Influence of Heat Flux Density on Combustion Efficiency in the Initial Torch Section

Neven Krystev, Ivan Petrov

Abstract **- Proper understanding of combustion and heat transfer in combustion devices, and clarification of combustion and thermal efficiency, requires accurate measurement of heat flux density. In the present work, the influence of heat flux density on the combustion process in the initial section of the flare, under separate fuel and oxidant feed in the injected swirl conditions, is investigated using computer simulation. The possibility of determining the basic parameters of the combustion process and its control is shown. Basic characteristics of the heat transfer between the burning flare and the boundary surfaces of the combustion chamber are investigated, including heat transfer by both radiation and convection. The effect of changing the heat flux density through the cylindrical surface of the combustion chamber is analysed for four cases - under adiabatic conditions and in the presence of heat transfer, using Ansys/CFX software. It presents the possibilities of achieving optimal results in combustion processes by changing the geometrical, initial, boundary conditions of the combustion model.**

Keywords **– Harmful emissions, combustion processes, gaseous fuel, heat flux density, numerical simulation.**

I. Introduction

Combustion is a complex physical and chemical process that involves a number of sequential and parallel physical and chemical steps. A distinction is made between complete combustion i.e. without heat loss and incomplete combustion i.e. with heat loss. In complete combustion, all combustible substances of the fuel participate in the oxidation processes and only dioxides are formed - CO_2, SO_2, H_2O. The receipt of CO_2 in the exhaust gas signals that the combustion of the hydrocarbon fuel is complete. Chemical under burning occurs in the case of chemically incomplete oxidation of carbon - containing compounds to form carbon monoxide CO. Lower CO and larger CO_2 reflect more efficient fuel use and improved combustion quality [1], [2].

The combustion of gaseous fuel produces nitrogen oxides NO_X which are environmental pollutants. For gaseous fuel that does not contain organically bound nitrogen. They have a thermal origin and largely depend on the organisation of the combustion process [5], [14].

In industry, all flame-technical equipment using gaseous fuel operates under characteristic conditions of heat exchange between the burning flare and the boundary surfaces of the combustion chamber, which is a complex thermophysical process involving heat transfer by both radiation and convection [4].

Convective processes are an integral part of many natural phenomena as well as currents implemented in various technological devices. This determines a great interest in their experimental and numerical study. The study of the formation of convective currents through natural experiment requires a large resource of time and materials. The numerical approach makes it possible to vary a number of important parameters of the problem at hand [3], [7], [8].

Proper understanding of combustion and heat transfer in combustion devices, and clarification of combustion and thermal efficiency requires accurate measurement of heat flux density. Accurate knowledge of the heat flux distribution is also necessary for better control of the combustion process and design of new combustion devices.[6].

II. Substantive Part

The objective of this work is to investigate the possibility of influencing the combustion process efficiency by computer simulation. By investigating the basic characteristics of the heat transfer between the burning flare and the combustor boundary surfaces, including heat transfer through the combustor wall. The influence of the heat flux density through the cylindrical surface of the combustion chamber is analysed for four cases - under adiabatic conditions and in the presence of heat transfer, using Ansys/CFX software. The Ansys/CFX software package allows to model the combustion process taking into account turbulence, heat exchange and chemical reactions.

The Ansys/CFX model has been applied to study combustion processes and the formation of emissions of technically pure propane C_3H_8. Propane as a fuel has advantages over liquid fuels due to its good chemical and physical properties, such as a low carbon number in the fuel component [1].

The geometrical dimensions of an existing laboratory installation fig. 1 were used as the starting data for the numerical experiment. The laboratory installation consists of a seamless steel pipe with a hydraulically smooth surface, a rotating apparatus with four nozzles attached to it. The rotating apparatus allows to change the angle of the nozzles in intervals $\beta = 0 \div 60°$, over an interval of 15° [11], [14].

