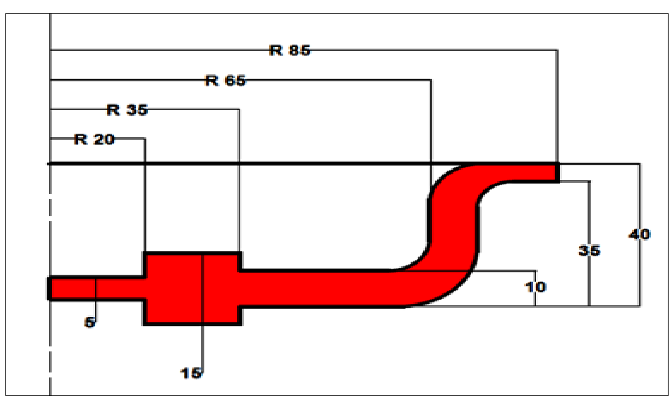

FIGURE 2. Final product with all dimensions.

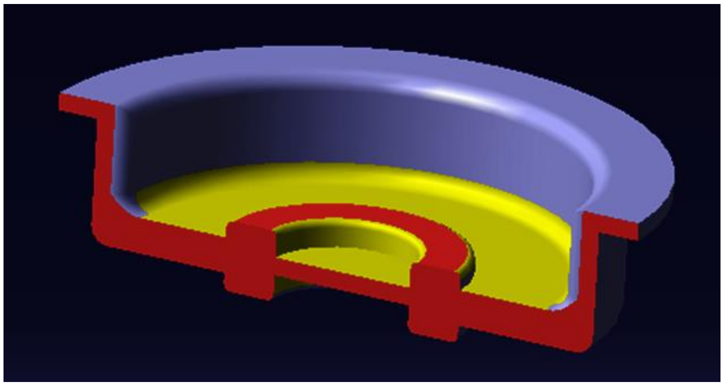

FIGURE 3. Front sectional view of the product in three dimensions.

The mechanical properties of aluminium work piece and the steel material used for punch and die are listed in **TABLE 1.** below.

TABLE 1. Mechanical properties of aluminium and steel used in this work.

Properties	**Values (Steel)**	**Values (Aluminium)**
Young's Modulus of Elasticity	200 Gpa	69 Gpa
Yield Stress	375 Mpa	111 Mpa
Expansion Coff.	0.00012	8.42 E-005
Poisson's Ratio	0.3	0.33
Density	7340 Kg/m3.	2700 Kg/m3.

The main parts of these models are consisting of deformable blank, rigid die and punch. The rounded blank dimensions are (25) mm in thickness and (110) mm in radius. The edge of the blank is rounded to facilitate the starting deformation process.

Modelling technique can show better understanding for this complex deformation process, and then this model can be exported to (ABAQUS) for analysis. For this purpose (CATIA/ R16) software has been adopted for this modelling process. "Figure 4" is a (3D) front sectional view for the assembly model during the last two final forging steps.

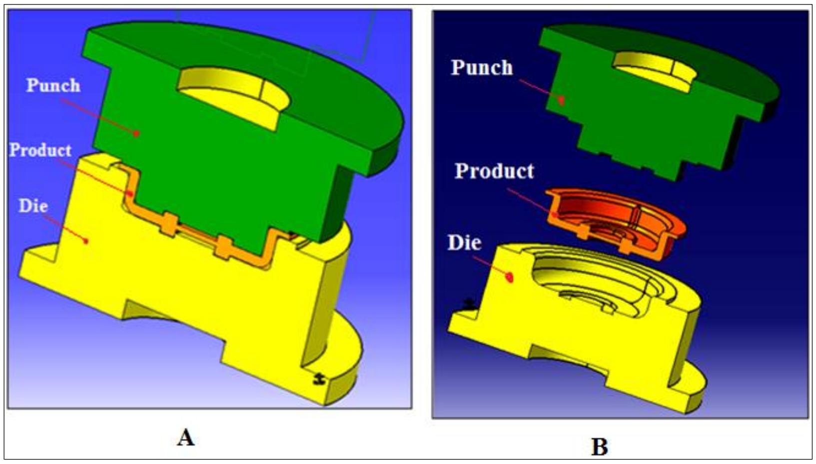

FIGURE 4. Sectional view for tools assembly. (A) during forging; (B) after forging.

