Journal for Technology of Plasticity, Vol. 38 (2013), Number 2

NUMERICAL MODELING OF IRONING PROCESS

Adamovic D.[1]*, Mandic V.[1], Zivkovic M.[1], Gulisija Z.[2], Stefanovic M.[1], Topalovic M.[1], Aleksandrovic S.[1]

[1]*Faculty of Mechanical Engineering Kragujevac, University of Kragujevac, Serbia*
[2]*Institute for Technology of Nuclear and other Raw Materials (ITNMS) Belgrade, Serbia*

ABSTRACT

Based on comprehensive experimental research and physical modeling of ironing, and detailed study of the material characteristics of the work piece and tool, as well as investigation of the contact friction conditions and measuring temperature generated by this contact friction, we were able to conduct a set of "numerical experiments" using finite element method implemented in the software Simufact.forming. The goal of numerical modeling was to use 3D visualization of the process, especially in deformation zone, to present strain, stress, velocity, and temperature fields which will enable more detailed analysis of the physics of the process. For that purpose a non-linear FE approach is applied, using solid 3D finite elements, which are optimized for metal forming simulations. This allows obtaining accurate simulation results with the evaluation of changes in the sheet thickness, the effects of residual stress and recurrent elastic strains.

Numerical modeling of the process, as well as results of FE analysis have allowed us to obtain important information about the ironing process such as the stress at the wall, strains, deformation forces, generation of temperature in the work piece as a result of plastic strains and the influence of contact friction and transfer of this temperature it to the die. All of these output parameters are evaluated depending on the angle of the die, the forces on the holder, lubrication conditions and the strain rate, analogous to the plan of experiments in physical modeling.

KEYWORDS: *Metal forming, ironing, numerical modeling, FEM*

1. INTRODUCTION

Numerical modeling of metal forming, based on knowledge of the physics of the process and validated experimental results is used to predict the strength of the material, determine the distribution of strain, stress, temperature, strain on tools, potential sources of defects and fractures, properties and microstructure of the product, as well as for estimation of the springback and residual stress [1, 2].

---

* Corresponding author E-mail address: adam@kg.ac.rs

The most widely used and very powerful "tool" for the numerical simulation of machining processes is finite element method (FEM). In the last ten years, thanks to the rapid development of computer technology, number of commercial software packages for solving problems in the processing of metals based on the finite element method was developed. The user interface is continuously improved in order to increase the possibility of industrial application, with options to define input parameters for FEM analysis based on the type of the process, used tools and machines, as well as processing conditions. In this way, engineers and designers can focused on modeling of machining processes, not on solving problems related to the FEM method [3, 4, 5].

SIMUFACT.forming software suite is a software originated as a unified variant of previous programs MSC.SuperForm and MSC.SuperForge, developed by SIMUFACT and MSC.Software for 2D and 3D computer simulation of industrial forming process, using an integrated FE (Finite Element) and FV (Finite Volume) technology. This package is a combination of complex analysis module (*solver*) and a simple user interface that is specially adapted for 3D simulation of metal forming. Researchers and industrial engineers use it all over the world with great efficiency. Finite volume method is fast and accurate because it does not apply *remeshing* (regenerating global mesh), and is used to monitor deformation of material and automatically refinement of natural small area (*facet*) on the free surfaces of the model. The finite element method is also integrated in the *solver* with the option of automatic generation of FE mesh and later *remesing*, for more demanding applications and analysis of stress in tool.

Using data obtained by experimental research and physical modeling of ironing [6, 7], and analysis of the material characteristics of the work piece and tool, as well as investigation of the contact friction conditions and measuring temperature generated by this contact friction, we conducted a set of "numerical experiments" using FE method implemented in the software SIMUFACT.forming. We used 3D visualization of the process, especially in deformation zone, to show strain, stress, velocity, and temperature fields which enabled us more detailed analysis of the physics of the process. Non-linear FE approach is applied, using solid 3D finite elements, which are optimized for metal forming simulations using "*2½ D sheet mesher*". Simulation results were obtained with great accuracy showing evaluation of changes in the sheet thickness, the effects of residual stress and recurrent elastic strains.

Important information about the ironing process such as the stress at the wall, strains, deformation forces, generation of temperature in the work piece as a result of plastic strains and the influence of contact friction and transfer of this temperature to the die were obtained using numerical modeling of the process, as well as results of FE analysis. These output parameters are functions of the angle of the die, the forces on the holder, lubrication conditions and the strain rate. In addition, thanks to the flexibility of numerical models, estimates were made of strain and temperature fields in the ironing process for a much wider range of input parameters than the physical experiments, e.g. strain rate [8, 9, 10, 11].

2. EXPERIMENTAL INVESTIGATIONS

2.1 Material characteristics

For experimental investigations in this paper the low carbon steel sheet, Č0148P3 (WN: 1.0336; DIN: DC 04 G1/Ust 4, Ust 14) was chosen. It belongs to a group of high quality sheets aimed for the ironing and it has properties prescribed by standard SRPS EN 10130:2004.

For the die and punch material the alloyed tool steel (TS) Č4750 (WN: 1.2601; DIN17006: X165CrMoV12; EN: X 160 CrMoV 12 1) was selected.

The special attention was devoted to material characteristics in the sheet rolling direction (0°), since the tested samples were cut in that way. (SRPS C.A4.002:1986) which was applied using specimens in rolling direction, material characteristics for test-piece were determined. Values are shown in Tab. 1. Tests have been performed under laboratory conditions (v = 20 mm/min, T=20°C).

