12th INTERNATIONAL SCIENTIFIC CONFERENCE ON DEFENSIVE TECHNOLOGIES
OTEH 2026
Serbia, Tara, 21 - 24 October 2026

CAE SIMULATION OF THE HOT FORGING PROCESS FOR A MILITARY VEHICLE STEEL COMPONENT

MARKO DELIĆ
Faculty of Engineering University of Kragujevac,Sestre Janjić 6, 34000 Kragujevac, Serbia; marko.delic@kg.ac.rs

NIKOLA KASTRATOVIĆ
Zastava Kovačnica A.D. Kragujevac, Kosovska 4b, 34000 Kragujevac, Serbia; nikolakastratovic93@gmail.com

VESNA MANDIĆ
Faculty of Engineering University of Kragujevac,Sestre Janjić 6, 34000 Kragujevac, Serbia; mandic@kg.ac.rs

DUŠAN ARSIĆ
Faculty of Engineering University of Kragujevac,Sestre Janjić 6, 34000 Kragujevac, Serbia; dusan.arsic@fink.rs

NIKOLA KOMATINA
Faculty of Engineering University of Kragujevac,Sestre Janjić 6, 34000 Kragujevac, Serbia; nkomatina@kg.ac.rs

Abstract: *Today, it is impossible to imagine mechanical parts in the automotive, mechanical or military industry without forgings, primarily because of their durability and safety in operation. However, in order to obtain a quality part, one must know exactly how the material flows through the tool and what happens during the deformation itself. That's why numerical simulations have become our main tool when developing or changing forging technology.*

Instead of expensive tests in the plant, we used a computer model to follow in detail the propagation of deformations, the change of stress and the movement of the forging force during the compression of the steel. We mainly focused on identifying critical regions in the tool that can adversely affect the geometry of the forging or cause unexpected production downtime.

The results from Simufact gave us in practice a "window" into the interior of the material during compaction. Based on that, we precisely assessed whether the proposed technological solution is good, monitoring how the mold is filled and how efficient the whole process actually is.
This engineering approach through CAE methods clearly proves its value in the industry, because it directly saves us time for new product development.

Keywords: *forging, Simufact Forming, material flow, military vehicle, CAE*

1. INTRODUCTION

The hot forging is one of the most important technologies for metal forming. New technologies like 3D metal printing have been developed, but hot forging is still very important in the production process of parts under high loads. Due to good mechanical properties of manufactured parts, hot forging has an important role on military and automotive industries. Lower values of the yield stress are obtained when the workpiece is heated up. But it is very difficult to lubricate because of the high temperature. In hot forging process, the tools are subjected to high temperature, friction due to sliding of material on tool surface and in case of hammer forging, impact load.

The ever-increasing costs of material, energy and especially manpower demand that forging processes and tooling be designed and developed with minimum amount of trial and error with shortest possible lead times. Hence, to keep up with the competition the cost-effective application of computer aided techniques i.e. CAD, CAM, CAE and especially Finite Element Analysis (FEA) and Finite Volume Analysis (FVA) based computer simulation are an absolute necessity. The practical use of these techniques requires knowledge of the principal variables of the forging process and their interaction.

CAE simulations of processes using software based on the finite element method are successfully applied to eliminate material flow defects, determine and optimize process parameters, determine tool stresses to prevent tool breakage, and predict workpiece properties through stress-strain analysis. In addition, it is very easy to make changes to the tool design and monitor the impact of these changes on process parameters, thus achieving optimal tool geometry at all stages of the process.

In hot forging processes, the lifetime of forging tools

Corresponding author: N. Kastratović;
E-mail: nikolakastratovic93@gmail.com

DOI: 10.5937/OTEH26134D

(upper and lower dies) is largely limited by the occurrence of abrasive wear, thermal fatigue and plastic deformation. Research shows that wear is the dominant failure mechanism, responsible for as much as 70% of all damage to tools. Validation of numerical tool wear models is performed by comparing simulation results with experimental measurements on three-dimensional measuring machines (CMM) [1].

Lee and Jou [2] applied Archard's wear law within a finite volume framework to estimate the service life of hot forging tools. Their approach enabled the prediction of wear development based on local contact conditions such as pressure, sliding velocity and temperature.

Hartley and Pillinger [3] highlighted the importance of remeshing techniques in forging simulations due to severe element distortion caused by large plastic deformations. Their findings demonstrated that adaptive mesh refinement is essential for maintaining numerical stability.

