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

NUMERICAL ANALYSIS OF THE HOT FORGING PROCESS FOR A STEEL DEFENSE INDUSTRY COMPONENT

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

MARKO DELIĆ
Faculty of Engineering University of Kragujevac,Sestre Janjić 6, 34000 Kragujevac, Serbia; marko.delic@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: *The paper deals with the numerical simulation of the hot forging process on a hammer. Today, it is hard to think of reliability in automotive, mechanical, or defense engineering without using forged parts. Their toughness and field safety are just too critical. But to get a high-quality component, you have to know exactly how the metal behaves and flows inside the die during heavy deformation. This is why numerical process simulations have become our go-to tool for updating or creating new forging technologies.*

Instead of burning budget on slow and expensive shop-floor trials, we built a finite volume model to map out deformation, shifting stresses, and forging forces while the steel is compressed under the hammer. We focused heavily on pinning down those weak spots in the tooling that can mess up the final geometry or trigger unexpected downtime.

The data from Simufact Forming basically gave us a clear window into the material's core during the hits. It let us judge whether the planned workflow actually works, showing us exactly how the die fills and how efficient the whole setup is. This CAE-driven method makes perfect sense for modern manufacturing because it cuts down product development time drastically.

Keywords: *hot forging, Simufact Forming, material flow, defense indsutry, numerical simulation*

1. INTRODUCTION

Hot forging is one of the most important technologies for metal forming. Modern technologies such as 3D metal printing have been developed, but hot forging plays a very important role in the manufacturing process of parts under high loads. Hot forging has an important role in military and automotive industries due to good mechanical properties of manufactured parts. The lower values of the yield stress are obtained when the workpiece is heated up. However, it is very difficult to lubricate because of the high temperatures. In hot forging process, the tools are loaded by high temperature, friction due to sliding of material on tool surface and in case of hammer forging, impact load.

Numerical simulations of processes using software based on the finite volumes 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.

Die life acts as an important factor on the productivity. In warm forging process, wear is the predominant factor affecting the die life since it sustains higher temperature and working pressure. Lee and Jou [1] reported that die life in warm forging is affected by wear in over 70%. This kind of failure affects not only the cost but also the tolerances of the forging. Developing methods which allow prediction of wear and which are suited to be used in the design stage is the most effective way in order to optimize the process.

In accordance with the methodological approach of Lee and Jou [1], the lifetime estimation of hot forging tools requires the application of Archard's wear law integrated within a finite volume numerical simulation. This

Corresponding author: N. Kastratović;
E-mail: nikolakastratovic93@gmail.com
DOI: 10.5937/OTEH26133K

approach enables a precise correlation between the local conditions at the interface (pressure, sliding speed, temperature) and the predicted wear in the most stressed zones of the engraving.

According to Hartley and Pillinger [2], the degradation of finite elements is one of the primary challenges in the simulation of the forging process, which is why the implementation of reliable re-meshing strategies is necessary to maintain the stability of the calculation. Accordingly, in this paper, a nonlinear refresh criterion based on an increase in effective plastic deformation of 0.4 was adopted.

The phenomena at the interface between the forging tool and the workpiece are extremely complex and involve the combined influence of high friction, heat generation and material flow. When defining the numerical model in this paper, the influence of imperfect lubrication was simulated using a friction factor of 0.65, which required additional optimization of the size of the final mesh size to 1.5 mm in order to avoid numerical anomalies.

For the numerical analysis of the hot forging process, the Simufact Forming software was used, which in recent research (Delić et al) [3] has been successfully applied for the prediction of tool wear and the analysis of deformation forces. The numerical model in this paper was additionally validated through the optimization of flow parameters under highly loaded friction conditions [3].

As pointed out by Hawryluk et al. [4], tribological conditions in the hot forging process are extremely dynamic and subject to changes due to lubricant degradation. In accordance with these findings, the friction factor μ=0.65 was applied in this paper, which covers realistic conditions of limited lubrication at the tool-workpiece interface.

Following the methodology applied by Davoudi et al. [5], numerical analysis of forging tools requires precise definition of local contact stresses. The application of fine discretization (mesh size 1.5 mm) in this work enabled the detection of critical zones of stress concentration on the engraving radii, which ensures high reliability of the prediction of tool behavior under the influence of large deformation forces.