Table 1 - Properties of tools and test piece materials

		Materials	**Mechanical properties**
Tool	**Die (D)**	TS – Tool steel, Č4750 (DIN17006: X165CrMoV12)	TS Hardness 60÷63 HRC
	Punch plate (P)	TS – Tool steel, Č4750 (DIN17006: X165CrMoV12)	
Test-piece		Č0148P3 (WN: 1.0336; DIN: DC 04 G1/Ust 4) Thickness: 2.0 mm Width: 18.6 mm	R_p = 186.2 MPa R_m = 283.4 MPa A_{80} = 37.3 % n = 0.2186, r = 1.31915

Experimentally obtained data for true stress - true strain relationship, that is flow curve, was fitted in exponential form. Equation 1 was finally used for further numerical description of material flow and behavior during FE ironing simulation, where K is true stress and φ is true strain. In Fig. 1 is shown the flow curve for the tested material in the rolling direction, obtained according to equation 1.

$$K = 491.6874 \cdot \varphi^{0.2186}, \text{ MPa} \tag{1}$$

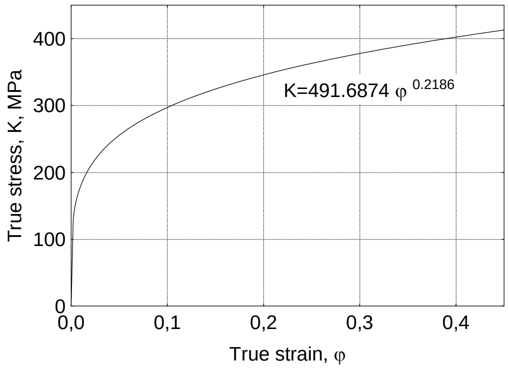


Fig. 1 *- The flow curve in the rolling direction (0°)*

2.2 Physical modeling

The original model of strip ironing device for experimental investigations used in this paper has been developed at Faculty of Engineering Kragujevac. It imitates the zone of contact with die and punch [7] with double-sided symmetry during modeling of ironing. This device enables the realization of high contact pressures and respects physical and geometrical conditions of real process (material of die and punch, contact surfaces topography, different semi-angle of die cone – α etc). The strip ironing device, with presentation of forces which act upon the workpeace, i.e. die and punch, as well as specimen shape is shown in Figure 2.

Strip ironing device is installed on the hydraulic press for investigation of thin sheet metals – ERICHSEN 142/12 (Figure 2). The main drive of the machine is used for production of ironing force (force F_{ir}), whereat the second action is the pressure on strip specimen (force F_D). Sheet

metal strip is bent and placed on the "punch". Dies are placed in supports, whereat the left support is motionless, and the right one is movable together with the die.
The divided punch consists of body and front which are inter-connected by gauge with measuring tapes. The strip is ironed between dies due to the effects of force F on the punch front. Throughout ironing, the outer surface of strip slides over die surfaces, which are skewed at an angle α. The inner surface of strip slides over plates, fixed onto the punch body. During the construction of strip ironing device, the main idea was to enable determination of friction coefficient, both on die side and on punch side at various contact conditions. Shape of sheet strip is shown in Fig. 3, before and after bending, and final dimensions of test-piece, too.

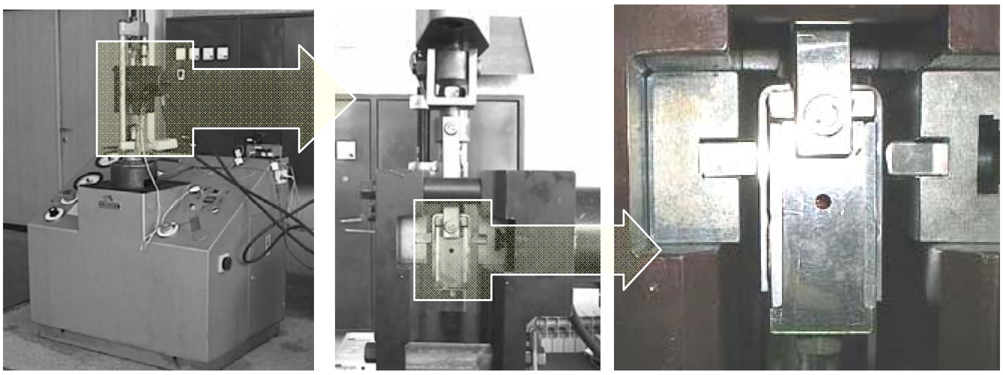
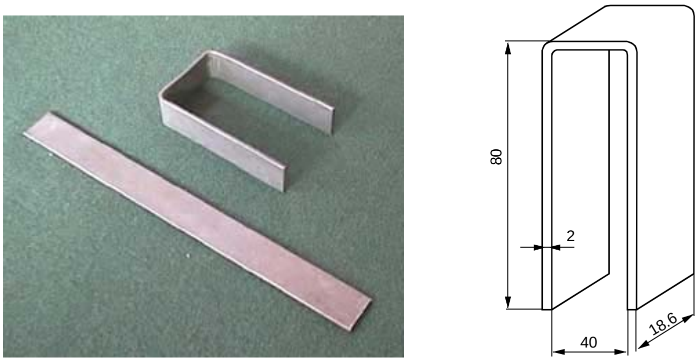


Fig. 2 *- Strip ironing device mounted on hydraulic press*



a) *b)*

Fig. 3 *- Shape (a) and dimension (b) of test-piece*

2.3 Numerical modeling

For accurate simulation of ironing process, numerical simulation of workpiece creation in the process of two-angle bending had to be done first. The goal was to provide the same strain history as it was observed in physical modeling experiments. For this purpose, we modeled bending tools and a metal strip with dimensions 200 × 20 × 2 mm. In order to calibrate the bending angle, elastic tool was used (*spring die*) with the aim of diminishing the effects of springback and obtaining the correct virtual workpiece, which dimensionally corresponds to those in the experiments. Figure 4 shows virtual models of bending tools and metal strips. The tools are considered rigid during the simulation. Based on workpiece geometry, initial FE hexagonal mesh with element size of 1 mm is generated, with 2 or 3 layers of elements through the thickness of sheet metal.

Considering that in the further numerical analysis of the ironing process, axial stress in the wall is monitored, we will show the stress state at the end of the process of workpiece bending with calibration, and stress state after removing workpiece from the tool, i.e. after calculation of effects of springback. Figure 5 shows the distribution of tensile stress in the wall of the specimen. So, "numerically shaped" specimens were used for further simulation of ironing.

