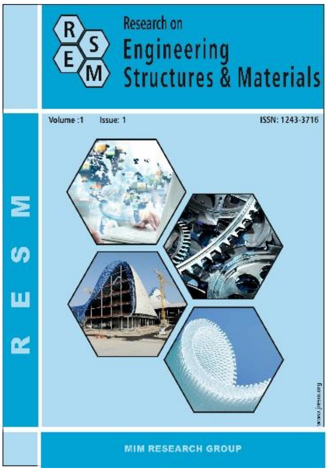
Research on
Engineering
Structures & Materials
journal homepage: http://jresm.org

RESEARCH
GROUP



Numerical analysis of the effect of conical turbulators to heat transfer performance of a liquid fuelled boiler

Bilal Sungur, Bahattin Topaloglu, Hakan Ozcan, Lutfu Namli

Online Publication Date: 20 Feb 2017
URL: http://dx.doi.org/10.17515/resm2016.53me0621
DOI: http://dx.doi.org/10.17515/resm2016.53me0621

Journal Abbreviation: *Res. Eng. Struct. Mat.*

To cite this article
Sungur B, Topaloglu B, Ozcan T, Namli L. Numerical analysis of the effect of conical turbulators to heat transfer performance of a liquid fuelled boiler. *Res. Eng. Struct. Mat.*, 2018; 4(2): 127-136.

Research on Engineering Structures & Materials
journal homepage: http://jresm.org
MIM RESEARCH GROUP

Research Article

Numerical analysis of the effect of conical turbulators to heat transfer performance of a liquid fuelled boiler

Bilal Sungur, Bahattin Topaloglu*, Hakan Ozcan, Lutfu Namli

[1] *Department of Mechanical Engineering, Ondokuz Mayis University, Samsun, Turkey*

Article Info

Article history:
Received 21 Jun 2016
Revised 17 Feb 2017
Accepted 17 Feb 2017

Keywords:
Spray combustion, Numerical modeling, Conical turbulators, Temperature distributions

Abstract

In this study, increasing the efficiency of liquid fuelled smoke tube boilers used for domestic heating was researched. In this context, turbulators with conical geometries placed to smoke tubes of boiler and effects on flame structure and heat transfer were investigated numerically. Calculations were carried out at two dimensional axisymmetric conditions and Fluent was used as the computational fluid dynamics software. In all cases, the standard k-ε model was used for modeling the turbulent flow and the species transport model was used for modeling the combustion. Based on the results of these turbulators were evaluated for each condition. Besides, temperature and stream function distribution and pressure drop were investigated according to the arrangement of turbulators and which number of turbulator would be the most appropriate at boilers was discussed. Increasing the number of turbulators from zero to three decreased the exhaust temperatures 15 K from 365 K to 350 K, and consequently the efficiency of the boiler was increased. Yet the turbulators inserted into smoke tube increased the pressure losses insignificantly from 20 to 23.5 Pa.

1. Introduction

Energy production, interest in alternative energy sources and the efficiently usage of energy are increasing with each passing day with population increment. Especially, in recent years various methods have been developed to save energy in the energy sector. One of the areas where energy savings can be applied is heat exchangers. Smoke tube boilers are also a heat exchanger used for heating purposes. The techniques used to increase the heat transfer in heat exchangers can be classified as active and passive. If heat transfer happens with additional power to the fluid or environment which provides improvements in heat transfer, it is called active method, and if heat transfer happens without additional power, it is called passive method. Increment of the surface areas increases the heat transfer. But with increasing surface, volume of the heat exchanger is growing. To prevent this, a turbulator inserted into the heat exchanger provides an increase on the heat transfer surface area and the heat exchangers volume will remain stationary. Turbulators placed into the pipe increase the turbulence and improve the heat transfer. In other words, turbulators are devices which improve the heat transfer between fluids at different temperatures. In the installed systems, increasing the efficiency of the existing system with passive heat transfer methods (demountable turbulators) is preferred rather than rebuild the entire system for heat exchangers or change necessary equipment’s with new one. Thus, these devices, which are unlikely to be used for other systems can be used again and manufacturing companies will have the opportunity to sell their products for a cheaper price and with a better quality.