Yin et al. [6], emphasize that incorrect positioning of the ingot (billet) in relation to the center of the die leads to eccentric loading and premature breakage of the central punch, which can be successfully corrected by optimizing the guides and the forging positioning process itself.

As Řiháček et al. [7] proved in their comparative analysis of numerical methods in the Simufact.forming software, the finite volume method (FVM) shows superior numerical stability compared to the classical finite element method (FEM) when modeling processes with large deformations and intensive material flow.

The network refresh strategy implemented in this paper, based on the plastic strain increment of 0.4, is fully aligned with the findings of Řiháček et al. [7], who point out that optimal accuracy and stability of calculations are achieved by a combination of a suitable numerical solver and controlled adaptation of the network model.

As pointed out by Chenot et al [8], numerical simulation is increasingly replacing expensive and time-consuming trial and error methods on real machines, allowing engineers to find optimal solutions faster. The integration of optimization modules with simulation software allows not only to reduce production costs, but also to improve the mechanical properties of the final part through controlled preform geometry.

2. SIMULATION CONDITIONS

A three-dimensional thermo-mechanical finite volumes model was developed using Simufact Forming software in order to analysis the upset forging process of an industrial component manufactured from 42CrMo4 steel.

Steel 42CrMo4 (according to the standard EN 10083) is a low alloy improvement steel, which is very appreciated in the forging industry because of its high strength, excellent toughness and good hardenability. The material contains a precise balance of chromium and molybdenum, resulting in 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. 42CrMo4 presents a stable behavior at high temperatures during the hot forging process, which allows to obtain complex geometries without the risk of internal defects, and after a proper thermal treatment it presents superior mechanical properties necessary to work in the most difficult operating 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 size of mesh.

The workpiece material was defined as $42CrMo_4$ steel using the material data available in the Simufact material database. The material behavior was considered temperature-dependent in order to accurately represent the hot forging conditions.

First case: The forging operation was simulated on a Hammer MPM-1000 machine. The initial workpiece temperature was set to 1200°C, while the initial die temperature was 250°C. The ambient temperature was defined as 20°C. Contact interactions between the workpiece and the dies were modelled using friction conditions 0.5.

Second case: The forging operation was simulated on a Hammer MPM-1000 machine. The initial workpiece temperature was set to 1100°C, while the initial die temperature was 250°C. The ambient temperature was defined as 20°C. Contact interactions between the workpiece and the dies were modelled using friction conditions 0.65.

The simulation was performed for the given thermo-mechanical conditions, considering the heat transfer between the workpiece and the tools, the temperature increase due to plastic deformation and the influence of temperature on the material flow behavior.

The aim of the numerical model was to analyze the material flow, distribution of the effective strain, stress state and development of the forging force during forming operation. The obtained results were used to evaluate the suitability of the selected technological solution and to indicate potential critical regions that could influence the quality of the final forging.

3. RESULTS AND DISCUSSION

The numerical simulation provided a detailed insight into the thermo-mechanical behavior of the workpiece during the hot die forging process. The obtained results enabled the evaluation of material flow, effective strain distribution, stress state, temperature field and forging force development throughout the forming operation.

Two numerical simulations of the hot die forging process were performed in order to investigate the influence of friction conditions on the material behavior during forming.

The first simulation was carried out using a shear friction factor of 0.50, while the second simulation considered an increased friction factor of 0.65. In addition, a finer mesh size was applied in the second case.

The obtained results showed that the increase in friction reduced the maximum effective strain from approximately 6.25 to 4.52, indicating a more restricted material flow. At the same time, local material compression increased, which was reflected in higher local density values and a greater number of remeshing operations during the simulation.

The maximum workpiece temperature remained within the expected range for both cases, although a slight reduction was observed for the higher friction condition.

The numerical results confirm that friction conditions significantly affect material flow behavior, deformation distribution and overall process stability during hot die forging.

The analysis confirmed that the material successfully filled the die cavity under the selected technological conditions. The distribution of effective plastic strain and stresses indicated the regions subjected to the highest deformation intensity, while the temperature field showed the thermal conditions during the forming process.

Special attention was paid to the influence of friction conditions on the material behavior. Comparative analysis of the investigated cases demonstrated that the change in friction coefficient affected the material flow pattern, deformation distribution and contact conditions between the workpiece and the dies.

In addition to the standard process parameters, the numerical model enabled the evaluation of potential critical regions by means of the Cockroft-Latham damage criterion and die wear prediction. These results provide useful information for assessing process stability and the quality of the final forging.

