**UDC:** 662.9:532.517.4

10.70769/3030-3214.SRT.4.3.2026.11



# CFD MODELING OF THE AEROHYDRODYNAMIC BEHAVIOR OF BIOMASS PYROLYSIS LIQUID AND AIRFLOW IN A COMBUSTION CHAMBER



***Davlonov Khayrulla Allamuratovich***

*PhD, Prof., Karshi State Technical University, Karshi, Uzbekistan*
*E-mail: davlonov80@mail.ru*
*ORCID ID: 0000-0001-7444-9853*
*Science ID: FQD-0825-0058*

***Uzboev Maftun Dusiyarovich***

*Karshi State Technical University, Karshi, Uzbekistan*
*E-mail: maftunuzboev@gmail.com*
*ORCID ID: 0000-0001-7444-9853*
*Science ID: MSN-0726-0094*

***Toshmamatov Bobir Mansurovich***

*PhD, Ass. Prof., Karshi State Technical University, Karshi, Uzbekistan*
*E-mail: bobir.toshmamatov@kstu.uz*
*ORCID ID: 0000-0001-7051-5307*
*Science ID: MSD-0825-0028*

***Abstract.*** *This study presents a numerical investigation of the aerohydrodynamic interaction between a liquid alternative fuel derived from biomass pyrolysis and airflow inside the combustion chamber of a power generation system. A two-dimensional computational model was developed in COMSOL Multiphysics using the finite element method. The turbulent motion of the airflow supplied to the combustion chamber was described using the Navier-Stokes and continuity equations together with a standard turbulence model, whereas the motion of the biomass pyrolysis liquid droplets was represented using a Lagrangian particle-tracking approach. The computational domain was discretized using free triangular elements. The numerical results demonstrated that optimizing the pyrolysis-liquid injection velocity, droplet diameter, inlet-flow ratio, and outlet-channel configuration can enhance mixing intensity, reduce the fraction of droplets impinging on the chamber walls, and facilitate the efficient organization of the fuel combustion process.*

***Keywords:*** *biomass pyrolysis, pyrolysis liquid, combustion chamber, CFD modeling, finite element method, multiphase flow.*

| Received: 21.07.2026 | Accepted: 15.08.2026 | Issue publication: 28.09.2026 |
|---|---|---|

# CFD МОДЕЛИРОВАНИЕ АЭРОГИДРОДИНАМИЧЕСКОГО СОСТОЯНИЯ ПИРОЛИЗНОЙ ЖИДКОСТИ БИОМАССЫ И ВОЗДУШНОГО ПОТОКА В КАМЕРЕ СГОРАНИЯ

***Давлонов Хайрулла Алламуротович***

*PhD, проф. Каршинский государственный технический университет, Карши, Узбекистан*

***Узбоев Мафтун Дусиярович***

*Соискатель, Каршинский государственный технический университет, Карши, Узбекистан*

***Тошмаматов Бобир Мансурович***

*PhD, доц. Каршинский государственный технический университет, Карши, Узбекистан*

***Аннотация.*** *В данной работе выполнено численное моделирование аэрогидродинамического взаимодействия жидкого альтернативного топлива, полученного в результате пиролиза биомассы, и воздушного потока в камере сгорания энергетической установки. Двумерная расчетная модель разработана в программном комплексе COMSOL Multiphysics с использованием метода конечных элементов. Турбулентное движение подаваемого в камеру сгорания воздушного потока описывалось уравнениями Навье-Стокса, неразрывности и стандартной моделью турбулентности, тогда как движение капель пиролизной жидкости - методом отслеживания частиц в лагранжевой постановке. Расчетная область была дискретизирована свободными треугольными элементами. Полученные результаты показали, что оптимизация скорости распыливания пиролизной жидкости, диаметра капель, соотношения входных потоков и конструкции выходных каналов позволяет повысить интенсивность смешения, уменьшить долю капель, осаждающихся на стенках, и обеспечить эффективную организацию процесса сжигания топлива.*
***Ключевые слова:*** *пиролиз биомассы, пиролизная жидкость, камера сгорания, CFD моделирование, метод конечных элементов, многофазное течение.*