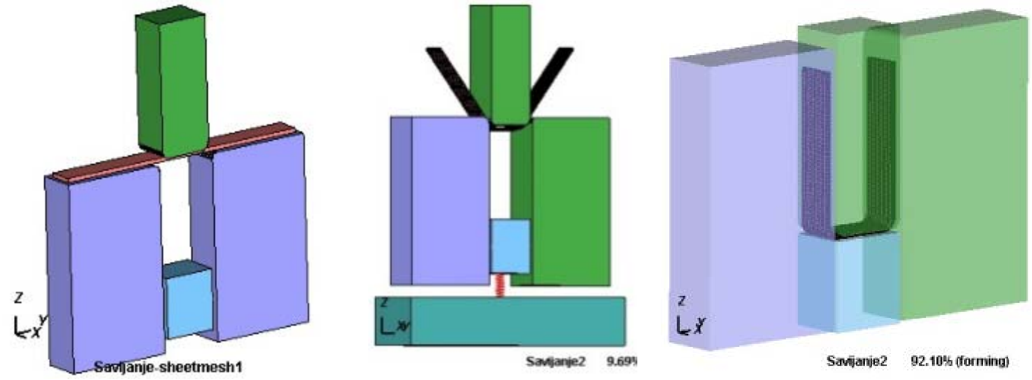
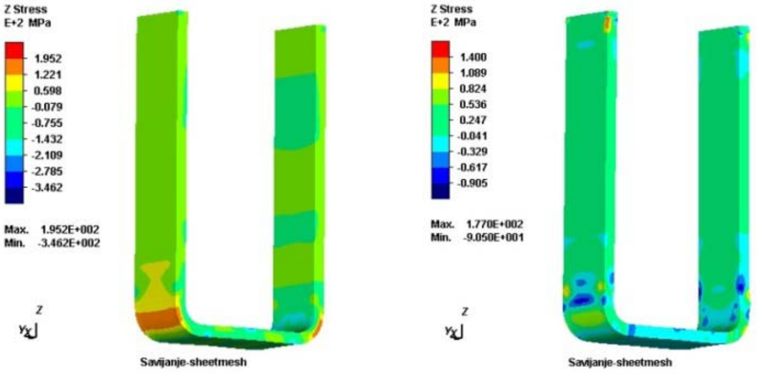


Fig. 4 - *Model of tool and workpiece for numerical FE analysis of workpiece bending (before, during and at the end of the bending)*



a) b)

Fig. 5 - *Distribution of the axial stress in the workpiece model a) at the end of the bending process and b) after removing from the tool (springback)*

Advanced features in the applied simulation software enables "transfer" of strain from the operation of the operation. Thus the strain and stress fields after unloading of specimen in bending operation are transferred to the next operation.

For numerical modeling of the ironing we prepared 3D models of die and punch in CATIA software. Using CAD interfaces these models were imported in SIMUFACT.forming software as STL files. All geometries applied in physical experiments were used in the numerical simulation as well. In order to obtain more precise results in deformation zone, slip condition of materials along the cone angle of die was applied. We also used options to improve tool surface (surface remesh) so that the geometry of the die and the surface of the exit die cone are modeled accurately, as shown in Figure 6.

For punch model no improvement of STL surface mesh was required, since the virtual workpiece was placed on the punch along its inner surface, and was in constant contact with the punch. Figure 7a shows the initial numerical model for ironing. For virtual workpiece after bending, with the complete history of deformation, before the installation of punch, initial FE mesh with 1 mm HEX elements was generated, with 2 or 3 layers in thickness (Fig. 7a). The left and right dies are placed symmetrically in relation to the punch, with the distance provided by the planned reduction of the wall thickness in each numerical experiment. Figure 7b shows the die position, punch and bent workpiece during the ironing process.

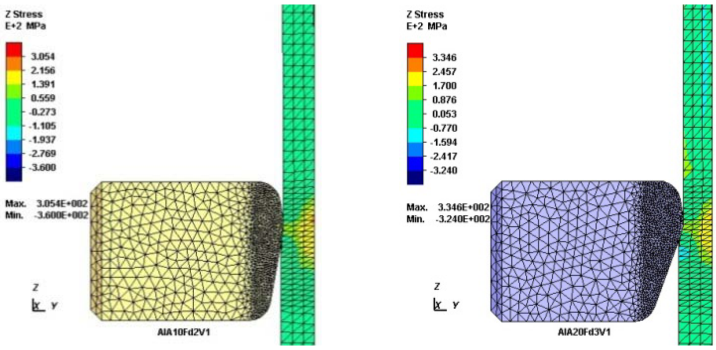
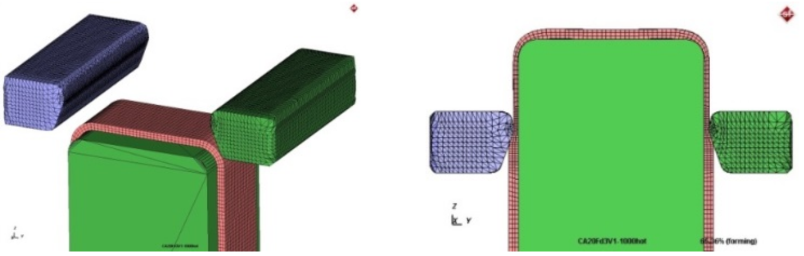


Fig. 6 *- The virtual die models with different cone angles and applied "surface remesh" option*



a) b)

Fig. 7 *- a) Virtual model of assembly tools for numerical modeling of ironing and b) die position in the process*

The numerical modeling is not done with full experimental plan as the physical, but with only those experiments that show the impact of changes in process parameters on the stresses in the wall and changes of the temperature during ironing are chosen. For selected experiments, these values correspond to those in the physical modeling in order to compare the results of both approaches. This is a good basis for further numerical experiments with values outside the range of the process parameters used in the laboratory. Selected numerical experiments illustrate well the impact of process parameters (angle of the die, holding force, the conditions of lubrication and contact surfaces, strain rate and degree of reduction of the wall) on the output parameters of the process (deformation force, the stress in the wall, temperature). Contact friction conditions are described in detail using the coefficients of friction measured in physical experiments for selected cases.

3. EXPERIMENTAL RESULTS

3.1 The results of numerical modeling - draw force

It is known that one of the measurable indicators for comparison and verification experimental-numerical modeling of various deformation processing operations is flow curve.

In SIMUFACT.forming software, after completing the analysis, one can get a diagram of forces in different directions (x, y, z) for all the tools involved in the process of shaping material. Given that the physical experiments registered forces on punch, they were used as relevant for comparison with numerical modeling.

Table 2 presents the process parameters used in the numerical experiments: variation of die angle (5°, 10°, 15°, 20°), holding force (17.4 kN and 26.1 kN), reduction of the wall thickness, friction conditions, the measured and calculated coefficients of friction on the punch (μ_P) and die (μ_D).

