EXPERIMENTAL AND NUMERICAL STRESS AND STRAIN ANALYSIS OF THE BOILER REVERSING CHAMBER TUBE PLATE

by

Milena N. RAJIĆ [a*], Dragoljub S. ŽIVKOVIĆ [a], Milan S. BANIĆ [a], Marko V. MANČIĆ [a], Taško Dj. MANESKI [b], Miloš S. MILOŠEVIĆ [c], and Nenad R. MITROVIĆ [b]

[a] Faculty of Mechanical Engineering, University of Nis, Nis, Serbia
[b] Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia
[c] Innovation Centre of Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia

Original scientific paper
https://doi.org/10.2298/TSCI210313207R

Boilers are one of the most used units for both heat generation plants and industry systems. Their operation is subjected to different working loads and maintenance requirements. Exploitation experience points out critical boiler zones where failures and break downs typically occur. This paper analyzes critical zones in hot water fire-tube boiler. Experimental procedure was performed on the model of this type of boilers and its critical element. The tube plate of hot water boiler was identified as the most critical one. Experimental analysis and numerical model verification were performed using Aramis system based on 3-D digital image correlation method. Numerical analysis was done in ANSYS software package and verification of results was done based on measurements obtained by strain gauges and local measurements performed by the Aramis system. Stress-strain analysis indicates the critical zones of boiler tube plate. The character of change parameters such as strain and stress occurring in the critical zones can be verified both by experimental and numerical data. The paper presents a novel approach in experimental and numerical analyses that can be conducted in similar units and used for existing unit optimization, as well as for new product testing on different loads and provide opportunity for further development and improvement for practical industrial application.

Key words*: hot water boilers, 3-D digital image correlation, finite element method*

Introduction

Boilers are typically used in heat generation plants and industrial heat plants. Boiler design depends on capacity, working fluid and working conditions. Hot water boilers can be of various designs. Hot water fire tube boilers have a fire tube wherein the combustion is performed, and can be fitted with water-cooled reversing chamber or without screened walls [1]. The boiler elements are exposed to high pressure and temperature of the working fluid, which imply application of strict operational requirements for their safe and reliable operation. In order to avoid any break downs and failures during the operation it is necessary to foreseen possible scenarios of accidental of critical boiler zones.

* Corresponding author, e-mail: milena.rajic@masfak.ni.ac.rs

Previous investigations and studies done in this field point out the lack of literature data on measurements conducted on the boiler itself or boiler elements. Analyses performed so far are based on recommendation data given in standards and technical norms [2-4]. The allowed stress values provided within the standard are based on simplified membrane stress analysis caused by the loads originating from pressure of working fluid only. Other load, such as the thermal load, causes different type of stress which is not included in safety norms and standards [2, 4-10]. It should be emphasized that the procedure of calculation given in existing standards introduces safety factors especially in calculation of wall thickness. A large value of the safety factor used in calculations can lead to an over-dimensioned element thickness, where they lead not only to increased costs, but also increased thermal stresses and may have an unfavourable impact to the strength of the structure.

Analytical analysis and application of elasticity method is not quite possible due to the complexity of the boiler structure. Therefore, it is not possible to obtain an analytical problem solution. For stress-strain analysis and calculation of boiler elements, finite element method (FEM) has been used more frequently recently. The FEM represents a numerical method that enables modelling and calculation of complex constructions and problems by dividing the structure into finite elements with correct geometric form, whose behaviour is possible to describe. Stress-strain analysis of hot water boiler indicates different application of FEM [11-18]. These analyses are based on data given in standards or in boiler manufacturer documentation, but present general averaged data. There is no evidence of experimental data that can be used to determine the construction temperature field of the boiler's element, neither stress-strain filed measured in any part in this type of boiler. One of the limitations of experimental procedure is high cost of experiments that should be performed. Boilers are expensive pressure vessel units, therefore, experiments are limited to certain part of the boiler that can be easily repaired without endangering the essential part of the unit. Secondly, this boiler type operates in high temperature and pressure regimes, hence security and safety level of the plant and its personnel involved in the experimental procedures is also one of the limitations. Finally, the third reason for not having this kind of experiments is the limitation of experimental equipment and quality of the collected data. Conventional methods of measuring stress-strains of the boiler construction would give us the data in singular point or many analyzed points, without stress-strain field distribution. In this manner, the distribution of stress filed would be also averaged. Having analyzed the aforementioned limitations, a decision was made to perform an experiment on the model in the laboratory where several measurement procedures can be conducted with different loads, measurement conditions*, etc.* The data from the complex experimental procedure will be presented in this paper. The measurement includes: temperature measurement of the critical element of the boiler model is performed using a thermocouple, strain measurement of the same critical element is performed using a strain gauge and the digital indicator watch comparator and the 3-D digital image correlation (3D-DIC) method is applied for stress-strain measurement. The numerical calculation of the boiler model with all real time parameters data was done and the comparison of given data is presented in this paper. Considering a large amount of data that is obtained in this experimental procedure, only the critical parts of the tube plate (as one of the most critical part of this type of boilers) in critical regimes will be presented.