Also "Fig. 5"   illustrate a two dimension view for last forging step during the product removal from the die.

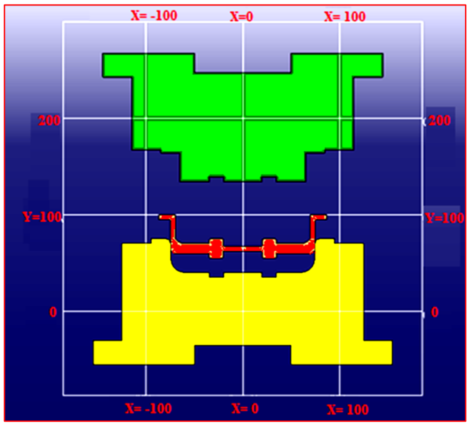

FIGURE 5. Sectional view for tools assembly. (A) during forging; (B) after forging.

Finite Element Simulation

Commercial software ABAQUS/Explicit (CAE) has been adopted for analysis and simulation this complex deformation process which involves a substantial material flow. The main parts of this finite element process are consisting of three models; Die, punch, and workpeice (blank). The blank material is aluminium and modelled as a deformable body and considered as a Von – Mises plastic elastic material. Die and punch material is steel and modelled as analytical rigid surfaces (revolved type). Simulation procedure by (ABAQUS) is consisting of some step as in "Fig. 6".

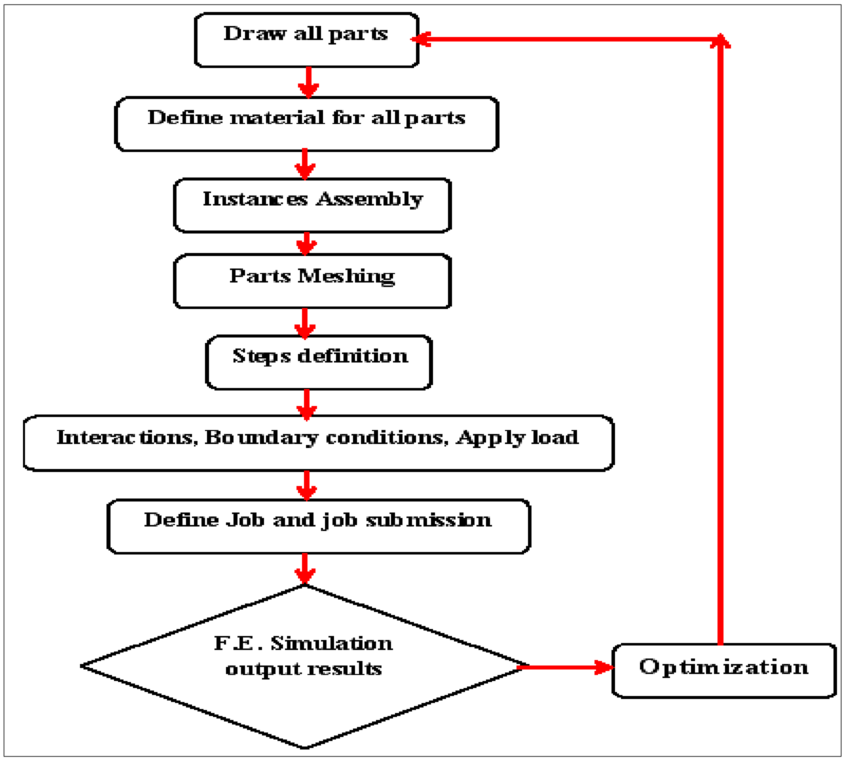

FIGURE 6. Analysis Flow Chart by ABAQUS.

In this modelling and simulation of forging process; both three-dimensional geometric models and axisymmetric model are used. The forging process is starting with placing the sheet metal inside the die cavity, as shown in "Fig. 7".and applying gravity loads with pressure to force the sheet agonist the die. The gap between the die and sheet metal should be modelled, because in the beginning it's very difficult to place the sheet inside the die in exact position.

