State of the art in the field of cold forging tools

Ilija Varničić[1*], Miloš Pjević[1], Mihajlo Popović[1]
[1] Department of Production Engineering, Faculty of Mechanical Engineering, University of Belgrade (Serbia)

This paper chronologically presents the development of the geometry of cold forging tools, as well as the factors that influence the tool lifespan. Tools for plastic deformation processing are of strategic importance for large-scale and mass production. Their lifespan and properties have a great impact on process productivity and product quality. Cold forging represents an efficient metal processing technique through plastic deformation with significant material savings. One of the main challenges of this processing technique is managing high contact normal stresses in the tool, which can lead to serious tool damage. In order to better understand tribological phenomena in the cold forging process, which involve tool wear, fatigue, and failure, this paper will present a practical example of how to improve tool lifespan from the perspective of tool design and construction. The example taken is a forging tool for manufacturing a screw in four operations. The distribution of contact pressure in the contact zone between the working tool and the workpiece was examined using the finite element method for each of the geometries tested. This allowed for a comparison of the service life of the matrix for each of the tested matrix geometries.

Keywords: Cold forging, Tool geometry, Wear, Finite element method

1. INTRODUCING

Metalworking tools for cold forging are very important both for production costs and for the quality of the final product. In fact, the service life of the tool is a fundamental factor that needs to be considered in the process of designing and optimizing the forging process. Cold forging is one of the most significant forging processes and is widely used for the production of small parts where high precision and good surface quality are required. The products resulting from the cold forging process are often used directly in operation after their production, without the need for further processing. In this process, the material is introduced and deformed inside the mold cavity under the force of the press. Depending on the complexity of the process and the main processing factors, cold forging can be performed in multiple operations.

2. THE INFLUENCE OF THE MATERIAL REDUCTION DESIGN ON THE SERVICE LIFE OF COLD FORGING TOOLS

When designing a cold forging tool, it is necessary to consider that the design solution is efficient and applicable, taking into account that depending on the complexity of the product, the cost of the tool amounts to approximately 5-15% of the total production cost. The initial damage to the tool is based on plastic deformation, wear, and fatigue of the working surfaces of the tool. The necessary information for analysing tool fatigue is:

- identification of the most stressed zones;
- material fatigue properties;
- micro-structural properties, grain size and presence of micro-cavities

The mentioned critical phenomena that occur during tool operation are caused by high loads, which in some cases amount to up to 2000 MPa, inadequate lubrication, material deposition due to adhesive friction, and high stress rates. Due to these critical phenomena, the materials used to make the dies and punches must have high tensile strength and wear resistance. Wear is the loss of material from both the tool and the processed material due to high normal stresses.

Dynamic cyclic loading of the tool during material forging causes the formation of cracks and early failure of the die, which is classified as fatigue failure. This is precisely why finite element simulation models are used for tool construction to make necessary design modifications and mold efficiency. With the development of software packages, methods for simulating metal forming, such as Deform, CFTC Form, QForm, have been improved. With the help of these software tools, it is possible to determine the stress distribution on tools and make certain modifications to the tool's construction. The application of these simulation models significantly speeds up the design process and results in significant savings on experimental testing of the constructed tools [1].

By using the finite element method to investigate the distribution of normal stresses through the application of Von-Mises theorem, it has been concluded that the stress in the mold was reduced from 1535 MPa to 1050 MPa by reducing the value of the radius in the forging matrix. An example of correcting the transition radius is presented in the case of a matrix made of an insert that is made of hardened metal or a steel carrier [2].

In the works [2], the effects of the surface finishing of tools on the tribological properties of the hard metal G55 were investigated. After EDM treatment, diamond paste polishing with a grain size of 15 μm was found to reduce the friction factor by up to 0.03, while polishing with a grain size of 1 μm reduced the effect of friction to a negligible level. However, it is necessary to emphasize that reducing surface roughness also reduces micro-dents where oil can accumulate, which can lead to a counter-effect and increase wear.