In the present work, the combustion process in the initial (mixing) section of the combustion chamber is considered, which has the dimensions of the outer diameter of the tube D = 76 mm and length L = 200 mm. Inclination of gas nozzles $\beta = 45°$ and nozzle diameter d = 0,8mm. As the

Received: 18.09.2024
Published: 30.09.2024
https://doi.org/10.47978/TUS.2024.74.03.005

Neven Krystev is with TU-Sofia, IPF - Sliven, Department "MMT", Sliven, Blvd. "Burgassko shose" №59 (nkrystev@tu-sofia.bg)
Ivan Petrov is with TU-Sofia, IPF - Sliven, Department "MMT", Sliven, Blvd. "Burgassko shose" №59 (ivan.apetrov@tu-sofia.bg)

mixing chamber is cooled under free convection conditions, with the temperature of the cooling fluid T_f = 300K. From reviewed literature data and sources [4], [9], [10], [12] to determine the heat transfer process between the boundary external surface and the surrounding fluid enveloping it due to density differences, the following criterion relation is used:

$$N_u = c(G_r P_r)^n \quad (1)$$

$$N_u = \frac{\alpha D}{\lambda} \quad (2)$$

$$q = \alpha\,(T_w - T_f) \quad (3)$$

TABLE 1

$G_r P_r$	$1.10^{-3} \div 5.10^3$	$5.10^3 \div 2.10^7$	$2.10^7 \div 1.10^{13}$
c	1,18	0,54	0,135
n	0,125	0,25	0,33

where: N_u, G_r, P_r – Nusselt criteria, Grashof criteria, Prandtl criteria; c and n – empirical coefficients; α – heat transfer coefficient [W/m^{2}K]; λ – coefficient of thermal conductivity (for steel 45 W/mK); D – characteristic geometric size, the outer diameter of the mixing chamber; T_w and T_f – mixing chamber wall temperature and coolant temperature respectively [K]; q – heat flux density [W/m^2].

Fig. 1. Schematic of laboratory installation

The computer simulation considered and analysed four cases of changing the heat flux density through the cylindrical surface of the mixing chamber - adiabatic conditions and the presence of heat transfer, setting different values of the heat transfer coefficient α. From literature sources and data [4], [9], [10], [12] it is known that α in different cases of convective heat transfer changes its values within the limits: in free convection for air from 5W/m^{2}K to 30 W/m^{2}K, under free convection for water from 10^2W/m^{2}K to 10^3 W/m^{2}K. As mentioned above, numerical simulations were performed for four cases - under adiabatic conditions (no heat transfer) the value of α = 0 W/m^{2}K and with the presence of heat exchange, taking the values of α = 30, 100, 1000 W/m^{2}K. As in previous computational experiments, results were obtained, which are not commented on in the present work, from a value of α = 5W/m^{2}K, where the analyzed results showed that at such value of heat transfer coefficient are very close to adiabatic conditions.

A system of Ansys/CFX differential equations for heat transfer, heat transfer and coolant movement, complemented by the continuity equation, describes an infinite number of convective heat transfer processes. For the mathematical description of a particular heat transfer process, the initial and boundary conditions of that process must be added to this system of equations. In this case, the solution of the described system of equations will allow to determine the values of velocities, temperatures, heat transfer coefficients and heat flux densities over the entire area considered [13], [16].

Fig. 2 shows a view of the geometric model in the Ansys/CFX environment, which is a combination of five cylindrical solids. The dimensions of the initial (mixing) section of the combustion chamber with outer diameter D = 0,76mm and length L = 200mm and four nozzles with diameter d = 0,8mm were used as initial geometrical data.

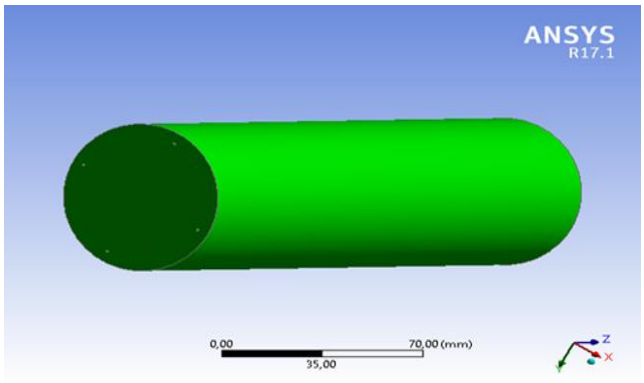

Fig. 2. Schematic of geometric model

Ansys/CFX uses a finite element network (numerical values at the nodes of the network), similar to that used in strength analysis, to discretize the domain Fig. 3.