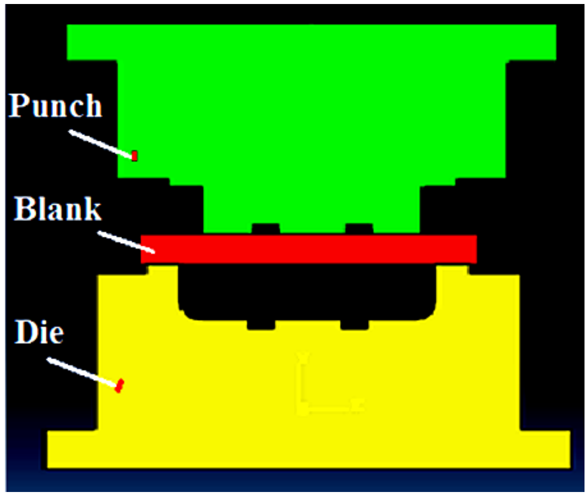

FIGURE 7. Placing all parts in contact before any action.

Simulation process consists of three main steps. In the first step, the forging punch will move downwards to place the sheet metal inside the die and the contact will happened. The boundary condition for this step includes fix and constraint the die in all directions to prevent any motion as shown in "Fig. 8". Also this step include applying a downward pressure and gravity load of (100) MPa. It's recommended in this step to avoid any tangential contact to keep an adequate stabilization. In the end of this step the contact stabilization will ramps down automatically by (ABAQUS) such as like there's no any contact remaining.

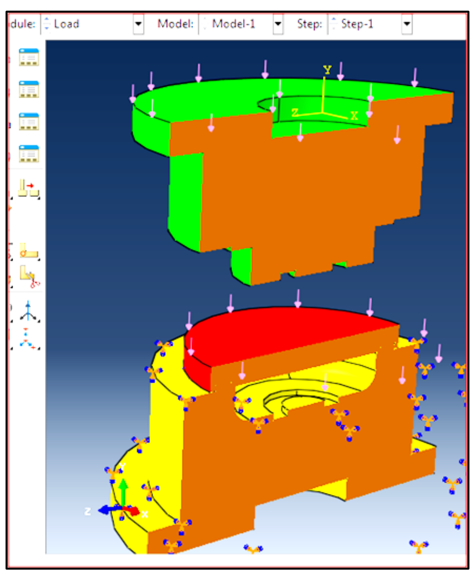

FIGURE 8. Applying load by the punch, blank and constrain die in all direction.

In F.E.M; Adaptive meshing technique are used to illustrates and simulate the behaviour of multiple geometrically complex dies during the forging process. Its can present and simulate the material flow, For 2D Axisymmetric model and due to the large amount of material flow during deformation and geometrical complicity; so the mesh is need to be refined especially in radial direction of the blank as shown in Fig. 9".

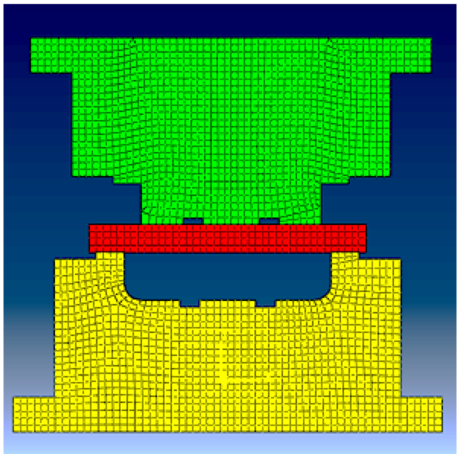

FIGURE 9. Mesh of 2D axisymmetric model.

In ABAQUS; the default definition of symmetry planes are considered as Lagrangian boundary regions and the type of elements used in blank mesh are (CAX4R) elements. However; due to the high amount of deformation in each step, and the elastic-plastic nature of this deformation, so sweep mesh and other controlling parameters will be suitable to simulate this type of deformation process as shown in "Fig. 10".

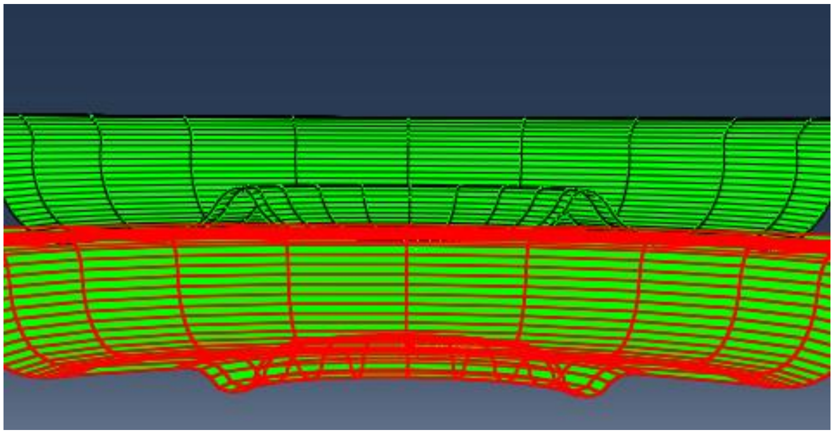

FIGURE 10. Elements of parts in contact during deformation process.