The construction model testing or its structure is of high importance especially in cases of prototype testing for solving problems of construction design [19]. Using the method of experimental analysis, a relevant data are usually obtained for assessing the load capacity and stability of the structure. Data related to the effect of local stress concentrations are particularly important, determining the area of plastic deformations, the fracture mechanisms, as well as the impact of changes of mechanical characteristics with respect to time and temperature. Experimental methods are of the highest importance especially in those cases where the theoretical model is almost impossible or very difficult to solve due to the complexity of the problem. Experimental methods are particularly important in cases where significant stress concentrations on the construction occur, which in operating conditions very often lead to failures and break-downs.

The 3-D-DIC method represents a novel method for full-field stress-strain measurement [20-23]. A single measurement results in big datasets replacing a large number of strain gauges in a conventional experiment, and therefore, significantly reduces experimental planning time and preparation as well as costs. Also, numerical methods enable results of full covered stress-strain field that can be easily verified by the data obtained from experiment. This 3-D-DIC method has high accuracy (up to 1 μm) and it has been used in variety of cases so far: different material testing, structure testing, model verification, fracture mechanism [24-37].

The paper presents a new approach of application of Kriging interpolation in coupled analysis in order to obtain the rigidness of the real boiler unit and suitable model for experimental procedure. The aim of this paper is to analyze critical elements of the hot water boiler by obtaining experimental data from its model and forming an improved numerical model. The investigation was performed in order to analyze constrains and loads the model is exposed to and provide stress-strain field in dependence of the different pressure loads obtained from experimental data.

Design and construction of a model for experimental analysis

For the purposes of a laboratory experiment, a model suitable for testing was designed. As industrial boiler units have large dimensions, it was necessary to scale the object to an appropriate model dimension, which could easily be installed in laboratory conditions. Having that in mind, the scaling of the large boiler unit was done by using numerical simulation, with purpose to preserve the rigidity of the real object and the model for experimental analysis. This was done, since stress-strain analysis of tube plate would be examined under different conditions. In order to form a suitable model for experimental analysis, with the same construction rigidity as the real object, the numerical simulation was done in ANSYS software package.

All dimensions were scaled 10 times compared to the real boiler unit, except the thickness of the tube plate, the diameter of the flue gas pipes and the thickness of the fire tube. In order to identify suitable dimensions of the previously mentioned elements, the scaled geometric model was examined by using FEM. The numerical simulation was performed where individual points were selected in the critical zone of the real object and the model. These critical points were located on the tube plate. Eight different points, distributed along the tube plate of the first and second reversing chambers were selected, where the rigidity of the model and the real object should be equivalent. A coupled analysis was performed, which was necessary, because it was impossible to provide a Fox corrugated fire tube with dimensions required for the model. A plain tube was selected, but the coupled analysis of the numerical simulation would give the information about the required thickness of the fire tube so that the rigidity of that element would be identical in the operating conditions of the real boiler unit.

The other experienced problem was that, due to the complexity of the distribution and the number of flue pipes of the real boiler object, in which there are 208 flue pipes, the scaled flue pipes, would give an extremely dense pipe network in the model. The scaled flue pipes were also impossible to provide due to their dimensions. Hence 44 *full tubes* or rods were adopted as geometrical solution, distributed on the tube plate of the model, following the schedule and distribution of the flue pipes on the real object. The coupled analysis would give the dimensions of these rods, with the rigidity of the structure remaining identical or substantially the same in the already defined critical points of the model and the real boiler unit. The coupled analysis is based on the fact that it is necessary to transfer the same loads of the real boiler object to the model. Geometric model of the adopted experimental model is presented in fig. 1.

The geometric model was transformed into the discretized finite element model with the application of advanced meshing tool capable of creating adaptive discrete model. The discretized model consisted of 438128 nodes, which formed 352921 finite elements. The discretized model of the analyzed boiler is presented in fig. 2. The finite element mesh has an identical topology for thermal and structural analysis, but these meshes are formed by different types of finite elements. The resulting meshes are automatically unified into a unique hybrid finite element mesh in which the definite topology overlaps the finite elements relevant to individual analyzes.