*Corresponding author: Kraljice Marije 16, 11120 Belgrade, Serbia, ilijavarnica@gmail.com

By testing the process of forging with multi-stage tools, which is performed by transferring the part formed at the first station to the next work station using movable grippers and further forming it inside the tool in the next cycle, it has been found that besides the rounding radius, the material reduction angle also affects the service life of the die.

In Figure 1, the sequence of material deformation from the blank to the final shape is shown, which is carried out in five stages.

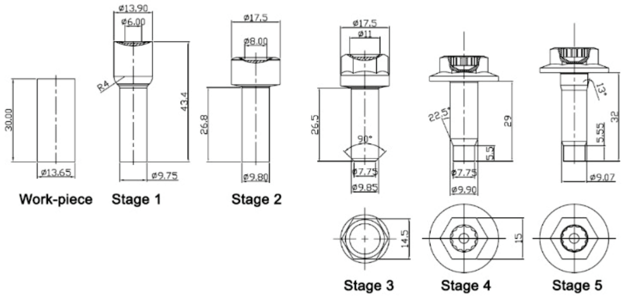
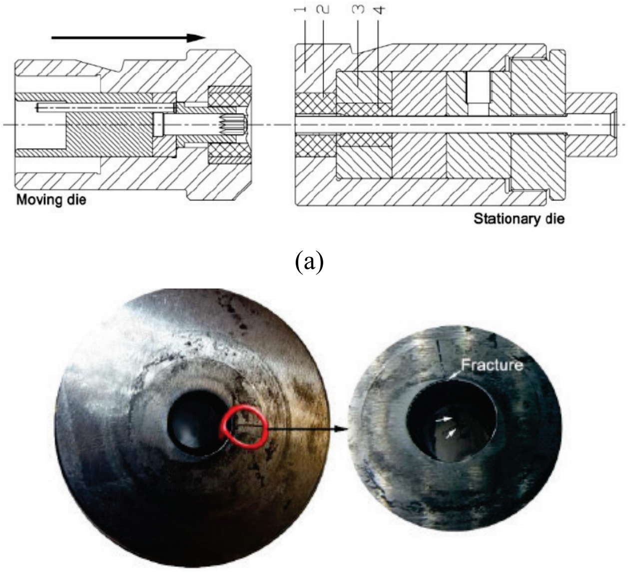


Figure 1: Sequence of material deformation in five stages

In this case of forging a screw with complex geometry and reliefs in a practical example, a critical tool failure occurred in the fourth stage of forging.

For better understanding and analysis of this problem, Figure 2a. shows a set of tools for the fourth forging phase, consisting of a movable part that forms the head of the screw and an immovable part where the shaft of the screw is formed, and where defects often occur. Figure 2b. shows a faulty tool with a broken detail at the transition radius, which further extends in a radial direction along the entire inside of the tool. This defect on the tool is a result of excessive load, therefore it is necessary to perform an analysis and optimization of the tool geometry.



(a)

b)

Figure 2. Example one stage tools [2]

The optimization and reduction of normal stresses in the tool is carried out by [1] numerical modeling of the tool assembly using the Simufact finite element software package. Thermal disturbances in the working zone due to increased tool friction, which affect the change in the flow stress of the material, have also been taken into account.

In figure 3a, a simulation model of the tool assembly is shown, as well as the distribution of normal stresses in figure 3b, at critical points of the working tool prior to structural changes. Figure 4 depicts the optimized tool model that was achieved by dividing the tool into two parts that together form the finished part during the forging process, identical to the initial tool design. The modified design increases the tool's lifespan, eightfold in terms of cycles, from 35,000 cycles per tool to 135,000 cycles per tool.