*Corresponding author: btopal@omu.edu.tr
DOI: http://dx.doi.org/10.17515/resm2016.53me0621

Many researchers have realized research on the heat transfer effects of turbulators which are inserted in the pipes. Muthusamy et al. [1] investigated the heat transfer, friction factor and thermal performance of conical turbulators with inner fins on flow direction and on the opposite flow direction experimentally. They stated that the turbulators inserted on the flow direction had better results. Yakut and Sahin [2] investigated experimentally the effect of coiled wire turbulators inserted to the tubes on the heat transfer and the friction factor. Kahraman et al. [3] investigated experimentally and numerically the heat transfer performances of turbulators with two different fin gap (b=0.1 and 0.2 m) and three different angles (θ=30°, 45° and 60°). They reported that in all cases, using turbulators in tubes increased the Nusselt number. Akansu [4] analyzed numerically the effect of turbulators like porous-ring shaped on the heat transfer and the pressure drop. He used Fluent software in calculations. He used k-ω model as turbulence model and air as a fluid. Karakaya and Durmuş [5] experimentally investigated the effect of conical spring turbulators on the performance of the heat transfer and the pressure drop in a pipe. They observed that increasing the heat transfer increased the pressure losses and therefore an optimization should be made for pressure losses.

Numerical modeling of the combustion in the boilers was investigated by many researchers. Saario et al. [6] investigated the heavy fuel oil combustion in the furnace experimentally and numerically. They used Fluent software for numerical simulation and they tried standard k-ε and Reynolds Stress Model (RSM) separately as turbulence model. The mixture fraction approach and the probability density function approach were used as combustion model. They said that predicted gas concentrations were in satisfactory agreement with measurement results, but they found that there were conflicting results near the burner and near the furnace axis. Hu [7] researched the liquid fuel combustion at different fluid spray burners experimentally and numerically. He announced that numerical results provided a good agreement with experimental results. Ing et al. [8] numerically analyzed the spray characteristics of biodiesel-diesel mixtures at different mixing ratios in their work and compared these mixtures to pure diesel. They used FLUENT package program for numerical modeling. They stated that k-ε model was used as turbulence model and this model successfully predicted the spray characteristics of all biodiesel blends.

Numerical evaluation of the effect of turbulators on the heat transfer performance of a boiler is very important to improve the heat transfer performance of this kind of systems for energy savings. In this study a liquid fuelled flame tube boiler was modeled at two-dimensional conditions with four different geometries. The effect of conical turbulators in various numbers which were inserted to smoke tubes on the heat transfer was investigated. The first geometry was the basic geometry without a turbulator in smoke tubes. The other three were with one, two and three turbulators. As a result, temperature and stream function contours, velocity vectors and exhaust gas temperatures were investigated and results were discussed.

2. Materials and Methods

Computational Fluid Dynamics (CFD) is a field of fluid mechanics, which solves the problems related to heat transfer, fluid motion, particle motion, droplet motion and combustion with numerical methods and algorithms via a computer. In recent years, developments on the technology and computer science have increased the importance of CFD software. Using physical experiments to get essential engineering data for design can be expensive and time consuming. CFD simulations are relatively cheap and can be executed in a short period of time compared to experiments. Currently, there are various commercial CFD codes in the market and FLUENT software is one of them.

Modeling the movement of the gas phase was made by Eulerian perspective to solve the mass, momentum, and energy conservation equations for scalar variables. These differential equations were solved using the appropriate boundary conditions on the problem. The general forms of the steady state governing equations (continuity, momentum, energy and species) for the 2D turbulent reactive flow under cylindrical coordinates are expressed as follows:

$$\frac{\partial}{\partial x}(\rho u\phi)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v\phi)=\frac{\partial}{\partial x}\left(\Gamma\frac{\partial\phi}{\partial x}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(r\Gamma\frac{\partial\phi}{\partial r}\right)+S_{\phi} \quad (1)$$

where ϕ denotes for the dependent variables, Γ_{ϕ} is the transport coefficient of the variable ϕ and S_{ϕ} is the source term of the transport equation for ϕ [9]. If ϕ is taken as unity 1, axial velocity u, radial velocity v and enthalpy h; the continuity equation, the momentum equation in the axial direction x, the momentum equation in the radial direction r and the energy equation are obtained, respectively. ρ represents the gas density.

