# ASSESSMENT OF THE DRUM REMAINING LIFETIME IN THERMAL POWER PLANT

by

***Miroslav M. ŽIVKOVIĆ[a*], Snežana D. VULOVIĆ[b], and Rodoljub S. VUJANAC[a]***

[a] Faculty of Mechanical Engineering,University of Kragujevac, Kragujevac, Serbia
[b] Faculty of Information Technology, University of Belgrade, Belgrade, Serbia

Original scientific paper
UDC: 621.182.2
DOI: 10.2298/TSC100507030Z

*In this paper analysis of stress and thermal-elastic-plastic strain of the drum is performed. Influence of modified thickness, yield stress and finite element model of welded joint between pipe and drum on assessment of the remaining lifetime of the drum in the thermal power plant is analyzed. Two analyses are compared. In the first, drum is modeled by shell and by 3-D finite elements with projected geometrical and material data of drum. Then, the drum is modeled by shell and by 3-D finite elements with modified thickness and yield stress. The analysis show that detailed modeling of stress concentration zones is necessary. Adequate modeling gives lower maximal effective plastic strain and increased number of cycles and, in that case, 3-D finite elements are better comparing to shell finite elements.*

Key words*: boiler drum, plastic strain, life time, finite element method*

## Introduction

Most of the plants used in the transport industry, electric power industry, process industry, *etc.*, within their structures include welded construction, *i. e.* welded joints. Welded joints are very sensitive parts of each construction as it is made and operates in complex metallurgical and stress conditions. Assessment of the lifetime exploitation of these structures is of practical importance for the industry. Assessment of the remaining lifetime after regular maintenance, when cracks are detected on the construction, is of a great importance.

This paper presents the assessment methodology of the drum remaining lifetime in the thermal power plant. The effect of modeling methods to estimate the drum remaining lifetime was analyzed. Drum is modeled by shell finite elements and by 3-D elements, and hanging is modeled by general beam finite element. Model with 3-D elements accurately describes the actual geometry of the drum and pipe joints and leads to more realistic modeling of welded joints, taking into account the change of material properties in the area of welded

* Corresponding author; e-mail: zile@kg.ac.rs

joints. To determine the impact of reduced thickness and changes in material characteristics [1], both models have been corrected and additional calculation has been done. Calculations of construction are carried out using program PAK [2], which is used for general linear and non-linear analysis of structures, thermal conductivity, fluid mechanics, coupled problems, fracture mechanics and fatigue.

## Description of the drum model

The analysis of drum geometry shows two planes of symmetry and quarter drum is modeled. The analysis takes into account the holes with diameter above 108 mm, and ignores the holes of smaller diameter. Symmetry exists in terms of supports, self weight, water weight and pressure. On the basis of measurement it is adopted that temperature field is symmetrical. Drum is modeled by shell finite elements (chapter 3) and by 3-D elements (chapter 4). In both cases two analyses were done: the first with the designed geometrical and material data and the other with corrected thickness and corrected yield stress. According to the study [1], there was a reduction in thickness of the drum by 2% due to the internal surface grinding of the structures in order to remove initial cracks. Yield stress was reduced by 20% due to ageing of materials and due to many years of use of the structure.

## Finite Element Model – Shell and Beams

Drum is modeled by four node shell finite elements, with enhancements [2-4]. The model is shown in figure 1. Fine mesh was used around the hole in order to take into account the stress concentration. Supports are modeled by beam finite elements (general beam element with dotted segment [2-5]). Boundary conditions consider the symmetry and the conditions adopted in the supports, which means that the drum is free to expand in all directions.

