Int. J. Adv. Eng. Pure Sci. 2019, 1: 53-61
DOI: 10.7240/jeps.467682

RESEARCH ARTICLE / ARAŞTIRMA MAKALESİ

Numerical Analysis and Experimental Validation of a Mono-block Type Natural Gas Burner

Mono-Blok Tipte Bir Doğal Gaz Yakıcısının Sayısal Analizi ve Deneysel Doğrulanması

Barış YILMAZ[1], Engin UZA[2]

[1] Marmara University, Faculty of Engineering, Mechanical Eng. Dept.,34722,Kadıköy, Istanbul, Turkey

[2] ECOSTAR Combustion Systems, 59860 Çorlu, Tekirdag, Turkey

Abstract

Fossil fuels are still one the major sources of the energy production. Fuels, utilized in the power generation, especially in industrial scale applications, are almost hydrocarbons in gaseous, liquid or solid phases. In combustion of gaseous fossil fuels, besides the several other emissions carbon monoxide and nitric oxide which are identified as the main suspects of the environmental pollution are produced.

In this study, the flow field and the emission statistics of an industrial scale gas burner were investigated numerically. The burner and the assembled combustion chamber have been modelled using ANSYS-Fluent software. The standards for flue gas emissions of gas burners defined in TSE EN 676:2000 Class-3 have been taken into account. One of the most important emissions of NO_x level given in that standard, limited to be less than 80 mg/kWh, has been initially computed for the existing burner and then for the modified burner designs. Both the velocity and temperature fields are calculated and represented in terms of contour plots. The emission results are collected at the exit of the combustion chamber and the chamber wall temperatures have been also computed. The numerical model results were validated with the available experimental measurements obtained from on-site test facilities.

Keywords: Combustion, NO_x, CFD, Burner, Natural Gas

Öz

Fosil yakıtlar halen enerji üretiminin en önemli ana kaynaklarıdır. Enerji üretiminde, özellikle endüstriyel ölçekte uygulamalarda kullanılan yakıtlar büyük oranda gaz, sıvı veya katı fazlarda bulunan hidrokarbonlardır. Gaz fazındaki fosil yakıtların yanmasında, diğer bazı emisyonların yanı sıra çevre kirliliğinin ana şüpheli olarak nitelendirilen karbon monoksit ve nitrik oksit de üretilmektedir.

Bu çalışmada, endüstriyel ölçekte bir gaz yakıcısının akış alanı ve emisyon istatistikleri sayısal olarak incelenmiştir. Yakıcı ve buna bağlı yanma odası ANSYS-Fluent yazılımı kullanılarak modellenmiştir. Gaz yakıcılarının gaz emisyonları için tanımlanan TSE EN 676:2000 Class-3 standardı göz önünde bulundurulmuştur. Bu standartta yer alan en önemli salımlardan birisi olan NO_x seviyesi, ki 80 mg/kWh'den az olacak şekilde sınırlandırılmıştır, öncelikle varolan sonrasında ise iyileştirilen yakıcı tasarımları için hesaplanmıştır. Hız ve sıcaklık alan değerleri hesaplanmış ve kontur grafikleri olarak çizdirilmiştir. Salım sonuçları yanma odası çıkışında toplanmıştır ve ayrıca yanma odası duvar sıcaklıkları hesaplanmıştır. Sayısal model sonuçları firma bünyesinde var olan deneysel ölçme imkanları ile elde edilen verilerle doğrulanmıştır.

Anahtar Kelimeler: Yanma, NO_x, HAD, Yakıcı, Doğal Gas

I.INTRODUCTION

Fossil fuel usage remains still as the major method of energy utilization. Nowadays, most of the energy is still being generated by fossil fuel burning technologies both in household and industrial scale applications. The steady increase of energy demand and consumption results in the increase in the fossil fuels usage in, especially, energy production which causes unpleasant effects on the environment. Therefore, the utilization of energy more efficiently and the decrease in release of flue

Sorumlu yazar/Corresponding Author: Barış YILMAZ, Tel: +902163480292/1554, e-posta: byilmaz@marmara.edu.tr **Gönderilme/Submitted:** 05.10.2018, **Düzenleme/Revised:** 18.03.2019,, **Kabul/Accepted:** 18.03.2019.

gas emissions are the most important topics in combustion science and engineering [1-4].

In general, gaseous hydrocarbon fuels such as natural gas produce less emission compared to liquid and solid ones. Therefore, the use of gaseous fuels in power generation and household applications is promoted in many countries, in recent decades. These fuels are mainly burned in combustion chambers equipped with single or multiple burners[4].

In gas fuel combustion applications, combustion is classified in terms of state of reactants and oxidizers at the beginning. In premixed combustion, fuel and oxidizer are mixed before entering the combustion chamber. Therefore, it is also known as homogeneous combustion. On the other hand, they do not mix before the initiation of the combustion in non-premixed or diffusion (heterogeneous) combustion. There is also another mode, partially premixed; representing mixing of both types if there are some heterogeneous regions, rich in fuel or oxidizer, observed within the chamber. The combustion models vary for these modes due to involving different physical phenomena [4-6].