For modeling the movement of the liquid phase Lagrange perspective was used. Fuel was sprayed by the pump into combustion air delivered by a fan or compressor. Different velocity and droplet size were formed as a result of spraying. Due to different initial values, droplets tracked on different trajectories, heated and evaporated with different velocities, then mixed with air and burned. For the determination of droplet spectrum, Fluent offers dispersion models such as linear dispersion, and Rosin Rammler dispersion models [10]. The motion of the gas and liquid phases were taken into account with successive iterations.

It's necessary to define combustion model, turbulence model and radiation model for numerical modeling of the combustion. In this study, species transport-eddy dissipation model was used as combustion model, standard k-ε model was used as turbulence model, P1 approach model was used as radiation model. Liquid fuel was injected from the center of the burner. The conical turbulator which was inserted into smoke tubes was shown in Fig. 1. Calculations were made for four different situations with Geometry 1, which is the basic geometry without turbulator and Geometry 2-1, Geometry 2-2 and Geometry 2-3 as shown in Fig. 2. Injector spray pressures was adjusted to 8 bars. Group injection model was used as the injection type, the injection half angle was given as 30° and the injector diameter was given as 0.4 mm. Rosin-Rammler dispersion model was used for modeling the dispersion of droplets. $C_{10}H_{22}$ fuel was accepted as diesel fuel and the fuel viscosity was entered as 0.0024 kg/m s.

