INNOVATIVE MECHANICAL ENGINEERINGISSN 2812-9229 (Online)
University of Niš, Faculty ofMechanical Engineering
VOL. 3, NO 2, 2024, PP. 1-13

Original scientific paper*

INVESTIGATION OF HOT FORGING PROCESS BY FEM SIMULATION

Saša Ranđelović[1], Tanja Miladinović[2], Srđan Mladenović[1], Vladislav Blagojević[1], Nikola Vitković[1], Predrag Janković[1], NikolaKostić[1]

[1]University of Niš, Faculty of Mechanical Engineering, st.Aleksandra Medvedeva 14, Niš, Serbia, [2]MING Kovačnica, st. Bulevar 12. Februar 95, 18000Niš, Serbia

Abstract.*The process of hot forging of steel is one of the most common industrial technologies. Designing this technology is a challenge because today complex geometries of finished products are obtained. The goal of FEM simulation is to generate the most realistic contact between the tool and the deformable material, both in volume and in the immediate vicinity of the boundary surfaces of the tool. FEM simulation by Qformof the hot forging process is a good way to prevent possible errors and problems of tool construction. Using the example of a complex forging, the paper analyzes the geometrical conditions of contact, the stress-deformation state within the deformable volume, and the constructive solution of the tool for hot forging in a closed die.*

Key words:*Hot forging, Contact nodes, Nonlinear FEM simulation, Closed die, Qform.*

1. INTRODUCTION

FEM simulation of the hot forging process represents a software method of developing and designing hot forging technology that today gives the best results in industrial conditions.The feasibility and accuracy of the FEM method are highly dependent on how node spacing is controlled within the domain. The density of the nodes must be controllable over the entire meshed region. Geometric constraints on node boundaries must be satisfied as it is often necessary to specify that some nodes be located precisely on specific points and curves inside the domain, such as in cases where some boundary conditions will be imposed on these geometries in analysis.Therefore, it is indispensable precisely for complex prismatic forgings, where large amounts of excess material are needed in order to fill even the outer most volume inside the mold cavity.The modeling of the contact between the deformable material and the rigid surface of the tool comes to a special expression.Precisely that moment when the tool is completely filled

---

***Received: June14, 2024 / AcceptedSeptember30, 2024.**

Corresponding author: Saša Ranđelović
University of Niš, Faculty of Mechanical Engineering
E-mail:sassa@masfak.ni.ac.rs

and when the forging material at elevated temperatures begins to flow over the transition bridge is the most interesting for designing.This gives the exact geometry of the forging, which is a prerequisite for obtaining the designed finished part.

Due to the finite dimensional nature offinite element analysis and the approximation of the true domain used to represent the dieand workpiece geometries, there will always be approximation errors in the solution. In CAD systems, curved geometries are typically represented as parametric curves and surfaces.On the other hand, a method based on FEM analysis requires a different type of geometric representation mesh. Toimprove the overall solution accuracy and make the simulations realistic, these errors need tobe detected and a new mesh must be generated when these errors become large. The keysources of errors associated with the finite space approximation in the modeling of forming problems [1] are:

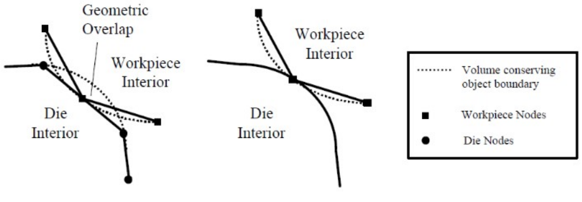
- geometric approximation errors,
- element distortion errors,
- mesh discretization errors, and
- mesh rezoning errors.

In the following sections, we will examine each of these types of errors and metrics to indicate when the error is becoming large and provide techniques to improve the accuracy and make FEM simulations more realistic. FEM simulation continuous remeshing is essential in regions of high deformation and strain/stress rates which may shift, as the original mesh becomestoo thin to yield accurate results. From the description presented in this chapter, it will be clear that the FEM solver can monitor the error indicators due to geometric approximation, element distortion and mesh discretization as the simulation progresses. Hence, they can be used as the basis to decide if the workpiece needs to be remeshed. The remeshing capability allows one to regenerate and/or modify a mesh only in desired regions while the other portions of the mesh remain unchanged. The reason why this capability is practically important lies in the fact that the amount of computation required for local remeshing is less than that for remeshing the whole domain, and particular mesh pattern, or node-spacing distributions, are known to work well for certain types of geometric features, so these pre-registered mesh patterns can be easily superimposed in the mesh.The decision to remesh will depend on the criteria used to detect errorsand activate a remesh.