Several regulations are developed and deployed for the control of combustion products including CO, NO_x and SOx etc. in combustion based applications such as power generation and internal combustion engines. The NO_x limitations are varied depending on the type and the capacity of the application. The standard for automatic forced draught burners for gaseous fuels is defined by EN 676:2008[7]. The proposed emission classes in that standard are shown in Table 1 in line with the CO emission value of less than 100 mg/kWh.

Table 1. EN 676:2008 NO_x Limitations-Emission Class

CLASS	mg/kWh	mg/m³	ppm
1	170	169.59	82.56
2	120	119.71	58.28
3	80	79.1	38.85

The level of emissions can be reduced by increasing the efficiency of the combustion. The efficiency is related to several factors such as flow field generated inside the combustion chamber, mixing level of reactants, geometry etc. In order to design and optimize the geometry of both the burner and the combustion chamber, CFD analysis becomes a very effective tool in recent decades with the increasing capabilities of the computers. Using CFD shortens the design time, easy to modify the geometry and reduces product development and testing costs. The developed model should be verified experimentally due to numerical and computational uncertainties in modeling studies. On the other hand, the agreement between the experimental and the computational results relays on detailed knowledge of the utilized flow field and combustion models.

There are several studies in literature using CFD to examine the emission levels of burners having various capacities and the results are verified experimentally. Šarlej et al. [8] focused on CFD computation of an experimental low – NO_x burner with a two-stage natural gas burner for capacities between 0.5 MW and 1 MW. Several alternatives were introduced to minimize NO_x emissions. They utilized k-ω turbulence model and chemical reactions were modelled by two-step simple methane combustion reactions with Eddy-dissipation model (EDM) and radiative heat transfer was considered and solved with the discrete ordinates (DO) model.

Chacon et al. [9] studied on a new methodology for the design and optimization of a 2 MW low NO_x – CO mono-block natural gas burner that is located in a superheated steam boiler using ANSYS-Fluent software. Numerical model was validated with experimental prototype burner. Several air/fuel ratios have been studied with hexahedral meshes of 350.000 cells. Turbulent flow was modelled with the well-known k-ε turbulence model. Flamelet combustion model and P-1 radiation model were used in the numerical studies. A prototype burner has been developed based on the results of numerical modeling studies.

Spangelo [10] investigated the influence of modifications of 20 kW burner geometry on the NO_x emissions and fuel supply pressure on a swirl burner. The experimental, theoretical and numerical studies have been performed and the burner was solved with a 2D geometry assumption with k-ε turbulence model. Several combustion models including Eddy Dissipation Model, the Equilibrium PDF model and Flamelet PDF model were tried for combustion modeling. Flamelet PDF model was found to be most suitable model for combustion modelling. The NO_x emission values corrected for 3% O_2 reference conditions have been measured to be 25 and 45 ppmv for methane and propane as fuel respectively for patented burner concept. Internal gas recirculation, rapid air and fuel mixing cases were also tested for NO_x reduction. Optimized burner was scaled successfully to 200 kW and 370 kW by using constant velocity scaling criteria which is commonly used for industrial burners.

Saripalli [11] studied on the combustion and flow field statistics for industrial steam boiler application burning methane to identify how to increase the boiler efficiency and reduce the flue gas emissions. 3D computational geometry was modelled using ANSYS-Fluent. Turbulent flow was modelled by using k-ε turbulence model. Constant wall temperature was defined for the outer shells of the boiler. In addition, radiation was modelled by using the Rosseland radiation model. A single step reaction with finite-rate chemistry model has been implemented for methane combustion. The simulations were performed for 19 industrial steam boilers.

Khanafer et al. [12] investigated effect of swirl velocity and burner wall temperature on the NO_x formation for an industrial swirl burner. It was found that swirl enhanced the mixing of air-fuel streams and was leading to reduce flue gas emissions. It was concluded that the level of emissions such as CO and unburned hydrocarbons can be reduced about 3-5 times depending on swirl velocity. In addition, it was observed that the concentration of NO_x at the outlet did not decrease with swirl.

Cellek and Pınarbaşı [13] have experimentally and numerically modelled the combustion in a natural gas industrial scale low swirl burner. The effect of the hydrogen addition to natural gas by varying H_2 content from 25% to 100% on the combustion performance and the released emissions have been examined. It is concluded that using hydrogen-enriched natural gas instead of natural gas in the burner has decreased CO and CO_2 emissions remarkably, as expected. However, NO_x emissions have significantly increased due to high temperature conditions.