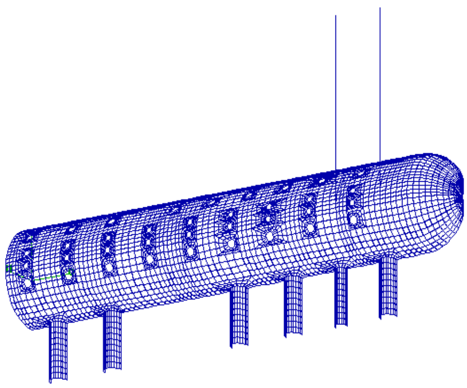

**Figure 1. Finite element model of the drum**

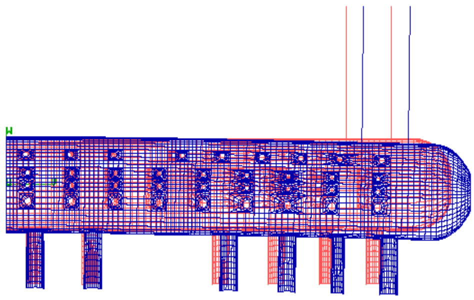

**Figure 2. Deformed configuration**

The construction is exposed to static loads of self weight, weight of water, variable internal pressure and variable thermal loads. The analysis used projected maximum working pressure p = 16.5 MPa and maximum operating temperature T = 350°C. The calculation was performed with the value of pressure p = 15.5 MPa, which is the maximum pressure measured, and the impact of reducing the pressure on the lifetime of the structure was analyzed, too. As the temperature goes up to 350 °C, the construction is analyzed using the thermo-elastic-plastic material model, and creep deformation is ignored [6, 7].

### *Thermo-elastic-plastic analysis*

The analysis was performed using isotropic thermo-elastic-plastic material model in which yield function is described with equation [6]:

$$\bar{\sigma} = \sigma_{yv} + C(M\bar{e}^{p})^{n} \tag{1}$$

where $\sigma_{yv}$ is initial yield stress which depends on the temperature, $C$ = 373.7 MPa and $n$ = 0.294 are constants, $M$ = 1 is the ratio of mixed hardening, and $\bar{e}^{p}$ is the effective plastic strain. Other constants are: modulus of elasticity $E = 1.93 \cdot 10^5$ MPa; Poisson's ratio is $\nu$ = 0.2642; linear thermal expansion coefficient $\alpha$ = 12.5·10-6/°C; reference temperature Tref = 20 °C.

Deformed configuration is shown in figure 2, wherefrom we can see that the structure deforms freely and that the bending deformation is small (maximum movement is 45mm). Effective stress field in the outer and inner surface are shown in figures 3 and 4. It is obvious that the stresses inside is larger, and the maximum effective stress in the construction is $\sigma_{max}$ = 270.5 MPa. The maximum value of effective plastic strain, when corrected both thickness and material, is $\bar{e}^{p}_{max} = 3.063 \cdot 10^{-3}$. The field of effective plastic strain, on both the inner surface and around the hole are given in figure 5, which also shows that plastic strain occurs only around the joint of drum and pipes, as a result of stress concentration.

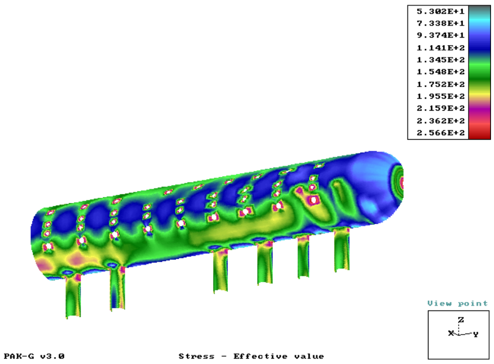


**Figure 3. Stress field in the outer surface**

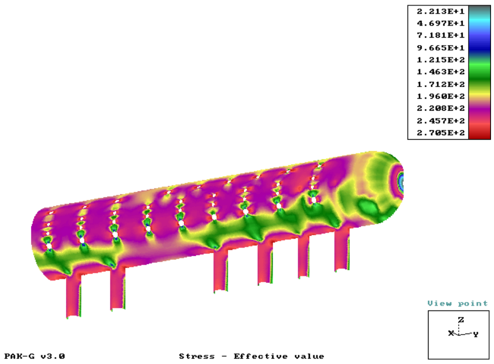


**Figure 4. Stress field in the inner surface**

### *Assessment of the structure lifetime*

Figure 5 shows that the material enters the zone of plastic deformation, and low-cycle fatigue analysis is performed. To assess lifetime of structures the chart that shows number of cycles depending on the accumulated effective plastic strain can be used. This diagram was obtained from the manufacturer of the drum SES Tlmače, Slovakia [1]. This dependence has a character shown on diagram in figure 6, and can be written in an analytical form as:

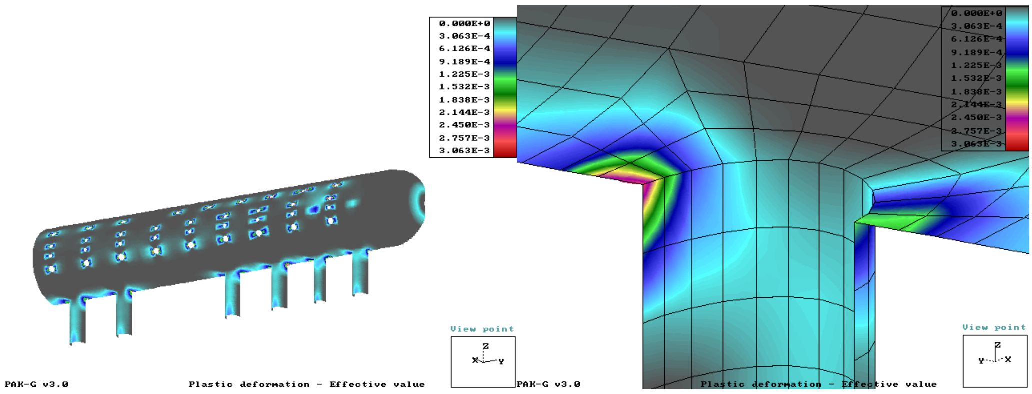


**Figure 5. Field of effective plastic strain on the inner surface and around hole**

$$e^{p} = c / N^{m} \tag{2}$$

where $e^p$ – the value of the effective plastic strain amplitude, $N$ – number of cycles, $c$ and $m$ – constants given by drum manufacturer.

Calculated values of maximum effective stress, the corresponding maximum plastic strain and the number of cycles to crack initiation are presented in tab. 1, for the analysis described in Chapter 2. The analysis results show that the reduction in thickness of 2% and reduction of the yield stress of 20%, causing an increase of the maximum effective plastic strain of 62% and reducing the number of cycles to 2.8 times of the projected model. Figure 7 shows the calculated values of the cycle number to crack initiation of the drum inner surface, using the amplitude of the effective plastic strain shown in figure 5.

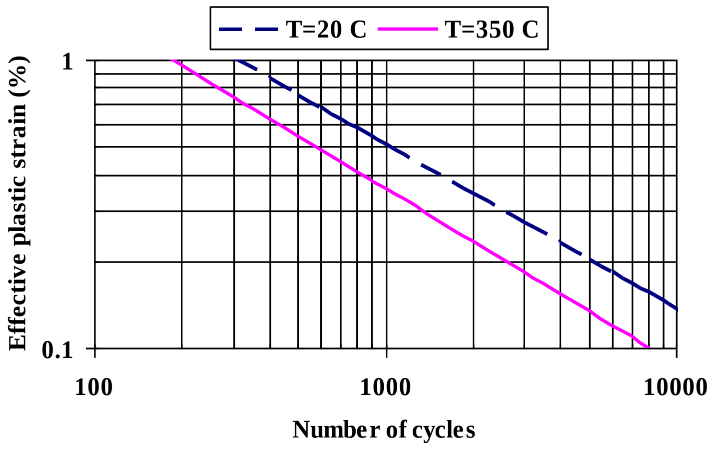


**Figure 6. Dependence of the cycle number of effective plastic strain**

**Table 1. Lifetime of the drum (cycle number)**

| Type of analysis | Maximum effective stress [MPa] | Maximum effective plastic strain [%] | Number of cycles |
|---|---|---|---|
| Projected model | 312.7 | 0.1886 | 3598 |
| Changed material (– 20%) and Corrected thickness ( – 2%) | 270.5 | 0.3063 | 1292 |

