Improvement of the Combustion Chamber

Jan Novosád^{1,*}, Pavel Peukert¹, and Marcel Havrda²

¹VÚTS, a.s., Svárovská 619, Liberec, Czech Republic

²Brano, a.s., R&D Heaters, Jablonec nad Nisou, Czech Republic

Abstract. The subject of this work is the analysis of an independent heater combustion chamber. The main goals are the analysis of the current state and modification of the combustion chamber design, while the CFD methods are applied. The standard methodology considering the model and grid preparation and the ANSYS Fluent solver setup were provided to obtain the velocity, thermal and species concentration fields as results. These results were used for the analysis of the flame stability as well as the heat load of the individual component of the combustion chamber. The initial results leads to the identification of areas for possible improvement due to the uneven thermal fields and high thermal stress. Then, several modifications were designed and tested. Finally, the results for modified design were analysed to evaluate the variant suitable for future experimental testing.

1 Introduction

A combustion chamber is the part of an internal combustion device in which the air-fuel mixture is burned. In our case, the combustion chamber of an independent heater was investigated. The independent heater is a device using the heat generated by burning fuel for heating of air.

In the long run, the development in the field of small combustion devices is based on prototyping. Hence, the prototypes are experimentally tested to obtain the operating parameters, e.g. flow rates, temperature, heat power, and flue gases composition. These parameters are usually measured outside the combustion chamber area, i.e. at the inlets and outlets. Following these, several indirect (external) methods have been found and used to identify heat transfer and thermal load inside the combustion devices, described for instance by Lipkea et al. [1], Franco and Martorano [2], Enomoto and Furuhamu [3], and many others. The experimental investigation of flow and thermal fields directly inside the combustion chamber is quite difficult, however. In recent years, the non-contact, especially optical, methods are applied for identifying the fields inside the engine cylinder. Different optical systems have been used, e.g. the LIF imaging by Kranz et al. [4], Vanhaelst et al. [5] developed and used the IR sensor and finally Muhammad et al. [6] designed the UV endoscopic imaging system. The disadvantage of the optical system is the system prize, generally. Furthermore, the costs of prototype manufacturing are still essential.

On the other hand, the simulation and numerical testing could be used for the combustion chamber research. The paper of Kranz [4] includes the CFD simulation and its results

* Corresponding author: jan.novosad@vuts.cz

in comparison with the experimental work. Adámek et al. [7], followed the following the work of Merker [8], investigated the combustion chamber of the independent heater by CFD methods. They have prepared and verified the numerical model for a complex case with fluid flow, oil combustion and heat transfer. The advantage of such models is the possibility to see the influence of individual design parameters on the flow rates, temperature and thermal output. Additionally, the research and development of heaters with limited production of real prototypes could be realised.

2 Combustion chamber of the independent heater

The independent heater and its combustion chamber which we are aimed in this work is a commonly produced type. The main parts are illustrated in Figure 1 as the representation of a simplified model only. Firstly, the air (7) driven by the blower (2) is mixed with the fuel (8) inside the mixing chamber (3). Then the air-fuel mixture enters through the nozzle (5) the combustion chamber (4) inside which it mainly burns. Finally the generated heat is transferred via the aluminium heat exchanger (1) wall which is the interface between flue gases (inside) and heated air (outside) to the rest of the hvac system.

Fig. 1. Combustion chamber model: a) 3D overview, cross-section through the spark plug axis.

For our case, the experience indicate some limitations and risk factors when operating outside the nominal conditions, i.e. the operation modes at very low (minimum) or extremely high (maximum) power levels. To be specific, the high sensitivity of the flame stability have been found at minimum power level, when the small amount of fuel is injected and the minimum heat flux is gained. Otherwise, the heat generated at the maximum level causes extremely high heat load of the inner components. To conclude, the operation mode significantly affect lifetime and maintenance intervals of the heater. The investigation and analyses of the phenomena inside the whole combustion chamber should give us probably the hints for the improvement of inner composition. Therefore, the longer lifetime and greater reliability in wider range of operation modes could be expected as the result of our research.

3 Combustion modelling

The CFD methods were chosen for the problem analysis using ANSYS Fluent software. The expected fluctuations of the flow and thermal fields should be neglected, hence the computations are set as steady-state. The $k-\epsilon$ realizable model with the non-equilibrium wall functions in the near wall region was used to consider the viscous flow effects.

The flow is modelled as multiphase where the continuous phase represents the combustion air as well as the flue gases. The gas mixture is composed by N_2 , O_2 , CO , CO_2 , $C_{12}H_{23}$, CH_4 , OH , H_2O , and H , additionally. The fuel is injected as DPM evaporating particles. The fuel properties were substituted by properties of kerosene $C_{12}H_{23}$ from the ANSYS Fluent material library. The solid bulk was the aluminium heat exchanger for which the aluminium properties from the default material library was applied.

The combustion process is modelled using the Partially Premixed Combustion model [9], which is the hybrid model based on the Non-Premixed Combustion model and the Premixed Combustion model. The density weighted mean scalars (such as species fractions and temperature) are calculated from the probability density function (PDF). The PDF table values were calculated from the thermo-physical properties of gas species in the range of operating temperature and pressure.