Increase in wear: When moving from μ =0.5 to μ =0.65, the maximum die wear (Max die wear) increased from 1.3E-11 to 8.54E-11. This is almost 6.5 times the higher value, confirming that even small changes in the tribological conditions at the tool-workpiece interface drastically shorten tool life.

Drastic jump in pressure: The Max pressure value jumps from 4691.77 MPa to an incredible 22829.8 MPa Such high pressure values indicate that with greater friction, the material moves much harder through the forging engraving, which creates an extreme load on the surface of the tool.

Deformation parameters: It is interesting that the Max effective plastic strain decreases from 6.25 to 4.52 at higher friction. This is the expected behavior because higher friction restricts the free flow of the material which directly confirms the theoretical proposition that friction dictates the deformation mechanism in hot forging.

Table 1. Comparation of results

Parameter	***μ=0.5***	***μ=0.65***
Max effective plastic strain	6.25	4.52
Max pressure	4691.77 MPa	22829.8 MPa
Max plastic strain rate	29235.5	194846
Max die wear	1.3E-11	8.54E-11
Max temperature	1286.11°C	1263.25°C

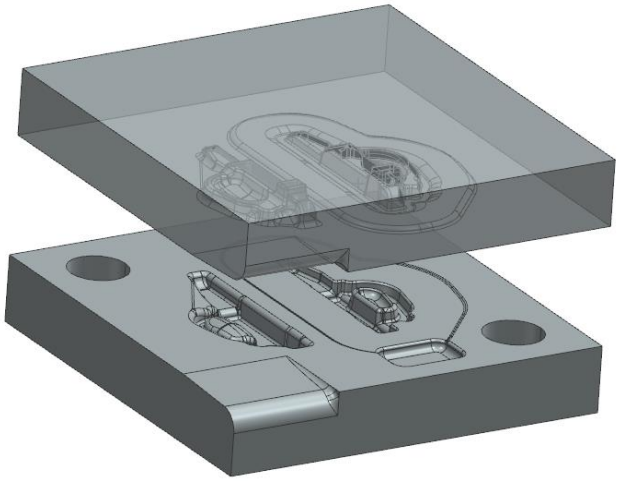


Figure 1. Upper and lower die for forging

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.

The forging process of the analyzed industrial component consists of operations: the preforming operation and the finishing operation, which are necessary to achieve the required geometric characteristics and precision of the

part. The initial step includes the operation of compressing (upsetting) the preform, which prepares the raw material for the deformation process, followed by the final stage, in which the material is completely pressed into the engraving of the tool under the influence of high pressures and temperatures. This sequential division into preparatory and final operations enables a more even material flow, reducing the risk of internal defects and optimizing the use of energy during hammer forging.