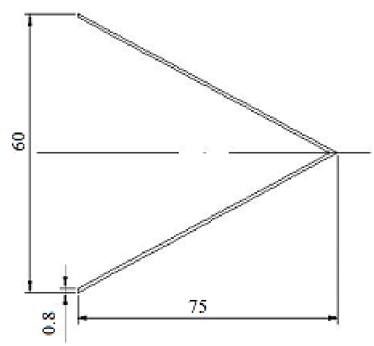


Fig. 1. Conical turbulator geometry

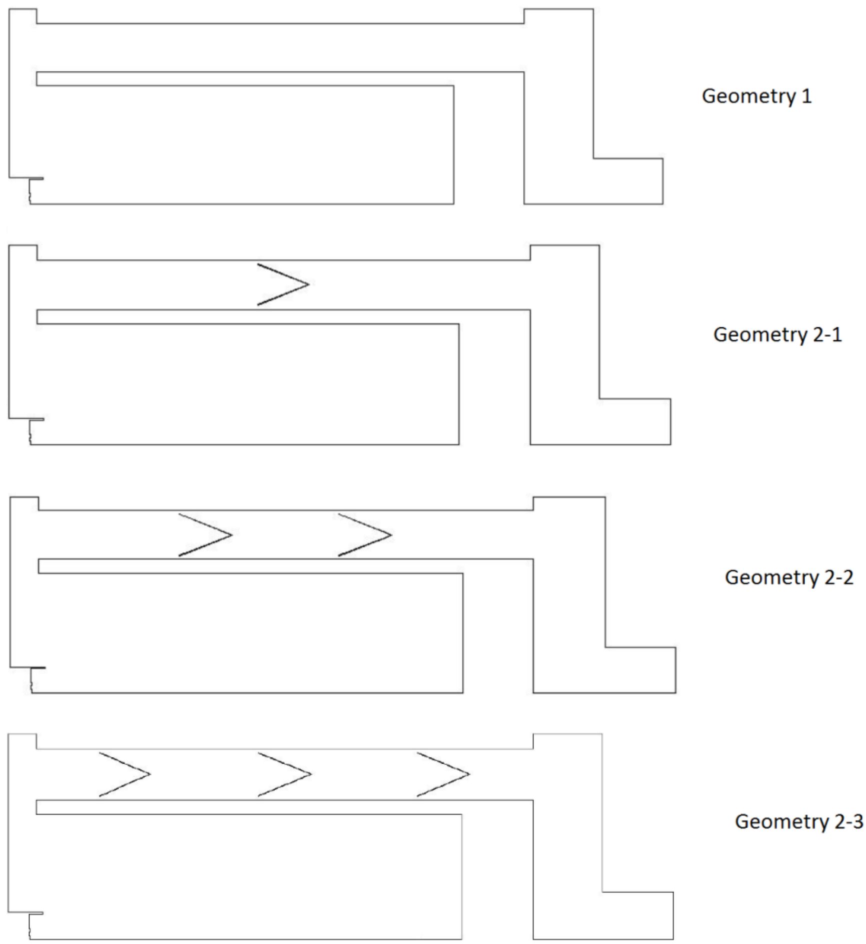


Fig. 2. Boilers with different number of turbulators

Fig. 3 shows the schematic diagram of the reverse flame fire tube boiler and the mesh structure of this boiler (Geometry 1) was given in Fig. 4. This mesh had a total of approximately 230,000 cells. Also, more grid structures were tested to see the grid's effect on the solution. Both fine grid and coarser grid were used for calculations. However, in these calculations with more grids, there were no meaningful changes in the results. The other geometries (Geometry 2-1, Geometry 2-2 and Geometry 2-3) had almost the same mesh sizes and all calculations were performed with the grid 230,000 meshes. All surfaces in contact with water were defined as the wall with temperature of 353 K. Exit zone was defined as the pressure outlet, and air inlet was defined as velocity inlet.