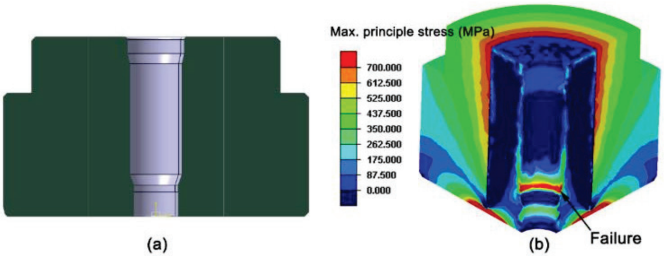


Figure 3. Example of tool before modification [2]

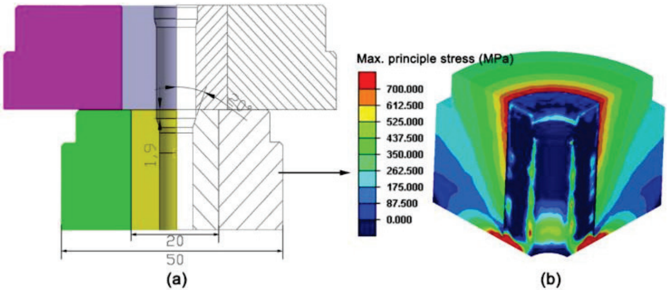


Figure 4. Tool design after modification [2]

For a better understanding of the stress state in the reduction zone, a brief analysis of the stress distribution in different reduction stages is presented.

The forging process originally constitutes a very delicate fracture mechanics problem, due to the cyclic multi-axial loading conditions and the complex mixed-mode loading. For simplification, it is assumed that the velocities of the process are relatively low, so that the problem can be changed to rather straightforward static FE analysis.

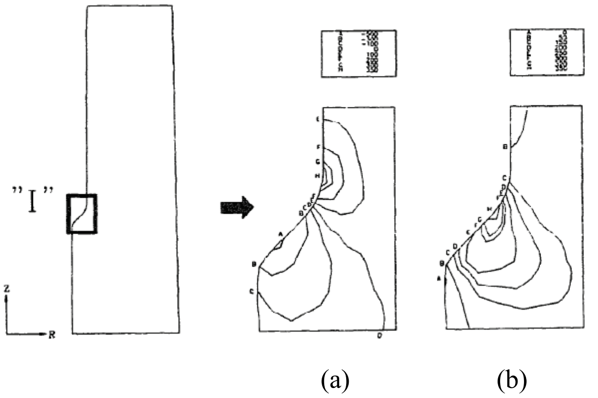


Figure 5. Stress distribution: (a) axial stress, (b) shear stress

The position of fatigue crack initiation is assumed to be in the region where the maximum stress occurs in the die during the extrusion process, as suggested in several studies [7, 8]. Whilst the crack initiation can be calculated in a single analysis from a pre-existing surface crack, the simulation of fatigue crack propagation should be described with an incremental procedure. A considerable amount of work has been done with the application of the FEM in fatigue crack propagation analyses. The main goal of these researches has focused on the determination of the stress intensity factor. If a special element is introduced in order to consider the singularity of the stress-strain in the vicinity of the crack tip, as employed in some notable studies, a more accurate solution can be obtained. For this reason, the so-called quarter point technique is used in this study. Applying isoperimetric standard elements with mid-side nodes at the crack tip according to this method, the singularity found there can be modeled easily by moving the mid-side nodes of the element towards a quarter point position in front of the crack tip. This is the way that an accurate stress intensity factor value can be obtained with a rather coarse mesh. Although the method of stress intensity factor evaluation may be carried out in various ways, it has been found in previous studies that the direct method of calculating the stress intensity factor from the displacement field at the vicinity of the crack tip leads to good results [6, 7]. The stress intensity factor, therefore, is calculated in a similar way in this study.

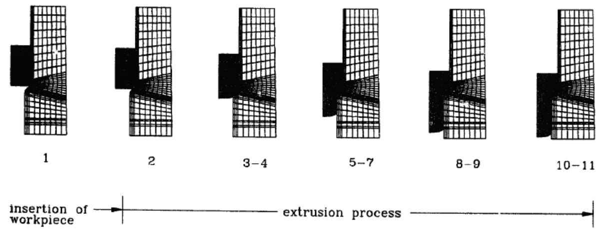


Figure 6. The stages of forming of most interest in calculating the stress intensity factor