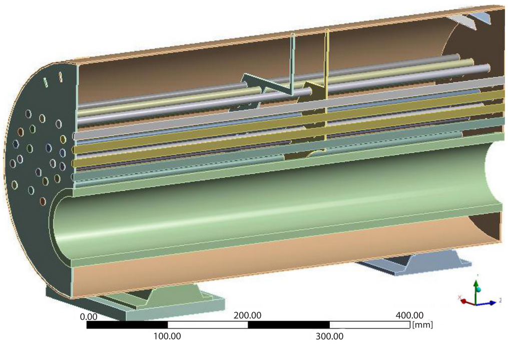

Figure 1. Symmetrical geometric model

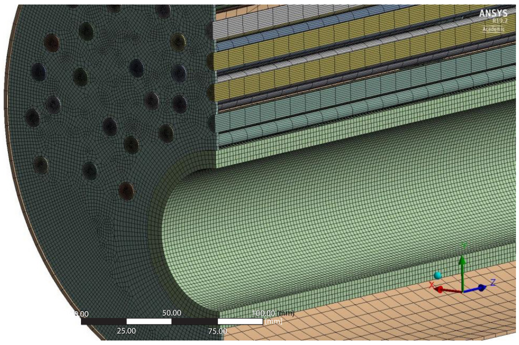

Figure 2. Numerical model of discretized structure of the model

Working loads were also defined to simulate the experimental loads. With the adopted reference points, at the same positions as the real object, the numerical simulation was performed. The resulting criteria were to determine the thickness of the tube plate of the model and the diameter of the rods, while preserving the rigidity of the model, which means that the rigidity of the model and the real object can be considered equivalent. As a first step in this process, the parameterization of the required geometric model was introduced. The maximal and minimal values of the thickness of the tube plate and the diameter of the rods were defined

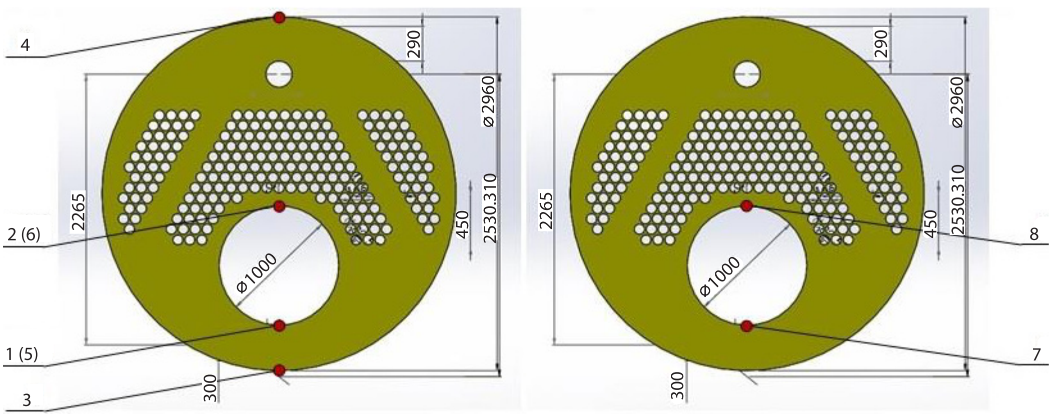

Figure 3. Marked reference points of numerical calculation of the real boiler unit

Table 1. Values of equivalent stress and deformation in reference points of the real object

Reference point	Character of the variable	Value
1	Equivalent stress [MPa]	178.49
2	Equivalent stress [MPa]	193.41
3	Equivalent stress [MPa]	126.65
4	Equivalent stress [MPa]	0.14681
5	Directional deformation [mm]	4.4101
6	Directional deformation [mm]	2.1768
7	Equivalent stress [MPa]	98.373
8	Equivalent stress [MPa]	111.79

as the boundaries of these parameters. Parameterization of geometric measures is performed in the CAD software, the SolidWorks package, after which the parameterized geometry is loaded into the Ansys software package for FEM analysis. After pre-processing (which includes the material selection, model discretization, defining the contacts as bonded in the model, the loads definition and setting the boundary conditions), the numerical solving of the equation system was performed. The output values of the numerical simulation were the stress values at the six selected points on the tube plate and the deformation at the two selected points of the tube plate, fig. 3. Values in reference points are given in tab. 1.

After defining the output parameters, a virtual numerical experiment was defined through the design of experiment software module. Within this phase of optimization, the approximate functions were found that gave the dependence of the previously defined output parameters from the input geometric parameters.

After determining approximate functions using the optimization procedure, optimal values of the input geometric parameters are required – the thickness of the fire tube and the diameter of the rods. The optimization process provides more possible solutions, so the final choice of dimensions depends on the decision of the designer or user.