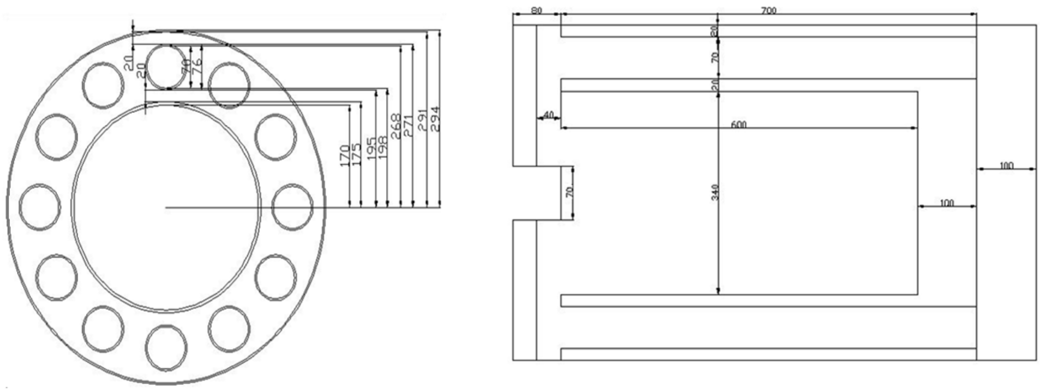


Fig. 3. Schematic diagram of the boiler

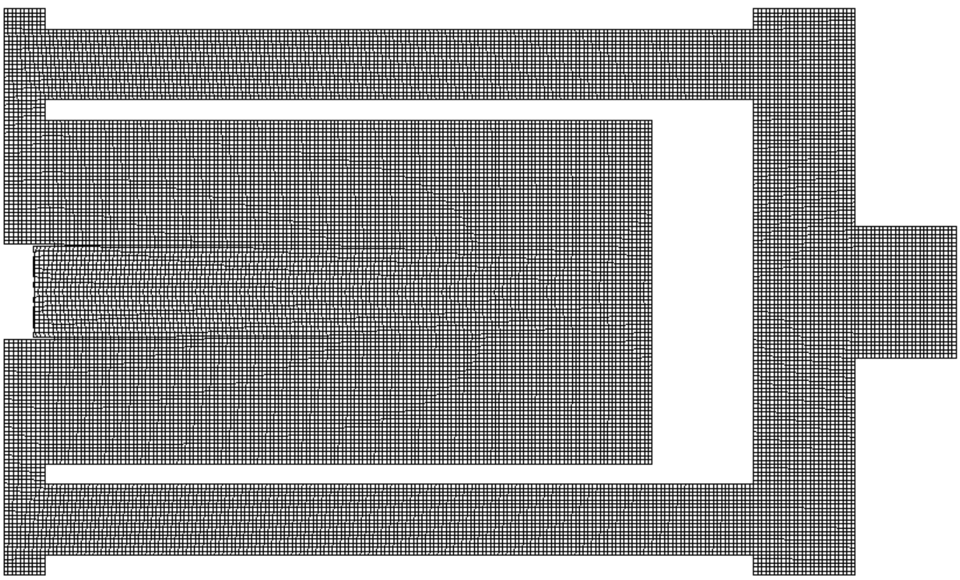


Fig. 4. The mesh structure of the solution domain

3. Results and Discussion

Fig. 5 represents the droplet trajectories. The maximum and minimum droplet diameters of the droplet spectrum according to the Rosin-Rammler model were 100 μm and 5 μm, respectively. As seen from this figure, the droplets dispersed in the gas flow according to injection conditions with a half angle of up to 30 degree. Their trajectories remained fairly straight and were not influenced much from the gas phase flow, mainly because of high impulses of droplets in relation to the gas phase. The smaller droplets warmed up, evaporated and burned very fast. The bigger droplets, on the contrary, needed more time for the same processes. Only the Geometry 1 was shown because the behavior of the droplets was similar at all solutions.

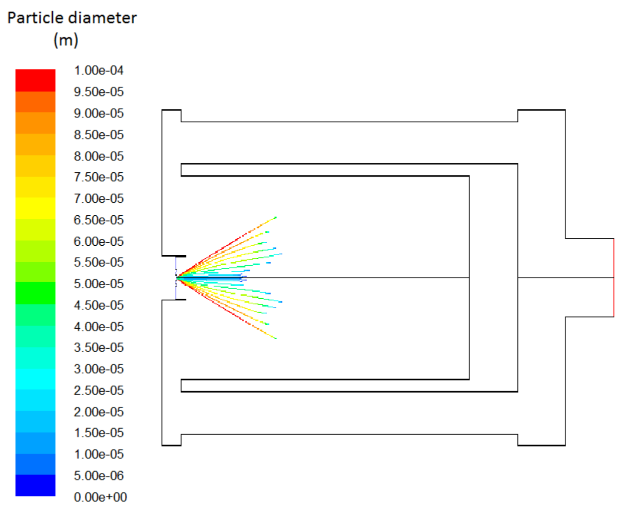


Fig. 5. Droplet trajectories in the boiler

After the injection of the droplets, the flame formed in the combustion chamber with a temperature increase in the surrounding of the burning droplets. Fig. 6 shows the temperature contours in the boiler for four different geometries. From this figure it can be seen that temperature decreased along the flame tube and smoke tube which were surrounded with water; therefore the heat of combustion was transferred to these regions. The maximum flame temperature was nearly 1650 K and occurred in the center of the combustion chamber. Increasing the number of turbulators in smoke tube decreased the temperatures along the smoke tube as shown in Fig. 6.