As a result of this approach used in Ansys/CFX, finite-volume equations are generated that ensure conservation of flow values, which is a prerequisite for the exact solution of the problem [15].

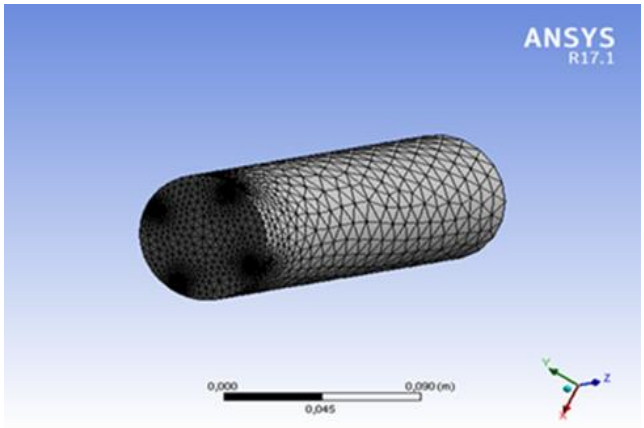

Fig. 3. Discretisation of the geometric model

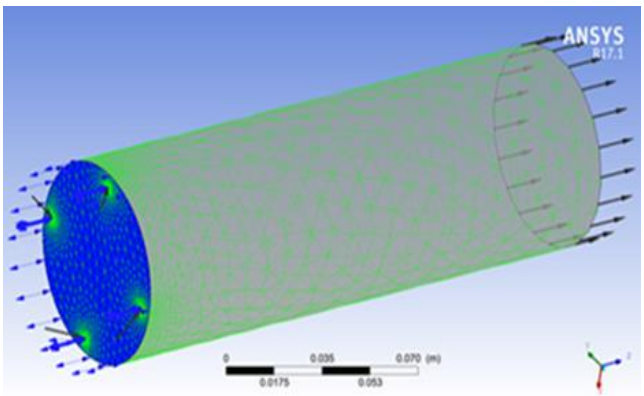

Fig. 4. Scheme of fluid spaces

Fig. 4 shows the fluid spaces of the mixing chamber and the four nozzles. Diffusion combustion of propane gas is simulated, which flows out through the openings of cylindrical nozzles that are positioned at an angle $\beta = 45°$. The directions of the vectors are depicted, and it is observed that active gas jets entrain the necessary amount of air required for combustion.

III. Results

Fig. 5, Fig. 6, Fig. 7 and Fig. 8 show the temperature distribution in the combustion flare itself along its central longitudinal axis at different heat transfer coefficients.

Fig. 9, Fig. 10, Fig. 11 and Fig. 12 show graphically the temperature variation in the flare along the axis of the mixing chamber at different values of the heat transfer coefficient α. The comparative analysis shows that as the value of α (analogous to the heat flux density) increases, the cooling intensity of the mixing chamber walls decreases, the average temperature level in the flare decreases.

Fig. 13, Fig. 14, Fig. 15 and Fig. 16 show graphically the variation of NO_X concentration of combustion products at different values of heat transfer coefficient α.

Fig. 17, Fig. 18, Fig. 19 and Fig. 20 show graphically the variation of CO_2 concentration in the combustion products at different values of the heat transfer coefficient. α. It is observed that as the value of α increases, the concentration of CO_2 continuously increases.