Hawryluk et al. [4] investigated tribological conditions in hot forging processes and showed that friction conditions continuously evolve as a consequence of lubricant degradation and elevated temperatures.

Davoudi et al. [5] emphasized the importance of accurate contact stress prediction in forging tool analysis. Their work demonstrated that fine mesh discretization improves the identification of stress concentration zones and increases prediction accuracy.

Yin et al. [6] reported that improper billet positioning may result in eccentric loading and premature tool failure, highlighting the importance of accurate billet alignment during forging operations.

Řiháček et al. [7] compared different numerical approaches implemented in Simufact Forming and concluded that the finite volume method provides superior numerical stability in simulations involving severe plastic deformation and intensive material flow.

Chenot et al. [8] demonstrated that numerical simulation can significantly reduce the need for costly experimental trials by enabling process optimization in a virtual environment before industrial implementation.

Studies on hot deformation behavior of Cr-alloyed steels have shown that dynamic recrystallization is the dominant softening mechanism at elevated temperatures. The flow stress response is strongly dependent on temperature and strain rate, making accurate material modelling essential for reliable numerical simulations [9].

2. SIMULATION CONDITIONS

A three-dimensional thermo-mechanical finite element model was developed using Simufact Forming software in order to analysis the upset forging process of an industrial component manufactured from 38HS steel.

Steel 38HS (according to the standard GOST 4543) is an improved steel which is highly valued in the forging industry due to its high strength, excellent toughness and good hardenability. The material has a precise balance of chromium and molybdenum, which gives a superior resistance to material fatigue and impact loads and is the ideal choice for the manufacture of components that are highly stressed in the defense and automotive industries. 38HS exhibits a stable behavior at high temperatures during the hot forging process, enabling the production of complex geometries without the risk of internal defects. After a suitable thermal treatment, 38HS shows superior mechanical properties required to operate in the most challenging working conditions.

The simulation model consisted of three main geometrical entities: the workpiece, the upper die and the lower die. The workpiece geometry was imported as a CAD model and discretized using an adaptive finite element mesh.

The workpiece material was defined as 38HS. The material behavior was considered temperature-dependent in order to accurately represent the hot forging conditions.

The forging process was simulated through three consecutive stages: upsetting, pre-forging and final forging. The upsetting and pre-forging operations were analyzed using a single mesh configuration, since their primary purpose was to generate the intermediate billet geometry required for the final forging stage.

A mesh sensitivity analysis was performed exclusively for the final forging operation, which represents the most critical stage of the process due to the complex die geometry and intensive material flow. Three mesh sizes were investigated: 3.0 mm, 2.2 mm and 1.5 mm. All other process parameters, including billet temperature, die temperature, friction factor and press characteristics, were kept constant throughout the analysis.

This approach enabled the influence of mesh density on numerical stability, computational time and predicted process parameters to be evaluated while maintaining identical initial conditions obtained from the upsetting and pre-forging simulations.

The simulations were performed using the finite volume (FV) solver in Simufact Forming. The initial billet temperature was set to 1250 °C, while the die temperature was maintained at 250 °C. A shear friction model with a friction factor of m = 0.5 was applied at the workpiece–die interface. The forging operation was simulated using a Forging hammer with an impact energy of 35 kJ and an efficiency factor of 0.7. The total forging stroke was 70 mm and each simulation was divided into 100 equal increments.

Therefore, the only variable parameter among the analyzed cases was the mesh density, allowing the determination of the influence of element size on numerical stability, computational time and predicted process parameters to be evaluated.

3. RESULTS AND DISCUSSION

The numerical simulation of the hot forging process was carried out using the finite volume solver implemented in Simufact Forming. The objective of the analysis was to evaluate material flow, strain distribution, contact conditions and the influence of mesh density on the numerical stability of the obtained results.

The simulation results confirmed complete filling of the die cavity and successful material flow throughout the forging operation. The highest values of effective plastic strain were recorded in the flash region and in areas characterized by intensive material redistribution. Such behavior is expected in hot forging processes due to the constrained flow conditions near the die parting line.

The computational cost increased significantly with mesh refinement. The simulation time increased from approximately 34 minutes for the 3.0 mm mesh to 90 minutes for the 2.2 mm mesh and 111 minutes for the 1.5 mm mesh. In addition, the number of remeshing operations increased from four to six, indicating a higher degree of mesh distortion during the forming process.