In second simulation step; the boundary condition includes a fully constrained for the die to prevent this part from any motion in any direction, while the punch will move (60) mm downward in (Y) direction with constant velocity to complete the deformation process as in "Fig. 11". The default definition of contact surfaces are sliding boundary regions, and the definition of symmetry planes will be as a Lagrangian boundary regions.

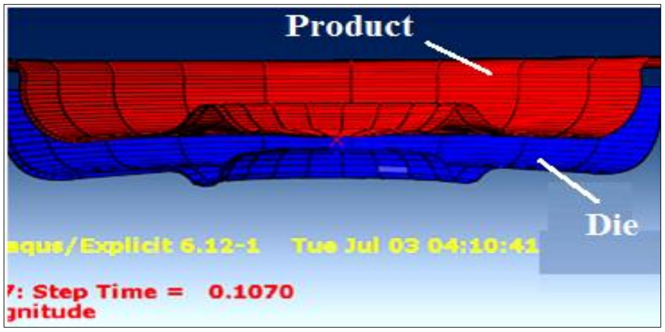

FIGURE 11. Product forging is completed in second step.

However; it’s very important to avoid the force discontinuity in the previous step to decrease the generation of viscous force and principle stresses to the minimum. Finally and in the last step, the pressure will remove and the springback of the sheet metal will be calculated. "Figure 12"   show a cross section view of the final step in product removal.

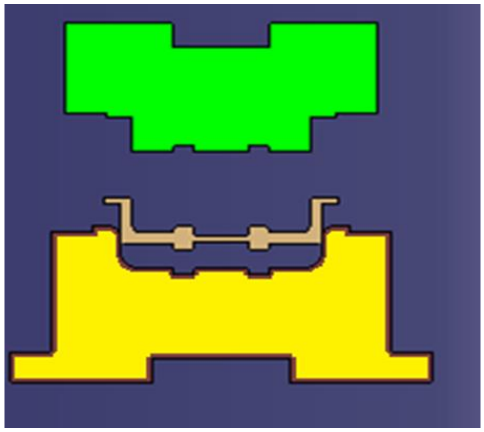

FIGURE 12. Section View of the product during removal in final step.

In experimental part of this work; some workpiece samples before and after forging process has been tested by Scanning Electron Microscope (S.E.M) to show the effects of this process on development and optimization in microstructure.

Results and Discussion

Many parameters have been estimated as extracted results from this simulation. This nonlinear with elastic plastic deformation has a lot of contact surfaces including many interactions with different boundary conditions.

The nature and types of element mesh is very important to estimate the accurate results.   During simulation process in the first two steps; the blank elements still seem well shaped and kept their shape during their travelling through the die with some exception near the edges, which become coarse due to high strain hardening in this area. Due to high friction between the contact surfaces; nodal elements of the workpiece will gain high energy and move downward with high generation of plastic strain. "Figure 13"   shows the contour of nodal velocity in (Y) direction. In this velocity contour its very clear that the workpiece nodal was in maximum speed due to high amount of particles dislocation in the end of second step.

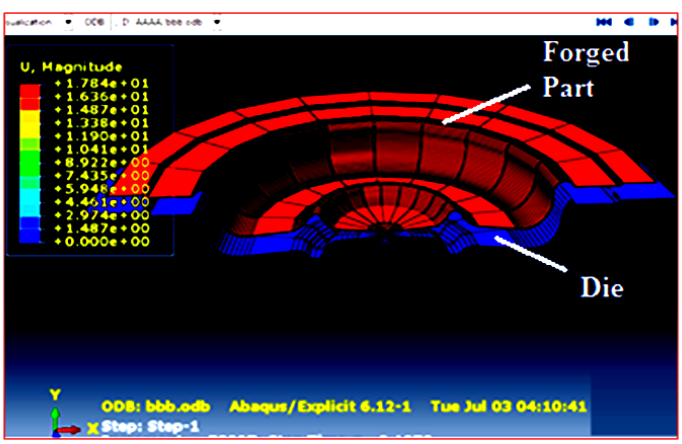

FIGURE 13. Contour of nodal speed in (Y) direction.