Table 2 - Process parameters for numerical modeling

Numerical experiment	**Die angle (°)**	**Holding force (kN)**	**Reduction of the wall thickness (mm)**	μ_P	μ_D
CA05Fd2V1	5	17.4	0.12	0.144	0.093
CA10Fd2V1	10	17.4	0.28	0.106	0.071
CA15Fd3V1	15	26.1	0.66	0.256	0.080
CA20Fd3V1	20	26.1	0.95	0.191	0.095

Comparative diagrams of experimentally measured ironing force and their numerical estimates are shown in Figure 8. It can be seen that the trends and values of forces are approximately equal in the experimental measurements and numerical evaluation.

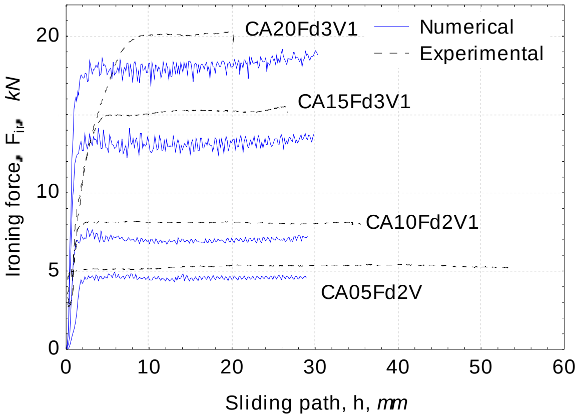


Fig. 8 *- Comparative diagrams of ironing force for experimental and numerical modeling*

3.2 The results of numerical modeling - tensile stress in the wall

For a numerical estimation of the stress in the wall in the ironing process and comparison with obtained experimental results we chose 4 numerical experiments. Material behavior is described by flow curves, and the contact conditions are described by measured and calculated values of the friction coefficient of punch and die for each experiment. Selected numerical experiments are designed to demonstrate the effect of different angles of the die, reduction of wall thickness, holding force, and contact friction conditions on the stress distribution in the wall during ironing. Although the numerical modeling based on the finite element method can present and analyze a multitude of output results, such as stress, strain, strain rate, temperature fields, flow curves, in this case we will only show the axial stress distribution in the direction of z axis, which also represents the stress in the wall, calculated based on experimental research.

Figure 9 shows the distribution of stresses in the wall obtained by numerical experiments in the yz plane in deformation zone near the exit cone of the die [11]. Trends of stress increase in the contact zone are identical to the laboratory experiments: with increasing of the die angle and reduction of wall thickness, an increase in stress values in the wall occurs.

Unlike laboratory experiments, where the stress at the wall is calculated as the mean value in relation to force on punch F_z [12], in the numerical modeling, it is possible to calculate and show the stress in the wall at any point of the process, at any position in deformation zone, and at any cross section [13].

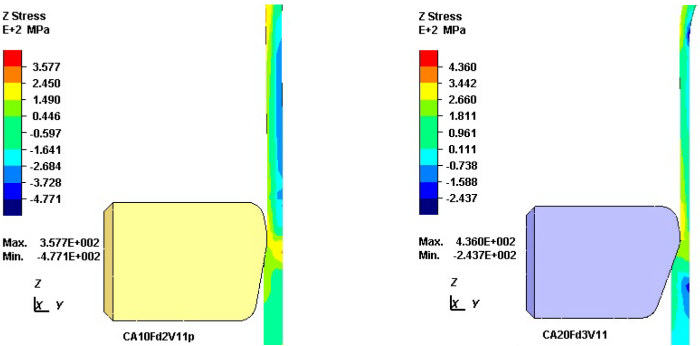
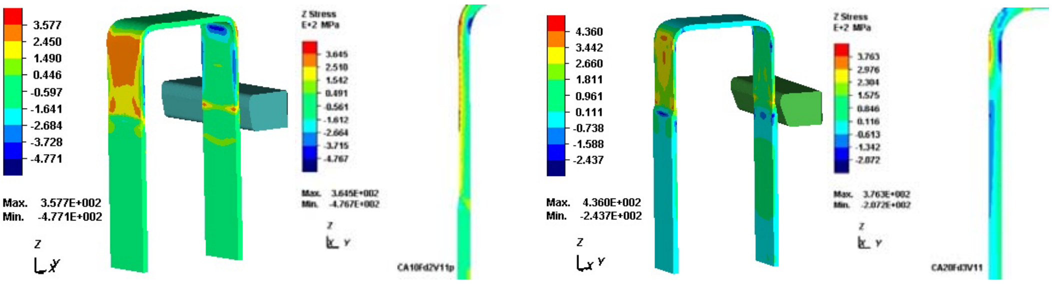


Fig. 9 *- Stresses in the wall in the zone of deformation, Č0148P3, y-z cross section*

In order to better understand the influence of process parameters on the stress in the wall, and further analysis, Figure 10 shows the distribution of stresses in the wall in 3D at the end of numerical experiments, as well as cross sections yz and xy cross section at the contact between the workpiece material and die. This last cross section enables analysis of the stress value along the wall thickness that is getting thinner, on the inside of the contact with the punch and on the outside of the contact with the die [10, 14]. In addition, comparison of the obtained results by numerical experiments can be used for the analysis of the effect of friction on the punch and the die surface and the stress distribution in cross section, and to obtain significant conclusions for the further management of this process, which is based on finding an optimal balance between the conditions of the punch and contact conditions in the contact die, i.e. tensile stress in the wall.



CA10Fd2V1 CA20Fd2V1

Fig. 10 *- Distribution of tensile stress in the wall*

This method of numerical modeling is a powerful tool for researchers and engineers in the design process of ironing in industrial applications with a variety of workpiece materials. Easy changes in numerical models and changing process parameters beyond those available in the range of laboratory equipment offers great opportunities to explore this specific and demanding metal forming processes.