However, others have considered this area and the tools developed in [2, 3] have beenrelied upon for the interpolation of solution variables onto the remeshed workpiece.Geometric approximation errors exist because a finite element mesh approximates theworkpiece and the dies during analysis. Such an approximation leads to aninaccurate representation of the die-workpiece interaction and the workpiece free surfaceduring the simulation. This could potentially have a serious impact on the material flowcomputations resulting in an inaccurate prediction of the workpiece deformation patternand die fill. Small forming defects such as laps (where the material folds onto itself)could go undetected, and workpiece remeshes based on the approximation could lead tovolume loss or gain. Each of these issues is explained below.

The impact of geometric approximation on die-workpiece interaction is shown in Fig.2. Typically, the contact algorithms in finite element solvers [2,3] ensure that workpiece nodes do not penetrate the mesh of the die boundary.

Hence, the contact with the true curved geometry could be initiated much before (or after) the analysiscode is able to detect it. A geometric overlapof the die and workpiece geometries is illustrated in Fig. 1a. Since plastic deformation is a path (strain history) dependent phenomenon, the use of an approximation could potentially modify the computeddeformation behavior for the workpiece.



(a) contact based on die mesh (b) contact based on die geometry

Fig. 1 Geometric approximation errors in the contact region

The reason for this is that, depending on thelocal die curvature and the discretization used to represent that region, portions of theworkpiece (near existing contact regions) are likely to be subjected to more (or less)deformation before the solver determines new contact regions. This is illustrated in Fig. 2, which shows the additional die-stroke for the top die before the free surface ofthe workpiece contacts the bottom die (exaggerated to illustrate the point). This impliesthat some regions in the workpiece would have a different strain history when anapproximation of the die is used in the analysis as opposed to case when the true curvedgeometry is used during computations. A different strain history would result in adifferent future path of deformation (loads, material flow).

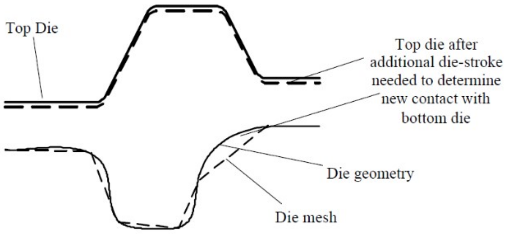


Fig. 2 Imact of die geometry approximation on process simulation

Thiseffect is illustrated for the 3-D analysis of a disk-forging problem in Fig. 3, whichshows the variation in the load-stroke curves (in the final stages of the analysis) when themesh and the geometry are used to represent the die during the analysis. The data hasbeen culled from simulation results performed with the software[2, 3] and discussed later in this document. The load-stroke curves differ more during the finalstages when the material is beginning to flash and all of the corners (curved regions of the die) begin to fill up. Note that the load-stroke curves do vary throughout the processsimulation though only the variation for the final 15% of the stroke is plotted in Fig. 3.

As the simulation progressesand the workpiece is remeshed several times, these errors could potentially result in a redistributionof workpiece volume causing die underfill or overfills and in a different loadstrokecurve for the process (as compared to the case when the true curved geometry ofthe die is used). Note that the issue of volume re-distribution refers to the variation in theworkpiece shape for the two (die mesh versus die geometry) scenarios. Obviously, thediscretization of the die affects this variation with a finer discretization reducing it.