The application of the Kriging interpolation determined the approximate functional dependences of the thickness of the fire tube and the diameter of the rods from the given stress values and deformation [4]. Kriging's interpolation was applied, because in this case, the minimum deviation of the approximate functions from the experimental data was achieved. Based on the results obtained in this analysis, it can be concluded that the results obtained by the approximate functions for the stress-deformation state at the defined points, tab. 1, perfectly match the results of the virtual experiment. The same conclusion can be made by analysis of the graph of the relationship between the results obtained by the approximate functions and the results of the virtual experiment that is shown in fig. 4.

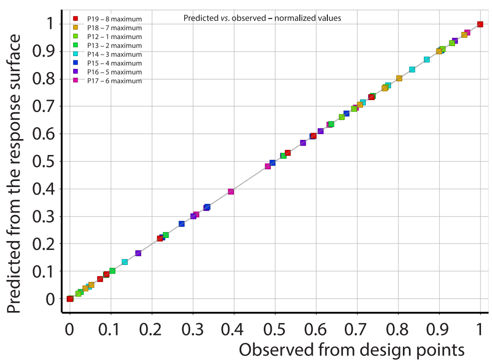

Figure 4. The ratio of the results obtained through the approximate functions and results of the virtual experiment

Based on the results of the numerical experiment, the resulting thicknesses are for fire tube 5.5 mm and for the rods 6.0 mm. These elements were installed more easily into the model. The obtained and adopted geometric model is further prepared for production.

Experimental procedure

Experimental testing had the aim to verify the numerical model. With verified numerical model we can examine similar models of similar constructions and with similar loads, without endangering the real objects installed within the plants.

The experimental methods that were used are strain gauge method and methods for contactless measurement of stress and deformation. The given load is symmetric, so it allows (along with the already existing symmetry of the construction itself) parallel measurement using the strain gauge and the Aramis system [38] based on 3-D-DIC. Having that in mind, four points were selected on the tube plate (two for strain gauges and two for Aramis measurement), which were used for symmetric parallel measurement. Two strain gauges are placed at the selected points on one half of the structure (the tube plate), and the other half was used to record corresponding fields with cameras, fig. 5.

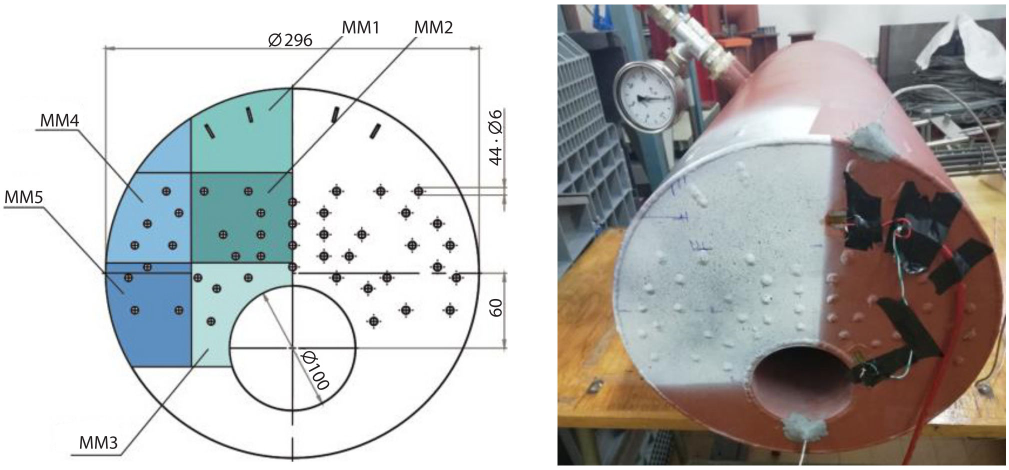

Figure 5. The strain gauges locations and the measuring area for 3-D-DIC system

For strain measurement using the DIC system, appropriate procedures are defined by producer and developed for experimental testing. The experiment was performed according to the following procedure: *sample preparation* (the measuring surface must have a pattern with contrast to clearly allocate the pixels in camera images), *measuring volume selection* (selecting the measuring volume is based on the sample size), *system calibration* (for certain measuring volume, the appropriate calibration panel is used for sensor configuration), *sample positioning* (the boiler model was positioned in a way that appropriate tube plate was fully exposed to cameras, the one side of the boiler was fixed as it is in real operating conditions), and *measurement* (after calibration, measurement procedure was performed). The pressure on the water side was gradually loaded, up to 5 bars. Digital images were recorded 60 seconds after the loading), data processing (recorded data is in a form of report, that can be further analyzed).