It is very important to note that, since the correct crack tip behavior is only modeled on the special elements on the crack tip, the length of these elements is significant [9, 10]. Therefore, an *L*/*n* ratio is selected to be between the scale from one to ten [4], where *a* and *L* are crack length and the element length at the crack tip respectively and *n* was number of finit element. Figure 7. shows the typical mesh system of the die containing a certain crack, which was used in the analysis. It is shown that a fine mesh is used around the region containing the crack for accurate analysis. For illustration purposes, the left side of the axi-symmetric model was added. For the case of 0.1 mm crack length, the stress distribution from the vicinity of the crack tip to the horizontally straight points in the radial direction is plotted in Fig. 7. As shown in this diagram the stress value around the crack tip is very much higher than at other points.

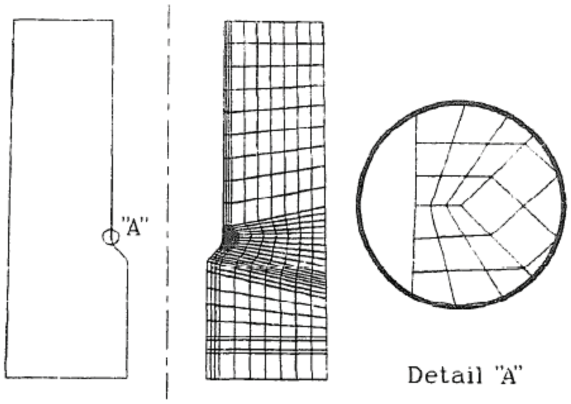


Figure 7. Mesh system for the fracture analysis by the FEM. [10]

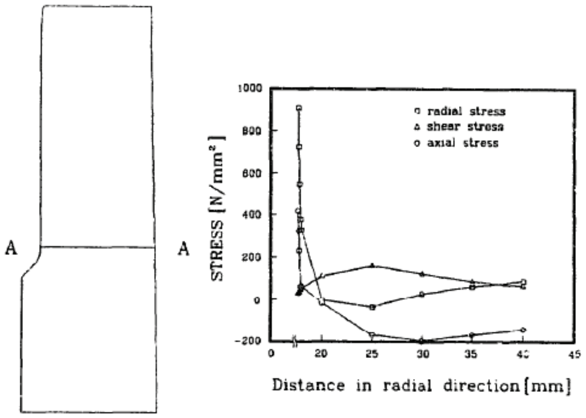


Figure 8. Stress distributions in the vicinity of the crack tip and other places for a 0.1 mm crack. [11]

The diagram of figure 9. shows examples of the stress intensity factor evaluation at the crack tip with increasing crack length. The graphs are obtained from the interpolation of all process stager instigated and show a function of process time for single loading cycle. It can be seen that the influence of mode I decreases as the crack propagates [11].

The effective stress intensity factor, which can be obtained by applying some results of figure 9. is calculated in advance to evaluate the fatigue life. Figure 10. shows the computed effective stress intensity factor exceeds the value of fracture toughness in the early stages but then drops below it with increasing crack length. From this date the crack growth rate, as well as the crack propagation direction, can be determined, suggesting the propagation of the crack for the following crack growth increment, which gives the new crack tip coordinates. For the next analysis run, remeshing of the FE model will be conducted.

The fatigue life obtained using the stress intensity factor and the effective stress intensity factor for a single crack length indicates part of the total fatigue life in the forging die. The desired fatigue life, therefore, is obtained by the summation of each result.

The surface failures of dies observed in the actual manufacturing process are about 0.01÷0.1 mm in depth. the first crack increment being taken as AN = 0.1 mm in consideration of this Fact. The fatigue life of the investigated extrusion die was evaluated. This result is shown in Fig. 10, from which it can be seen that l000-12000 cycles are required for the crack length to reach approximately 3.2 mm.

Although the difficult assumption of the initial crack increment and the ignored wedge effect of the penetrated workpiece and of lubricants flowing into the crack may affect the accuracy of result, the simulated result in this study is in good agreement will the results of experiment [11, 12] for fatigue crack propagation behavior in extrusion dies.