The estimated value of tensile stress in the wall, for all of the numerical experiments, are shown along strip length and in selected cross sections, indicate that the tensile stress in the wall has its maximum value in areas outside the strip deformation zone, beyond the contact with the exit cone of the die (e.g. from the value of 49 MPa to 325 MPa, for the angle of of 5 °, or 140 MPa to 430 MPa for the cone angle of 20 °, in the case of the ironing steel strip). Figure 11 shows the value of the minimum and maximum stress in the wall, depending on the angle of the die cone obtained by the numerical experiments. This must be taken into account in the design process in order have its successful implementation and achievement of the highest possible reduction in thickness without the occurrence of defects. At the same time, it explains the reason for the tearing of strips at higher wall thickness reductions, when tensile stress on the outer surface of the strip on the die side exceed the permissible stress for the given material, which appeared in some physical experiments.

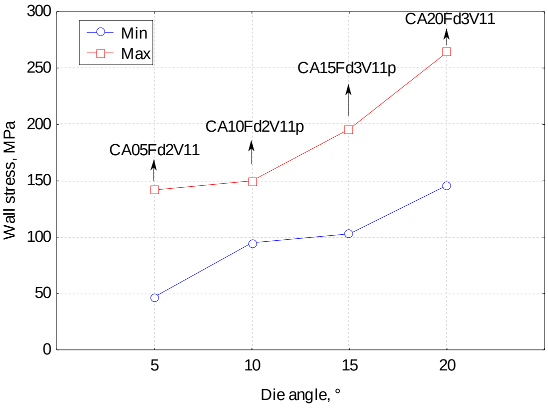


Fig. 11 *- Variations of the minimum and maximum values of tensile stress in the wall, depending on the angle of inclination of the die*

3.3 The results of numerical modeling - Temperatures in the contact zone between the die and the workpiece

Sometimes it is not practical to measure temperatures experimentally at operating pressures in the interface, it is easier to apply numerical simulation using finite elements with integrated thermo-mechanical coupling to estimate the temperature distribution in contact between workpiece and tool in the ironing process [9, 15].

The aim of this part of the research was to use unified thermo-mechanical FE analysis of ironing process to gain wider knowledge of the generation and transfer of heat in contact, at high speeds, which could not be applied in experimental research because of the limited speed range of laboratory equipment. Also, the experimental limitations in measuring contact pressures and temperatures are exceeded using numerical FE simulations.

In order to estimate temperature field in ironing process which originate from heat generated by plastic work and contact friction, as well as due to heat transfer from workpiece to tool and ambient, a series of numerical experiments was conducted using SIMUFACT.forming software. Coupled thermo-mechanical nonlinear FE analysis implemented in advanced MARC solver developed based on strain method was used. Ambient of process is also important for analysis of heat transfer from workpiece to tool and to ambient. Thermal properties of workpiece and tool must be defined, such as initial temperature, heat transfer coefficients to workpiece and ambient, emissivity for heat radiation. As in previous cases of numerical experiments, 3D (HEX) solid finite elements which are optimized for simulation of sheet metal processing are used. Mesh generation and later remeshing are performed using "*2½ D sheet mesher*".

Heat transfer from tool to die is modeled using rigid die with heat conduction effects. For such numerical analysis it is necessary to make initial FE mesh for tools as well. *Overlay hex mesher* is used with option *refinement box* in order to have fine FE mesh in the zone of exit die cone which would ease condition of contact sliding of material along die.

In coupled thermo-mechanical FE analysis critical input data are thermal characteristics of workpiece and tool materials, initial temperature, heat transfer coefficients to workpiece and ambient, emissivity for heat radiation as well as contact friction parameters. Table 3 shows input data which were used in numerical experiments.

Table 3 - Input data for FE numerical simulations

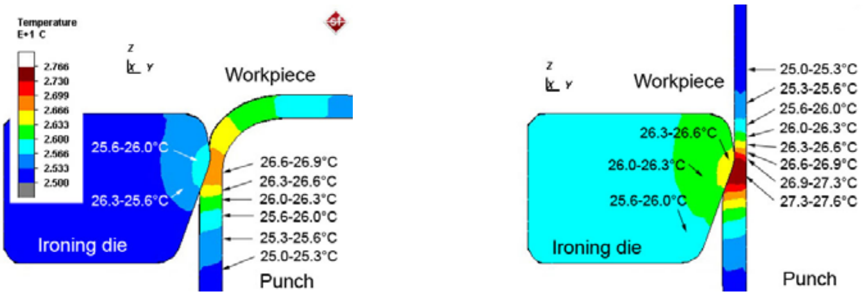
Input data for FE simulations	**Workpiece**	**Die**
Initial temperature (°C)	25	25
Thermal conductivity (Wt/m K)	50	-
Specific heat capacity (J/kg K)	478	-
Heat transfer coefficient to ambient (W/m2K)	50	77
Emissivity for heat radiation to ambient	0.25	0.25
Heat transfer coefficient to workpiece	-	6000

Six numerical experiments were conducted with the same input parameters as in physical experiments, except strain rate, which is shown in Table 4. First two of these six experiments were conducted with the same strain rates as in laboratory conditions (0.33 i 4.17 mm/s), while other four were conducted with increased strain rates in order to investigate influence of strain rate on temperature field in workpiece and die during ironing. In these cases, numerical modeling was performed with correction of conditions of contact friction, i.e. coefficient of friction, according to research presented in [16]. Table 4 provides the calculated values of the coefficients of friction for the applied strain rate in each of the numerical experiments, and the contact surface of the punch and die.

Table 4 - FE experiments' codes and applied friction parameters depending on ironing speed

FE experiment code	**Ironing speed (mm/s)**	**μ_P**		**μ_D**	
		% of change	recalculated	% of change	recalculated
CA20Fd3V1-0.33hot	0.33	0	0.1063	0	0.0707
CA20Fd3V1-4.17hot	4.17	+10% of 0.1063	0.1169	+10% of 0.0707	0.0777
CA20Fd3V1-100hot	100	-2% of 0.1169	0.1145	-2% of 0.0777	0.0761
CA20Fd3V1-1000hot	1000	-3% of 0.1169	0.1133	-3% of 0.0777	0.0753
CA20Fd3V1-5000hot	5000	-10% of 0.1169	0.1052	-10% of 0.0777	0.0699
CA20Fd3V1-10000hot	10000	-15% of 0.1169	0.0993	-15% of 0.0777	0.0660

