Analysis Of Hydrocarbon Gas Combustion Processes Through CFD Modeling Based On Theoretical And Experimental Approaches

Savriyev Sh.Sh.
PHD student at BSTU

Introduction. The growing demand for energy resources, the need for environmental safety, and the improvement of fuel efficiency have made the in-depth study of hydrocarbon gas combustion processes a pressing issue. These processes directly determine not only the generation of thermal energy but also the amount of exhaust gases produced. In particular, increasing the efficiency of combustion, gaining a deeper understanding of heat transfer mechanisms, and developing environmentally compliant technologies require a modern scientific and technical approach.

Nowadays, the use of Computational Fluid Dynamics (CFD) methods enables the development of mathematical and visual models of combustion processes, as well as the analysis of temperature distribution, gas flow, mixing phenomena, and chemical reactions. Moreover, when combined with experimental research, CFD modeling results contribute to the reliability and accuracy of the models. In this study, hydrocarbon gas combustion processes are investigated on both theoretical and experimental bases, modeled using CFD software, and the obtained results are analyzed. The findings of the research are of significant practical importance for the design and optimization of energy systems.

Research Methodology. The combustion processes of hydrocarbon gases represent complex physico-chemical phenomena involving heat and mass transfer, various reaction rates, turbulence, and radiation effects. In this study, the theoretical analysis of ideal combustion conditions of hydrocarbon gas mixed with air was conducted, identifying the key parameters for both laminar and turbulent regimes. A two-step reaction model, based on multiple elementary reactions, was employed as the kinetic model of the combustion process.

Experimental investigations were carried out under laboratory conditions. The combustion chamber had a cylindrical shape with an internal diameter of 150 mm and a height of 500 mm. A premixed gas-air mixture was introduced into the chamber. Temperature, pressure, and gas composition were measured using a thermocouple, a manometer, and a gas analyzer, respectively. During the experiments, the stability of the combustion process and the flame structure were observed.

CFD Modeling. The combustion process of hydrocarbon gases was modeled using Computational Fluid Dynamics (CFD) methods in the ANSYS Fluent software. The geometry and mesh were created using the ANSYS Meshing module. The total number of mesh elements was approximately 300,000, and a structured mesh was generated.

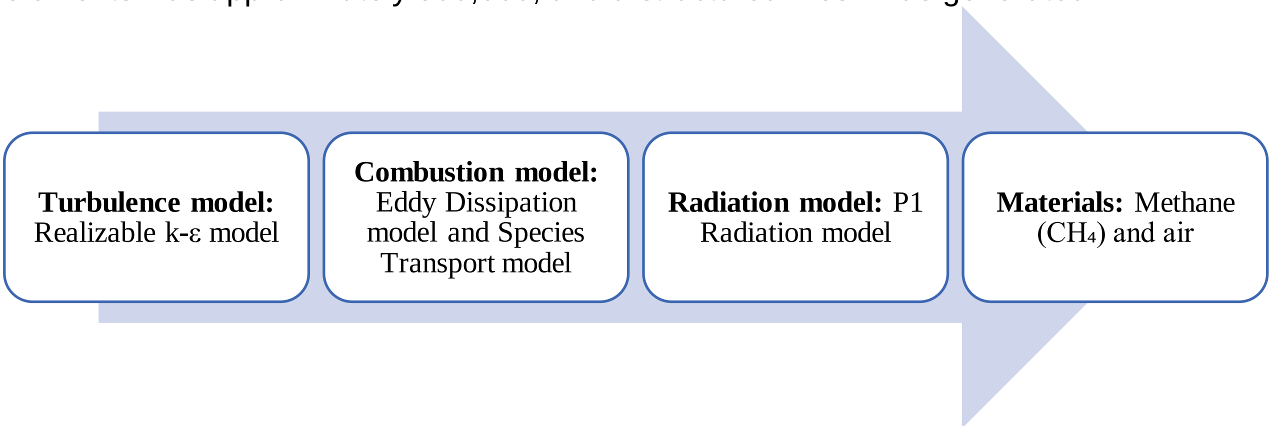

Figure 1. The following physical models were used in the simulation:

Boundary conditions were defined as follows: Inlet flow: Methane-air mixture at 1 atm pressure and 300 K temperature, Outlet: Pressure outlet, Walls: Adiabatic condition.
During the analysis, the following parameters were monitored: temperature distribution, flame structure, concentrations of CO_2 and NOx emissions, and heat flux.

Research Results and Analysis:
CFD Modeling Results: According to the CFD simulation results, the temperature distribution within the combustion chamber reached its maximum along the central axis. Due to the complete combustion of the methane–air mixture, peak temperatures were recorded in the range of 1980–2100 K. The combustion zone formed a symmetric shape, with steep temperature gradients observed at the flame front.
The velocity field distribution showed that high-temperature combustion products rose upward, generating strong convective motion. The maximum velocity was observed at the edges of the combustion zone, reaching approximately 12–15 m/s, indicating stable flame behavior and dynamic gas flow characteristics.
Exhaust Gas Analysis: The main exhaust components — carbon dioxide (CO_2) and nitrogen oxides (NOx) — were closely monitored. Based on the CFD model results:
- The **CO_2 concentration** peaked within the combustion zone, reaching up to **9–10%**.
- The **NOx emissions** increased due to high temperatures and an extended flame length, with concentrations estimated in the range of **~120–150 ppm**.

These values closely matched the experimental measurements, thereby confirming the reliability and accuracy of the CFD modeling results.
Comparison with Experimental Results
The temperature values measured during the experiment corresponded to the CFD simulation results with a deviation of approximately **5–7%**. This discrepancy may be attributed to several factors:
- Incomplete representation of **heat losses through the chamber walls** in the simulation model
- **Uncertainties in the actual composition** of the gas mixture
- **Accuracy limitations** of the measurement instruments

However, the overall trends — including flame shape, locations of maximum temperature, and concentrations of exhaust gases — were complementary and consistent between the CFD simulations and experimental results.

Conclusion. This study investigated the combustion processes of hydrocarbon gases using theoretical, experimental, and Computational Fluid Dynamics (CFD) methods. Through CFD simulation, the distributions of temperature, velocity, and exhaust gases were determined, and their physico-chemical characteristics were thoroughly analyzed.
Comparison with experimental results demonstrated that CFD modeling provides a high degree of accuracy and reliability. In particular, data obtained on flame stability, heat distribution, and exhaust components (CO_2, NOx) confirmed the effectiveness of the modeling approach.
The study yields the following conclusions:
- CFD models enable effective visual and mathematical analysis of hydrocarbon gas combustion processes;
- Optimal combustion conditions for methane-air mixtures were identified, with evaluations of high-temperature zones and exhaust emissions;

- The findings have practical significance for the design of industrial fuel systems, heat exchange devices, and environmentally safe technologies.

Future work plans include reducing high-temperature emissions, improving combustion efficiency, and conducting modeling studies based on new types of fuels such as biogas and hydrogen blends.

REFERENCES

Turns, S.R. An Introduction to Combustion: Concepts and Applications. 3rd Edition, McGraw-Hill, 2011.

Versteeg, H.K., Malalasekera, W. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education, 2007.

ANSYS Fluent Theory Guide, ANSYS Inc., Release 2023 R1.

Glassman, I., Yetter, R.A. Combustion. Academic Press, 4th Edition, 2008.

Poinsot, T., Veynante, D. Theoretical and Numerical Combustion. R.T. Edwards, 2005.

Wilcox, D.C. Turbulence Modeling for CFD. DCW Industries, 3rd Edition, 2006.