Funke et al.[14] have worked experimentally and numerically on the optimization of a low NO_x combustion chamber of the Micromix gas turbine. The developed method was aimed to be applied to industrial scale gas turbine chambers. The NO_x reduction potential for operation with hydrogen fuel has been studied by modifiying the combustion chamber. It is reported that 99% combustion efficiency has been reached and the NO_x emissions levels lower than 10% have been achieved under full-scale gas turbine conditions.

Enagi et al.[15] studied on design and development of a micro gas turbine combustion chamber numerically using ANSYS-Fluent 16.1 software. Various combustion models and parameters have been tested to obtain an optimum configuration. An experimental chamber has been also produced for validation. It has been reported that the low CO emissions of less than 100 ppm have been achieved by the optimized chamber geometry.

As summarized in previous paragraphs, there are still ongoing researches on reduction in the levels of emissions, especially NO_x and CO, by improving the combustion efficiency of the burners. CFD is utilized as a common method to test how the modifications on the geometry affect the combustion efficiency and the emissions. In addition, the parametric studies such as varying operation conditions, equivalence ratios etc. can be examined more comprehensively.

In the present study, the numerical analysis of an industrial scale natural gas burner has been performed in order to examine the outlet temperature and emission levels which are also measured experimentally. Both the flow field and flame statistics have been investigated. Moreover, using the validated numerical model, an optimized design is proposed to reduce NO_x levels.

I.METHODOLOGY

The flow and temperature fields of the non-premixed gas burner were investigated both numerically and experimentally.

1.1.Numerical Model

A modulating natural gas burner has been modelled numerically using CFD techniques. Ansys-Fluent software, a well-known commercial program, is utilized for simulations. Initially, the geometry of both the burner and the combustion chamber has been developed then it was meshed using the structured type cells. The modelling studies have started with the cold flow field analysis using k-ε turbulence model. Due to the complexity of the geometry, the flow is turbulent as in most of industrial applications. Then the reactive case studies have been performed to investigate the in-chamber temperature distribution and emission levels at the exit of the combustion chamber. The flamelet combustion model has been utilized in reactive flow analysis. The modeling results were validated by the experimental measurements obtained from the set up mentioned below. Moreover, a new burner design was proposed and modelled to achieve lower levels of NO_x emissions.

Burners have complicated components to obtain desired mixing of fuel and oxidizer and the flame shape and position. In order to model burner geometry, simplifications are needed to obtain converged solutions without affecting the flow field developed in the combustion chamber. Simplified burner and combustion chamber geometry are shown in Figure 1.

Simplifications made during modeling studies are as follows;

- Ignition electrode and ionization electrode were not modelled,
- Burner body and other equipment were not used in modelled geometry,
- Diffuser wings were modified,
- Fixing plates of the flame tube extension were not modelled.

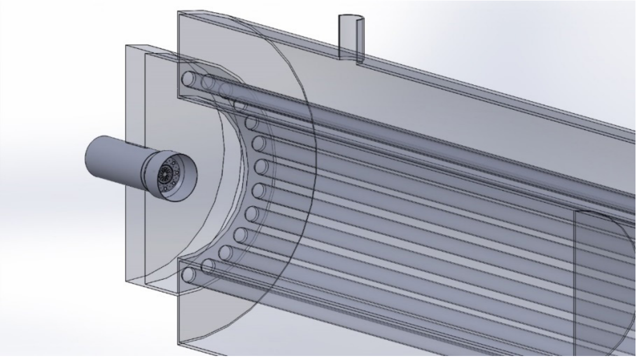


Figure 1. Simplified burner and combustion chamber geometry

Mesh independency studies were performed with four different mesh geometries. Details of the burner head were directly affected the mesh quality and total mesh number due to complexity in this part. During the mesh independency studies, element sizes in the burner head region and the adjacent combustion chamber parts where the combustion and high gradients occurred were varied. Details of the studied meshes are listed in Table 2.

Table 2. Mesh element number and mesh quality of the geometry

Mesh Name	Element Number	Skewness (Maksimum)	Orhogonal Quality (Minimum)
Mesh-1	5961246	0.84834	0.22609
Mesh-2	5522284	0.87171	0.21169
Mesh-3	4789363	0.89912	0.18363
Mesh-4	4399183	0.93495	0.08915

Initially, cold flow has been computed for the four-different mesh option given in Table 2. The comparison of the mean axial velocity in axial direction with four mesh cases is shown in Figure 2. The mean axial velocity profiles were found almost same when all flow domain is considered. However, some variations have been observed in the region near the exit of the burner corresponding to velocity core region. The mesh size and number coded as Mesh-3 is decided to be accurate enough and used in further studies. A detailed view of Mesh-3 is shown in Figure 3 where structured meshes were used in the combustion chamber and tetrahedral meshes were used in the region where the geometry is complex.