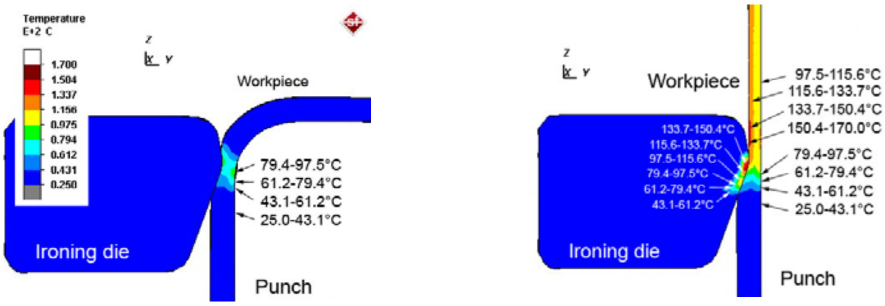
While in the laboratory experiments it was not possible to monitor changes of the temperature in the die in the area near the contact surface, the numerical modeling of process allows monitoring of thermal effects at any time, at any speed, both in the workpiece and in the tool. Temperature distribution in a numerical model of the ironing process with punch rate *v*=0.33 mm/s at different points in the process (punch stroke *h*=3 and 35 mm) is shown on Figure 12. In the deformation zone of the workpiece in direct contact with the tool, an increase in temperature due to plastic deformation and contact friction occurs. Part of the heat generated is transferred to the die. Due to the low strain rate, and therefore significant contact time between materials and tools, heat is transferred to the whole die. At the same time workpiece leaving the contact zone cools down due to the effects of radiation of heat to the ambient. From the initial 25°C, temperature in the workpiece reaches a maximum value of 27.6 °C, while in the die, at the position 0.3 mm from the contact surface, as in the experimental measurements, the temperature reaches a value of 26.6 °C.



h=3 mm *h*=35 mm

Fig. 12 *- Temperature variation in the die and the workpiece at ironing speed of v=0.33 mm/s*

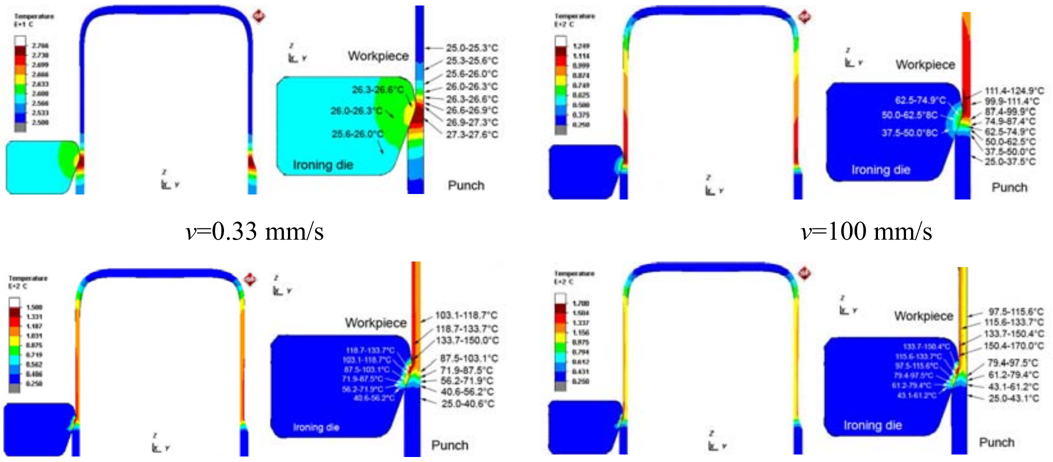
At significantly higher punch rate (*v*=10000 mm/s), corresponding to industrial conditions during production of 250 cans per minute, the temperature distribution is different, as is shown in Figure 13. Unlike in the case of previous numerical experiment, when punch stroke reaches 3mm after the first contact with the die, the temperature of the workpiece reaches a value of 97.5 °C due to high slip velocity in contact and influence of friction. Heat transfer to the tool due to the very short duration of the contact is evident after punch stroke of 15 mm. At the end of the process, maximum temperature reached in the workpiece is 166 °C, while in the die at sensor position temperature is 133 °C.



h=3 mm *h*=35 mm

Fig. 13 *- Temperature variation in the die and the workpiece at ironing speed of v=10000 mm/s*

Since the time of ironing process with the punch stroke of 35 mm is very short, especially at high ironing speeds, for other numerical experiments we have shown only the temperature distribution at the end of the simulation. Figure 14 shows the results of numerical simulation of FE at ironing speeds of 0.33 mm/s to 10000 mm/s. The temperature distribution around the work piece and the die is shown on the left and the development of the temperature field in the zone of contact with numerical values is shown on the right portion of the Figure 14.

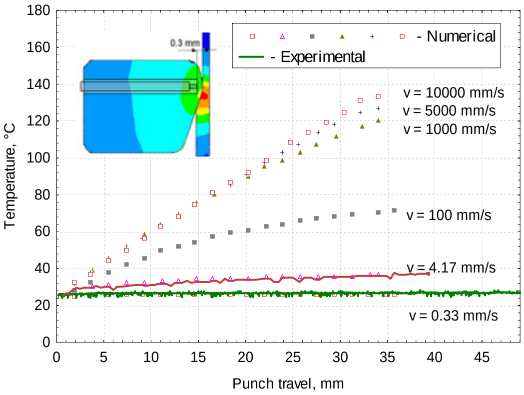


v=0.33 mm/s *v*=100 mm/s

v=1000 mm/s *v*=10000 mm/s

Fig. 14 *- Temperature variation in the die and the workpiece at various ironing speeds*

Increase of the strain rate leads to an increase in the temperature of the contact due to the fact that the heat transfer to the tool is less significant since duration of the process is small. Also the effects of surface heating issues and local generation of heat are essential for the workpiece due to tin coating and their compactness over the entire surface of the workpiece. On the other hand lubricating and cooling at high processing speeds must be chosen with great care, because, as shown in Figure 15, the radiation of heat to the ambient and cooling of the workpiece at those speeds is almost imperceptible.