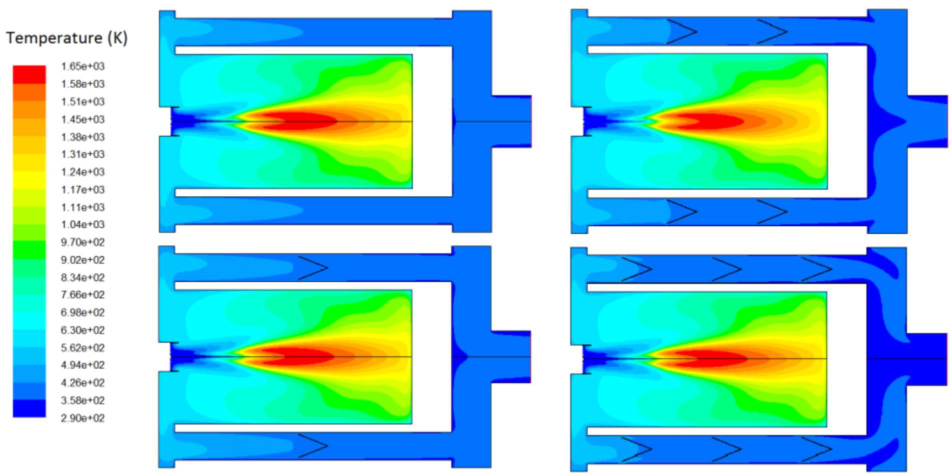


Fig. 6. The temperature contours in the boiler for different installation geometries

In Fig. 7 stream function contours were given. It can be seen from this figure that by using the turbulators, eddies occurred near the turbulators in the smoke tubes. Eddies caused the mixing and consequently enhanced the heat transfer.

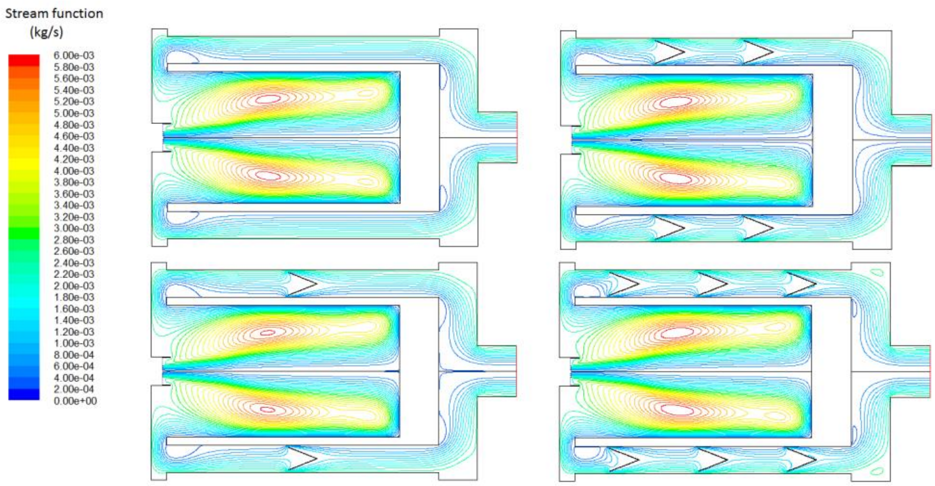


Fig. 7. Stream function contours in the boiler for different installation geometries

To see the eddies occurred in smoke tubes more clearly, the velocity vectors of the flow were displayed in Fig. 8, which was done only for the smoke tube of the Geometry 2-3. To have a closer look at the flow near the turbulators, velocity vectors were shown by zooming the smoke tube. From this figure, it can be seen that eddies formed in and around the turbulators. Between the turbulator and the smoke tube wall, the velocities first increased and then decreased forming eddies due to boundary layer separation of the flow from the turbulator wall.

