Proceedings of the 4th World Congress on Momentum, Heat and Mass Transfer (MHMT'19)
Rome, Italy – April, 2019
Paper No. ENFHT 109
DOI: 10.11159/enfht19.109

Feasibility of Using CFD Analysis for Improving the Gas Hobs Performance In terms of Efficiency and Emissions

Ehsan Amirabedin[1], Tuba Anık[1], Ali Karaduman[1]
[1]Candy Hoover, Organize Sanayi Bölgesi 8. Cadde No:1,
26110, Odunpazarı, Eskişehir/Turkey
eamirabedin@hoover.com.tr; tanik@hoover.com.tr; akaraduman@hoover.com.tr

Abstract **-** This paper presents the feasibility of using Computational Fluid Dynamic (CFD) simulation for gas hobs design with the aim of improving the efficiency and emissions (CO and CO_2) performances. CFD analysis of a hob including the combustion simulation can take notably long time considering that it shall be solved time dependent (transient). In this regard, a shortcut method based on comparison between different design/modifications of the model is developed to reduce the solving time. Combustion simulation was fulfilled for a quarter of a Candy Hoover Groups' hob with a semi rapid burner under the same laboratory conditions using FloEFD with Advanced Module for Combustion. For evaluation, distance changing between burner cap and pot bottom was studied. By increasing the distance, decreasing of the CO emission and the energy efficiency and increasing of the CO_2 emission were observed. These results of CFD analysis are in excellent agreement with experimental results from laboratory tests. Therefore, this study reveals that CFD analysis can be reliably used in the gas hob design and optimization.

Keywords**:** CFD analysis, Gas Hob, Gas Combustion, Feasibility, FloEFD, CO emission, CO_2 emission.

1. Introduction

Gas hobs/cookers are used in almost all households for cooking. Improvements in energy efficiency and emissions performances of the gas hobs, on the one hand, will have a significant impact on customer satisfaction, on the other hand, improving the combustion performance of the gas hobs will positively affect the climate change considering the fact that the inefficient combustion process produces high pollution emissions and greenhouse gases. Performance improvement of the hobs can be achieved by several conceptional design modifications which beside the industrial application difficulties, they must be validated by long-lasting experimental tests.

One alternative tool for the evaluation of the applied modifications, from the thermal-flow patterns and combustion performance point of view, is Computational Fluid Dynamic (CFD) simulation. Simulation can provide faster results with minimum setup for experiments comparing the real laboratory experiments [1] and [2].

CFD simulation uses numerical methods for solving equations of a system involving fluid flow, heat transfer and other associated phenomena (i.e. chemical reactions) with the discretization techniques such as the finite element/volume methods [3] and [4]. By using CFD, one can precisely obtain nearly all characteristics of a flow including temperature, pressure, heat transfer coefficient throughout the solution domain. During the simulation, boundary conditions or the design geometry can be easily changed to observe their effects on the system and consequently, to optimize the system performance. In addition, CFD is an essential engineering design and optimization tool which can significantly decrease the amount of experimentation necessary to solve problems and provide detailed parametric studies in order to reduce the development period [5].

Accordingly, CFD is increasingly being used by many researchers and engineers, in academia and industry, for the combustion simulation [6], [7], [8] and [9]. Particularly, for the gas hobs, some researchers [10] and [11] have used the CFD analysis for investigating the relationship between the flame characteristics of the burner and the generated power. Despite all these studies, there hasn't been very much related studies focusing on utilizing CFD on design and development of the gas hobs or cookers.

The major aim of this paper is to study the feasibility of using the state of the art of CFD simulation in design and optimization of domestic gas hobs. To accomplish this goal, three-dimensional geometry of one quarter of a Candy Hoover Group's Hob was modelled, then one pre-defined modification with certain effects on the system were implemented and following this, several simulations were performed using FloEFD CFD simulation software. Thereafter, a comparison between the obtained numerical results (before and after modification) and the results from experiments were made and the similarities and differences between them highlighted and discussed. However, it must be noted that the exactly calculation of the value of the system parameters have not been intended in this study. Moreover, a strategy to reduce the solving time of the transient combustion processes is investigated as the second aim of this study.

The results obtained in this work would help us to understand how efficiently we can implement the 3-D CFD simulation at design and development of domestic gas hobs.