Process simulation by FEM is a suitable instrument to evaluate stresses and strains of forming tools. For the deformation analysis of the workpiece and the die, rigid-plastic FE analysis and elasto-plastic FE analysis were performed. For more precise results of tool loading, an internal pressure distribution derived from FE simulation of the forming process was applied. Using these results, the fatigue crack behavior and the fatigue life of the extrusion die were investigated by the introduction of some criteria for LEFM.

Experimental research, according to the study [11], found that the highest stresses in the cold forging process occur at the beginning of the conical profile in the area of the transition radius, Figure 9.

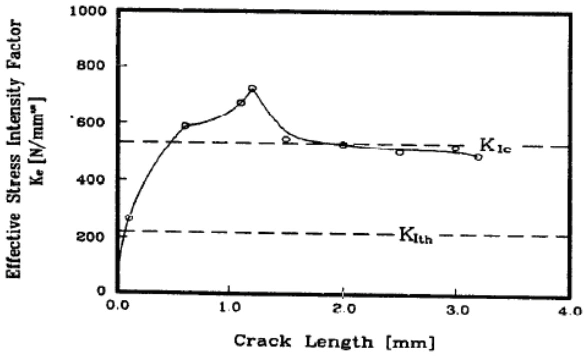


Figure 9. Effective stress intensity factor for increasing crack length [11]

As a consequence of the multiple repetition of the forging process in one tool, the stress that accumulates at the transition radius is distributed. Figure 10. shows the behavior of the crack at the transition radius of the reduction profile depending on the number of forging cycles

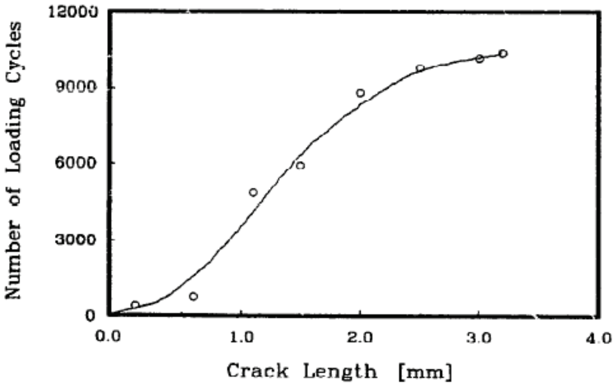


Figure 10. Computed fatigue life of the extrusion die [11]

In addition to modifications in the form of dividing the tool into segmental matrices based on research [11], the reduction of the impact of maximum normal stresses on the transition radius and reducing cones is achieved through the application of sigmoid or conical profile in the zone of intensive material deformation.

An example of the crack propagation path obtained from the above results is shown in figure 11. This diagram points out that the angle between the radial axis and the direction of crack propagation is about 30-40" and that the fatigue crack propagates in a zig-zag path along the radial direction [11].