During simulation by (ABAQUS); any initial penetrations happened due to contact pairs can solve automatically. However, and as a result, the pure stiffness are used in two-dimensional models and the formation of orthogonal kinematic strain hare used in modelling the three-dimensional geometric. According to the above; it's found that the plane stresses concentration will be in maximum values in the contact surfaces between workpiece and die, as shown in the contour of principle stresses in "Fig. 14".

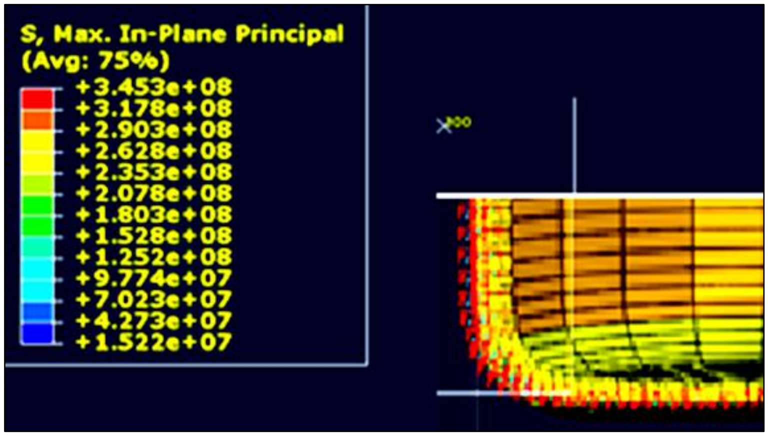

FIGURE 14. Contour of principle stresses.

Concentration of plane stresses will increase rapidly on the outer surfaces ((sticking surfaces between punch and workpiece) due to high friction and heat generation between contact surfaces. "Figure 15" show the contour of these stresses

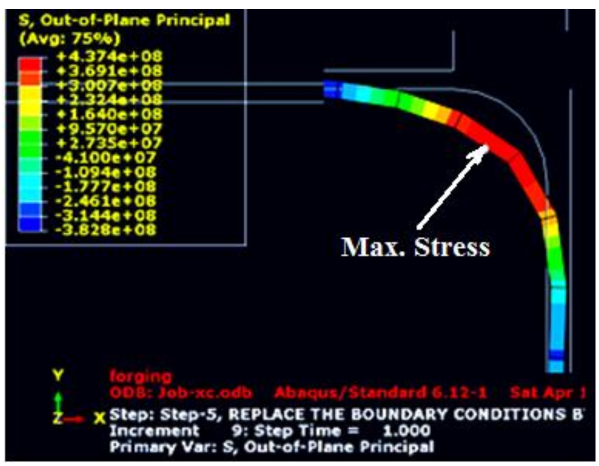

FIGURE 15. Contour of principles stresses out of the plane.

As a statistical from this analysis; it's found that the sheet mesh consists of (1110 S4R) shell elements, and the die is represented (987 R3D4) rigid elements. The value of flow stress in the mid-point is (1.0 × 109) Pa.

During forging process; plastic strain is increase and develop gradually to reach the maximum value at the final deformation step as shown in "Fig. 16".

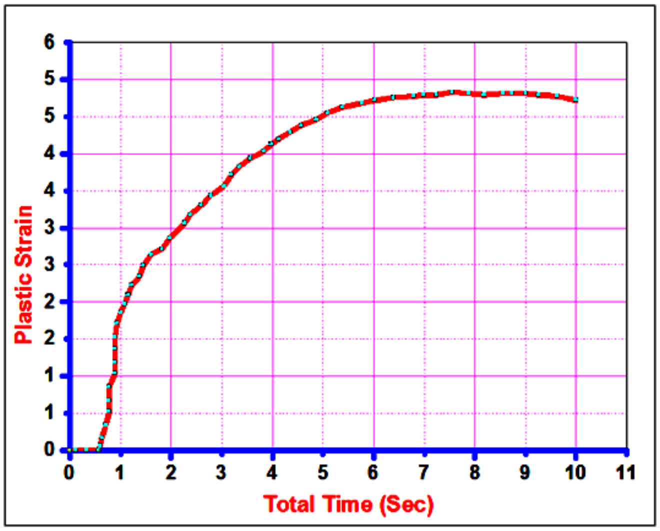

FIGURE 16. Plastic strain with time.