Because similar anomalies appeared in both refined meshes, it can be concluded that the observed behavior is related to local numerical effects rather than to the actual physical response of the material. Consequently, the 3.0 mm mesh provided the most stable and physically meaningful solution while maintaining acceptable computational efficiency.

Table 1 presents the influence of mesh density on the maximum effective plastic strain and maximum contact pressure obtained during the final forging simulation. The results indicate that mesh refinement leads to a gradual increase in both parameters. The maximum effective plastic strain increased from 1.92 for the 3.0 mm mesh to 2.07 and 2.09 for the 2.2 mm and 1.5 mm meshes, respectively. The relatively small difference between the 2.2 mm and 1.5 mm meshes suggests that the solution approaches numerical convergence with respect to plastic strain.

A similar trend was observed for the maximum contact pressure, which increased from 1215 MPa for the 3.0 mm mesh to 1499 MPa and 1790 MPa for the 2.2 mm and 1.5 mm meshes, respectively. This behavior is expected because finer meshes provide a more accurate representation of local contact conditions and stress concentrations, particularly in regions with complex die geometry.

The obtained results demonstrate that CAE-based process simulation is an effective tool for evaluating forging technology before industrial implementation. The performed analyses enabled detailed insight into material flow, strain development and die filling conditions, reducing the need for costly experimental trials and shortening the development cycle of forged components intended for military vehicle applications.

Figure 1 shows a 3D model of the tool assembly, which includes the upper and lower dies imported in the Simufact Forming software environment. The geometry of the tool is imported from the CAD system with the application of precise discretization, which ensures a reliable mapping of contact surfaces and interactions with the workpiece during the forging process. Figure 1 also show forged part.

Table 1. Comparation of results

Parameter	Mesh 3.0 mm	Mesh 2.2 mm	Mesh 1.5 mm
Max effective plastic strain	1.92	2.07	2.09
Max pressure (MPa)	1215	1499	1790
Max strain rate (s^{-1})	173982	402216	364102
Max die wear (mm)	1.67E-11	2.53E-11	2.06E-11
Max temperature (°C)	1351.52	1372.61	1537.31

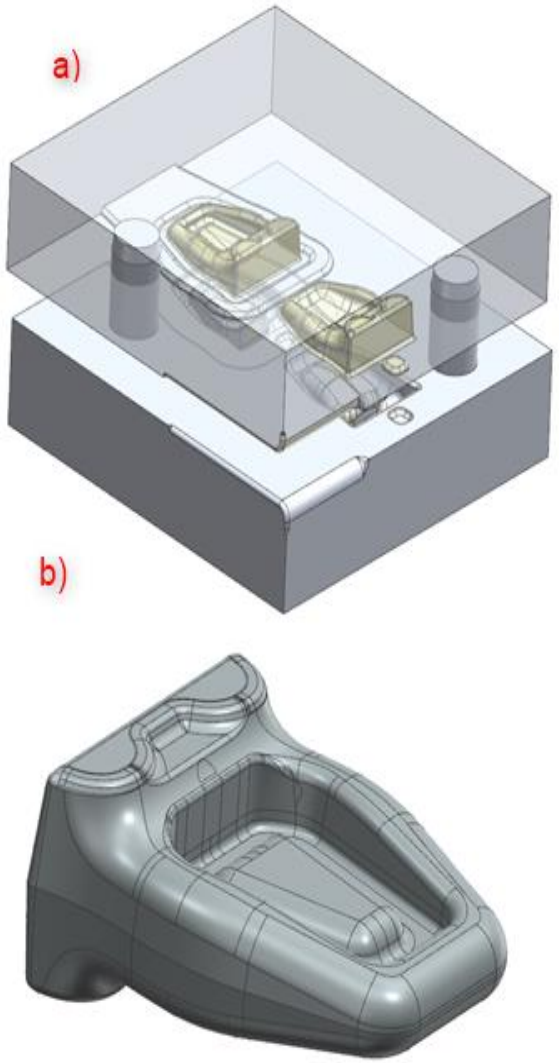


Figure 1. 3D model: a) Upper and lower die for forging; b) Forged part

Figure 2 presents the upsetting and pre-forging stages used to obtain the intermediate billet geometry for the final forging operation. These stages were identical for all analyzed cases and were simulated using the same process parameters and mesh configuration.

Figures 3, 4 and 5 illustrate the final forging stage

performed with different mesh densities in order to evaluate the influence of element size on the numerical results. Figure 3 shows the results obtained using a 3.0 mm mesh, Figure 4 presents the simulation performed with a 2.2 mm mesh, while Figure 5 corresponds to the finest analyzed mesh of 1.5 mm.