The experiment was performed for each measuring area (five marked measuring areas, fig. 6) in phases, with different loads-pressures on the water side (the boiler was previously filled with water). Filling the boiler with water and setting the pressure was carried out by a manual pump. The pump is connected to the model with the valve to allow constant pressure during the test. The measuring pressure was in interval 1-5 bar with 60 seconds stabilization time period between the measured pressure set.

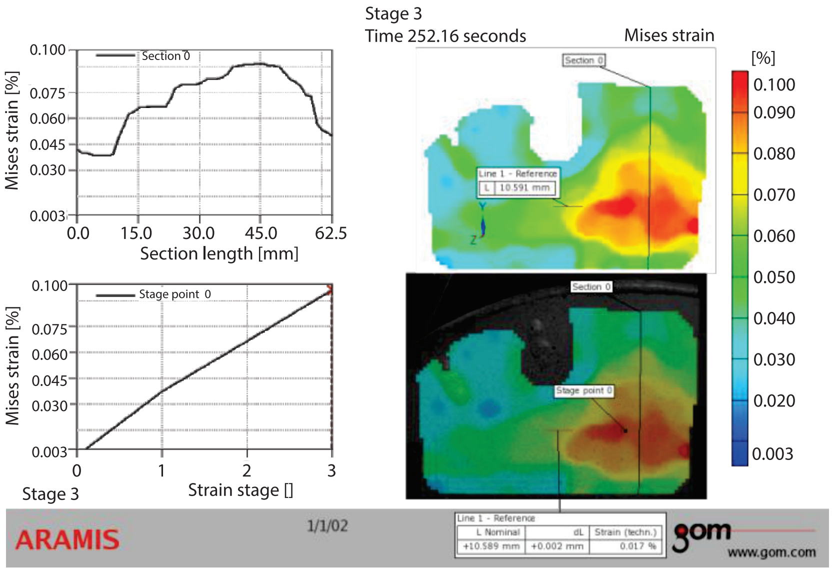

Figure 6. Experimental results of Von Mises strain for maximum test pressure of 5 bar; (a) Von Mises strain as a function of distance for marked Section, (b) Von Mises strain as a function of strain stage, (c) Von Mises strain field, and (d) sample image with overlaying Von Mises strain field (measuring Area 1).

Results and discussion

After the experimental procedure, the critical areas were identified, which will be presented. Von Mises strain results are presented for measuring Area 1, fig. 6. The examined condition was under max test pressure of 5 bar. Experimental data is presented graphically as function of section length. Scale in % is given on ordinate of fig. 6. The 3-D Von Mises strain field across the sample surface (measuring Area 1), figs. 6(c) and 6(d) presents the highest measured strains (orange color). Von Mises strain values are also given as section length function, figs. 6(a) and 6(b). Strain-stages (0-3) represent pressure increase, where stage 0 represents the beginning of the experiment and stage 3 maximum test pressure for the experiment, when pressure on the water side was 5 bar. Figure 6(d) shows a line which *imitates* virtual strain gauge. The Aramis system software provides ability to determine the distance between any two points in any moment of experiment. The horizontal line here is in the same place as the real strain gauge in order to verify results and compare them.

Table 2 presents the comparison of measured results by using strain gauge, DIC Aramis system and results from finite element method numerical model done in ANSYS software. The results presented are obtained using testing pressure of 3 bar. The results of certain pressures values are missing form DIC Aramis system because the measurement was conducted in stages of (2 bar, 3.5 bar, and 5 bar). Measured results from strain gauge under pressure of 3.5 bar is omitted, while the results for pressures of 4 and 5 bar are presented.

Table 2. Resulting strain value measured by strain gauge, DIC Aramis system, results of FEM

Pressure [bar]	Strain gauge	Aramis system	FEM ANSYS
1	$3.2931 \cdot 10^{-5}$	–	$3.0189 \cdot 10^{-5}$
2	$5.7071 \cdot 10^{-5}$	$9.44376 \cdot 10^{-5}$	$6.01 \cdot 10^{-5}$
3	$8.5409 \cdot 10^{-5}$	–	$8.9972 \cdot 10^{-5}$
3.5	–	$1.30344 \cdot 10^{-4}$	$1.009 \cdot 10^{-4}$
4	$1.47662 \cdot 10^{-4}$	–	$1.201 \cdot 10^{-4}$

As it can be observed, the results from the strain gauge and the numerical results match considerably. The numerical model can be verified under different pressure loads by the measurement results of strain gauge. The 3-D-DIC Aramis system has certain limitation, especially when there are small displacements. Here in experimental results there are deviations of around 30% compared to the numerical model or the measured strain by the strain gauge. The advantage of this measurement method is definitely the whole measured strain area. The results of the total deformation are presented in the Aramis system and the numerical model is done in Ansys software, as it can be seen on figs. 7 and 8. The pressure load was 2 bar on the water side. The distribution of total deformation field is quite the same as well as critical zones.