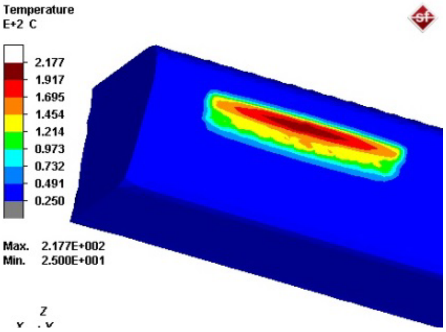


Fig. 15 *- Temperature variation at the die*

Fig. 16 *- Distribution of temperature in die during ironing process*

The diagram in Figure 15 summarizes the trend of changes in the maximum temperature in the die at sensor position (FE node which has distance of 0.3 mm from the die radius) during the numerical simulation of strain at different speeds. With the increase of strain rate heating of the die is greater. At a speed of 10000 mm/s, and the working punch stroke of 35 mm, it reaches a temperature of 135 °C in reference point for only 3.5 µs. However, on the surface of the die, in the area of the output angle temperature is higher (217 °C), as shown in Figure 16. In real industrial processes, with the use of coolants and lubricants we can expect lower temperature of the tool.

The trend of temperature rise in the workpiece, in the zone of contact with the die, where they are at their maximum during the ironing process at different speeds is shown in Figure 17. In contrast to the die, the maximum temperature is reached when the punch stroke reach 15-20 mm, at all speeds. Since the process is stationary after material pass trough die, with the same deformation, same strain rate, and "numerical constant value of the friction coefficient," which describes the contact conditions, these results are expected. However, each numerical experiment is carried out with the initial temperature of the workpiece and tool of 25 °C, which in the real process, despite the implementation of the cooling is not the case. One can therefore expected that numerically estimated value is slightly higher than values in manufacturing processes [17, 18].

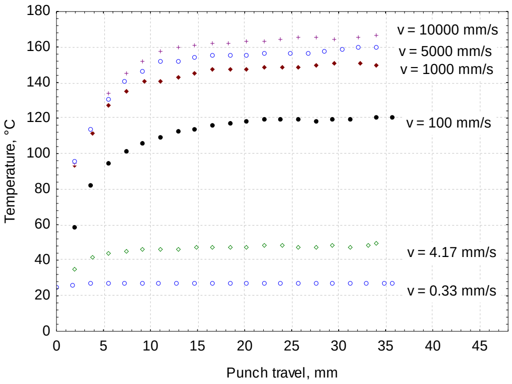


Fig. 17 - *Temperature variation at the workpiece*

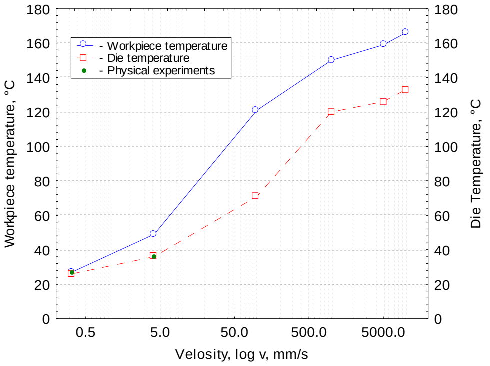


Fig. 18 - *Temperature variation at the workpiece with speed (numerical and physical experiments)*

Comparative diagram of the maximum achieved temperature in the die and in the workpiece, at all tested speeds, in the numerical and physical experiments, is shown in Figure 18. The difference in temperatures in the workpiece and the die is a consequence of the position from which the die temperature is read, in the numerical node which has distance of the ≈0.3 mm from the radius of the die in order to be able to compare numerical with experimental results.

4. CONCLUSION

In this paper, it is shown that the techniques of physical modeling of a laboratory device and numerical FE simulations can be successfully used for a comprehensive analysis of the ironing process. Given the limitations and advantages of experimental and numerical methods, their integrated application has complementary advantages i determination of the effect of process parameters (die angle, die force, friction conditions on punch and dies, velocity) to the output performance of processed parts. From this investigation, following conclusions can be summarized:

a) The physical experiment is necessary to define the precise input data for a comprehensive numerical analysis, so the combined experimental-numerical approach is recommended as the best method for research of ironing or similar deformation processing. Using the combination of physical and numerical experiments it is possible to know the residual stress distribution in each cross section and all over the sliding path, determine the course of deformation force in ironing process, as well as the temperature change in the random part and the tool.

b) The value of friction coefficient, which describes the state of contact, is determined by measuring the force of the punch and the die, and the total force of the process on the machine. Considering that during the ironing process coefficient of friction changes over time, for each numerical experiment the mean coefficient of friction for contact pairs are given. Although using a constant coefficient of friction is one of the drawbacks of numerical FE simulations, which means that there is no change in the conditions of contact friction, the results of experimental and numerical methods have good agreement.

c) By numerical simulations it is possible to know the distribution of wall stress in each cross section and the entire sliding path, as well as to determine strain forces of ironing process. Value of the wall tension stress, obtained by physical experiment, represent the mean value of the wall tension stress during sliding, because is not possible to determine distribution of wall stress in cross section by physical modeling. Unlike this, with the numerical simulation, it is possible to determine the spatial distribution of stresses at any moment.

d) As a result of plastic deformation of materials and friction between workpiece and processing tools increase of temperatures occurs, especially at higher strain rates. A combination of experimental and numerical studies was used to investigate the possible influence of the strain rate both in the laboratory and in industrial conditions on development and transfer of heat during the ironing process. Comparing the results obtained by physical and numerical modeling shows good fit of temperature values in tool at sensor position at speeds v=0.33mm/s and v=4.17mm/s. This is a good basis for further numerical estimate of the temperature in the ironing at higher speeds which are characteristics of industrial applications.

e) Using the experimental and numerical approach one can see all the important aspects of the ironing process, similar to the conditions that occur in the real process, regardless of the limitations of the physical experiment.

ACKNOWLEDGMENT
The part of this research is supported by Ministry of Education and Science, Republic of Serbia, Grant TR32036 and TR34002.

REFERENCES

[1] Deneuville P., Lecot R.: The study of friction in ironing process by physical and numerical modeling, Journal of Materials Processing Technology, 45, 1994., pp. 625-630.

[2] H.C.E. van der Aa, M.A.H. van der Aa, P.J.G. Schreurs, F.P.T. Baaijens, W.J. van Veenen, An experimental and numerical study of the wall ironing process of polymer coated sheet metal, Mechanics of Materials 32 (2000), 423-443.

[3] V. Mandić V.: Physical modeling and numerical simulations as the basis for a new tool design concept for hot bulk forming, Ph. D. Thesis, December 2002, Faculty of Mechanical Engineering, Kragujevac (in Serbian).