In hot forging process, even though each workpiece should be heated separately before the process, but the outside environment will be as a fast cooling and temperature will drop directly. However; the results approved that there is an inverse proportional between generation stresses and temperature values as shown in "Fig. 17".

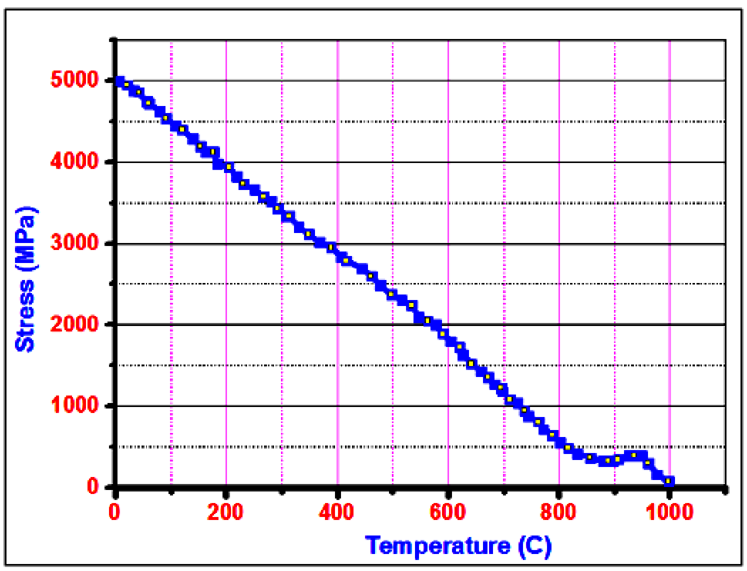

FIGURE 17. Stresses – temperature relationship.

As a comparison between simulation and experimental results regarding to the total deformation force with deflections, it's found that, there's some differences especially in the staring zone, and this gap can due to variation in boundary conditions as shown in "Fig. 18".

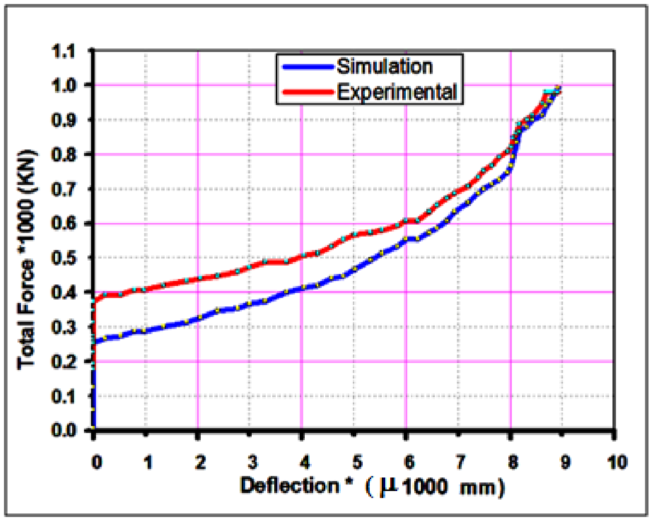

FIGURE 18. Comparison between simulation and experiment results.

Scanning electron microscope (S.E.M) micrographs show that, there are a lot of changes happened in microstructure. "Figure 19" presents the (S.E.M) micrographs of forged material before and after forging process. This micrographs reveals clearly good Optimization due to the new distribution and orientation of grains.