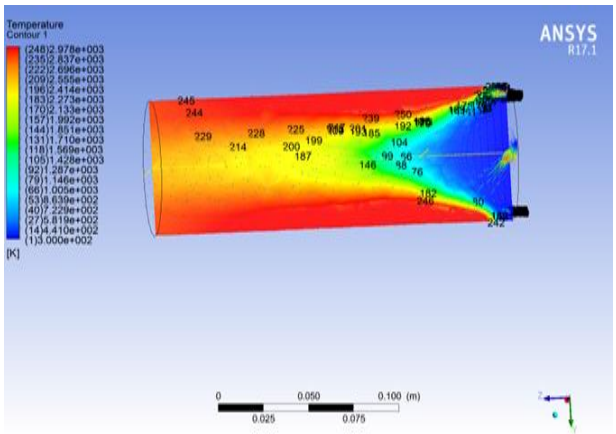

Fig. 5. Temperature distribution at α=0 W/m^{2}K

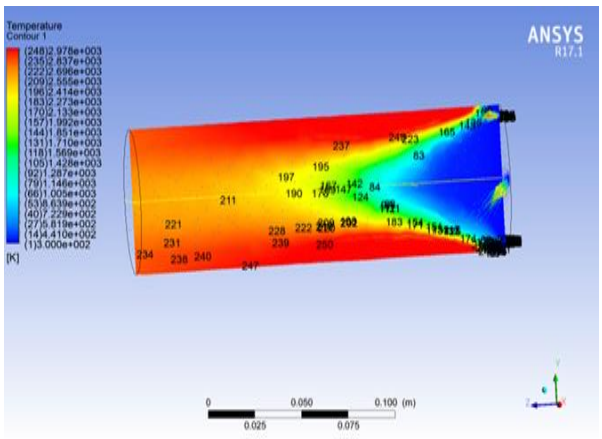

Fig. 6. Temperature distribution at α=30 W/m^{2}K

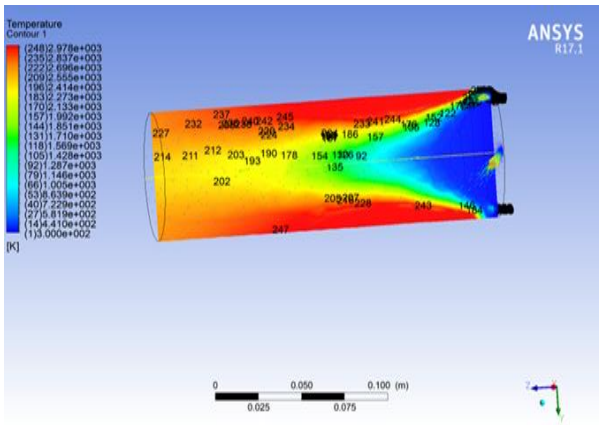

Fig.7. Temperature distribution at α=100 W/m^{2}K

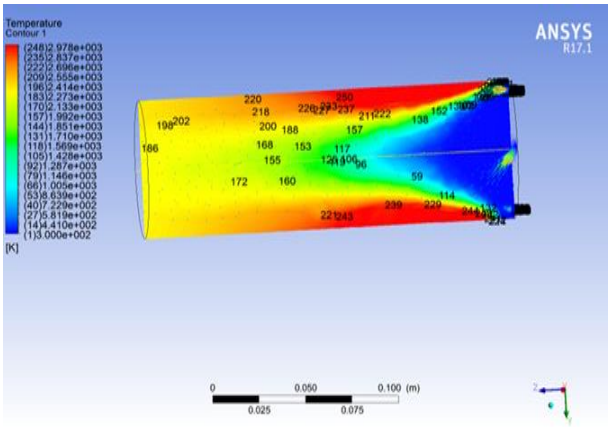

Fig. 8. Temperature distribution atα=1000 W/m^{2}K

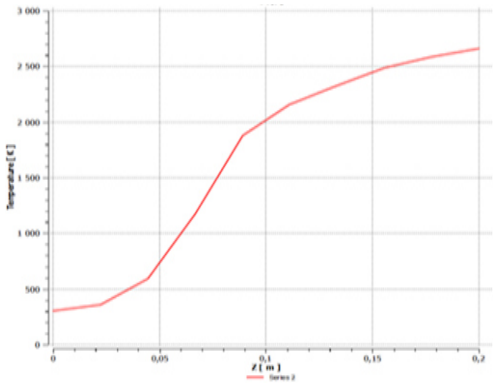

Fig. 9. Temperature variation along the mixing chamber axis at α=0 W/m^{2}K

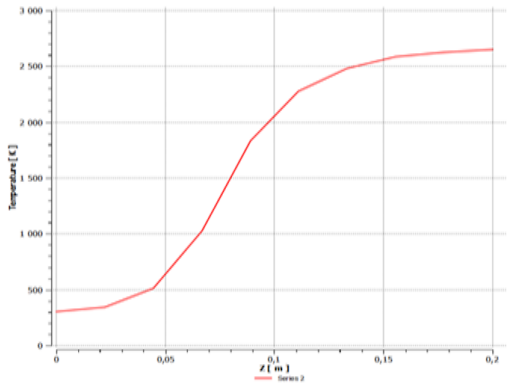

Fig. 10. Temperature variation along the mixing chamber axis at α=30 W/m^{2}K

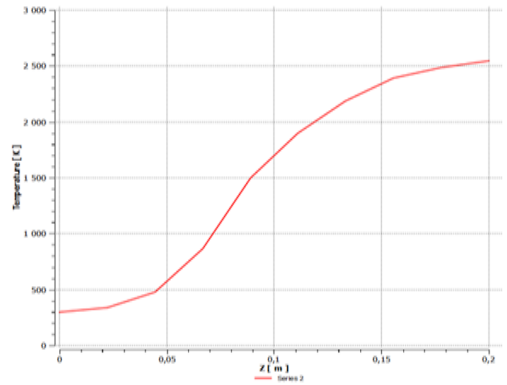

Fig. 11. Temperature variation along the mixing chamber axis at α=100 W/m^{2}K

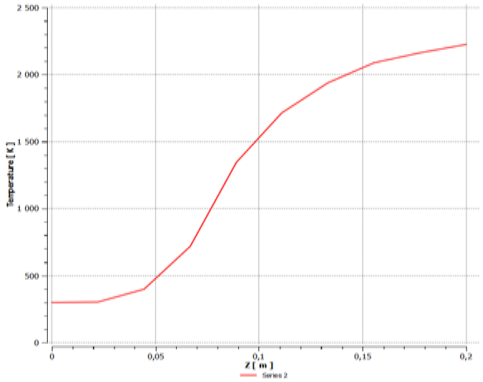