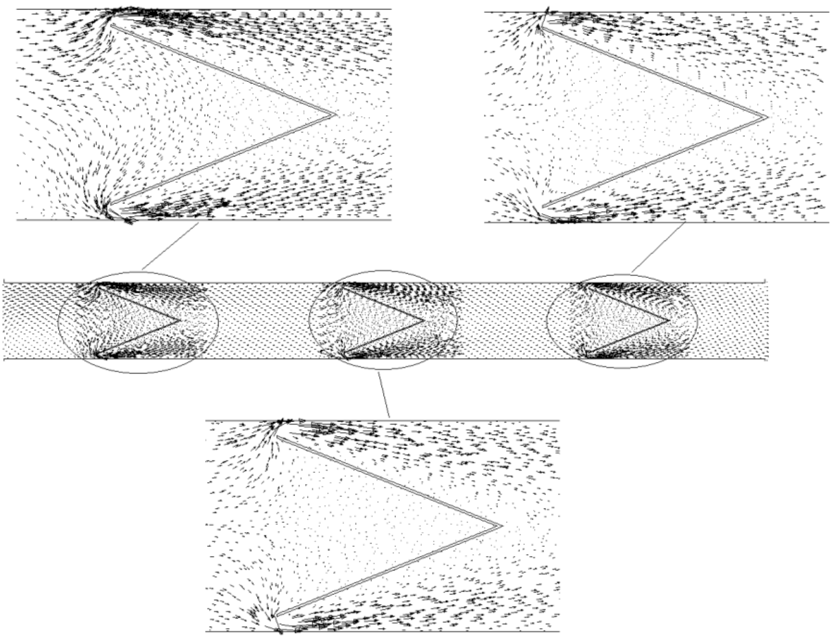


Fig. 8. Velocity vectors in Geometry 2-3

The efficiency of a boiler can be calculated with Eq. (2), if losses are neglected except sensible heat of flue gases [11]:

$$\eta = 1 - (1 + \lambda\, A_{sto})(T_{exh} - T_0)c_{p,exh} / H_U \quad (2)$$

In this equation λ is the excess air coefficient, A_{sto} is the stoichiometric air fuel ratio, $c_{p,exh}$ is the specific heat of exhaust gases, T_{exh} is the exhaust gas temperature, T_{amb} is the ambient air temperature and H_U is the lower heating value of the fuel.

From Eq. (2) it can be seen that to improve efficiency of the boiler, exhaust temperature should be decreased. Exhaust gas temperatures for four different installation geometries were given in Fig. 9. Exhaust gas temperatures decreased slightly with increasing number of turbulators in smoke tube. Geometry 1 had the maximum exhaust temperature which was 365 K. The temperature difference between Geometry 1 and Geometry 2-3 was almost 15 °C.

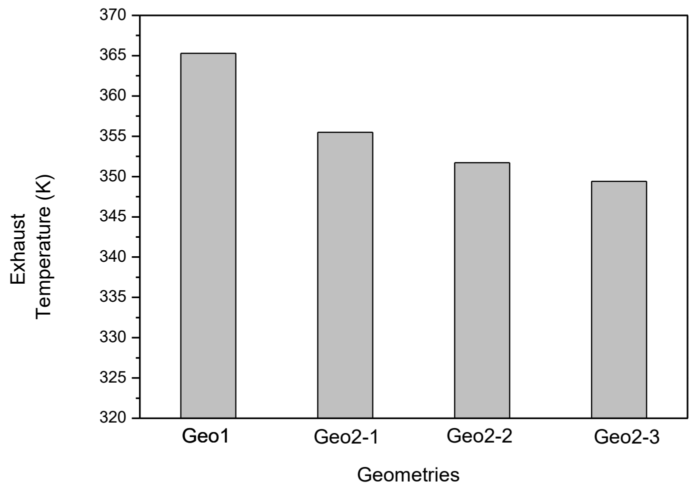


Fig. 9. Comparison of the exhaust temperatures for the different installation geometries

It is expected that turbulators inserted into smoke tubes will cause additional pressure losses. The comparison of the pressure losses was given in Fig. 10. As can be seen from this figure, increasing the number of turbulators increased the pressure losses. Geometry 1 had almost 20 Pa pressure losses and Geometry 2-3 had almost 23.5 Pa.