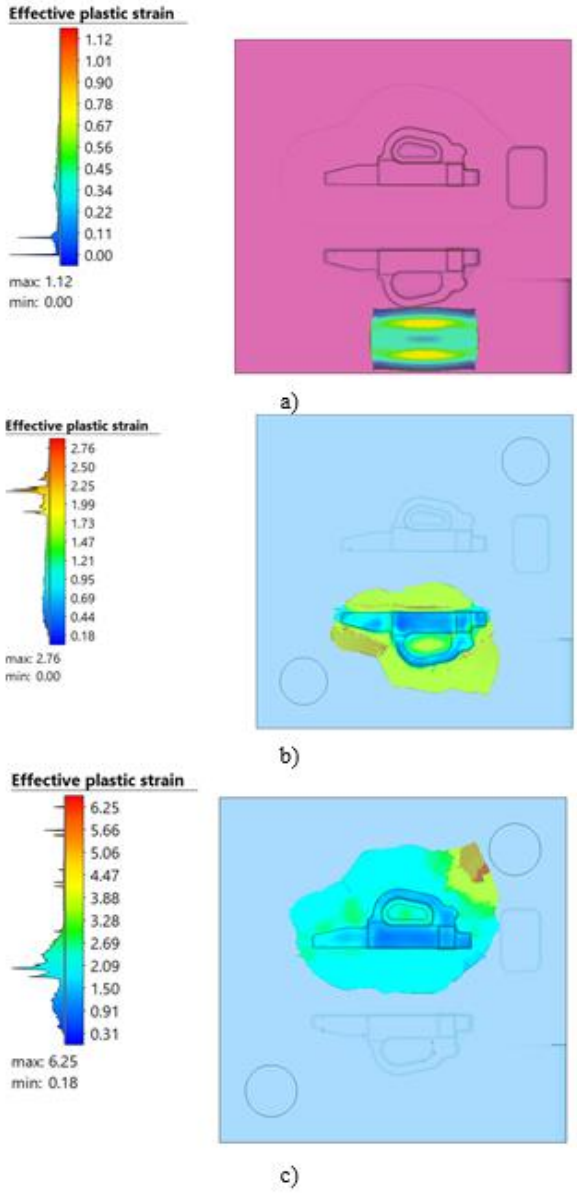


Figure 2. Value of Plastic Strain for first case: a) Upsetting; b) Pre-forging; c) Final forging

Figure 2 shows the distribution of the effective plastic deformation in the final phase of the forging hammer stroke (at 65 mm stroke) for the initial numerical model with a defined friction factor $\mu = 0.5$. The contour display indicates a characteristic inhomogeneous deformation field, typical for hot forging processes of asymmetric parts in open molds. The maximum value of the effective plastic deformation reaches a value of 6.25 and is strictly localized on the periphery of the forging, in the zone of crown formation (upper right segment figure 3c). This high value is a direct consequence of the intense shearing of the material that leaves the central engraving under pressure and enters the narrow rim channel. This peripheral ring of highly deformed material creates the necessary hydrostatic resistance, which forces the remaining volume of 42CrMo4 steel to fill the deepest segments of the tool, such as the trigger guard zone.

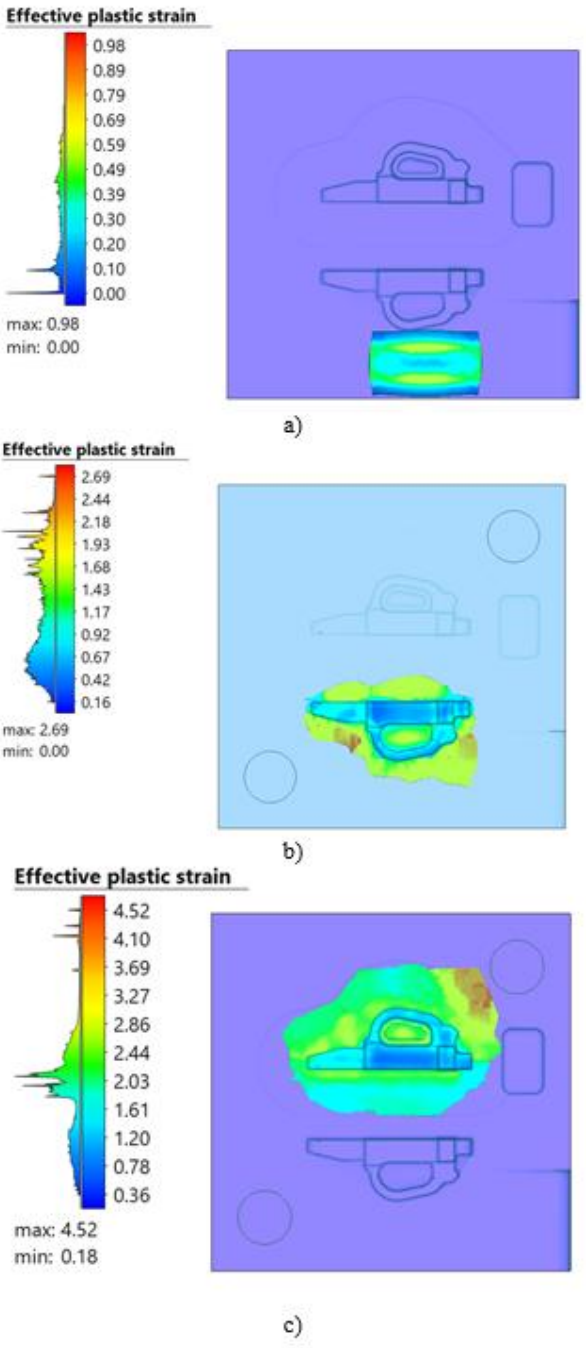


Figure 3. Value of Plastic Strain for second case: a) Upsetting; b) Pre-forging; c) Final forging

In contrast to the rim zone, the working part of the forging (central zone) shows a significantly lower and more homogeneous deformation profile. This uniform distribution in the core of the component indicates a stable material flow without the appearance of critical zones that could lead to internal defects, such as folds or cracks. However, although the deformation field in the central part is stable, this basic network model showed signs of numerical instability at the very end of the stroke

(Increment 2143). This served as the basis for further optimization of the remeshing conditions and the size of the final elements in subsequent iterations, especially in analyzes with more realistic, higher friction factors.