2. Methodology

In order to use of the CFD simulation in the development of the gas hobs and to predict the performance behaviour of the hobs during the customer utilizations, the simulation results shall be verified by the experimental findings. In this regard, methodology of this study involves both experimental and 3-D CFD simulation analyses. The study was initiated by creating the 3-D model and applying the CFD simulation of the gas hob based on the validated experimental test results. Prior to the CFD analysis, efficiency and emissions evaluation parameters were conducted in the R&D test laboratory according to the regulation EN 30-2-1:2015 [12].

2.1. Experimental

2.1.1. System description

Figure 1-b displays the investigated domestic gas hob which is one of the most popular models for household. Some major technical specifications of the gas hob adopted in the present study are as below:

- Automatic, electric ignition of 220/240 V
- Total weight of 10.80 kg
- External dimensions of 145 (mm) × 640 (mm) × 585 (mm)
- 1.75 kW of heating capacity and 0,167 m^3/h of volumetric flow rate for Semi Rapid Burner.

In order to shorten the analysing time, particularly during the CFD analysis, only one fourth of the hob containing a Semi Rapid Burner (SRB) head with natural aspiration system was investigated (figure 1-a).

In this type of burner, the primary air is drawn naturally into the bottom of the burner by pressure difference between the high velocity gas jet and the ambient air. The gas jet is sprayed with a nozzle of diameter of 1,5 mm and then the air-NG mixture flows out through the holes around the burner cap. There are 20 holes in the form of rectangular slots and located on the outer ring of the burner head of 65.5 mm diameter [13], [14], [15] and [11].

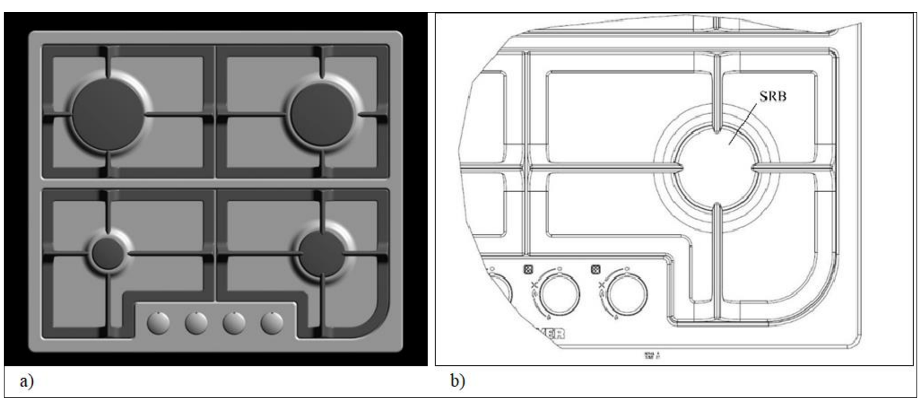

Fig. 1: a) Investigated Candy Hoover Group's gas hob, b) Schematic of one fourth of the hob containing a Semi Rapid Burner (SRB) head.

2.1.2. Laboratory and test standard conditions

For maximum data acquisition as specified by mentioned regulation for reference conditions, tests were carried out with a vessel (i.e. pan) under the laboratory condition of 298.15 K (25°C) and 101,500 Pa for temperature and absolute pressure, respectively.

SRB was supplied with G20 type Natural Gas (NG) of 50 MJ/m3 of Lower Heating value (LHV) at a nominal supply supply pressure of 20 mbar. Then it was adjusted within ± 2% to its nominal heat input of 1,75 kW.

An aluminium vessel with diameter of 240 mm with no handles and complying with the characteristics of EN 30-1-1:2008+A3:2013, C.1 and containing 4.8 kg of water was placed on a cast-iron grill just above the burner (figure 2).

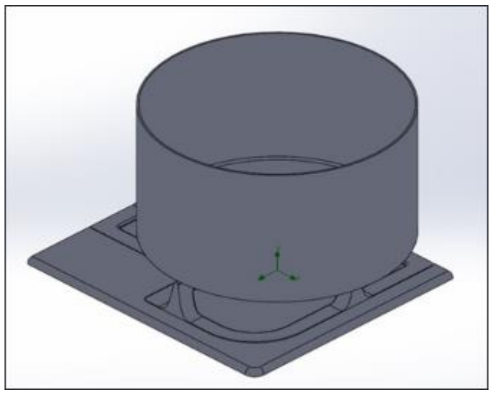

Fig. 2: Schematic of the aluminium vessel on one fourth of the hob.