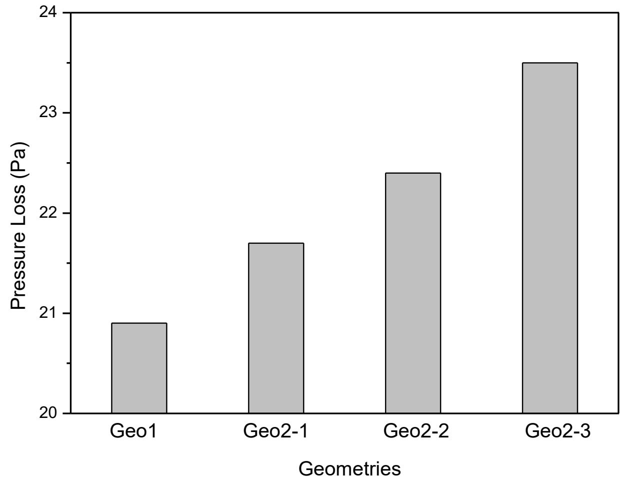


Fig. 10. Comparison of the pressure losses for the different installation geometries

4. Conclusions

In this study, a liquid fuelled flame tube boiler having conical turbulators was modelled at two-dimensional conditions. The basic geometry without turbulator (Geometry 1) and the geometries with one, two and three turbulators (Geometry 2-1, Geometry 2-2 and Geometry 2-3) were calculated and compared with each other. Increasing the number of turbulators from zero to three decreased the exhaust temperatures 15 K from 365 K to 350 K, and consequently the efficiency of the boiler was increased. Yet the turbulators inserted into smoke tube increased the pressure losses insignificantly from 20 to 23.5 Pa. Nevertheless, Geometry 2-3 had the best performance among all installation geometries.

References

[1] Muthusamy C, Vivar M, Skryabin I, and Srithar K. Effect of conical cut-out turbulators with internal fins in a circular tube on heat transfer and friction factor. International Communications in Heat and Mass Transfer, 2013; 44: 64-68. https://doi.org/10.1016/j.icheatmasstransfer.2013.03.004

[2] Yakut K and Sahin B. The Effects of Vortex Characteristics on Performance of Coiled Wire Turbulators Used for Heat Transfer Augmentation. Applied Thermal Engineering, 2004; 24: 2427–2438. https://doi.org/10.1016/j.applthermaleng.2004.03.008

[3] Kahraman N, Sekmen U, Çeper B, and Akansu SO. Boru içi akışlarda türbülatörlerin ısı transferine olan etkisinin sayısal incelenmesi. Isı Bilimi ve Tekniği Dergisi, 2008; 28: 51-59.

[4] Akansu SO. Heat transfer and pressure drops for porous-ring turbulators in a circular pipe. Applied Energy, 2006; 83: 280-298. https://doi.org/10.1016/j.apenergy.2005.02.003

[5] Karakaya H and Durmuş A. Heat transfer and exergy loss in conical spring turbulators. International Journal of Heat and Mass Transfer, 2013; 60:756-762. https://doi.org/10.1016/j.ijheatmasstransfer.2013.01.054

[6] Saario A, Rebola A, Coelho PJ, Costa M, and Oksanen A. Heavy fuel oil combustion in a cylindrical laboratory furnace: measurements and modelling. Fuel, 2005; 84: 359–369. https://doi.org/10.1016/j.fuel.2004.10.002

[7] Hu D. Comparison of numerical and experimental results of four liquid spray combustors, Master Thesis, Louisiana State University The Department of Mechanical Engineering, USA, 2002.

[8] Ing N.L., Jaafar M.N.M., Ishak M.S.A., Arizal M.A.A., 2010. Spray characteristic of palm biofuel blends, International Journal of Mechanical and Materials Engineering, 5, 214-221

[9] Ilbas M, Yılmaz İ, Veziroglu T.N., Kaplan Y., 2005. Hydrohen as burner fuel: modeling of hydrogen-hydrocarbon composite fuel combustion and NOx formation in a small burner. International Journal of Energy Research, 2005; 29: 973-990. https://doi.org/10.1002/er.1104

[10] FLUENT, Fluent User's Guide, Fluent Incorporated, Lebanon, NH. 2006.

[11] Sungur B, Topaloglu B, Ozcan H. Effects of nanoparticle additives to diesel on the combustion performance and emissions of a flame tube boiler. Energy, 2016; 113: 44-51. https://doi.org/10.1016/j.energy.2016.07.040