In order to achieve more realistic forging conditions, which include imperfect lubrication and high flow resistance in contact with the tool, an optimized numerical model with a friction factor of μ = 0.65 was created. Figure 4 shows the distribution of effective plastic deformation at the very end of the process (at 65 mm stroke). In order to preserve the stability of the calculation under such rigorous conditions, a fine spatial discretization was applied (mesh size of 1.5 mm) with a stricter nonlinear criterion of mesh refresh (remesh) at a plsatic strain increment of 0.4.

Unlike the base model, the maximum recorded effective plastic strain value is now 4.52. This drop in the maximum value is an indicator of the exceptional stability of the solver; namely, the finer mesh and stricter remesh criterion successfully eliminated local numerical anomalies and "lock-in" effects at sharp radii, enabling smooth mathematical tracking of material flow.

As expected, the zone of maximum deformation (red spectrum) is still localized in the peripheral ring of the forging part. The main progress of this model is, however, presented in the centre, working zone of forging. The deformation field in the region of the rifle body and trigger guard is significantly more homogeneous.

This stable and visually continuous material flow under high friction conditions proves the robustness of the applied meshing strategy. The obtained results confirm that the optimized model is free from non-physical peaks, thus providing highly reliable initial data for any further thermo-mechanical analysis of forging tools and the assessment of their service life.

The obtained simulation results confirm that numerical analysis represents an efficient tool for the investigation and optimization of industrial forging processes. The developed model made it possible to evaluate the influence of selected technological parameters without the need for costly experimental trials.

The comparison of the analyzed cases indicates that friction conditions have a significant influence on the material flow and deformation behavior during the forging process. Variations in friction affect the contact conditions between the dies and the workpiece, which consequently influence the stress distribution, effective strain and overall process performance.

The application of damage and die wear criteria contributes to a better understanding of potential critical zones and provides additional information for improving tool design and technological parameters.

The presented approach demonstrates that CAE methods can support the development of industrial forging technologies by reducing development time, improving process reliability and contributing to higher product quality.

For a more detailed analysis (Figure 4 and 5) of the state of deformation, mapping of the effective plastic deformation was carried out using measuring points placed at characteristic places inside the working part of the forging. This approach made it possible to precisely monitor the intensity of deformation in the zones of interest, which eliminates the influence of numerical extremes localized on the rim and gives a more realistic insight into the distribution of deformation within the piece itself.

Case 1 ($\mu=0.5$), Figure 4: Application of a higher friction coefficient resulted in a more homogeneous deformation profile in the same zones, with values ranging from min 0.38 to max 2.98. The reduction of the maximum deformation values in the working part of the forging confirms that the decreased friction limits the free flow of the material, which achieves a more stable filling of the engraving and reduces the risk of internal defects due to uneven flow.

Case 2 ($\mu=0.65$), Figure 5: In the central zones of the forging, recorded values of effective plastic deformation range from min 0.36 to max 3.37. This distribution indicates a more intensive penetration of the material into the engraving of the tool, but also potentially greater variations in the deformation field due to bigger frictional resistance.