All simulations were carried out under identical technological conditions, including the initial billet temperature of 1250 °C, die temperature of 250 °C, friction factor of μ = 0.5 and hammer energy of 35 kJ. Therefore, the only varying parameter among the analyzed cases was the mesh density used in the final forging stage.

The comparison of the obtained results enabled the evaluation of mesh sensitivity, computational efficiency and numerical stability, while maintaining identical initial conditions generated during the upsetting and pre-forging operations.

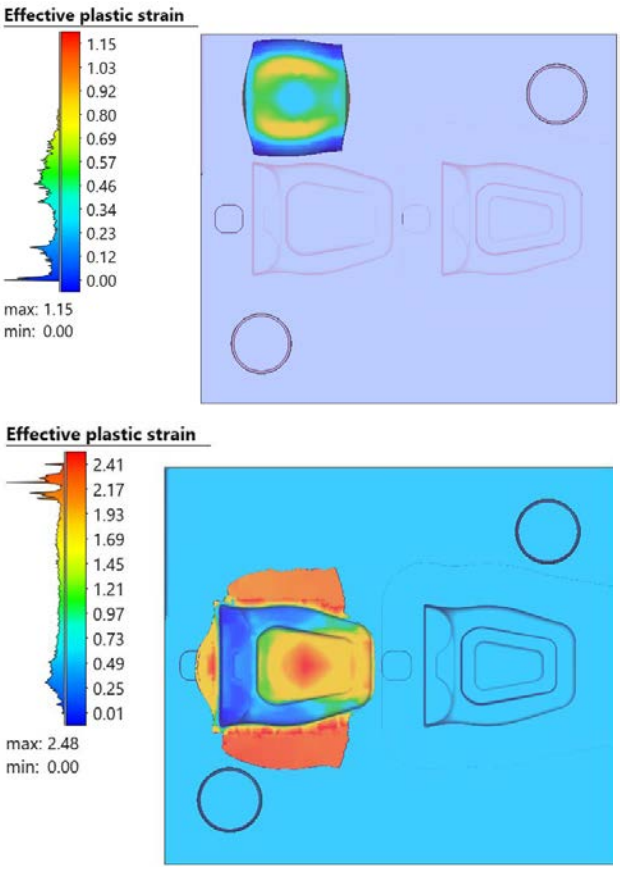


Figure 2. Upsetting and pre-forging

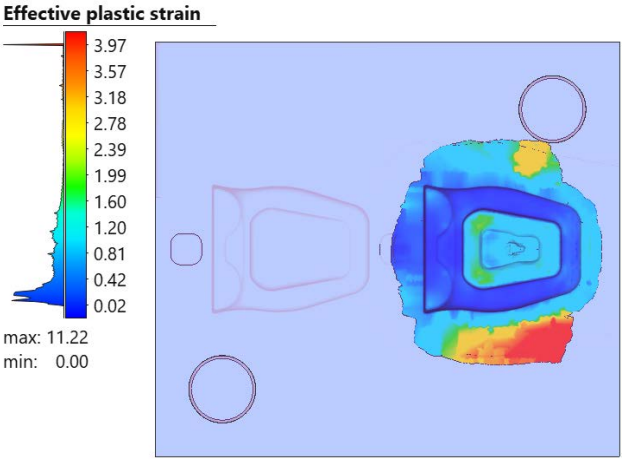


Figure 3. Value of effective plastic strain mesh 3 mm

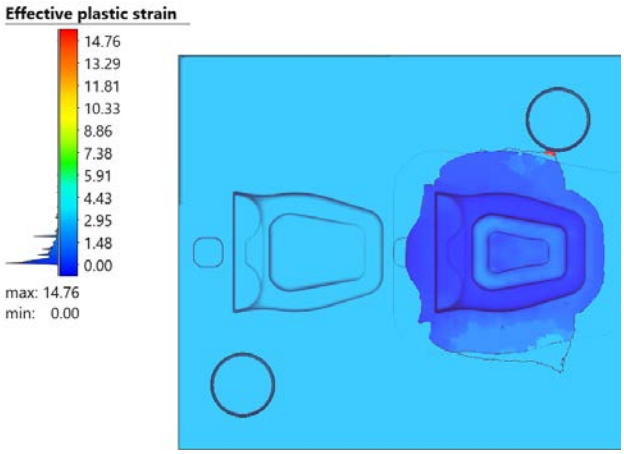


Figure 4. Value of effective plastic strain mesh 2.2 mm

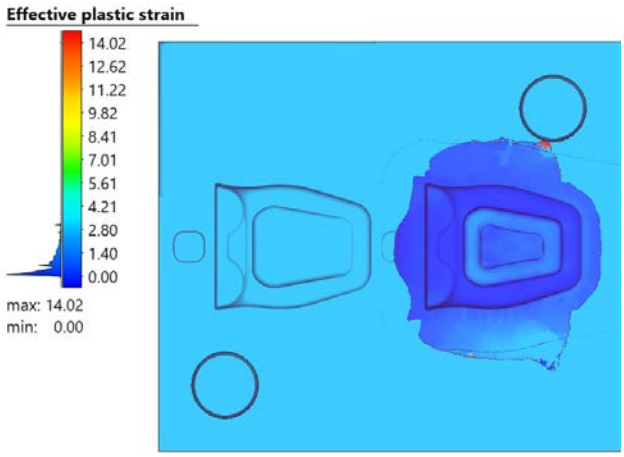


Figure 5. Value of effective plastic strain mesh 1.5 mm

Figure 6 shows the distribution of effective plastic strain for the three mesh configurations investigated. In all cases, the highest values are observed in the forgings wreath region, where the material