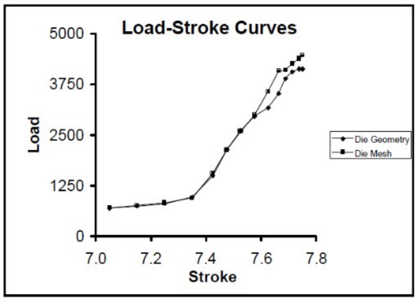


Fig. 3 Load stroke curves (final stages) for disk forging problem

An example of a potential situation where an inaccurate representation of the workpiecefree surface during the simulation causes a forging lap to be missed is illustrated in Fig. 4. The geometry of the free surface of the workpiece shows the formation of thelap. However, the discretization generated does not capture this feature on the workpiece boundary.Further simulation with this discretization will result in an erroneousprediction of a process that does not produce any defects [4, 5].

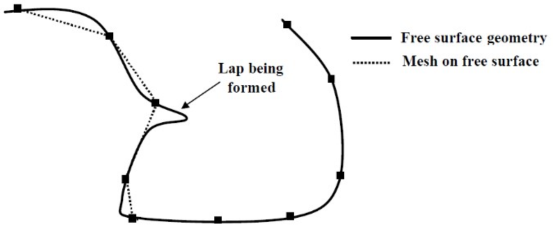


Fig. 4 Workpiece free surface approximation in vicinty of a lap

The use of the finite element mesh of the workpiece as the basis for generating the newworkpiece mesh (during a remesh) can potentially result in volume loss/gain. Anextreme scenario of volume loss is illustrated in Fig.5below. It is clear from this figurethat each remesh can potentially change the workpiece volume. Obviously, this can havea very serious impact on the accuracy of all predicted simulation parameters. The issueof volume loss during remeshing is discussed further in the next chapter dealing with howto remesh the deformed workpiece [3, 4].

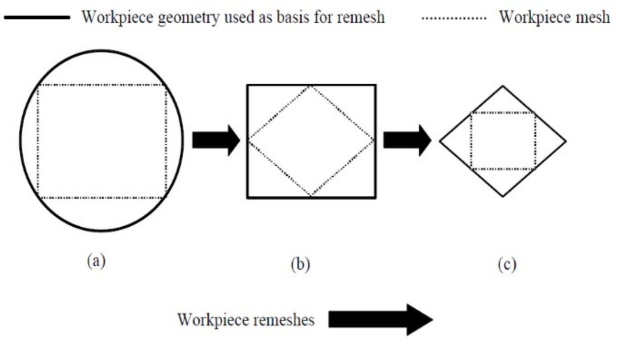


Fig. 5 Workpiece volume change due to remeshing.

In an application where one of the primary goals is the prediction of workpiece materialflow during deformation, geometric approximation errors have the potential of yieldinginaccurate results. In a typical 3-D modeling environment where the workpiece and diesare represented as mesh models with straight edged elements, the severity of thegeometric approximation errors strongly depends on how the discretizations of the workpiece and the dies are controlled. Having identified the situations that can beattributed to geometric approximation errors, the rest of this section focuses on actionsthat will reduce these errors thus making the simulations more realistic [4, 5].

Geometric approximation errors can be reduced, for example, by using the true geometryof the die as its representation during the analysis. This implies that the die-workpieceinteraction illustrated in Fig. 1a and resulting in geometric overlap would beeliminated and represented by the model shown in Fig. 1b. This can beaccomplished by using the geometric definition (lines, arcs, splines in 2-D and planes,complex surfaces in 3-D) of the boundary of the dies (instead of the mesh boundary) in the geometric checks performed during contact computations in the solvers. Thesegeometric checks determine the proximity of a node to the boundary of a die. The use of the truegeometry of the die is typically implemented in most commercial 2-D solvers [4, 5] andin the 2-D automated systems discussed in published literature [5, 6]. However, this isnot common in 3-D solvers due to the complexity of the algorithms to perform thegeometric checks with complex surfaces. None of the published articles describeimplementations of 3-D automated systems [7,8] with the capability of dealing withthe geometry of the die. All these systems assume that the geometry of the die isdefined by its finite element mesh. However, with the open environments provided by commercial CAD systems, procedures to query the geometric model and performgeometric computations are available and hence, do not have to be duplicated. For thisstrategy to work, the contact algorithm in the solver must be modified to perform contact checks using the die geometry from the CAD system. Hence, the contact algorithmshould now check for contact between workpiece nodes and the geometric model of thedie (instead of the mesh model). This approach has been implemented within theautomated system developed in this research. The main goals of this paper is the modelling geometry transformation by the finite element method at the hot forging technology.