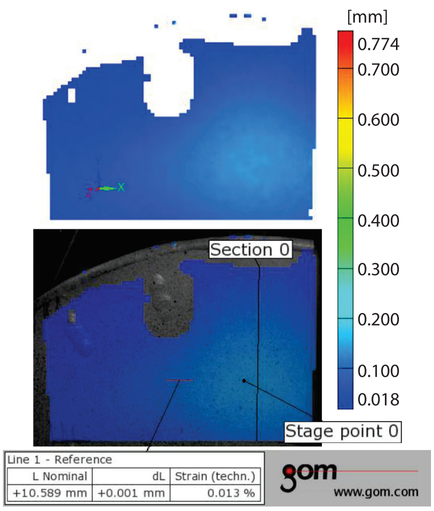

Figure 7. **Results of total deformation measured by DIC Aramis system; pressure load 2 bar**

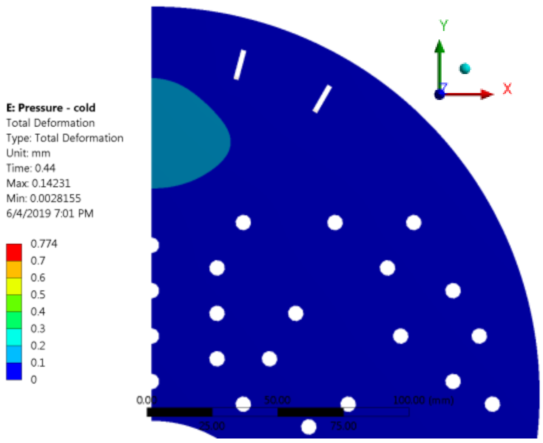

Figure 8. Results of total deformation done numerically by Ansys software package; pressure load 2 bar

Results of measured total deformation by Aramis system and obtained numerically by finite element method is quite the same. The critical zones are identified as well as the measured total deformations are exactly the same (in critical zone 0.1-0.2 mm). More precisely, the minimal total deformation under the test pressure of 2 bar is $2.8155 \cdot 10^{-3}$ mm and maximal is 0.14231 mm.

Next experimental stage was under pressure load of 3.5 bar. The result of total deformation done by Aramis and numerical method by ANSYS is presented in figs. 9 and 10.

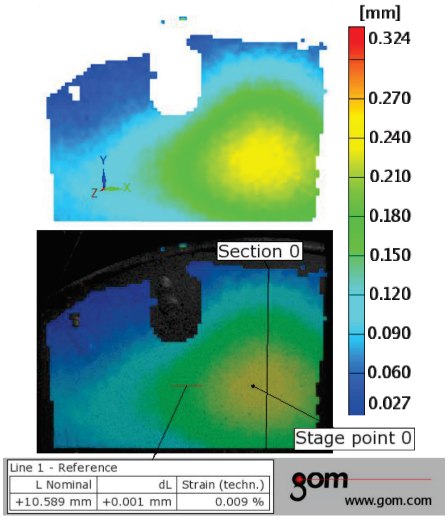

Figure 9. Results of total deformation measured by DIC Aramis system; pressure load 3.5 bar

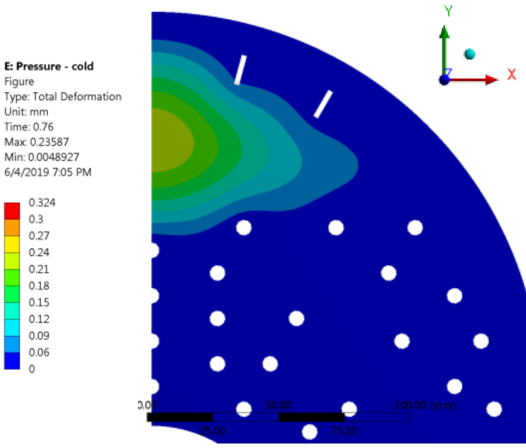

Figure 10. Results of total deformation done numerically by Ansys software package; pressure load 3.5 bar

The critical zone is now wider and with the exact value in both cases (in critical zones 0.21-0.24 mm). The distribution of the total deformation field has also the same position. Minimum total deformation in this stage is $4.8927 \cdot 10^{-3}$ mm and maximum 0.23587 mm.

In both cases the values obtained by the measured system 3-D-DIC Aramis system shows significant matches with values obtained using finite element method by ANSYS software package. In this case, it can be said that numerical model has the same behaviour under the same loads as the real model.