3.1 Computational domain

The computational model was based on the CAD file supplied by the manufacturer. The original geometry of the independent heater was limited to the area close to the combustion chamber including the devices important for the simulation. The computational domain consist of both fluid and solid regions. These reduced model was then simplified by neglecting the small size elements. Hence, several surfaces were prepared for boundary conditions definition. The final computational model with labelling of named selections is shown in Figure 2.

Fig. 2. Combustion chamber model.

The model discretization was done using ANSYS Fluent Meshing and its functions to create a polyhedral mesh. The refining in the near wall region was applied. As a result, the computational grid consists of 3.5 million elements (see Figure 3).

Fig. 3. Computational grid.

3.2 Boundary conditions

The operation parameters of the independent heater were used for the specification of the boundary conditions. Two operation modes, such as the minimum and maximum power, were taken into account for simulation. The rotation of the blower rotor was given as well as the flow rate of fuel (see Table 1.). The flow rate of the combustion air corresponds to the blower speed. Following the labels of boundaries in Figure 2 the boundary conditions were defined in the way shown in Table 2. Operating pressure was set to be $p_{op} = 101\,325\text{ Pa}$.

Table 1. Combustion chamber - operation modes parameters.

	1-Minimum power	2 – Maximum power
Blower rotation (RPM)	2760	7500
Fuel flow rate (kg/s)	$2.23 \cdot 10^{-5}$	$7.14 \cdot 10^{-5}$
Air flow rate (kg/s)	$6.5 \cdot 10^{-4}$	$2.08 \cdot 10^{-3}$

Table 2. Boundary condition types.

Label	Type	Value
Blower rotor	rotating wall (MRF)	Blower rotation (see Table 1)
Air inlet	mass flow inlet	Air flow rate (see Table 1)
Fuel inlet	wall, DPM injection	Fuel flow rate (see Table 1)
Exhaust	pressure outlet	$p = 0\text{ Pa}$
Heat exchanger	wall	Convection $\alpha = 20\text{ W}\cdot\text{m}^2\cdot\text{K}^{-1}$

3.3 Solver setup

The Coupled solver has been used including the second order schemes except the using first order schemes for momentum equation and turbulent quantities. Based on the previously solved cases the computations were provided for maximally 10 000 iterations to reach the convergence. The area weighted average of O₂ concentration, temperature and mass flow rate at the outlet (Exhaust) are monitored during computation hence the simulation is stopped after these values achieved steady state. The computation is quite time consuming, one case takes about 20 hours on 20 CPU-cores.

3.4 Results assessment

Inside the computational domain there were defined two cross section planes (see Figure 4). Plane A is spanned by the axis of combustion chamber and the spark plug axis. Plane B is perpendicular to plane A. The contours of pressure, velocity and temperature such as the concentration profiles in plane A and B were obtained and analysed as results. The thermal load of the combustion chamber were analysed based on the temperature profiles of the combustion chamber pipe.

Fig. 4. Combustion chamber – planes for results evaluation.

4 Initial issue results

The original CAD design were used for the initial analysis. The simulation were provided following the methodology described in chapter 3. Two sets of results were obtained for the minimum and maximum heater power level.

The selected thermal field contours for the maximum power case are shown in Figure 5. The temperature distribution shows the flame position and the thermal load of combustion chamber pipe. For the initial case the flame and thermal load are evenly distributed in plane B, otherwise the distribution of high temperature area in plane A is a little bit concentrated to the upper side of the combustion chamber. The significant thermal load affects the upper part of the chamber nozzle (marked with a red ellipse in Figure 5a). Additionally, the end part of the combustion chamber is unevenly thermally stressed (see Figure 6a). The contours of O₂ concentration in Figure 6b show again some uneven distribution in the upper part of the combustion chamber, similarly to the temperature field.

Fig. 5. Initial issue – maximum power level, contours of temperature in plane A (a) and plane B (b).

Fig. 6. Initial issue results – maximum power level: temperature of combustion chamber pipe (a), concentration of O₂ in plane A (b) and plane B (c).

These inhomogeneity is probably caused by the location of the spark plug and the fuel inlet, i.e. the components that cannot be moved in the design. Also the inner shape of the combustion chamber influenced the fluid flow, hence the modification could lead to the improvement of the thermal distribution in the combustion chamber. Due to the identified thermal load of the nozzle shown in Figure 5a the nozzle was selected for modifications.

5 Simulation of the modified design

The modifications of the nozzle were designed. The main goal was to even out the thermal field distribution. Also the rotation of air/fuel mixture was taken into account for improving the burning process.

Several steps were done to find the correct solution. Three stages of the design process took place, finally the fourth stage of the design process was selected for numerical testing. The newly designed nozzle is made of a sheet metal where the inner part is a circular hole drawn into a throat. The outer space is filled with few lanced parts. Several configurations were tested. The sketch of variants V04a-e are shown in Figure 7. The throat diameter D is the same as for the initial design. The green areas illustrated the newly placed lanced parts.