2. FEM AND CONTACT MODELLING

Geometric approximation errors can also be reduced by building a smooth geometricrepresentation of the workpiece (from the deformed finite element mesh data) and usingit as a basis for generating the new mesh. With reference to Fig.5, this would implythat before the mesh in Fig. 5b is generated, we would build a smooth geometric representation based on the deformed mesh of Fig. 5a, thus recreating the circularboundary in this case. Thus, when the smooth representation is discretized, there wouldbe no loss of volume. This issue of building a smooth representation for the workpiece isdiscussed further in the next chapter, which deals with how to remesh the workpiece.

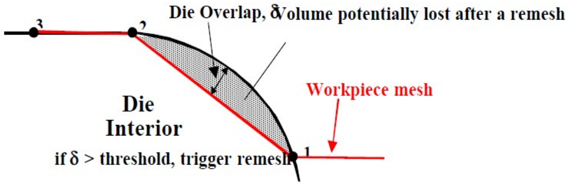


Fig. 6 Monitoring die overlap

Monitoring the “die overlap” can further reduce geometric approximation errors. The dieoverlap measures the overlap between the workpiece (usually represented by its finiteelement mesh) and the die (mesh or geometry). A typical situation with die geometry andstraight-edged workpiece mesh combination is illustrated in Fig. 6. After theworkpiece is remeshed, some or all the volume represented by the shaded region in Fig. 6 may be lost (or gained depending on the local die curvature) since nodes on themesh representing this contact portion of the workpiece are forced to be in contact withthe die. Hence, to avoid drastic changes to workpiece volume, such situations must bedetected [4, 5].

A simple procedure to monitor die overlap in 2-D simulations is explained with reference to Fig. 6. The procedure would check the distance between the mid-point of the finiteelement edge e1, whose endpoints (nodes 1and 2) are in contact with a die, and the diegeometry. If this overlap, δ, is greater than a threshold, a remesh is triggered. It isimportant to note that this procedure only attempts to keep volume loss (due toremeshing) under control by careful monitoring and does not ensure volumeconservation. Though it is not illustrated in Fig. 6, monitoring die overlap will alsoreduce geometric approximation errors when a finite element mesh represents the dies [4, 5].

The procedure described above is a simple procedure to estimate the overlap and wouldclearly be unsuitable in situations such as the one illustrated in Fig. 7 where the dieoverlap would be computed as zero. However, (it is assumed that) these types of geometric situations occur infrequently in the simulation of an industrial forming processsince the workpiece mesh is likely to be finer than the features on the die. Also, the diestypically do not have small features since they can present die fill problems and result inan increase in forming loads. This algorithm can easily be extended to and implementedfor complex 3-D geometries due to the availability of the modeling functionality via theopen environments provided by the commercial CAD systems. The die overlapmonitoringalgorithm described above has been implemented within software [5-7].

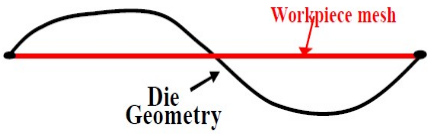


Fig. 7 Problem geometry for the overlap-monitoring algorithm

Another technique that helps to reduce the geometric approximation errors is meshrefinement around separation points of the contact region. This results in a finer mesh inthe regions that are most likely to come in contact with a die as the simulation (formingoperation) progresses and in a better representation of the workpiece as the material flowsaround the curved surfaces on the die. Brahme et al. havedevised a 2-Dremeshing scheme that implements this approach[8]. However, this is an a posteriori fix tominimize the geometric approximation errors for the next phase of the simulation. Abetter scheme for minimizing approximation errors would be to develop a procedure that“looks ahead” and refines the mesh in regions that are likely to come in contact[9]. For example, Fig. 8 shows the possible introduction of a refinement region where a finer discretization is generated using the center of the region as a refinement point. Such aprocedure would “scan” the boundary of the workpiece (only the free surface) to checkfor proximity to die surface. Portions of the workpiece free surface within a reasonabledistance from the die can be refined to capture the die curvature as the workpiece flowsaround the curved die surface. The algorithm can also be made smarter by tracking nodalvelocity directions to infer regions likely to come in contact with a die[10, 11].