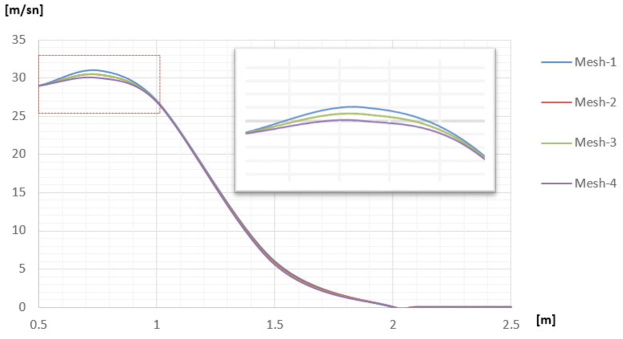


Figure 2. Axial velocity distribution in axial direction. [m]

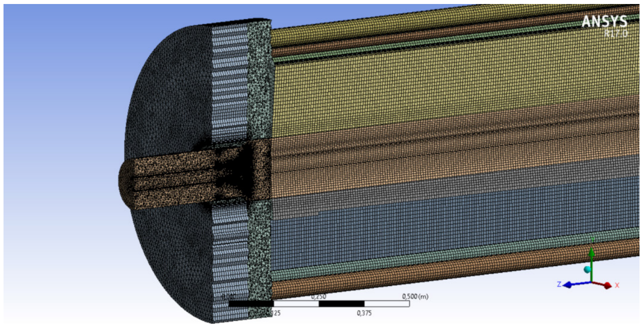


Figure 3. Detailed view of chosen mesh case, Mesh-3

For the boundary conditions, mass flow inlet condition was applied for both fuel inlet and air inlet. In addition, constant wall temperature was defined at the surface of the combustion chamber since heat is rejected by water flowing around the chamber. The pressure outlet was defined as the outlet boundary condition.

2.2. Experimental Setup

In this study, a commercial burner is considered. It is a, monoblock modulating, natural gas/LPG and non-premixed type since air and gas flow separately in the burner.

Figure 4. General view of the experimental setup

Experimental measurements were conducted to determine the emissions and the temperature at the outlet of the combustion chamber by varying several parameters. The airflow ratio can be adjusted to examine the effect of the equivalence/excess air ratio. General view of the experimental setup and test burner is given in Figure 4.

The technical specifications of the experimental model are shown in Figure 5. The combustion chamber is cooled down by using a water jacket around it. The inlet and the outlet temperature of the water were monitored to determine amount of heat rejection through the chamber walls.

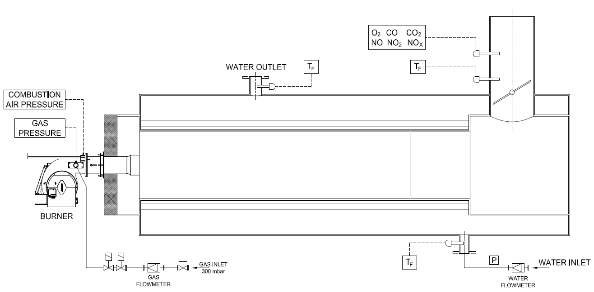


Figure 5. Test setup and measured operating parameters of the test setup

The natural gas burner is connected to the combustion chamber test set up equipped with several measurement devices. The following operating parameters are being measured during the tests in line with PLC controlled test units;

- Fuel flow rate, fuel supply temperature and pressure,
- Cooling water flow rate, cooling water inlet and outlet temperature,
- Flue gas emissions (CO, CO_2, O_2, NO_x, SO_x)
- Flue gas temperature,
- Boiler back pressure,
- Fuel and combustion air pressure,

Experimental test conditions are listed in Table 3. Measured experimental values were used for comparison with numerical studies.

Table 3. Test conditions for the experimental studies

Air/Gas Side			Water Side		
Air Temperature	10	**°C**	**Water Inlet Temperature**	30	**°C**
Gas Temperature	13	**°C**	**Water Outlet Temperature**	50	**°C**
Gas Inlet Pressure	300	**mbar**	**Water Flow Rate**	16.5	**m³/h**

General specifications of the burner and its modified versions based on the capacity are given in Table 4.

Table 4. Burner Technical Specifications

Fan Motor Power [kW]	1.5	**Natural Gas Consumption High Load [Nm³/h]**	125.1
Fuel Type	Natural Gas-LPG	**Ignition**	Direct Ignition
Operating Type	Modulating	**Flame Control**	Ionization

II.RESULTS

1.2.Determination of Theoretical Amount of Air

The modeling of the burner has been performed after determination of the required flow rates operating at two different air to fuel ratio conditions.