is subjected to severe compression as it flows into the narrow flash gap. These localized maxima are expected and represent the most intensive plastic deformation occurring during the forging process rather than the deformation of the forged component itself. Therefore, the effective plastic strain values reported in Table 1 were measured directly on the forged part, excluding the forgings wreath region, in order

to provide a representative comparison of the deformation state within the final product. The results indicate only minor differences between the investigated mesh sizes, confirming that mesh refinement has a limited influence on the predicted deformation of the forged component, while mainly affecting the resolution of localized deformation in the forgings wreath region.

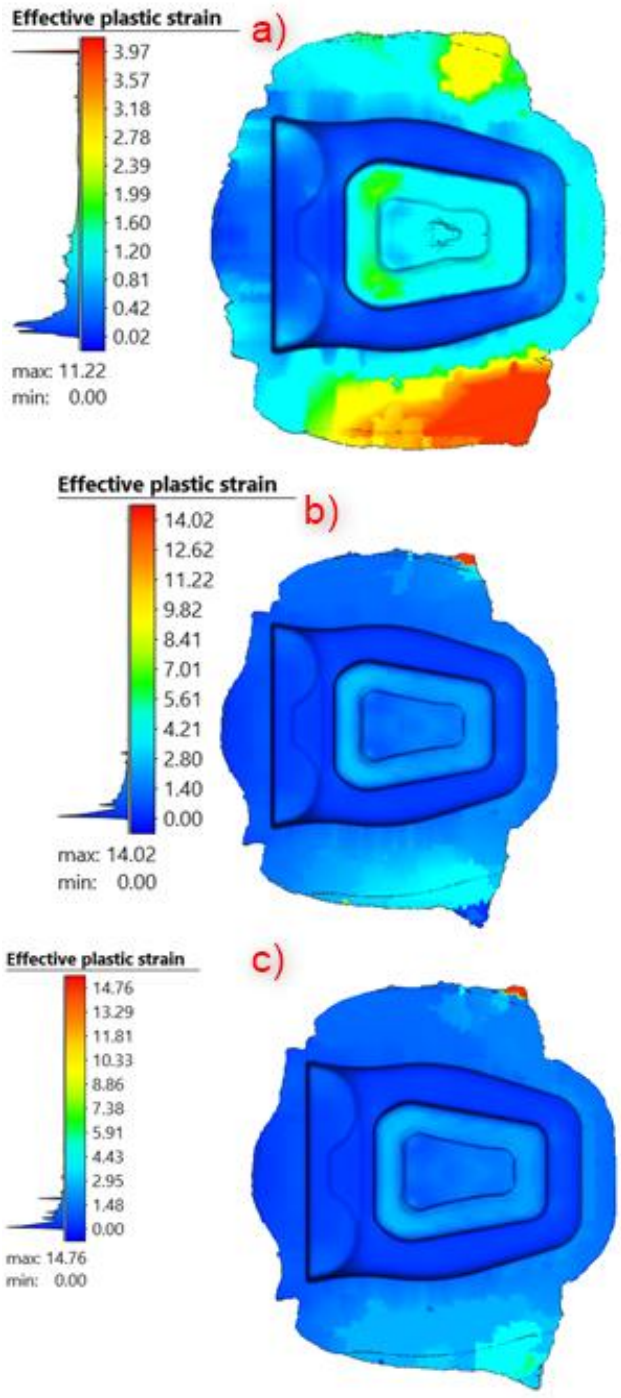


Figure 6. Forged part: a) Mesh 3.0 mm b) Mesh 2.2 mm c) Mesh 1.5 mm

5. CONCLUSION

In this paper, a numerical analysis of the forging process was carried out, where the finite volume method (FVM) was applied and showed exceptional stability and reliability in the simulation of large plastic deformations.