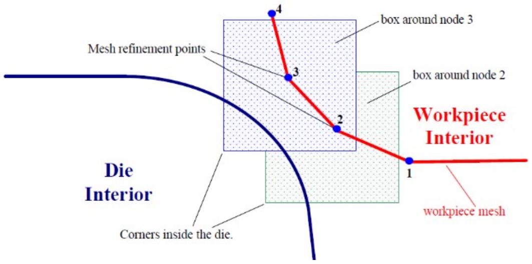


Fig. 8 Illustration of the “look-ahead” algorithm

One such scheme to check proximity with a die boundary would be to define a region, ofpre-defined size and centered at the given node location and check if a portion of this region lies inside a die[12, 13]. The region could be a square/circle in 2-D and a cube/sphere for 3-D applications. Fig. 8 shows a schematic illustration of this scheme using a squarefor the region. If a corner of this box lies inside a die (as shown in Fig. 8 for nodes 2and 3), the location of the node is treated as a refinement point when the new mesh isgenerated[14]. Hence, in Fig.8, the locations of nodes 2 and 3 on the workpiece boundarybecome points in whose vicinity the mesh is refined. This offers a good solutionconsidering the fact material flow cannot be predicted with certainty by evaluation of thecurrent die-workpiece configuration. A similar "look ahead" algorithm, that examinesthe proximity of midpoints of edges on the workpiece free surface, has been implementedwithin the software[15-17].

3. FEM SIMULATION OF HOT FORGING

Hot forging of a pre-bent workpiece is the final technological operation that gives the required shape and required geometry of the forging. The selected product under consideration is a load-bearing cross member used in road vehicles (trucks) and some agricultural machinery.The material of the part is steel S355J2 with initial dimensions of rod material Ø80×550mm, which is heated to 1050°C.The mechanical characteristics of the selected steel are given in table 1.

Table 1 Mechanical characteristics

D[mm]	to 3mm	3-100	100-150
R_m[N/mm^2]	510-680	470-630	450-600
$R_{0.2}$[N/mm^2]	355	320	280

The curve change in true flow stress at elevated temperatures is given by the curves of hardening (Fig. 9)

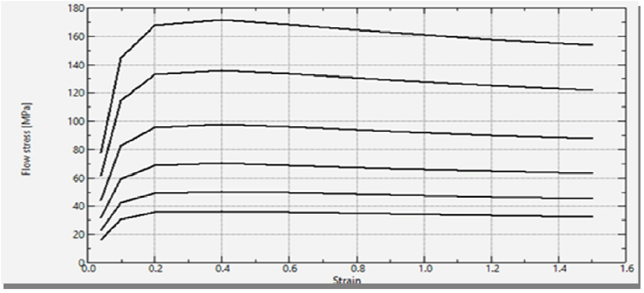


Fig. 9 Flow stress for material S355J2 on temperature 1100 °C

Due to the shortening of the design and process analysis time, the FEM simulation was carried out with half of the preparation, which is quite sufficient to recognize all critical parameters (Fig. 10).

The FEM simulation of the hot forging process shows the uniform filling of the mold cavity and the excess material necessary to obtain the required geometry and accuracy. A key role is played by the transition bridge of the tool, which determines the quality of the forging with its parameters (Fig. 11).