The parameter Lambda (λ) is defined to be the ratio of the actual air-fuel ratio to the stoichiometric air-fuel ratio. $\lambda>1$ corresponds to lean mixture in fuel and $\lambda<1$ refers to rich mixture in fuel. Increasing the excess air ratio decreases both the flame and combustion chamber temperatures. In order to establish the same load for excess air conditions, more fuel is needed to be burned. Lambda is also defined as the reciprocal of the equivalence ratio as given in following equation;

$$\lambda = \frac{(Air/Fuel)_{actual}}{(Air/Fuel)_{stoichiometric}} = \frac{1}{\phi} \quad (1)$$

Theoretical air-fuel and combustion product concentrations were calculated for methane at two air to fuel ratio conditions. They are also obtained for natural gas at the same operating conditions. Densities for both fuel types were calculated by using an online calculator [16,17]. Reference temperature and pressure were chosen to be 15°C 1.01325 bar, respectively. The density of natural gas at the same operating conditions and the required amount of air for these conditions are calculated similarly. The results of the calculations for both fuel types are given in Table 5.

Table 5. Density calculation for Methane and Natural Gas

		Methane	Nat. Gas
Temperature	°C	15	15
Pressure	MPa	0.101325	0.101325
Mass density	kg/m³	0.68	0.711
Compression factor	-	0.998	0.9979
Molar mass	kg/kmole	16.043	17.775
Molar density	kmole/m³	0.04238	0.0424

Natural gas composition that was used in calculations are shown in Table 6.

Table 6. Natural gas composition utilized in calculations

Methane	%	96.064
Ethane	%	2.133
Propane	%	0.748
Butane	%	0.233
Nitrogen	%	0.659
Carbon dioxide	%	0.163

The required amount of methane and air has been theoretically calculated for λ=1 and λ=1.2 conditions. Total air demand was found that for 1 m³/h methane, 9.52 m³/h of air was needed. On the other hand, for the latter condition, the required amount of air was 11.424 m³/h. Similarly, for the natural gas fuel case, the required amount of air for λ=1 and λ=1.2 conditions were also calculated which are 9.52 m³/h and 11.7 m³/h, respectively.

3.2. Reactive Flow Studies

Influence of several parameters on combustion characteristics in the chamber were examined both numerically and experimentally. Reactive flow analyses were performed for different excess air ratio values at high load conditions. It is accomplished by varying the air flow rate while the amount of natural gas fuel is kept constant. The numerical computations were performed for both the natural gas and 100% methane fuel conditions for 125.1 Nm³/h gas consumption flow rate. Similarly, the experiments were conducted for the same operating conditions of natural gas. The comparison of numerical and experimental results for the first reactive case study with natural gas is shown in Table 7, Figure 6 and Figure 7.

As seen in Table 7, O_2 and CO_2 emissions are predicted in a good agreement with the experiments. On the other hand, the difference is found slightly higher, however, within an acceptable range of less than 10% for NO_x emissions.

Table 7. The comparison of the first reactive case studies with experiments

Case No/Type		Fuel Type	Gas Consumption (Nm³/h)	λ (-)	O_2 (%)	CO_2 (%)	NO_x (ppm)
1	**Numerical**	Natural Gas	125.1	1.1	1.61	10.7	61.2
	Exp.	Natural Gas	125.1	1.1	2	10.41	66
2	**Numerical**	Natural Gas	125.1	1.2	3.3	9.6	61.6
	Exp.	Natural Gas	125.1	1.2	3.5	9.76	63
3	**Numerical**	Natural Gas	125.1	1.3	4.26	9.03	53.32
	Exp.	Natural Gas	125.1	1.3	4.9	8.77	57
4	**Numerical**	Natural Gas	125.1	1.4	5.61	8.11	55.53
	Exp.	Natural Gas	125.1	1.4	6	8.18	51

In addition, the comparison of numerical and experimental studies at the same λ value of 1.2, corresponding to the second reactive case studies of natural gas and methane, is shown in Table 8. The results of numerical studies with natural gas are observed better than those of with the methane case when both are compared to the experimental data.

Table 8. The comparison of the emissions for the second reactive case numerical studies with experiments.

Case No /Type		Fuel Type	Gas Consumption (Nm³/h)	λ (-)	O_2 (%)	CO_2 (%)	NO_x (ppm)
2	**Numerical**	Natural Gas	125.1	1.2	3.3	9.6	61.6
5	**Numerical**	Methane	125.1	1.2	3.25	9.56	55.52
4	**Experimental**	Natural Gas	125.1	1.2	3.5	9.76	63

The influence of the load is investigated by keeping the excess air value constant, for λ=1.2, in the third reactive analysis with natural gas. The numerical and experimental results are compared in Table 9.

Table 9. The comparison of the third reactive case results with experiments for natural gas.