2.2. Modification

As it is mentioned in section 1, the main object of this study is to understand how exactly CFD simulation can predict the behaviour of the system especially after applying a modification on it. In this regard, one conventional design modification of the gas hobs was implemented to the system both in CAD and laboratory environments in order to observe the effect of this modification. In this modification, the distance between the burner cap and bottom surface of the aluminium vessel was increased about 2 mm by increasing the height of the cast iron grid (figure 3).

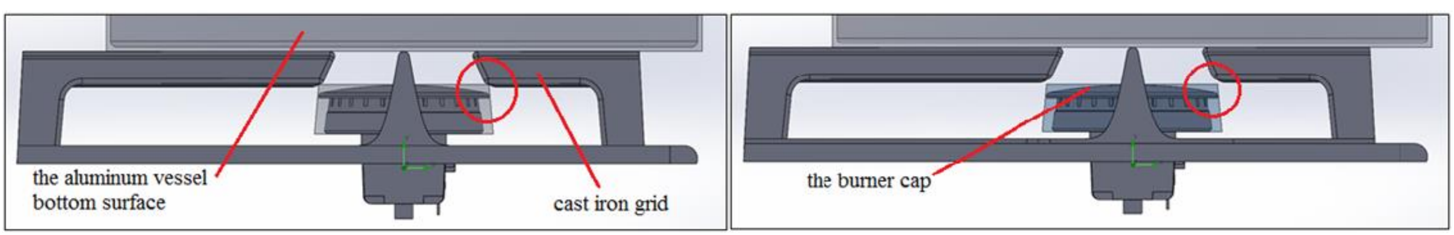

Fig. 3: 2 mm increasing of the distance between the burner cap and the vessel bottom surface.

2.3. CFD Analyses

The software used in this study is Mentor Graphics FloEFD which is based on the finite volumes method. This software solves formulations of mass, momentum and energy conservation laws for fluid flows (the Favre-averaged Navier-Stokes equations) with modified k-ε turbulent model with the damping functions [16] and with the laminar/turbulent transition [17].

It also utilizes the Van Driest's universal profiles and "Two-Scale wall functions" approach to describe the turbulent boundary layers. With help of these approaches, FloEFD will be able to ignore the CFD codes conventional restriction of using very fine mesh in boundary layers and therefore it can use the cartesian meshes for all geometries [18] and [19]. As a consequence of these feature, time of the meshing and simulation will be decreased significantly.

2.3.1. Model creation and boundary conditions

After the modelling of the gas hob in CAD program, the simulation of the combustion of G20 gas in the SRB and combustion products' flow around the burner and vessel starts-up with FloEFD Wizard command. By help of this definition of the boundary and initial conditions are fulfilled very fast. Then, creation of the calculation domain and of the analysis type (internal or external) are made. For this study, an external analysis type has been setup. This type of analysis (i.e. external) provides the ultimate accuracy results since it considers the air flows around the investigated model and includes the all heat transfer types (i.e. conduction, convection, and radiation) from the solid bodies to the moving flow.

For radiation heat transfer, the emissivity properties of the all surfaces including aluminium vessel and hob were specified.

2.3.2. CFD grid

Since the quality of the mesh affects tremendously the simulation results, a denser mesh was used in regions with combustion and near to the vessel bottom surface, while a coarse mesh was used in other regions [20]. Figure 5 shows a schematic diagram of the system with mesh structure.

The number of total solid and fluid cartesian cells are 4,630,960. Figure 4 shows the details of the created cartesian grid.