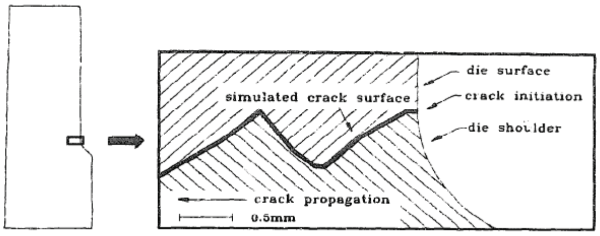


Figure 11. The zone of occurrence and propagation of the crack

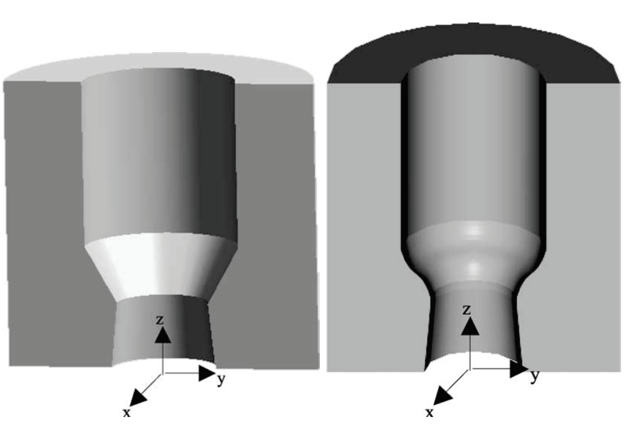


Figure 12. a) Conical profile b) Sigmoid profile

On Figures 12a and 13 is shown the reduction profile that has a conical chamfered reduction zone with an angle of 30°. The modified profile shown in figures 12b and 14 has a sigmoid profile instead of a conical reduction zone. The sigmoid profile is similar to the conical one in terms of the length of the profile and the degree of material reduction, but it is also less favorable compared to the conical profile in terms of complexity and production costs. The advantages of a sigmoid profile over a conical one lie in avoiding sharp variations in material flow and reducing the maximum value of contact stress at the beginning of the conical zone [10].

In the direct extrusion process four steps can be identified: workpiece insertion, start of the extrusion, stationary extrusion condition, workpiece expulsion. The stationary phase is the most onerous for the die resistance. This phase was simulated in axial-symmetric conditions, the matrix and the workpiece were meshed by 1000 and 800 elements, respectively. The simulation results provided the contact pressure distribution along the profile for both dies. In the case of conical matrix, the most stressed zone was at the beginning of the reduction zone where the maximum pressure value was equal to p_{max} ¼ 450 Mpa. In the case of sigmoid profile, the peak values were located at the flex point where there was the maximum value (p_{max} ¼ 417 MPa) and at the end of the tapered zone (p ¼ 409 MPa), Figures 15 and 16.

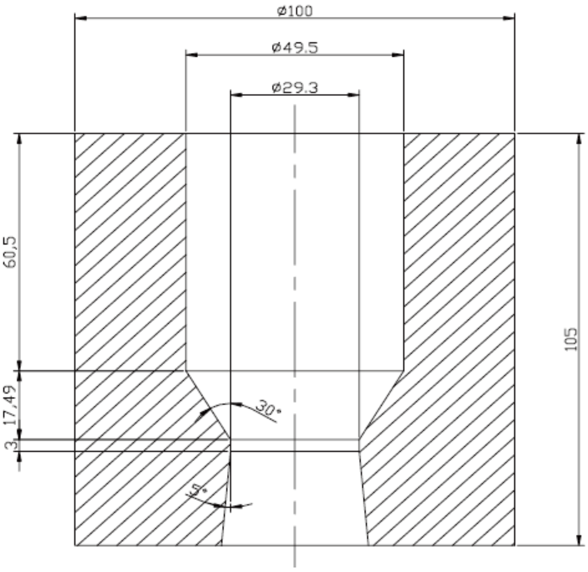


Figure 13. Truncated-conical geometry

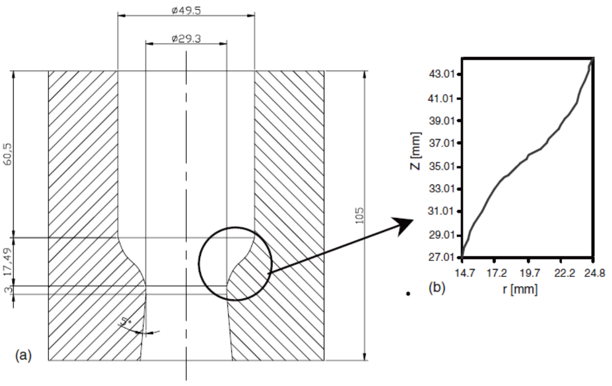


Fig. 14. Sigmoidal die (a); profile detail (b).

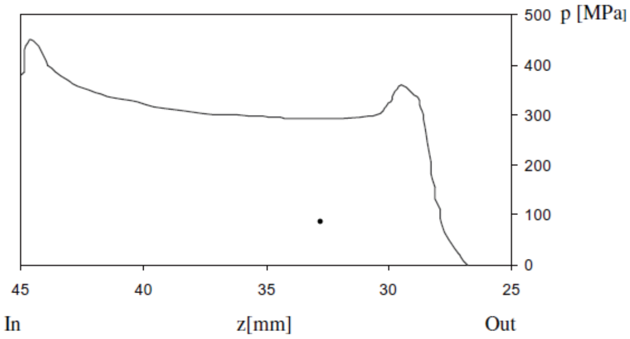


Fig. 15. Contact pressure distribution of the conical matrix.