Case No		Gas Consumption	λ	O_2	CO_2	NO_x
		Nm³/h	-	%	%	ppm
6	**Numerical**	50	1.2	3.5	9.48	46.54
	Experimental	50	1.2	3.5	9.44	57
7	**Numerical**	75	1.2	3.5	9.83	56.86
	Experimental	75	1.2	3.4	9.72	65
8	**Numerical**	100	1.2	3.5	9.5	49.7
	Experimental	100	1.2	3.5	9.55	61
2	**Numerical**	125.1	1.2	3.3	9.6	61.6
	Experimental	125.1	1.2	3.5	9.76	63

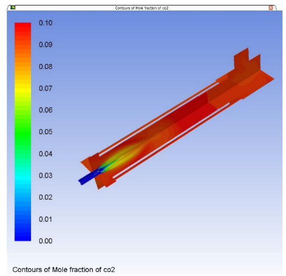
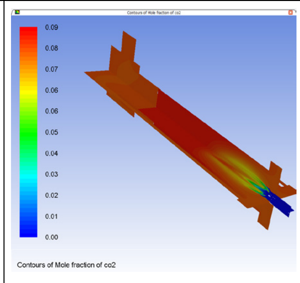
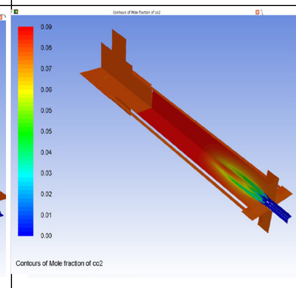
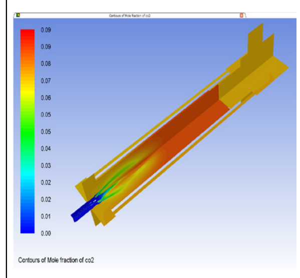
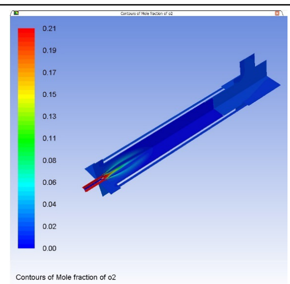
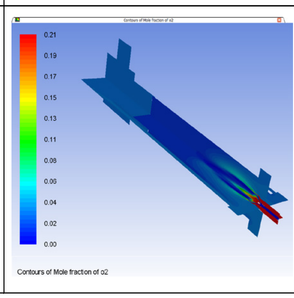
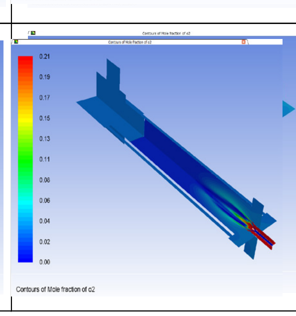
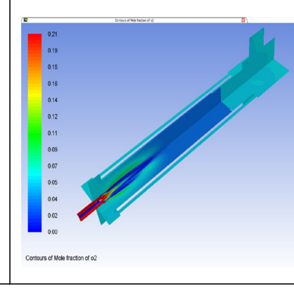
Lambda (λ)	λ=1.1	λ=1.2	λ=1.3	λ=1.4
Fuel Consumption [Nm³/h]	125.1	125.1	125.1	125.1
Contours of Mole Fraction of CO2 [%]				
Contours of Mole Fraction of O2 [%]				

Figure 6. Contour plots of CO2 and O2 mole fraction distributions with varying excess air ratio.

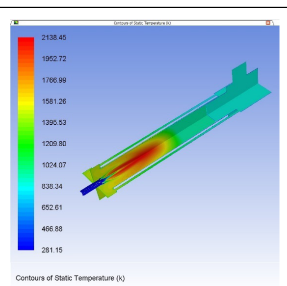
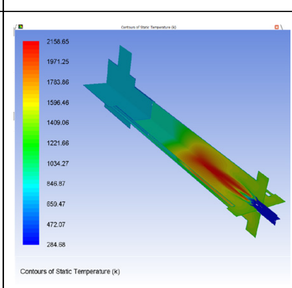
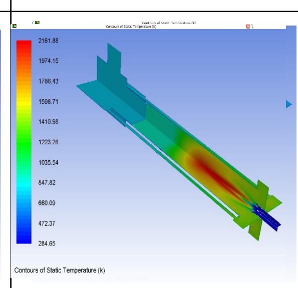
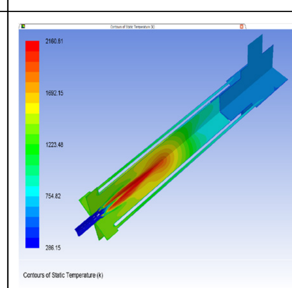
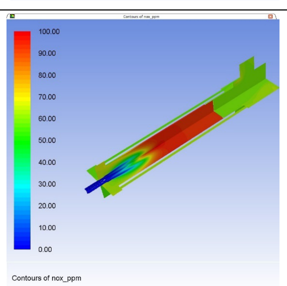
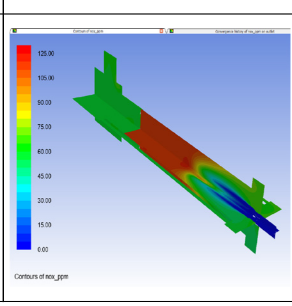
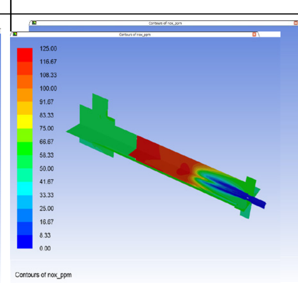
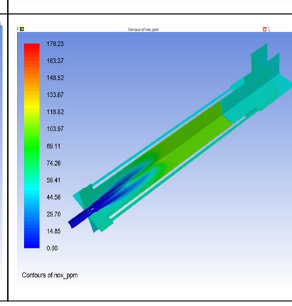
Lambda (λ)	λ=1.1	λ=1.2	λ=1.3	λ=1.4
Fuel Consumption [Nm³/h]	125.1	125.1	125.1	125.1
Contours of Static Temperature [K]				
Contours of NOx [ppm]				