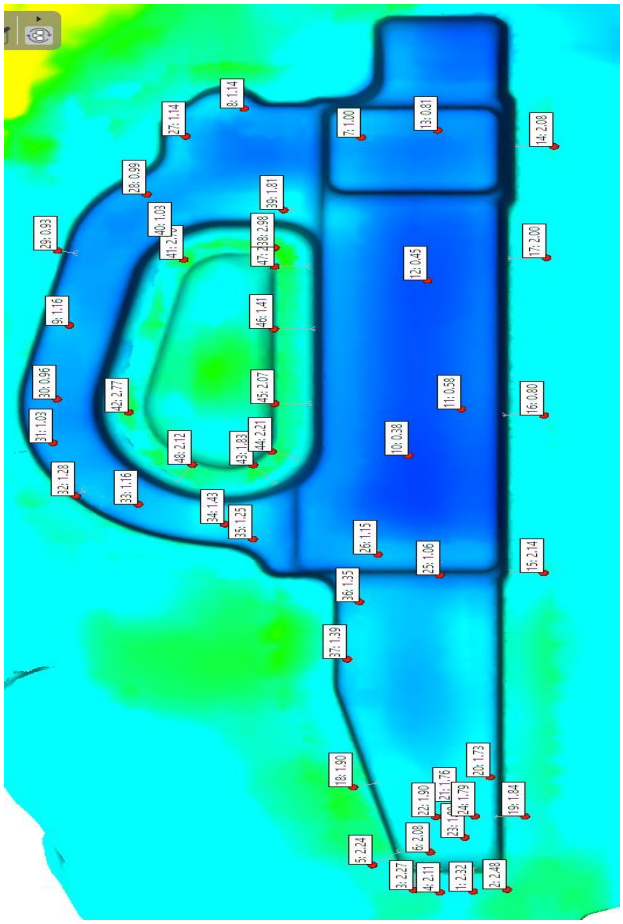


Figure 4. Plastic Strain point on forging part first case

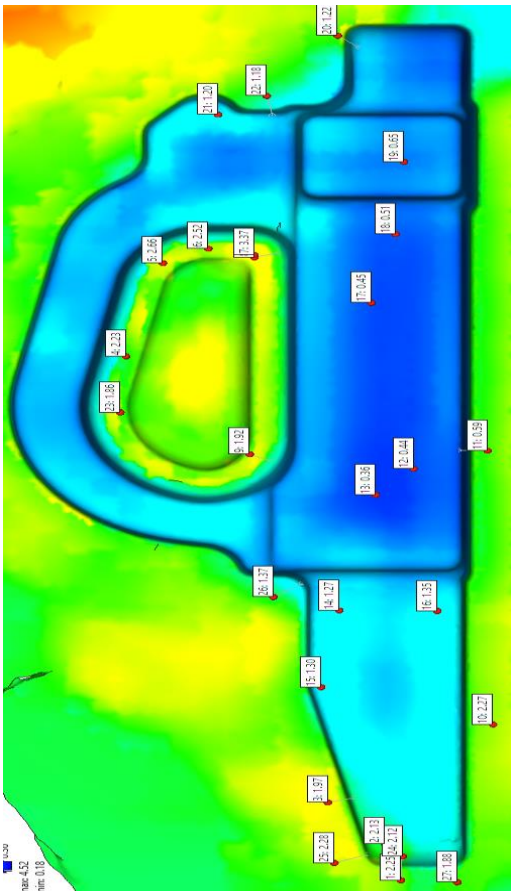


Figure 5. Plastic Strain point on forging part second case

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. The comparative analysis indicates that the results obtained by the FVM solver are fully comparable with the experimental data, thus confirming their applicability in industrial tool design. The key contributions of the research are reflected in the following: It has been confirmed that the precise definition of friction conditions, using the shear model with the factor μ =0.65, is the basis for a reliable simulation of wear. The application of the Simufact wear model enables the early detection of critical zones on the tool, which directly affects the increase of the service life of the dies. The analysis showed that the independent control of the movement of the punch significantly reduces the risk of material defects, such as "cold joints" (cold shut), which significantly improves the quality of the final product. Based on the above, it can be concluded that the integration of modern numerical simulations and experimental verification provides a comprehensive tool for forging process optimization. This approach enables the reduction of development costs, faster production and guaranteed high-quality production of parts with complex geometry.

The results of this study show that the conditions of friction have a major influence on the hot forging process. A higher friction factor results in a more restricted material flow, which increases the contact pressures at the interface between the tool and the workpiece, and in turn results in higher tool wear. Simultaneously the maximum effective plastic strain is reduced due to the reduced ability of the material to flow freely in the die cavity. The findings confirm that friction is one of the most important technological parameters affecting the stability, efficiency and the general quality of the hot forging process.

Future research may include a more extensive parametric analysis involving different process conditions, as well as the validation of numerical results by comparison with experimental and industrial production data. Such an approach would further improve the accuracy of the simulation model and contribute to the development of more efficient forging technologies.

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

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

[3] DELIĆ, M., DELIĆ, M.: *Analysis of the influence of hot forging process parameters on tool wear using the finite element method,* Faculty of Engineering, Kragujevac, 2025.

[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.