Fig. 12. Temperature variation along the mixing chamber axis at α=1000 W/m^{2}K

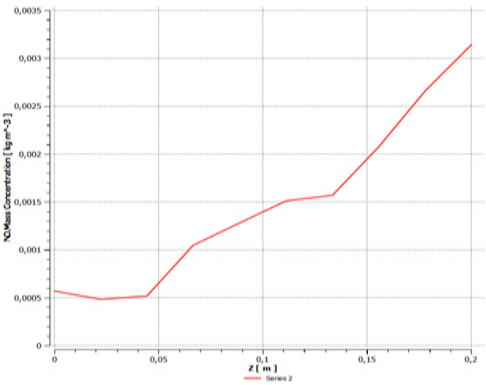

Fig. 13. Mass concentration of NO_X α=0 W/m^{2}K

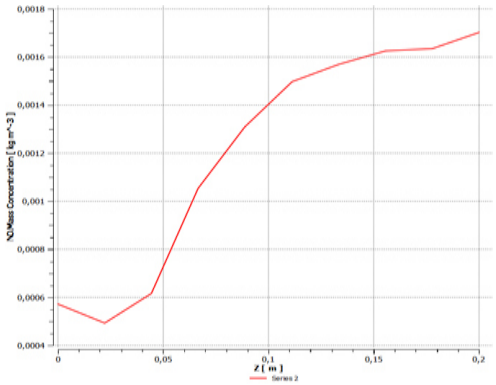

Fig. 14. Mass concentration of NO_X α=30 W/m^{2}K

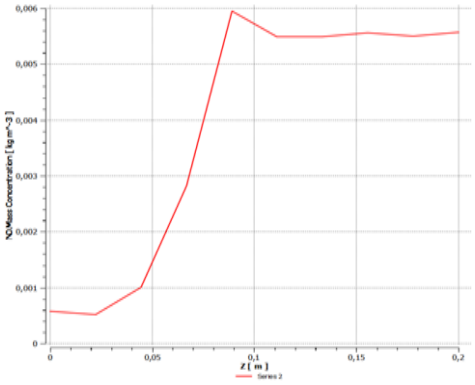

Fig. 15. Mass concentration of NO_X α=100 W/m^{2}K

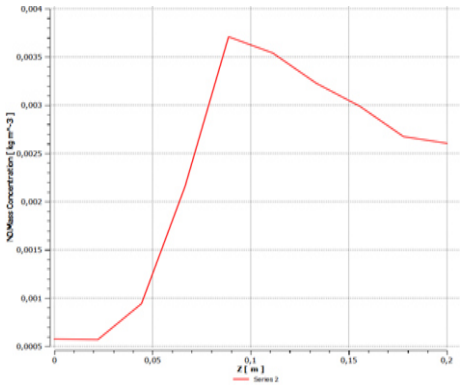

Fig. 16. Mass concentration of NO_X α=1000 W/m^{2}K

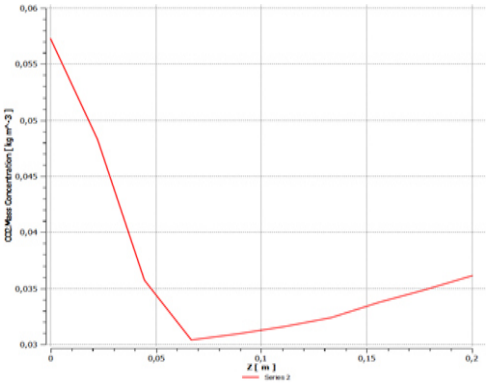

Fig. 17. Mass concentration of CO_2 α=0 W/m^{2}K

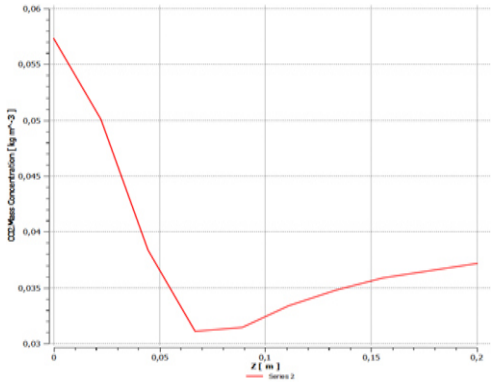

Fig. 18. Mass concentration of CO_2 α=30 W/m^{2}K

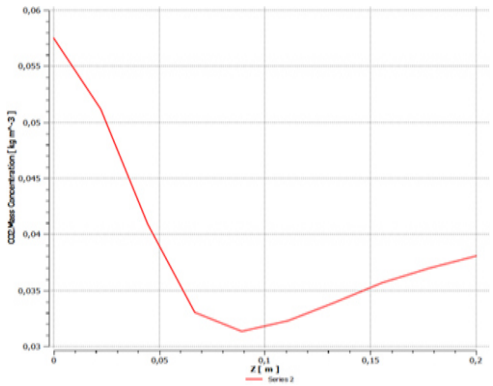

Fig. 19. Mass concentration of CO_2 α=100 W/m^{2}K

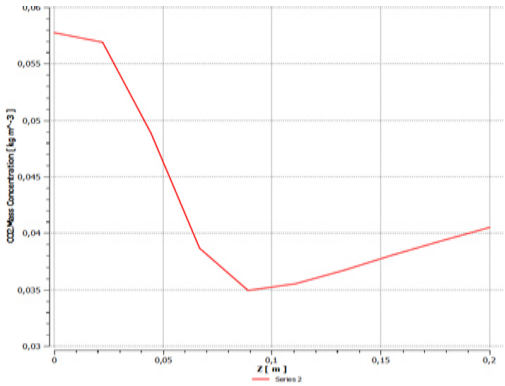

Fig. 20. Mass concentration of CO_2 α=1000 W/m^{2}K

IV. Conclusion

1. In diffusion combustion of a rotating gas flare in a confined space, the heat flux density parameter through the walls of the mixing chamber significantly affects the size and temperature of the high-temperature core, as well as the concentration of nitrogen oxides and carbon dioxide in the combustion products.

2. The presented numerical model of the combustion process can be used in the design or reconstruction of combustion plants, looking for an optimal option between combustion completeness and emission formation.

3. The algorithm of the stimulation model can be used in the solution of similar problems in the training of students in heat engineering disciplines.