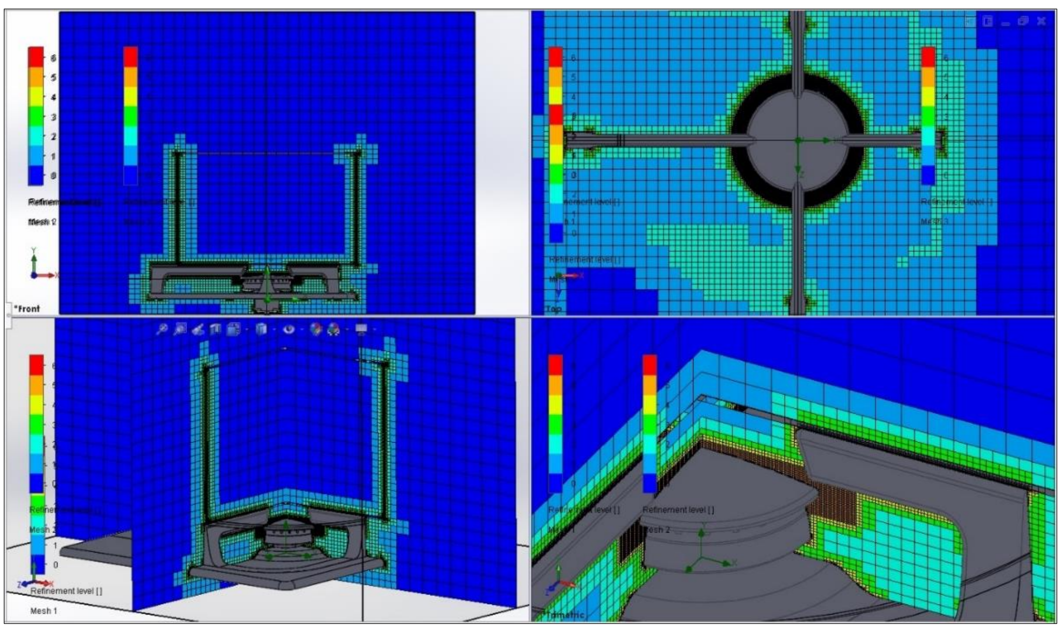

Fig. 4: Schematic diagram of the system with mesh structure.

2.3.3. Transient simulation

Simulation of the combustion of a gas hob including radiative heat transfer model and several chemical reactions must be fulfilled in the transient regime. In the scope of the second aim of this study, energy and combustion test time were divided in two time-steps:

Step one: Flame developing step

Step two: Steady combustion step (after the first step up to the end of the test)

In **the first step**, simulation initiated with a very small-time step of 0.005 second. This step represents the physical time from ignition to complete development and spread of the flame around the burner (approximately 3 seconds) [21].

At **the second step**, simulations continued with a higher time step of 0.01 second since the flame reached the steady state. This means that calculation will progress faster with stable flame characteristics but changing fluid properties such as as temperature, velocity and solid surfaces temperature distribution. At this step, there isn't any time limit for calculation, and simulation can continue for example up to the real test time which takes more than 45 minutes. However, as explained explained before, since in this study, it's not aimed to calculate the exact values of the test after the full time, and it's only only tried to compare the behaviour of the model before and after modification and the simulation results with the real laboratory test results, step two was limited to 30 seconds. After this time, beside the combustion emission results, the temperature of the water inside the vessel was also recorded and compared.

2.3.4. Results monitoring

For emissions monitoring: one virtual truncated cone around the burner has been created and average values of CO_2 and CO emissions within this volume before and after the modification at the end of the first step (i.e. at 3rd seconds) were recorded (figure 5). Approximately bulk volume is 50 cm^3.

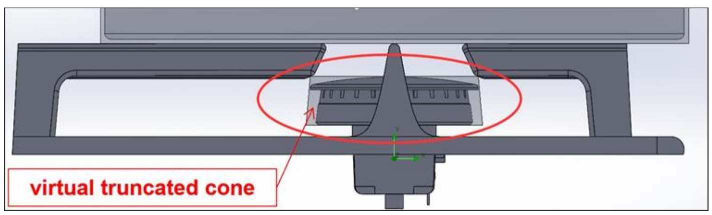

Fig. 5: virtual truncated cone around the burner.

For efficiency monitoring: as it explained before, to accurate calculation of efficiency, simulation must be implemented for physical time of 45 min which would take several months for finalizing. To overcome this problem, after the stablishing of the flame and at the end of the second step (i.e. at 30^{th} seconds), average temperature of the water inside the vessel was recorded before and after modification (2 mm distance increasing). If the temperature of the water increases, it can be interpreted as an improvement in the energy efficiency. If it decreases, then it can be said that energy efficiency of the system derogates.

3. Results

Experimental and CFD studies were developed to determine the corresponding detailed emissions and thermal properties around the burner for a domestic gas hob.

3.1. Experimental Results

Laboratory test results showed that 2 mm increasing of the distance between the burner cap and bottom surface of the vessel could increase the CO_2 emission. On the other hand, this modification caused the reduction of CO emission and consequently decrease of the energy efficiency of the system. Detailed results of the experiment are given in table 1.