Figure 7 shows that the minimum cycle number is 1292, which is about 30% more than all number of cycles so far completed in exploitation. In areas of high stress concentration around holes, sharp edges and welded joints are the largest accumulation of effective plastic strain, as shown in figure 5.

Since the measurements [1], found that the maximum working pressure is not exceeding 15.5 MPa, analysis of impact to reduce the maximum working pressure on the life of structures was done, for the most critical case. When the thickness and material are corrected, the results are shown in tab. 2. In this case, with reduction of maximum working pressure, maximum value of effective plastic strain is reduced to 22%, while the number of cycles increased by 28%, which is important in terms of safety.

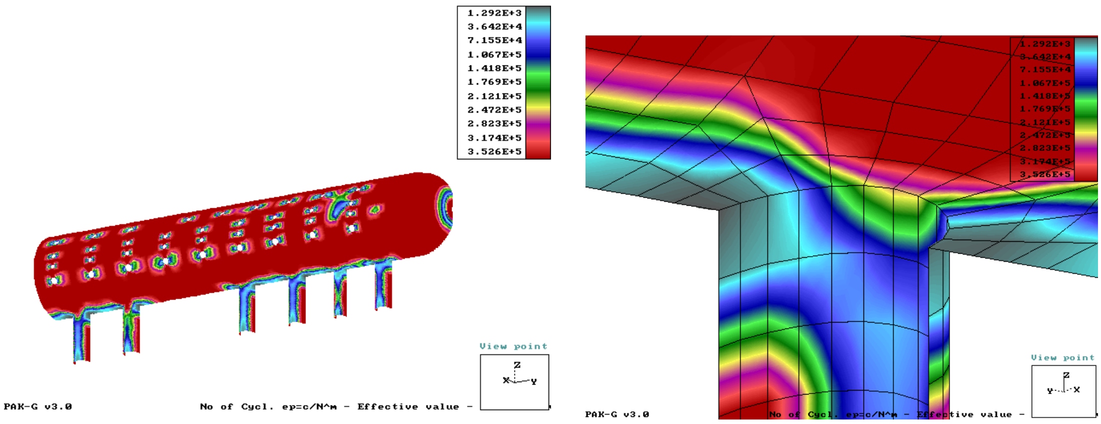


**Figure 7. The cycle number to crack initiation**

**Table 2. Dependence of the cycle number on the maximum working pressure**

| Maximum working pressure [MPa] | Maximum effective stress [MPa] | Maximum effective plastic strain [%] | Number of cycles |
|---|---|---|---|
| 16.5 | 270.5 | 0.3063 | 1292 |
| 15.5 | 266.6 | 0.2504 | 1802 |

## Finite element model 3-D and beam

Drum was modeled by 8 – node 3-D finite element with enhancements [2-4]. The drum is modeled with the same mesh density as in the previous model, while the thickness was modeled using 2 elements. Supports are modeled by beam elements (general beam element with dotted segment [2-5]). Boundary conditions and loads are the same as in the previous chapter.

### *Thermo-elastic-plastic analysis*

The analysis was performed using isotropic thermo-elastic-plastic material model, and the corresponding material data are given in chapter 3.1. In addition to basic material, two more materials are introduced in the welded joints zones of pipes and drum, material of heat affected zone and material of weld [8]. For the material in the heat affected zone material properties were reduced by 20%, compared to the base material, while the weld material properties increased by 20% compared to the basic materials.