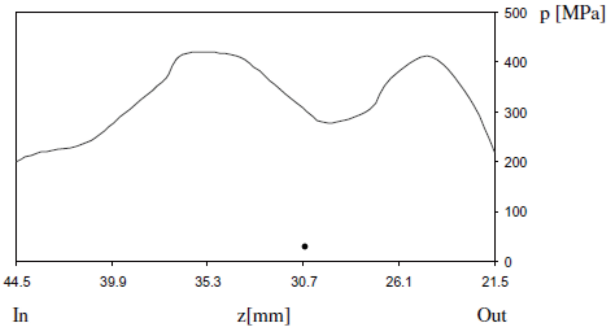


Fig. 16. Contact pressure distribution of the sigmoid matrix

3. CONCLUSION

In this paper, the influence of the material reduction design in cold forging on the die service life was presented. It was found that the reduction profile affects the material flow. As a result of the modification of the forging reduction zone profile, the maximum normal stresses are reduced, which directly affects the intensity of the die wear and, consequently, its service life. Another important factor justifying the modification of the die reduction profile is the improvement of the reduction process. It can be concluded that in the process of designing cold forging tools, the most important aspect is the design of the reduction pressure due to the concentration of stresses in the material reduction zone.

ACKNOWLEDGEMENTS

This work was supported by the Ministry of Education, Science and Technological Development (by contract No.: 451-03-47/2023-01/200105, from the 3th of February 2023.)

REFERENCES

[1] T.-W. Ku and B.-S. Kang, "Tool design for inner race cold forging with skew-type cross ball grooves", Journal of Materials Processing Technology, Vol. 214(8), pp. 1482-1502, (2014)

[2] K. Wagner, A. Putz, and U. Engel, "Improvement of tool life in cold forging by locally optimized surfaces",

Journal of Materials Processing Technology, Vol. 177(1-3), pp. 206-209, (2006)

[3] B. He, "Failure and Protective Measures on Punch & Die for Cold Extrusion", The 2nd International Conference on Computer Application and System Modelling, (2012)

[4] K. Andreas and M. Merklein, "Influence of Surface Integrity on the Tribological Performance of Cold Forging Tools", Procedia CIRP, Vol. 13, pp. 61-66, (2014)

[5] T.W. Ku and B.-S. Kang, "Tool design and experimental verification for multi-stage cold forging process of the outer race", International Journal of Precision Engineering and Manufacturing, Vol. 15(9), pp. 1995-2004, (2014)

[6] S. Yurtdaş, U. İnce, C. Kılıçaslan, and H. Yıldız, "A Case Study for Improving Tool Life in Cold Forging: Carbon Fiber Composite Reinforced Dies", Res. Eng. Struct. Mat., Vol. 3(1), pp. 65-75, (2017)

[7] K. Lange, W. Reiss, H. Arndt, A Study of Tool Fracture in Cold Extrusion, Proc. 17th NAMRC, Columbus, Ohio. (USA), (1989)

[8] K. Lange, A. Hettig, and M. Knoerr, “Increasing tool life in cold forging through advanced design and tool manufacturing technique”, J. Mater. Process. Technol, Vol. 35, pp. 495-513, (1992)

[9] D.R.J. Owen and A.J. Fawkes, “Engineering Fracture Mechanics: Numerical Methods and Applications”, Pineridge Press, Swansea (UK), (1953)

[10] C. Cosenza, L. Fratini, A. Pasta and F. Micari, “Damage and fracture study of cold extrusion dies”, Engineering Fracture Mechanics, Vol. 71(7-8), pp. 1021-1033, (2004)

[11] R.S. Barsoum, “On the use of isoparametric finite finite elements in linear fracture mechanics”, Int. J. Numer. Meth. Eng. Vol. 10, pp. 25-37, (1976)