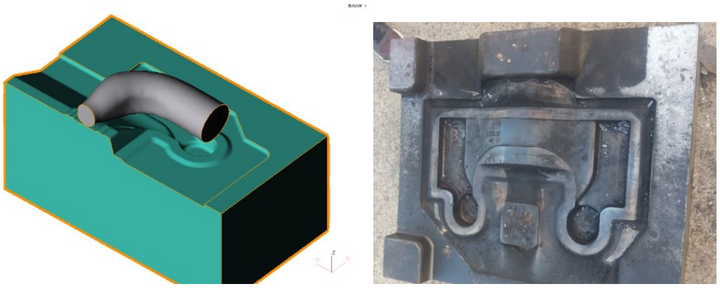


Fig. 10 Setup the bent workpiece in the hot forging tool

These parameters are the transition radius r=3mm at the exit from the engraving tool to the transition bridge as well as the width of the bridge b=12mm, i.e. its height h=3mm at the end of the forging process after 4 strokes. If the quality and accuracy of the forging is not satisfactory, that is, if the mold cavity is not completely filled, the hot forging process itself can correct these three parameters in terms of easier and faster material extraction. The opposite effects can be achieved by making it difficult and preventing the material from protruding (by reducing the radius r and increasing the width of the bridge b) if some part of the volume of the tool is not completely filled (Fig. 12).

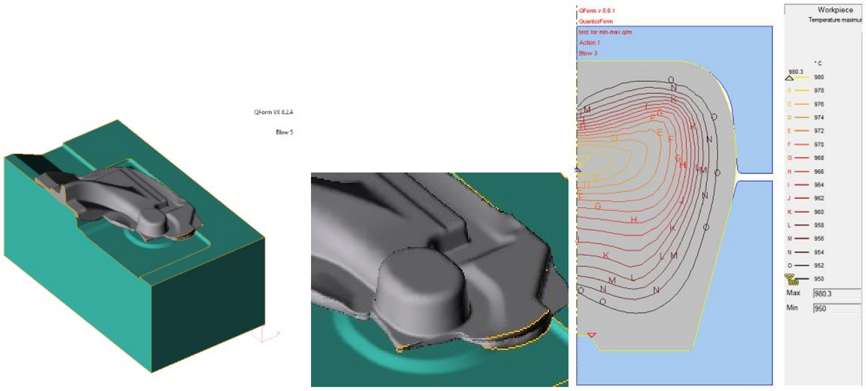


Fig.11 FEM volumetric model of finished part and temperature field

The FEM analysis of the upper part of the tool should indicate the critical places in the tool that suffer the greatest mechanical and temperature loads after a given number of production cycles, cc. 5000. The densest FEM network is at the transition radii and in the zones where the intense flow of forging material is expectedat high forging temperatures. The images show the generated mesh of finite elements by volume, i.e. the mesh of the forging tool itself, since the Qform software enables non-linear FEM simulation of the hot forging process itself, as well as FEM analysis of the load of the tools themselves in the area of elastic deformations.If these loads exceed the limit of elasticity, i.e. contact stresses exceed the strength of the tool, the software warns the designer that damage and breakage of the tool will occur.

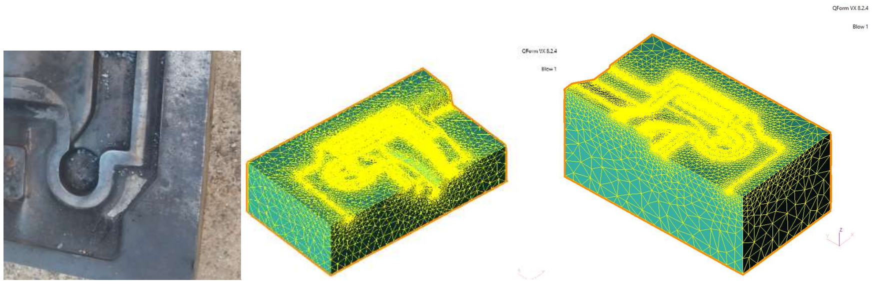


Fig. 12 FEM model for upper and lower tool

Fig. 13shows the change in energy through the first four impacts, i.e. the maximum energy consumption of 112kJ during an impact lasting 0.014s.It is to be expected that the energy during impact remains constant, i.e. that the stroke decreases and the nominal force onthe presser increases as the filling of the tool increases.