Table 1: Comparison between the combustion and emission results of the gas hob before and after modification.

Parameter	Modification	Bulk Average
Temperature [°C]	Before Modification	30.15
Temperature [°C]	After Modification	28.32
Temperature [°C]	Difference (Before-After)	-1.82

3.2. CFD Results

Using the commercially available FloEFD software, computational runs were carried out on two times on the model:

1- Before the modification (distance between cap and vessel is in original level)

2- After the modification (vessel was taken 2 mm higher than previous level)

In each time, first step (for emission measurement and 3 seconds of physical time) took approximately 8 hours and second step (i.e. simulation run for water temperature measurement and for 30 seconds of physical time) took about 4 Specification of the used WorkStation are as below:

- Processor: Intel Xenon 36 Core 2.3 GHz
- RAM: 64 GB

3.2.1. CFD Emission Results

Figure 7 shows the distribution of the CO_2 emission mass fraction on a diametrical vertical plane before and after modification. First, in both cases, relatively higher concentrations of CO_2 occur in the region below the vessel. The reason may be the longer residence time of the emissions that accelerate the CO conversion to CO_2 which is usually a slow process [11]. However, 2 mm increase of the distance exhibits increase of CO_2 mass fraction.

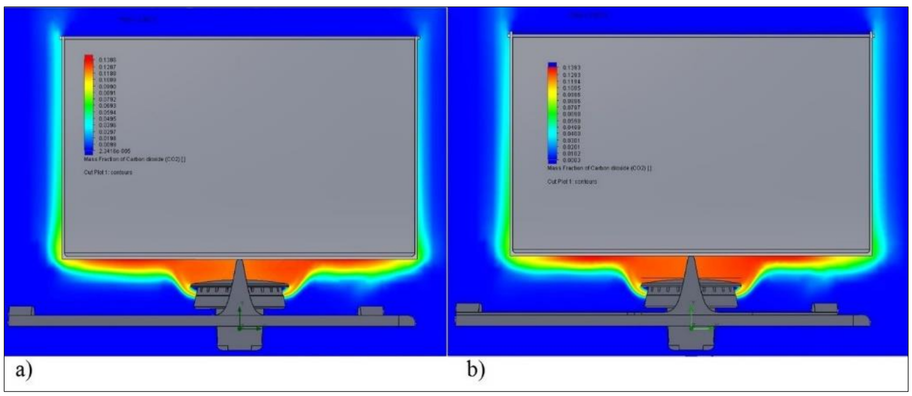

Fig. 7: CO_2 Emissions distribution after 3 seconds of physical time a) original position, and b) after modification.

The predicted distribution of the CO emission mass fraction is shown in Figure 8 which shows almost opposite results to those of the CO_2. By increasing the distance between two surface CO concentration reaches the higher value.

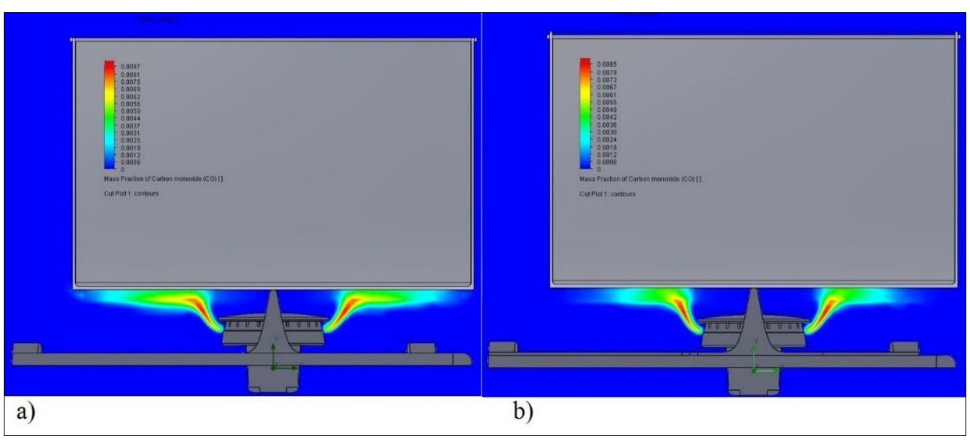

Fig. 8: CO Emissions distribution after 3 seconds of physical time a) original position, and b) after modification.