The bending angle in the direction of nozzle axis were set to 30° for V04a, then 20° for V04b,c,e and 15°for V04d.

Fig. 7. Nozzle modifications – different lancing (V04a-d) and lower throat diameter $d = 2/3D$ (V04e).

The numerical testing was performed for all the cases V04a-e following the methodology used for the initial issue. The selected results shown in Figure 8 illustrate the velocity fields at maximum power level. The bending angle of the lanced parts affect significantly the flow field as well as the throat diameter. The unfavourable combination of angle and throat diameter cause negative effects such as the increase of velocity in the upper part of the throat and non-uniform velocity profile in the throat.

Comparison with initial issue results shows the flame (high temperature area) located closer to the chamber axis for all the cases V04a-e (see Figure 9), so there has been an improvement over the original design. The change of the flame position leads to the evenly distributed heat load of the combustion chamber pipe. However, in the case V04b and V04e, the thermal stress on the throat of the nozzle is still very high. The high temperature area (highlighted by red circle in Figure 9) is located also inside the channel created by the lanced sheets, while the distance from the wall should prevent damage.

The combustion stability were analysed based on the thermal fields shown in Figure 10. The higher temperature regions in cases V04a and V04e demonstrate a satisfactory and stable flame, the rest cases are unsuitable for continuous operation. To sum up, cases V04a and V04e seem to be suitable for further use. These improvements should lead to less thermal stress on the components and thus longer heater life while maintaining stable operation.

Fig. 8. Velocity contours – maximum power level, section in plane A.

Fig. 9. Temperature contours – maximum power level, section in plane A.

Fig. 10. Temperature contours – minimum power level, section in plane A.

6 Conclusion

The combustion chamber of the independent heating unit was numerically analysed for potential design improvements. Based on the specification given by the manufacturer, the numerical model was prepared for simulation of combustion inside the combustion chamber. The simulation setup and boundary conditions specification were defined in accordance with the operating parameters.

The initial issue was tested numerically to obtain the velocity and thermal fields. The data about species concentration and progress variables were also found. Two operation modes have been analysed, represented by the minimum and maximum power level. The analysis of results shows potentially interesting areas for modifications, which could improve the combustion process or extend the service life. The uneven flame position inside the volume of the combustion chamber have been found as well as the increased thermal stress of the nozzle throat. These results were verified by the experience of the manufacturer and the reason why high refractory material have been applied to these regions.

Then the combustion chamber nozzle has been taken for modifications. Few design steps were performed, with version V04 selected for numerical testing. The tested nozzle was designed as a circular throat connected on its outer diameter with a sheet with several lanced parts. The sheet bending angle and throat diameter as a parameters define several variants for testing. Five geometrical variants were prepared, while 10 cases were computed, totally. The problem is computationally expensive, over 20 hours on 20 CPU-cores per each case.

The modified design do not show the significant influence of the individual design parameters. Due to the interconnection of the nozzle parts, i.e. the throat and lanced sheets, it is necessary to assess the influence of both geometric parameters together.

The analysis revealed a reduction in the thermal stress of the individual components, especially the combustion chamber end for some cases due to the repositioning of the flame closer to the axis of the chamber. Otherwise, some cases are not suitable because of the flame instability. Finally, two variants (V04a, V04e) were assessed as suitable for the improved combustion chamber. The recommendation for the prototype manufacturing and experimental testing have been issued for both variants, while the combustion parameters and lifetime testing should be realized.

The research was realized within the project FV20373 supported by the Czech Ministry of Industry and Trade. This publication was supported by the Czech Ministry of Industry and Trade in the framework of the institutional support for long-term conceptual development of research organization - recipient VÚTS, a.s.

References

1. W. H. Lipkea, J. H. Johnson, C. T. Vuk, SAE Transactions **87** (1978)
2. A. Franco, L. Martorano, SAE International Journal of Engines. **108**. (1999), DOI: 10.4271/1999-01-1252
3. Y. Enomoto, S. Furuhashi, Bulletin of JSME, Vol. **28**, Issue 235 (1985), DOI: 10.1299/jsme1958.28.108
4. P. Kranz, D. Fuhrmann, M. Goschütz, S. Kaiser, et al., SAE Int. J. Engines **11** (2018), DOI: 10.4271/2018-01-0633
5. R. Vanhaelst, O. Thiele, T. Berg, B. Hahne, H.-P. Stellet, F. Wildhagen, W. Hentschel, C. Jördens, J. Czajka, K. Wislocki, Optical infrared-sensor inside the cylinder to determine the EGR- and residual gas rate in diesel engine, 11th Congress Engine Combustion Processes, 2013
6. S. Muhammad, S. Kaiser, M. Schütte, T. Berg, Characterization of UV endoscopic imaging systems for combustion applications, 9th European combustion meeting (2019)
7. K. Adámek, Int. J. of Mechanical Engineering and Applications. **3** (2015), DOI: 10.11648/j.ijmea.s.2015030101.18
8. G. P. Merker, Simulating combustion: simulation of combustion and pollutant formation for engine-development. Springer (2006)
9. Ansys Inc. ANSYS Fluent Theory Guide. Release 2019 R2