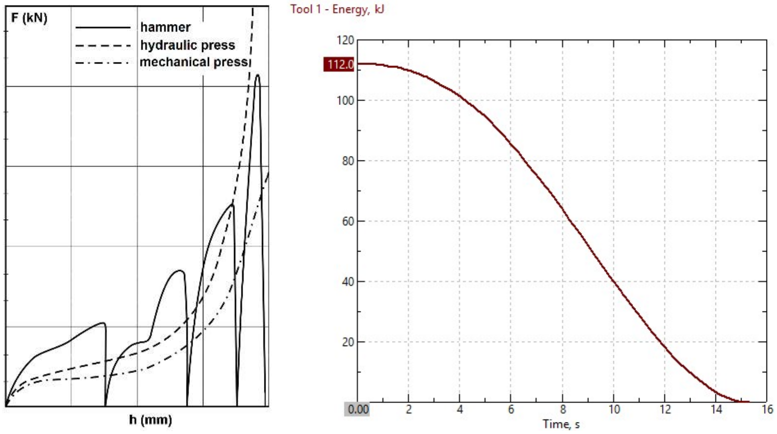


Fig. 13 Forging energy dependence of time, stroke 1

Then, a section is made at impact 4, and the graph of energy consumption changes, starting with 145kJ, itself being of a different shape. At the first forging blow, the tool is not in full contact with the undeformed preparation, and then the largest penetration into the deformable volume is achieved with the smallest achieveddeformation force (Fig.14).

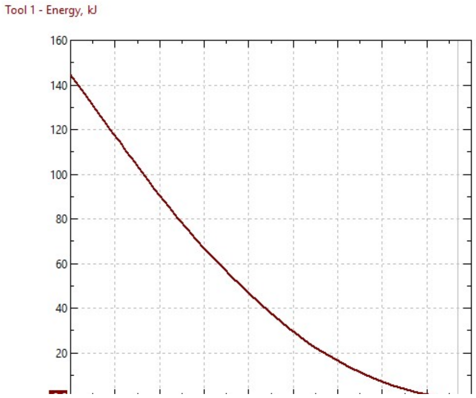


Fig. 14 Forging energy dependence of time, stroke 4

With each subsequent blow, the forging material fills the mold cavity better, the deformation force increases with decreasing travel of the upper part of the tool.

The temperature field on the 3D model of the forging indicates that the highest temperatures are reached in the part of the volume where the most intense plastic flow it is the part near of the transition bridge (Fig. 15).Through it, all the excess volume of the forging material flows into the excess material warehouse, and due to radial flow with intense friction and an increase in tangential stresses, parts of the forging material reach temperatures of up to 1120°C.

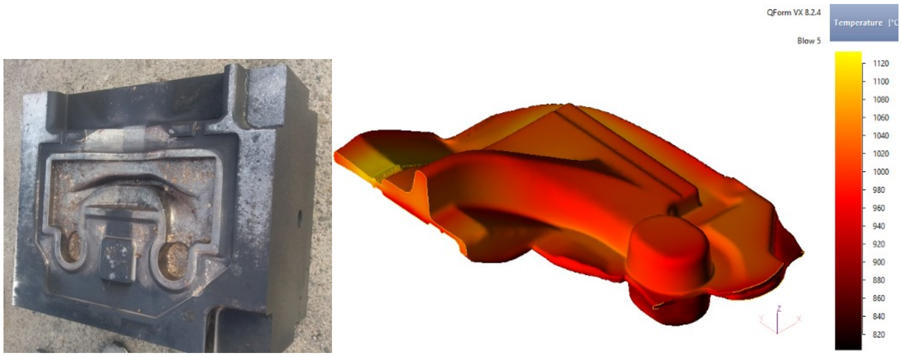


Fig. 15 Temperature field per volume at the end forging

The distribution of the mean stress per volume of the finished piece is an interesting parameter that indicates stress loads.The appearance of tensile stress would indicate a bad design solution of the tool, which is not permissible with such technologies.The highest values of mean compressive stress (negative values) are marked in blue inside the mold cavity indicating universal contact (Fig. 16).In the vicinity of the rim itself, we have lower values of the mean compressive stresses because there is an intense radial flow towards the warehouse of excess material, i.e. slight tensile stresses.