Table 2 presents the detailed calculated results of the CO_2 and CO emissions within the virtual truncated cone. As it can be seen from the table 2, increasing the distance between the flame and pot results in reduction of CO and increase of CO_2. Same trends were obtained at experimental tests in R&D laboratory as it explained in section 3.1.

Table 2: Comparison between emission results of the gas hob before and after modification.

Parameter	Modification	Bulk Average
Mass Fraction of Carbon dioxide (CO_2)	Before Modification	0.0329
Mass Fraction of Carbon monoxide (CO)		0.000163
Mass Fraction of Carbon dioxide (CO_2)	After Modification	0.0345
Mass Fraction of Carbon monoxide (CO)		0.000161
Mass Fraction of Carbon dioxide (CO_2)	Difference (After - Before)	+0.0017
Mass Fraction of Carbon monoxide (CO)		-0.000002

3.2.2. CFD Temperature Results

The distribution of the temperature counters is shown in figure 9. These results were obtained at the end of the second steps (after 30 seconds). Figure 9.a is before modification and 9.b is after the modifications.

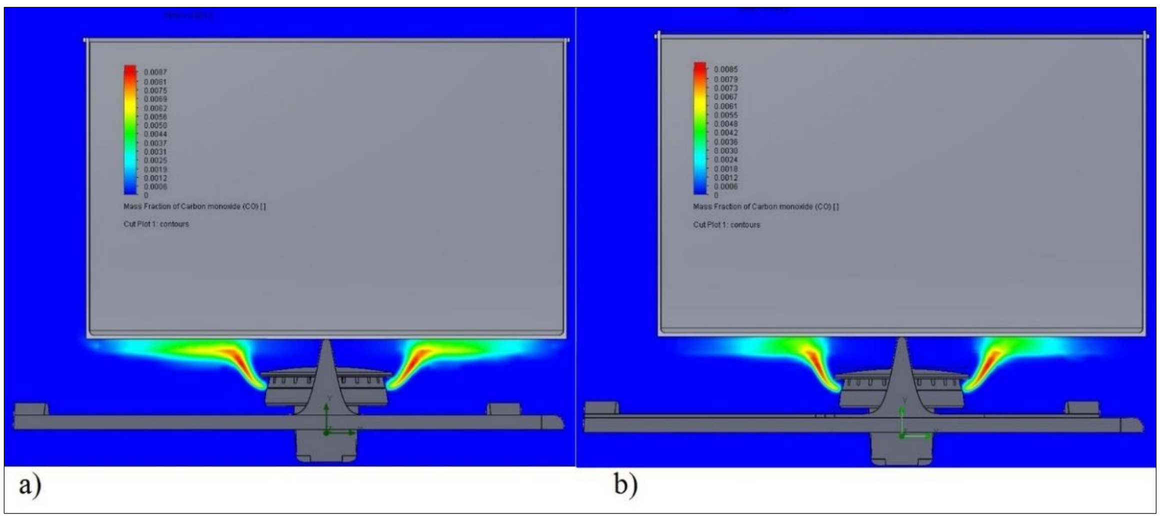

Fig. 9: Temperature distribution after 30 seconds of physical time a) original position, and b) after modification.

Table 3 presents the average water temperature at 30th second of simulation for original position and after the modification. As it can be seen from the table, increasing 2 mm of the distance between the burner cap and aluminium vessel results in reduction of the average temperature of the water of the vessel. Stated in other words, moving up the vessel decrease the burner (i.e. hob) efficiency because after the exactly same time (30 seconds) temperature reaches to 28.3°C while before modification it calculated as 30.15°C.

Table 3: Comparison between water temperature increase after 30 second for before and after modification.

Parameter	Modification	Bulk Average
Temperature [°C]	Before Modification	30.15
Temperature [°C]	After Modification	28.32
Temperature [°C]	Difference (Before-After)	-1.82

4. Conclusion

The feasibility of using CFD for design and optimization of the gas hobs was investigated from the energy efficiency and CO and CO_2 emissions points of view. For comparison, 2 mm distance increase between the burner cap and the bottom surface was selected, and for validation the simulation results, experiment results from laboratory tests was used.

Simulations were divided in two time-steps. One for the flame stabling and one for the steady combustion. With the help of this method, the solving time was reduced considerably.

For a SRB, with increasing the distance between the burner cap and the vessel bottom surface, the CO_2 emission inside the virtual truncated cone increases, but the CO emission decreases, leading to a decrease of thermal efficiency. On the other hand, the average water temperature inside the vessel was also decreased after the modification which validate emission results.