This study presented a numerical simulation of the hot forging process of a 38HS steel component intended for military vehicle applications using Simufact Forming software. The conducted CAE analysis enabled a detailed investigation of material flow, strain distribution, temperature evolution and die filling behavior throughout the forging operation.

The simulation results confirmed complete die cavity filling and demonstrated that the proposed forging technology is capable of producing the required component geometry without critical defects that could compromise the manufacturing process. The highest deformation values were observed in the flash region, which is consistent with the expected material flow characteristics of hot forging processes.

A mesh sensitivity analysis was performed using element sizes of 3.0 mm, 2.2 mm and 1.5 mm. Although finer meshes provided a more detailed representation of local deformation phenomena, they significantly increased computational time and introduced numerical instabilities manifested through nonphysical pressure, density and temperature peaks. Among the analyzed models, the 3.0 mm mesh provided the most stable and computationally efficient solution while maintaining physically meaningful results.

The obtained findings demonstrate that CAE-based simulation represents a valuable engineering tool for the design and optimization of forging technologies. By enabling virtual process verification prior to industrial implementation, numerical simulation reduces development costs, minimizes experimental trials and shortens product development time.

The methodology applied in this study can be successfully extended to other forged components manufactured from 38HS steel and may serve as a basis for future research focused on process optimization, tool design improvement and advanced thermo-mechanical analysis of hot forging operations.

Future work may involve the investigation of additional process parameters and their impact on forging performance. The incorporation of experimental validation and industrial production data would allow for further calibration of the numerical model, leading to improved prediction accuracy and enhanced process optimization in industrial forging applications.

Acknowledgements

The authors gratefully acknowledge Zastava Kovačnica A.D., Kragujevac, Serbia, for providing the technical resources, engineering infrastructure, and support necessary for carrying out the numerical simulations and research presented in this paper.

References

[1] NARESH, P., BABU, A. V. H., MADHAVA, V., REDDY, M. S.: *Design and Analysis of a Hot Forging Dies*, Journal of Mechanics and MEMS

(JMM), 2016.

[2] LEE, R.S., JOU, J.L.: *Application of numerical simulation for wear analysis of warm forging die*, Journal of Materials Processing Technology, 140, 2003.

[3] HARTLEY, P., PILLINGER, I.: *Numerical simulation of the forging process*, Comput. Methods Appl. Mech. Engrg., 195, 2006.

[4] HAWRYLUK, M., GRONOSTAJSKI, Z., ZIEMBA, J., DWORZAK, Ł., JABŁOŃSKI, P., RYCHLIK, M.: *Analysis of the influence of lubrication conditions on tool wear used in hot die forging processes*, Eksploatacja i Niezawodnosc - Maintenance and Reliability, 20, 2018.

[5] DAVOUDI, M., NEJAD, A.F., KOLOOR, S.S.R., PETRŮ, M.: *Investigation of effective geometrical parameters on wear of hot forging die*, Journal of Materials Research and Technology, 15, 2021.

[6] YIN, J., HU, R., SHU, X.: *Closed-die forging process of copper alloy valve body: finite element simulation and experiments*, Journal of Materials Research and Technology, 10, 2021.

[7] ŘIHÁČEK, J., PETERKOVA, E., CISAROVA, M., KUBICEK, J.: *Comparison of FEM and FVM for the numerical simulation of forging process*, MM Science Journal, 2020.

[8] CHENOT, J.-L., BOUCHARD, P.-O., FOURMENT, L., LASNE, P.: *Numerical Simulation and Optimization of the Forging Process*, International Cold Forging Congress 12th ICFC, Stuttgart, 2011.

[9] MENGARONI, S., CIANETTI, F., CALDERINI, M., EVANGELISTA, E., DI SCHINO, A., MCQUEEN, H.: *Tool Steels: Forging Simulation and Microstructure Evolution of Large Scale Ingot*, Acta Physica Polonica A, 2015.