Conclusions

In this paper, experimental and numerical analysis of boiler tube plate was presented. The suitable model of the real boiler object is calculated and produced. The aim was to verify numerical model by comparison with the measured values of strain or deformation. Apart from application of the conventional method for strain measurement (as strain gauge), a novel approach was applied using the 3-D-DIC System for measurement. The obtained results have the significant matches in both cases. These strain values are compared to results conducted by finite element method using ANSYS software package. In this paper, a novel approach of analysis of full-field experimental DIC method and numerical data, used especially on critical boiler element is applied. Qualitative comparison of experimental and numerical results shows that the strain values and distribution pattern in the critical measured area can be considered satisfactory. Therefore, the formed numerical model is completely verified and can be used for other model testing in order to improve the performances and the design of the real construction of boilers with similar geometry.

Future research should involve experiment planning with a suitable heater since it can be simulated with similar conditions of heating model construction. The planed experiment should also include different pressure loads on the water side, as well as adjustable heating of the fire tube of the model.

Acknowledgment

This research was financially supported by the Ministry of Education, Science and Technological Development of the Republic of Serbia (Contract No. 451-03-9/2021-14/200109).

References

[1] Nazeer,W. A., *et al*., In-situ Species, Temperature and Velocity Measurements in a Pulverized Coal Flame, *Combustion Sciences and Technology*, *143* (1999), 2, pp. 63-77
[2] Gulič, M., *et al*., *Steam Boilers* (in Serbian), Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia, 1991
[3] Gaćeša, B., Numerical-Experimental Analysis of Strength of Boiler Structures, Ph. D. thesis, Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia, 2011
[4] ***, EN 12953 – Shell Boilers – General, Materials for Pressure Parts of Boilers and Accessories, Design and Calculation for Pressure Parts, Requirements for Equipment for the Boiler, 2012
[5] Rajic, M., *et al*., Construction Optimization of Hot Water Fire-Tube Boiler Using Thermo Mechanical Finite Element Analysis, *Thermal Science*, *22* (2018), Suppl.5, pp. S1511-S1523
[6] Qian, C. F., *et al*., Finite Element Analysis and Experimental Investigation of Tubesheet Structure, *Journal of Pressure Vessel Technology*, *131* (2009), 1, pp. 111-114
[7] Yong Ling, Y., JU, Y., The Analysis of the Stress and Shift of Tube plate and Edge of Manhole of Boiler, *Journal of Dalian Fisheries University*, *1* (2000), 1, pp.71-75
[8] Čukić, R., Maneski, T., Thermomechanical Stress Analysis of the Hot-water Boiler by FEM, *Proceedings*, 3rd International Congress of Thermal Stress 99, Cracow, Poland, 1999
[9] Liu-Juan, Z., *et al*., Finite Element Analysis of Overall Strength of Shell and Tube Waste Heat Recovery Boiler, *Industrial Boiler*, *1* (2009), 1, pp. 19-22
[10] Yong-Ling, J., Min, T., Simplification of Models and Stress Calculation of Smoke Tubes for Boilers, *Journal of Anshan Institute of Iron and Steel Technology*, *4* (2000), 23, pp. 282-286
[11] Gong-Ping, W., *et al*., The Cause and Prevention of the Tube Plate Crack of one Gas-Fired Boiler, *Industrial Boiler*, *1* (2009), 1, 54
[12] Ji, Z., *et al.*, Thermo-Plastic Finite Element Analysis for Metal Honeycomb Structure, *Thermal Science*, *17* (2013), 5, pp. 1285-1291
[13] Gaćeša, B., Thermomechanical Analysis of Behaviour and Improvement of Steam Block-Boiler with Three Flue Gas-Flows Producing Smaller Steam, *Procesna Tehnika*, *18* (2002), 1, pp. 111-114
[14] Todorovic, M., *et al*., Application of Energy and Exergy Analysis to Increase Efficiency of a Hot Water Gas Fired Boiler, *Chem. Ind. Chem. Eng. Q*., *20* (2014), 4, pp. 511-521
[15] Gaćeša, B., *et al*., Numerical and Experimental Strength Analysis of Fire-Tube Boiler Construction, *Tehnički vjesnik*, *18* (2011), 2, pp. 237-242
[16] Gaćeša, B., *et al*., Influence of Furnace Tube Shape on Thermal Strain of Fire-Tube Boilers, *Thermal Science*, *18* (2014), Suppl. 1, pp. S29-S47
[17] Živković, D., *et al*., Thermomechanical Finite Element Analysis of Hot Water Boiler Structure, *Thermal Science*, *16* (2012), Suppl. 2, pp. S443-S456
[18] Živković, D., *et al*., Numerical Method Application for Thermo-Mechanical Analysis of Hot Water Boilers Construction, *Proceedings*, 24th International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems – ECOS 2011, Novi Sad, Serbia, 2011, pp. 1351 - 1362
[19] Milćić, D., *et al.*, Finite Element Thermal Analysis of Hot Water Boilers, *Proceedings*, 14th Symposium on Thermal Science and Engineering of Serbia – SIMTERM 2009, Sokobanja, Serbia, 2009, pp. 692-697
[20] Jian, H., YuPeter G. D., Dynamic Impact Deformation Analysis Using High-Speed Cameras and ARAMIS Photogrammetry Software, Army Research Laboratory, Arligton, Tex., USA, ARL Report Number ARL-TR-5212, 2010
[21] Orteu, J., The 3-D Computer Vision in Experimental Mechanics, *Opt. Lasers Eng*., *47* (2009), 3-4, pp. 282-291
[22] Pan, B., *et al*., Optimization of a 3-D Digital Image Correlation System for Deformation Measurements in Extreme Environments, *Appl. Opt*., *51* (2012), 19, pp. 440-449
[23] Sutton, M., *et al*., *Image Correlation for Shape*, Motion and Deformation Measurements: Basic Concepts, Theory and Applications, Springer, Berlin, Germany, 2009
[24] Mitrović, N., *et al.*, Experimental and Numerical Study of Globe Valve Housing, *Chemical Industry*, *71* (2017), 3, 1, pp. 251-257