Acknowledgements

This work was supported by the Candy Hoover Group.

References

[1] T. Döbler, "Simulation und Visualisierung in der Produktentwicklung," *FAZIT-Schriftenreihe,* MFG Stiftung Baden-Württemberg, 2008, Band 12.
[2] K. Stefan and R. Gunther, "CFD-Simulations in the Early Product Development," *Procedia CIRP*, 2016, pp. 443-448.
[3] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*, London: Pearson Education, 2007.
[4] T. Chung, *Computational fluid dynamics*. New York: Cambridge University Press, 2010.
[5] C. E. L. Pinho, J. M. P. Q. Delgado, V. Ferreira, R. Pilão and C. Pinho "Influence of Burner Geometry on Flame Characteristics of Propane-Air Mixture: Experimental and Numerical Studies," *Defect and Diffusion Forum*, vol. 273-276, pp. 162-167, 2008.
[6] P. J. Stopford, "Recent applications of CFD modelling in the power generation and combustion industries," *Applied Mathematical Modelling*, vol. 26, pp. 351-374, 2002.
[7] Z. Riahi, M. A. Mergheni, J. C. Sautet and S. B. Nasrallah "Numerical Study of Turbulent Normal Diffusion Flame CH4-Air Stabilized by Coaxial Burner," *Thermal Science*, vol. 14, pp. 1-20, 2012.
[8] A. Kumar, R. Kumar and S. Jain, "Application of Computational Fluid Dynamics for Different Fire Strengths in a Compartment Using Combustion Modelling," *Fire Science and Technology*, vol. 33, pp. 35-46, 2014.
[9] L. Gattei, "A Study on The Fluid Dynamics of Domestic Gas Burners", Ph.D. dissertation, University of Bologna, Bologna
[10] F. Sanz-Serrano, C. Saguesa and S. Llorente, "Inverse modelling of pan heating in domestic cookers," *Applied Thermal Engineering*, vol. 92, pp.137-148, 2015.
[11] İ. B. Özdemir and M. Kantaş. "Investigation of partially-premixed combustion in a household cooker-top burner," *Fuel Processing Technology*, vol. 151, pp.107-116, 2016.
[12] CEN/TC, T. C. (2015, 06 03). Domestic cooking appliances burning gas - Part 2-1: Rational use of energy - General. Estonian Centre for Standardisation.
[13] Y. C. Ko, T. H. Lin, "Emissions and efficiency of a domestic gas stove burning natural gases with various compositions", *Energy Conversion and Management*, vol. 44, pp. 3001-3014, 2003.
[14] A. Y. Wagner and P. G. Kristensen, "Particle Emissions from Domestic Gas Cookers," *Combustion Science and Technology*, vol. 182, pp. 1511-1527, 2010.
[15] J. B. Stubington, "Emissions and efficiency from production cooktop burners firing natural gas," *J. Inst. Energy*, pp. 143-155, 1994.
[16] C. A. Lam, "Modified Form of Model for Predicting Wall Turbulence," *ASME Journal of Fluids Engineering*, pp. 456-460, 1981.

[17] T. V. Trebunskikh, A. V. Ivanov and G. E. Dumnov, “FloEFD simulation of micro-turbine engine,” *Mentor Graphics* Moscow, Russia. [Online]. Available: http://cadflo.com/fileadmin/user_upload/Trebunskikh__FloEFD_simulation_of_micro-turbine_engine.pdf
[18] M. G. Corporation, (2011). Enhanced turbulence modelling in FloEFD. [Online]. Available: www.cadflo.com: http://www.cadflo.com/fileadmin/user_upload/mentorpaper_65206.pdf
[19] G. A. Kalitzin, “Turbulence Modeling in an Immersed-Boundary RANS method”, *Centre for Turbulence Research Annual Research Briefs*, 2002, pp. 415-426. [Online]. Available: https://web.stanford.edu/group/ctr/ResBriefs02/kalitzin.pdf
[20] E. Yasa, “The Evaluation of the Effects of Different Building Forms and Settlement Areas on the Thermal Comfort of Buildings,” *Procedia Engineering*, pp. 3267-3276, 2017.
[21] F. A. Williams, *Modern Developments in Energy, Combustion and Spectroscopy*. *In Honor of S. S. Penner*, Elsevier. 1993, pp. 15-27.