Figure 7. Contour plots of Static Temperature and NOx mole fraction distributions with varying excess air ratio.

Differences between the experiments and the numerical studies are observed to be 5%, 2%, 8% for O_2, CO_2 and NO_x emissions, respectively. As in the previous case, O_2 and CO_2 emission results are found similar to the experiments, less than 2%. Therefore, these emissions are predicted with a good agreement and the results are quite satisfactory. On the other hand, NO_x levels are computed slightly different than the experiments. Except for the case 2, for all other tabulated cases the difference is observed more than 10%. It is observed that NO_x emissions are predicted relatively more accurate at high load conditions.

Based on the results obtained from case studies, two new burner geometries were developed with modifications on some of the burner components. Then, the developed geometries were numerically modelled for the sake of decreasing NO_x emissions. New Geometry-1 (NG-1) was modelled to see how flame tube extension affect the flame shape and the modifications were done on the geometry for gas nozzles in which the nozzle shape is modified with radius form to decrease pressure lost.

With modified gas nozzles, pressure lost inside the combustion head was improved but without flame tube extension the resultant flame shape became too spherical and impinged to the combustion chamber wall. This may cause local temperature increase so that increase in NO_x emissions and damage on the chamber wall. Therefore, modifications needed to be applied to obtain a longer flame. Thus, another model , coded as New Geometry-2 (NG-2), has been developed based on NG-1 model. Modified gas nozzle in NG-1 was utilized and gas outlets were added in the center of the combustion head of the burner. A conical part was also added to the flame tube to improve the flame shape.

Mesh details of NG-2 is shown in Figure 8. Geometry has 2070285 elements with the maximum skewness of 0.87.

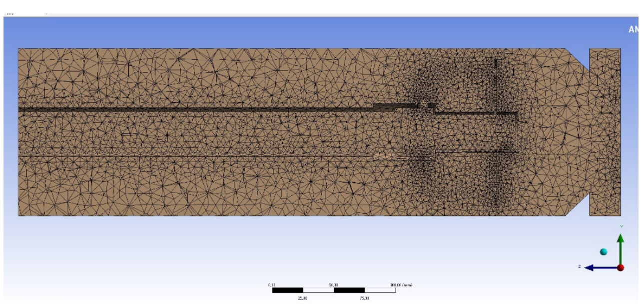


Figure 9. Mesh details of the NG-2 burner part

The reactive case simulations were performed with NG-2 for the natural gas consumption of 125.1 Nm³/h at λ=1.2 air to fuel ratio. The modelling results are shown in Table 10. It is found that with the modified geometry, coded as Case-9, nearly 25 ppm decrease in NO_x emissions for the same load conditions was observed compared to Case-2 in numerical studies.

Table 10. Reactive analysis results for the natural for the NG-2 in λ=1.2.

Case No		Fuel Type	Gas Consumption	λ	O_2	CO_2	NO_x
			Nm³/h	-	%	%	ppm
9	**Numerical**	Natural Gas	125.1	1.2	3.58	9.45	34.93

In addition, the flame position was found longer compared to the original geometry in length. Temperature contours of the reactive analysis result are presented in Figure 9. It is seen that the temperature levels are lower than the other cases verifying less level of NO_x emissions.

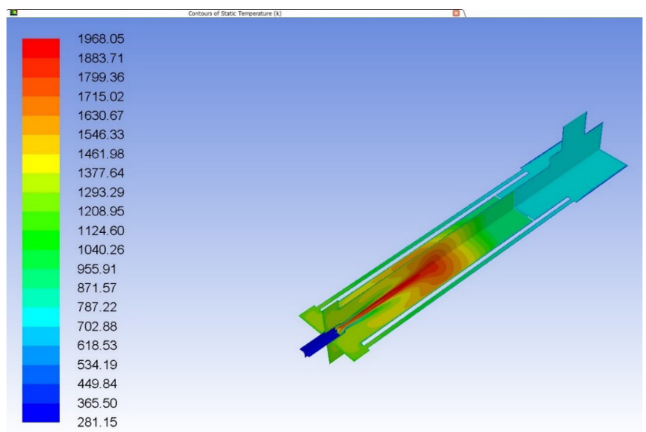


Figure 9. Temperature [K] contour plot of the NG-2

III.CONCLUSIONS

An industrial scale burner and its modified geometries have been studied numerically. Experimental measurements are also conducted for the same burners. The numerical studies have been performed initially for the mesh independency. The accurate enough mesh number and size have been determined by comparing four mesh cases. Then reactive case studies have been performed with chosen mesh.