[25] Milosevic, M., *et al*., Digital Image Correlation in Analysis of Stiffness in Local Zones of Welded Joints, *Technical Gazette*, *23* (2016), 1, pp. 19-24
[26] Mitrovic, N., *et al*., Strain Measurement of Pressure Equipment Components Using 3-D DIC Method, *Proceedings*, Structural Integrity Procedia 13, 22nd European Conference on Fracture – ECF22, Belgrade, Serbia, 2018, pp. 1605-1608
[27] Milosevic, M., *et al*., Measurement of Local Tensile Properties of Welded Joint Using Digital Image Correlation Method, *Chemicke Listy*, *106* (2012), Jan., pp. 485-488
[28] Lezaja, M., *et al*., Bond Strength of Restorative Materials to Hydroxyapatite Inserts and Dimensional Changes of Insert-Containing Restorations during Polymerization, *Dental Materials*, *31* (2015), 2, pp. 171-181
[29] Milosevic, M., *et al*., Digital Image Correlation Analysis of Biomaterials, *Proceedings*, 15th IEEE International Conference on Intelligent Engineering Systems, 2011, Poprad, Slovakia, pp. 421-425
[30] Tanasic, I., *et al*., Enhanced in-Vivo Bone Formation by Bone Marrow Differentiated Mesenchymal Stem Cells Grown in Chitosan Scaffold, *Journal Bioengineer & Biomedical Sci.*, *2* (2012), 1, pp. 1-6
[31] Mitrovic, A., *et al*., Thermal and Mechanical Characteristics of Dual Cure Self-Etching, Self-Adhesive Resin Based Cement, *Experimental and Numerical Investigations in Materials Science and Engineering*, *54* (2018), Sept., pp. 3-15
[32] Milosevic, M., *et al*., Analysis of Composite Shrinkage Stresses on 3-D Premolar Models with Different Cavity Design using Finite Element Method, *Key Engineering Materials*, *586* (2014), Sept., pp. 202-205
[33] Tanasic, I., *et al*., Analysing Displacement in the Posterior Mandible using Digital Image Correlation Method, *Journal Biochip Tissue Chip*, *S1* (2011), Jan., pp. 1-6
[34] Petrovic, S., *et al*., Rapid Evaluation of Maintenance Process Using Statistical Process Control and Simulation, *Int. Journal of Simulation Modelling*, *17* (2018), 1, pp. 119-132
[35] Sadowski, T., *et al*., Modelling and Experimental Study of Parallel Cracks Propagation in an Orthotropic Elastic Material, *Comput. Mater. Sci*., *52* (2012), pp. 231-235
[36] Balac, M., *et al*., Numerical Predictions of Crack Growth in a Pressure Vessel with Welded Nozzles, *Structural Integ. Lif*e., *15* (2015), Jan., pp. 55-61
[37] Rajic, M., *et al*., Experimental and Numerical Analysis of Stress-Strain Field of the Modelled Boiler Element, Experimental and Computational Investigations in Engineering, CNNTech 2020, Lecture Notes in Networks and Systems, Springer, *Cham*, *153* (2021), Sept., pp. 257-273
[38] ***, Aramis – The 3-D Deformation Analysis, GOM, Germany, http://www.gom.com/metrology-systems/systemoverview/aramis.html

Paper submitted: March 13, 2021
Paper revised: May 7, 2021
Paper accepted: May 7, 2021