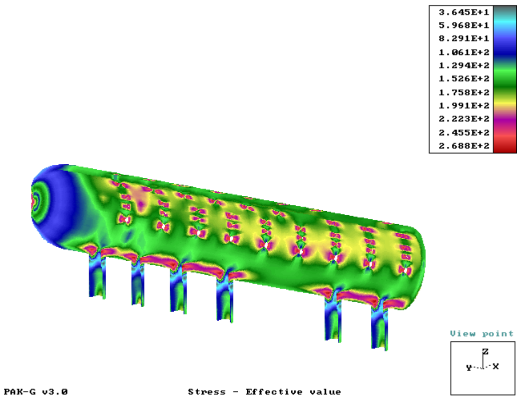


**Figure 8. Field stress on the inner surface**

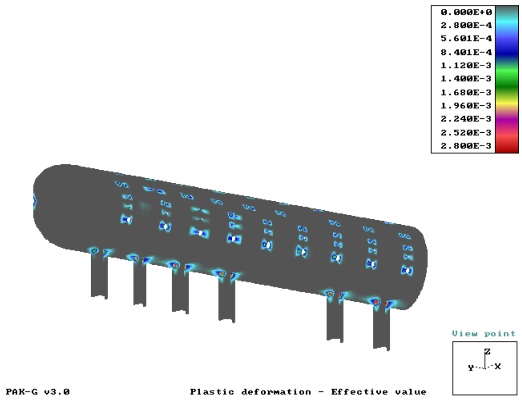


**Figure 9. Effective plastic strain field**

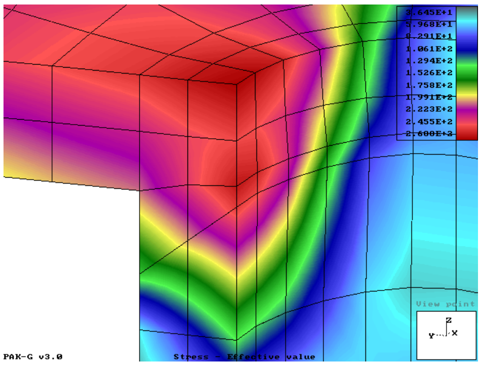


**Figure 10. Stress near the welded joint**

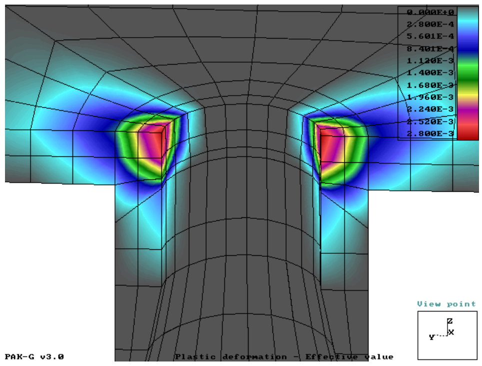


**Figure 11. Effective plastic strain field**

The field of effective stress on the structure inner surface is shown in figure 8 wherefrom it is seen that the maximum effective stress in the construction is $\sigma_{max}$ = 268.8 MPa. The maximum value of effective plastic strain is $\bar{e}^{p}_{max} = 2.8 \cdot 10^{-3}$. The field of effective plastic strain on the inner surface and around the holes and welded joints between the pipes and drum are presented in figures 9 and 11. The analysis results show that the material enters the zone of plastic deformation in places of stress concentration around holes, sharp edges and welded joints.

This analysis confirmed the validity of welded joints modeling, as it can clearly be seen in figure 10, that the highest stress concentration occurs around sharp edge of the holes and around connection of weld material with the material from the heat affected zone.

### *Assessment of the structure lifetime*

As the material enters the zone of plastic deformation, low-cycle fatigue analysis is performed [9, 10]. To assess lifetime of the structure dependence diagram of the cycle number on the accumulated effective plastic strain is shown in figure 6.

In tab. 3 are given calculated values of maximum effective stress, the corresponding maximum plastic strain and the number of cycles to crack initiation, for the analysis described in chapter 4.1.