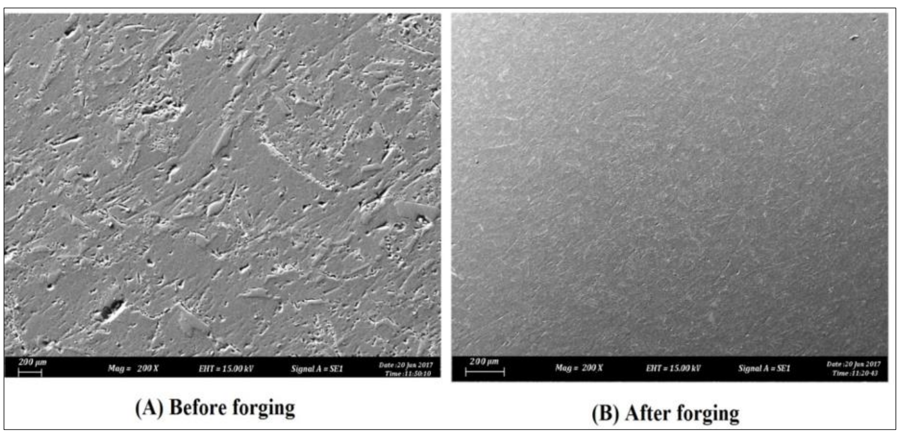

FIGURE 19. (S.E.M) micrographs before and after the process.

Conclusions

Some conclusion has been listed from the above work, and can be summarised as below:-

- For optimization approach, the parameters should be selected carefully, and first of all it should consider and specified the design constrains, because in this step the manufacturing process in not start yet. The second step is to match the design parameters with manufacturing parameters to insure a successful production procedure.
- The physical analysis and modelling by computer can make the forging process more creative, and the most interesting in all forging process planning is die and tools design.

- In simulation process, and in order to provide an accurate solution; the intensity of adaptive meshing must be increased and single adaptive mesh must be consider in each single step.
- The initial deformations in this process is elastic, but gradually with increasing load and high amount of friction forces, this will lowered until reach the plastic deformation. At the end, the load will remove and gradually the tools will cool and leaving the product with some of springback.
- If there's not enough boundary conditions applied on the workpiece, a big convergence problems may be happened, especially when the workpiece is initially placed far from the dies.

REFRENCES

1. K.J. Barnett. (2000). "Research initiatives for the forging industry". Journal of Materials Processing Technologies 98 162-164.

2. B. Tomov. (September 2007). "Hot closed die forging – State-of-Art and future development". Journal of Achievements in Materials and Manufacturing Engineering. VOLUME 24 ISSUE 1. University of Rousse, Bulgaria.

3. Murat Ozturk, Sinem Kocaoglan, Fazil O. Sonmez. (2016). "Concurrent design and process optimization of forging". Computers and Structures 167, 24–36 www.elsevier.com/locate/compstruc. Bogazici University, Istanbul 34342, Turkey.

4. K. Hyunkee, T. Yagi, M. Yamanake, (2000). "F.E Simulation as a must tool in cold/warm forging process tool design". Journal of Materials Processing Technology (143-149).

5. V.B. Nguyen a,c,n, D.J.Mynors b, C.J.Wang b, M.A.Castellucci a, M.A.English. (2016). "Analysis and design of cold formed dimpled steel columns using Finite Element techniques". Finite Elements in Analysis and Design 108.(22–31). www.elsevier.com/locate/finel.
6. Manas Shirgaokar, Gracious Ngaile, Gangshu Shen. (2005). "Process Modeling in Impression-Die Forging Using Finite-Element Analysis". © 2005 ASM International. All Rights Reserved. Cold and Hot Forging: Fundamentals and Applications. www.asminternational.org.

7. Lu B, Ou H. (2011). " Stochastic finite–element modelling and optimization for netshape forging of three-dimensional aero-engine blades". Proc Inst Mech Eng;225:71–85.

8. Stéphane Andrietti. (2016). "Benefits of Virtual Manufacturing for Modern Forging Companies" . Technical Reference Bulletin. http://www.transvalor.com/fr/.

9. Fayuan Zhu, ZhaohuiWang, andMi Lv.(2015). "Multi objective Optimization of Precision Forging Process Parameters Based on Response Surface Method". Advances in Materials Science and Engineering. Volume 2015, Article ID 893730, http://dx.doi.org/10.1155/2015/893730.

10. Lu B, Ou H, Long H. (2011). Die shape optimisation for net-shape accuracy in metal forming using direct search and localised response surface methods. Structural and Multidisciplinary Optimization. Volume 44, Issue 4, pp 529–545. https://link.springer.com/article/10.1007%2Fs00158-011-0635-x.

11. Rith Ly, Christophe Giraud-Audine, Gabriel Abba, Régis Bigot. (2009). "Experimentally valided approach for the simulation of the forging process using mechanical vibration". International Journal of Material Forming, Springer Verlag, pp.133-136. https://hal.archives-ouvertes.fr/hal-00781874.