[4] V. Mandić, P. Ćosić: Integrated product and process development in collaborative virtual engineering environment, Technical Gazette, vol.18, No3, 2011, pp.369-378.

[5] V. Mandić: Integrated virtual engineering approach for product and process development, Keynote paper, 3rd Management Technology - step to Sustainable Production-MOTSP 2011, Bol, Croatia, Proceed., pp. 7-22.

[6] D. Adamović, M. Stefanović, V. Mandić: Modeling of ironing process, Monograph, Faculty of Engineering Science, University of Kragujevac, Kragujevac, 2012.

[7] Adamovic D.: Conduct of materials in contact at processes of plastic forming with high working pressures, Doctoral thesis, Faculty of Mechanical Engineering, Universtity of Kragujevac, Kragujevac, (in Serbian).

[8] D. Adamović, V. Mandić, M. Stefanović, Z. Jurković, M. Živković, D. Pavletić, M. Perinić: Numerical and Experimental Analysis of the Wall Tensile Stress in Ironing, International Conference Maintenance and Production Engineering "KODIP – 2012", 26-29. June 2012. Budva, Montenegro, Proceed. pp. 23-32.

[9] D. Adamović, V. Mandić, M. Stefanović, S. Aleksandrović, M. Živković: Analysis of the temperature change on the tool and work piece during the ironing process, 11th International Conference on Accomplishments in Electrical and Mechanical Engineering and Information Technology - DEMI 2013, Banja Luka, Proceed. pp. 299-306.

[10] V. Mandić, D. Adamović, Z. Jurković, M. Stefanović, M. Živković, S. Ranđelović, T. Marinković: CAE Analysis of Ironing Process with Experimental Verification, MOTSP 2010 International Scientific Conference management of Technology – Step to Sustainable Production, 2010, Rovinj, FSB Zagreb, Hrvatska, Procced. - CD.

[11] V. Mandić, D. Adamović, Z. Jurković, M. Stefanović, M. Živković, S. Ranđelović, T. Marinković: Numericall FE Modelling of the Ironing Process of Aluminium Alloy and its Experimental Verification, Transactions of FAMENA No 4, vol.34 (2010), pp. 59-69.,

[12] D. Adamović, M. Stefanović, M. Živković, M. Devedžić, G.: Influence of ironing parameters on the tensile stress of the wall pieces, 33th Conference on Production Engineering in Serbia with international participation, Belgrade, 2009., Proceed., pp. 109-112., (in Serbian).

[13] D. Adamović, V. Mandic, M. Stefanovic, S. Aleksandrovic, M. Živkovic, Z. Gulušija, S. Randjelovic: Experimental and numerical determination of the tensile stresses in the wall during steel sheet ironing, ICMEN - 4th International Conference on Manufacturing Engineering, Proceedings, 2011, Thessaloniki, Proceed., pp. 621-630.

[14] D. Adamović, V. Mandic, Z. Jurkovic, B. Grizelj, M. Stefanovic, T. Marinkovic, S. Aleksandrovic: An experimental modelling and numerical FE analysis of steel-strip ironing process, Tehnički vjesnik/Technical Gazette, Vol.17, No.4, 2010. pp 435-444.

[15] D. Adamović, M. Stefanović, A. Davinić, B. Nedić: Temperature changes in the contact zone of the ironing process, 32th Conference on Production Engineering in Serbia with international participation, Novi Sad, 2008, Proceed., pp. 149-152., (in Serbian).

[16] R. Sauer: High speed ironing of tinplate beverage cans, Metall, 48/12, 1994.

[17] Sauer R.: Abstreckgleityiehen yur Fertigung von Weiβblechdosen, XI International Colloquium, Industrial and Automotive Lubrication, Esslingen, 1998., Proceed., pp. 2387-2398.

[18] S. Chandrasekharan, H. Palaniswamy, N. Jaina, G. Ngaile, T. Altan: Evaluation of stamping lubricants at various temperature levels using the ironing test, International Journal of Machine Tools & Manufacture 45 (2005), 379–388.

NUMERIČKO MODELIRANJE PROCESA DUBOKOG IZVLAČENJA SA STANJENJEM DEBLJINE ZIDA

Adamovic D.[1], Mandic V.[1], Zivkovic M.[1], Gulisija Z.[2], Stefanovic M.[1], Topalovic M.[1], Aleksandrovic S.[1]
Fakultet inženjerskih nauka Kragujevac, Univerzitet u Kragujevcu, Srbija
Institut za tehnologiju nuklearnih i drugih mineralnih sirovina, Beograd, Srbija

REZIME

Na bazi veoma opsežnih eksperimentalnih istraživanja i fizičkog modeliranja procesa dubokog izvlačenja sa stanjenjem debljine zida, kao i detaljno ispitanih karakteristika materijala obratka i alata, uslova kontaktnog trenja i merenja razvoja temperature u tom kontaktu bilo je moguće realizovati čitav set „numeričkih eksperimenata“ primenom metode konačnih elemenata u softveru Simufact.forming. Cilj numeričkog modeliranja je bio da se kroz 3D vizualizaciju samog procesa, posebno u deformacionoj zoni, kroz prikaz deformacionih, naponskih, brzinskih i temperaturnih polja izvrši detaljnija analiza same fizike procesa. U tu svrhu primenjen je nelinearni FE pristup korišćenjem 3D solid konačnih elemenata, optimiziranih za simulaciju procesa obrade lima. To omogućava dobijanje preciznih rezultata simulacija sa procenom promena u debljini lima, zaostalih napona i efekata elastične povratnosti.

Numerički modeli procesa, kao rezultati FE modeliranja, omogućili su dobijanje važnih informacija o procesu kao što su napon u zidu, deformacije, deformacione sile, razvoj temperature u obratku i prenos na matricu, kao posledica plastičnog deformisanja i uticaja kontaktnog trenja. Svi ti izlazni parametri su procenjivani u zavisnosti od ugla matrice, sile na držaču, uslova podmazivanja i brzine deformisanja, analogno planu eksperimenata pri fizičkom modeliranju.

Ključne reči: *obrada deformisanjem, duboko izvlačenje sa stanjenjem debljine zida, numeričko modeliranje, MKE*