**Table 3. Lifetime of the drum (cycle number)**

| Type of analysis | Maximum effective stress [MPa] | Maximum effective plastic strain [%] | Number of cycles |
|---|---|---|---|
| Projected model | 314.2 | 0.1912 | 2795 |
| Changed material (– 20%) and corrected thickness (– 2%) | 268.8 | 0.2800 | 1495 |

Comparing the results of the analysis with the projected geometrical and material data obtained with the shell elements, it is obvious that they give maximum effective plastic strain higher for 1.3%, and a smaller cycle number for 22%. Also, it is obvious that 3-D finite elements describe more realistic geometry of joints drum and pipes and that is why the maximum stress concentration occurs in the sharp inner edge of the hole and in the welded joint zone, figure 10.

The result of analysis shows that the reduction of thickness of 2% and yield stress of 20%, cause increasing of the maximum effective plastic strain of 5% and reduction of the cycle number of 9%. Figures 12 and 13 show the calculated values of the cycle number to crack initiation, using the amplitude of the effective plastic strain, which is shown in figures 9 and 11, as well as the diagram in figure 6.

Figures 12 and 13 show that the minimum number of cycles is 1495, which is 50% higher than all the cycle number of exploitation so far. In areas of high stress concentration around holes, sharp edges and welded joints accumulation of effective plastic strain is the largest, which can be seen in the figures 9 and 11.

Also, analysis of the impact of reduction of the maximum working pressure on the lifetime of structures was done for this model, for the most critical case when the corrected thickness and material, and the results are shown in tab. 4. In this case, with reduction of maximum working pressure, the value of maximum effective plastic strain is reduced by 19%, while the number of cycles increased by 24%, which is obviously important for safety.

**Table 4. Dependence of the cycle number on the maximum working pressure**

| Maximum working pressure (MPa) | Maximum effective stress (MPa) | Maximum effective plastic strain (%) | Number of cycles |
|---|---|---|---|
| 16.5 | 268.8 | 0.2800 | 1495 |
| 15.5 | 265.5 | 0.2360 | 1980 |

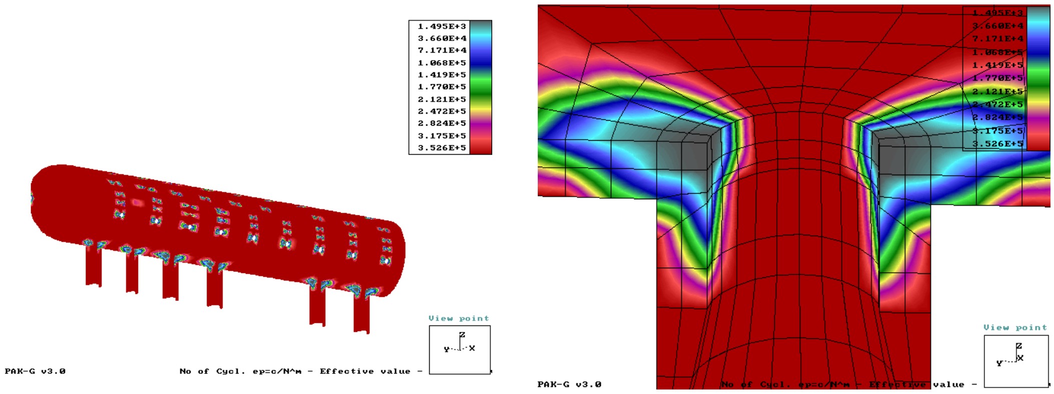

**Figure 12. The number of cycles to crack initiation**

## Conclusions

Assessments of remaining strength and lifetime are based on the analysis of the construction current state and diagnosis of its behavior. The paper describes the methodology of assessment of the drum remaining lifetime in thermal power plant after regular maintenance. During maintenance of the drum the initial cracks were observed, which had been repaired. Corrected thickness of the material after the repair was determined and reduction of yield stress due to aging of materials was estimated. An assessment analysis of the remaining lifetime with the projected geometric and material characteristics and the corrected thickness and yield stress was performed.