References

1. Windarto C., Lim O., „A comprehensive study of the effects of spark discharge duration on low-carbon combustion of high-pressure direct-injection propane: Factors affecting combustion, in-cylinder performance, and emissions“, International Journal of Hydrogen Energy, Volume 49, Part B, 2024, Pages 796-815, ISSN 0360-3199, https://doi.org/10.1016/j.ijhydene.2023.09.090.
2. Valchev G., „Gorivna tehnika i tehnologii“, Akademichno izdatelstvo na VIHVP, Plovdiv, 2001g.
3. Kuznetsov V.A., „Chislennoe issledovanie gorenia i teploobmena pri obzhige keramzita vo vrashtayushteysya pechi“, Vestnik BGTU im. V.G. Shuhova, , no. 1, 2017, pp. 170-174, DOI: 10.12737/23534
4. Kostov P., Krystev N., „Vliyanie na platnostta na toplinnia potok prez stenite na gorivnata kamera varhu obrazuvaneto na azotni oksidi pri izgaryane na gazoobrazno gorivo“, sp.” Mehanika na mashinite”, Izd. na TU-Varna, kn. 72, ISSN 0861-9727, s. 53, 2007g.
5. Kostov P., Krystev N., Atanasov K., „Modelirane na termichnite uslovia za obrazuvane na azotni oksidi pri izgaryane na gazovo gorivo“, sp.” Mehanika na mashinite”, Izd. na TU-Varna, kn. 62, ISSN 0861-9727, s. 52, 2006 g.
6. Arai N., Matsunami A., Stuart W. Churchill, „A review of measurements of heat flux density applicable to the field of combustion “, Experimental Thermal and Fluid Science, Volume 12, Issue 4, 1996, Pages 452-460, ISSN 0894-1777, https://doi.org/10.1016/0894-1777(95)00136-0.
7. Hamdamov M.M, „Matematicheskoe modelirovanie protsessov v tsilindricheskoy kamere sgorania v programmnom pakete Ansys fluent”, Journal of integrated education and research, ISSN: 2181-3558, 2022.
8. Jodeau M., Chamoun S., Feng J., Cesare G., and Anton J. Schleiss, „Numerical Modeling of turbidity currents with Ansys CFX and Telemac 3D“, E3S Web of Conferences 40, 03014 (2018), https://doi.org/10.1051/e3sconf/20184003014.
9. Kostov P., „Toplo – i masoprenasyane“, ISBN-954-8732-05-X, Sliven, 2004.
10. Kozhuharov I., Dochev Ya, Kostov P., „Toplotehnika“, izdatelstvo „Zemizdat“, Sofia, 1990.
11. Krystev, N., Kostov, P. „Vazmozhnosti za upravlenie na protsesa gorene v ogranichena inzhektirana zavartyana struya“. Toplotehnika, ISSN1314-2550, Izdatelstvo na TU-Varna, 2017
12. Gromova E.N., „Visokotemperaturnie teplotehnologicheskie protsessi i ustanovki: uchebnoe posobie“ /VShTE SPbGUPTD. – SPb., Sankt-Peterburg, 2020g., Chasty 1. – 88 s.
13. Kotovskiy N.V., „Teploperedacha: konspekt lektsiy“, Redaktsionno izdatelyskim sovetom MGTU GA, Moskva, 2015. – 76 s., ISBN 978-5-86311-999-1.
14. Krystev N., „Vliyanie na koefitsienta na izlishak na vazduha na inzhektsionna i ploskoplamachna gorelka varhu emisiite NO_X“, sp.” Mehanika na mashinite”, Izd. na TU-Varna, kn. 62, ISSN 0861-9727, s. 59, 2006.
15. Jiyuan Tu, Guan-Heng Yeoh, Chaoqun Liu, Chapter 2 - CFD Solution Procedure: A Beginning, Computational Fluid Dynamics (Third Edition), Butterworth-

Heinemann, 2018, Pages 33-63, ISBN 9780081011270, https://doi.org/10.1016/B978-0-08-101127-0.00002-7.
16. Tu J., Yeoh G-H., Liu C., Chapter 3 - Governing Equations for CFD: Fundamentals, Computational Fluid Dynamics (Third Edition), Butterworth-Heinemann,2018, Pages 65 - 124, ISBN9780081011270, https://doi.org/10.1016/B978-0-08-101127-0.00003-9.