In the first reactive case analysis, the influence of the air to fuel ratio on the product emissions has been investigated with natural gas. It is observed that CO_2 and O_2 emissions are found in a good agreement with experimental data. In addition, the simulations were performed with 100% methane gas as well. It is observed that temperature distribution and CO_2 , O_2 emission values are almost similar when 100% methane and natural gas results are compared. However, calculated NO_x values with natural gas are found closer to the experimental data. In other cases, the simulations were performed with natural gas.

In the third reactive case analysis and experiments gas consumption ratios were varied to examine the effect of load on the emissions. From comparison of experiments and reactive studies it is observed that increasing the load increases slightly the NO_x emissions in experiments which is expected due to the increase in the temperature. Similar trend has been also observed in simulations however the difference between the measurements and the simulations is observed higher at lower load conditions.

Based on the results obtained from the burner a modified version has been developed. With the modified burner geometry pressure drop of combustion head was decreased but flame shape was not suitable for a burner application since the flame impinges on the combustion chamber wall and too spherical. With the improvement on the burner geometry, NO_x values were obtained less than the original geometry and the flame is found longer than the first modified geometry. In addition temperature levels are observed lower compared to the other cases.

REFERENCES

[1] Charles, E.B., (2013). The John Zink Hamworthy Combustion Handbook Fundamentals Vol.1, 2nd Edition, CRC Press, Oklahoma.

[2] McAllister, S., Chen, J., Fernandez-Pello, A.C., (2011). Fundamentals of Combustion Proceses, 1st Ed., Springer, New York.

[3] Strehlow R.A., (1993). Combustion Fundamentals, 1st Ed., Krieger Pub Co.

[4] Basu, P., Kefa C., Jestin, L. (2000). Boilers and Burners Design and Theory, 1st Ed., Springer, New York.

[5] Poinsot, T., Veynante, D., (2001). Theoretical and Numerical Combustion, 1st Ed., Edwards, Philadelphia

[6] Ansys Inc. (2016). Fluent Combustion Training Notes, 1st Edition, Ansys Inc., Canonsburg

[7] Standart EN 67 (2008) Automatic Forced Draught Burners for Gaseous Fuels.

[8] Sarlej, M., Petr, P., Hajek, J., Stehlik, P. (2007). Computational Support in Experimental Burner Design Optimisation. *Applied Thermal Engineering* 27, 2727–2731.

[9] Chacon J., Sala J.M., Blanco J.M., (2007). Investigation on the Design and Optimization of a Low NO_x-CO Emission Burner Both Experimentally and through Computational Fluid Dynamics (CFD) Simulations, *Energy&Fuels*, 21, 42-58.

[10] Spangelo, O., (2004). Experimental and Theoretical Studies of a Low NO_x Swirl Burner, PhD Thesis, The Norwegian University of Science and Technology, Norway.

[11] Saripalli, R., (2004). Simulation of Combustion and Thermal Flow Inside an Industrial Boiler, PhD Thesis, University of New Orleans, USA.

[12] Khanafer, K., Aithal, S.M., (2011). Fluid-dynamic and NO_x Computation in Swirl Burners, International *Journal of Heat and Mass Transfer,* 54, 5030-5038.

[13] Cellek, M.S. and Pınarbaşı, A., (2018). Investigations on performance and emission characteristics of an industrial low swirl burner while burning natural gas, methane, hydrogen-enriched natural gas and hydrogen as fuels, *Int. J. of Hydrogen Energy,* 43(2), 1194-1207.

[14] Funke, H.H.W., Beckmann, N., Abanteriba, S.,(2019). An overview on dry low NO_x micromix combustor development for hydrogen-rich gas turbine applications, *Int. J. of Hydrogen Energy,* 44(13) 6978-6990.

[15] Enagi, I. I., Al-attab, K.A., Zainal, Z.A.,(2017). Combustion chamber design and performance for micro gas turbine application, *Fuel Processing Technology,* 166, 258-268.

[16] Natural Gas Density Calculator, http://www.unitrove.com/engineering/tools/gas/natural-gas-density. Erişim tarihi: Mayıs 2016.

[17] Natural Gas Calorific Value Calculator, http://www.unitrove.com/engineering/tools/gas/ natural-gas-calorific-value. Erişim tarihi: Mayıs 2016.