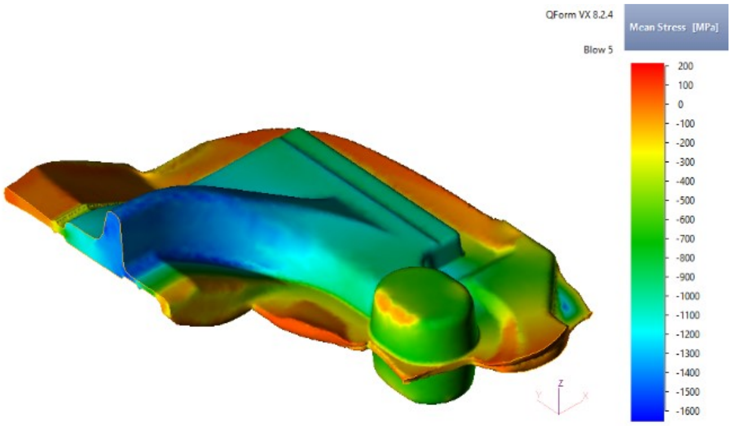


Fig. 16 Mean stress per volume at the end forging

4. CONCLUSION

The paper describes and illustrates the nonlinear FEM simulation, which is indispensable in research and commercial software today.The prerequisite is an ideal approximation of the contact of the deformable material of the preparation and the rigid surface of the tool in order to obtain the most accurate FEM simulation of the real process.Based on this simulation, it is possible to obtain the stress deformation field per volume of the deformable volume as well as the temperature field at high forging temperatures.

REFERENCES

1. Shephard, M. S., Baehmann, P. L., Collar, R. R., Hattagady, N. V. and Niu, Q.,*Automated Remodeling Techniques in Finite Element Analysis*, in Advances in CAD/CAE, Academic Press, 1993.
2. "*MSC. Marc, vol. A Theory and user information", MSC Software*, Simulating reality, Los Angeles, CA, U.S.A. 2018.
3. "*MSC. Marc, vol. B Element library", MSC Software*, Simulating reality, Los Angeles, CA, U.S.A. 2018.
4. *QForm, User guide*, VX 8.2.4, 2018.
5. *QForm, Manual refrence*, VX 8.2.4, 2018.
6. ChoiSh, HanSh, Chin Kg (2009) *Simulation of orientation gradients in a ferrite matrix containing hard martensite under plane-strain flange deformation*. Acta Mater 57(6):1947–1958.
7. Berbenni S., Favier V., Berveiller M., (2007) *Micro–macro modelling of the effects of the grain size distri-bution on the plastic flow stress of heterogeneous materials*. Comp Mater Sci 39(1):96–105
8. Brahme A., Winning M., Raabe D., (2009) *Prediction of cold rolling texture of steels using an artificial neural network*. Comput Mater Sci 46(4): pp. 800–804
9. Equbal I. Md., Kumar R., Shamim M., (2014) *A grey-based Taguchi method to optimize hot forging process*. (ICMPC 2014) Procedia Mater Scie (6):1495-1504.
10. OyekanmiB., Hughes T., Bramley A., (1990) *A microstructural evaluation technique for deformation studies in metal forming processes*. J Mater Process Technol 21(1): pp. 79–89
11. Altan T., Henning H. J. (1972): *Closed-Die Forging of Round Shapes. Flash Design and material Savings*, Metallurgia and Metal Forming, 39, pp.83-88.

12.Altan T., Ngaile G., Shen G. (2005): *Cold and Hot Forging Fundamentals and Applications*, © 2005 ASM International, Materials Park, USA
13.Doege E., Bohnsack R. (2000). *Closed die technologies for hot forging*, J.Mat. Proc. Technol., 98, pp. 165-170.
14. Tomov B., Radev R., Gagov V. (2004): *Influence of flash design upon process parameters of hot die for-ging*, J. Mat. Proc. Technol., 157-158, pp. 620-623.
15. Tomov B. (2007): *Hot closed die forming-State-of-art and future development*, J. of Achievements in Mtrol. and Mafg. Engng., 24, pp.444-449.
16.Jolgaf M., Sulaiman S. B., Ariffin M. K. A., Faieza A. A. (2008): *Closed Die Forging Geometrical Para-meters Optimization for Al-MMC*, American J. Engng. And Applied Sciences, 1, pp.1-6.
17.Sleeckx E., Kruth J. P. (1992): *Review of flash design rules for closed-die forgings*, J. Mat. Proc. Technol., 31, pp.119-134.