Based on previous analysis, detailed modeling of the zone of high stress concentration (holes, sharp edges, and welded joints) is necessary, as are the joints of pipes and drum, because the maximum effective plastic strain gets lower and number of possible cycle gets higher. Significant stress concentration occurs in the welded joints, on the transition between the weld material and material from the heat affected zone. Analysis by 3-D finite elements confirmed the validity of modeling of welded joints, as it can clearly be seen that the highest stress concentration occurs except around sharp edges of the hole and in connection of weld material with the material from the heat affected zone. Based on the above, the conclusion is that this type of analysis should be performed by 3-D finite elements.

Assessment obtained by the previous analysis can be accepted only under the condition that the initial crack on the inside surface structures has been removed on time.

## Acknowledgment

The research results summarized herein are obtained in the framework of research project TR12005 ''Development of software for explicit non-linear dynamic analysis PAK-Explicit'' which is financially supported by Ministry of Science and Technological Development of Serbia.

## Nomenclature

$C$ – constant, [MPa]
$E$ – modulus of elasticity, [MPa]
$\bar{e}^p$ – effective plastic strain
$N$ – exponent, [–]
$M$ – ratio of mixed hardening, [–]
$P$ – pressure, [MPa]
$T$ – temperature, [°C]
$T_{ref}$ – reference temperature, [°C]

*Greek letters*

$\alpha$ – linear thermal expansion coefficient, [°C$^{-1}$]
$\nu$ – Poisson's ratio, [–]
$\bar{\sigma}$ – stress, [MPa]
$\sigma_{yv}$ – initial yield stress [MPa]

*Subscript*

yv – yield stress
ref – reference temperature

*Superscript*

$p$ – plastic strain

## References

[1] Živković, M., *et al.*, Development of Software for Assessment of Remaining Lifetime of Basic Equipment of Thermal Blocks in Electric Power Industry of Serbia Using Fracture Mechanics Methods, Faculty of Mechanical Engineering, University of Kragujevac, Kragujevac, Serbia, 2005
[2] Kojić, M., *et al.*, PAK – Program for Linear and Non-linear Structural Analysis, Faculty of Mechanical Engineering, University of Kragujevac, Kragujevac, Serbia, 2005
[3] Živković, M., Non-linear Structural Analysis, Monograph, Faculty of Mechanical Engineering, University of Kragujevac, Kragujevac, Serbia, 2006
[4] Kojić, M., *et al.*, Finite Element Method I – Linear Analysis, Faculty of Mechanical Engineering, University of Kragujevac, Kragujevac, Serbia, 1998
[5] Živković, M., *et al.*, A General Beam Finite Element with Deformable Cross-Section, *Computer Methods in Applied Mechanics and Engineering, 190* (2001), 22-21, pp. 2651-2680
[6] Kojic, M., Bathe, K. J., Inelastic Analysis of Solids and Structures, Springer, Berlin-Heidelberg, Germany, 2005
[7] ***, Československa Statni Norma, OCEL 15223 Mn-Mo, ČSN 41 5223, 1977
[8] Lazić, V., *et al.*, Theoretical-Experimental Determining of Cooling Time (t8/5) in Hard Facing of Steels for Forging Dies, *Thermal Science, 14* (2010), 1, pp. 235-246
[9] Jovičić, G., *et al.*, Residual Life Estimation of a Thermal Power Plant Component – the High-Pressure Turbine Housing Case, *Thermal Science, 13* (2009), 4, pp. 99-106
[10] Jovičić, G., Živković, M., Jovičić, N., Numerical Simulation of Crack Modeling Using Extend Finite Element Method, *Strojniski Vestnik, 55* (2009), 9, pp. 549-554

Paper submitted: May 7, 2010
Paper revised: June 30, 2010
Paper accepted: July 4, 2010