# BIOMASSA PIROLIZ SUYUQLIGI VA HAVO OQIMINING YONISH KAMERASIDAGI AEROGIDRODINAMIK HOLATINI CFD MODELLASHTIRISH

***Davlonov Xayrulla Allamuratovich***
*t.f.f.d., prof. Qarshi dalat texnika universiteti, Qarshi, O'zbekiston*

***Uzboyev Maftun Dusiyarovich***
*Tadqiqotchi, Qarshi dalat texnika universiteti, Qarshi, O'zbekiston*

***Toshmamatov Bobir Mansurovich***
*T.f.f.d., dots. Qarshi dalat texnika universiteti, Qarshi, O'zbekiston*

**Annotatsiya.** *Ushbu tadqiqot ishida biomassa pirolizi asosida olingan suyuq muqobil yoqilg'i va havo oqimining energetik qurilmalar yonish kamerasidagi aerogidrodinamik ta'siri sonli modellashtirildi. Tadqiqot jarayonida ikki o'lchamli hisoblash modeli COMSOL Multiphysics dasturida chekli elementlar usuli asosida amalga oshirildi. Yonish kamerasiga uzatilgan havo oqimining turbulent harakati Navye-Stoks, uzluksizlik hamda standart turbulentlik tenglamalari asosida, biomassa piroliz suyuqligi tomchilarining harakati esa Lagranj zarrachalarni kuzatish usuli yordamida tavsiflandi. CFD modellashtirish natijasida olingan natijalar biomassa pirolizi suyuqligini yonish kamerasiga purkash tezligi, tomchi diametri, kirish oqimlari nisbati va chiqish kanallari konstruksiyasini optimallashtirish orqali aralashish intensivligini oshirish, devorga uriladigan tomchilar ulushini kamaytirish, yoqilg'ini yonish jarayonini samarali tashkil etish mumkinligini ko'rsatdi.*
***Kalit so'zlar:*** *biomassa pirolizi, piroliz suyuqligi, yonish kamerasi, CFD modellashtirish, chekli elementlar usuli, ko'p fazali oqim.*

**Introduction.** Currently, due to the depletion of traditional fossil fuel and energy resources in the world, the increasing costs associated with their extraction, as well as the increasing emissions of greenhouse gases and harmful compounds resulting from the combustion of fuels, interest in the creation of energy-saving technologies based on renewable energy sources has increased significantly. Such energy sources include solar, wind, biomass energy and other natural sources [1-4]. In particular, alternative fuels obtained from biomass - gaseous (pyrolysis gas), liquid (bioethanol, biodiesel and pyrolysis liquids) and solid (pyrolysis coal) - are recognized as environmentally friendly, renewable and promising sources for ensuring energy independence. At the same time, the development and use of alternative fuels based on biomass plays an important role in reducing greenhouse gas emissions, reducing the volume of biomass waste, effectively using local energy resources, and strengthening the country's energy security and export potential [5].

(E) ISSN: 3030-3214
Volume 4, № 3
2026

**Methods.** In this research work, the authors studied the aerodynamic and thermal-physical processes in the combustion chamber of pyrolysis gas, pyrolysis liquid, and "water-coal" alternative fuels obtained from biomass pyrolysis.

In the research process, the aerodynamic properties of pyrolysis gas, pyrolysis liquid, and "water-coal" alternative fuels obtained from biomass pyrolysis were studied using computer and CFD modeling methods.

Aerodynamic, thermal-physical and mass transfer processes occurring in the combustion chamber of pyrolysis gas, pyrolysis liquid and "water-coal" alternative fuels obtained from biomass pyrolysis are one of the main factors determining the combustion efficiency, flame stability and heat of fuels. Modern numerical modeling methods are used to study the flow of air supplied to organize the combustion process in the combustion chamber of power plants, the flow of pyrolysis gas, the spraying of pyrolysis liquid and "water-coal" fuels, the evaporation of droplets, the transport of coal particles and their interaction with laminar and turbulent flow.

Analytical methods in the analysis of fuel combustion processes can be used only for simple structures and simplified physical conditions. However, since the processes of spraying alternative fuels, especially "water-coal" alternative fuels obtained from biomass pyrolysis, the formation of multiphase flow, evaporation of droplets, as well as the mixing of alternative fuels with laminar and turbulent air flow are complex and multiphase, analytical methods cannot fully describe such processes. This method allows the determination of the velocity field, pressure pulsation, turbulence, temperature field, combustion process and spatial distribution of component concentrations in the combustion chamber of power devices by numerically solving the Navier-Stokes equations and the laws of conservation of matter and energy. This method analyzes the motion of sprayed liquid alternative fuel droplets, their evaporation, and their interaction with the carrier gas or air flow with high accuracy [6-8].

**Results.** This approach allows for reliable modeling of combustion processes in various phase states (solid, liquid and gaseous) of alternative fuels obtained based on biomass pyrolysis and was chosen as the most effective calculation method for assessing the combustion efficiency of "water-coal fuel" obtained based on biomass pyrolysis in energy devices.

Figure 1 below presents a two-dimensional geometric model of the combustion chamber of the "water-coal" alternative fuel prepared based on biomass pyrolysis products and a finite element mesh diagram of the computational domain.

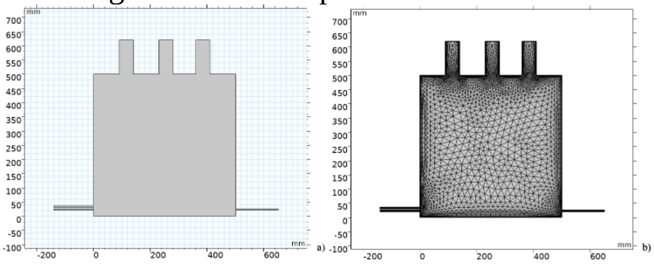

***Fig.1. Computational model of the combustion chamber:***

*a- two-dimensional geometric model of the combustion chamber, b- finite element mesh of the computational domain.*

In Figure 1, the two-dimensional geometric model of the combustion chamber under study was discretized using the finite element method. The computational domain was divided into a mesh using free triangular elements adapted to the dynamics of alternative fuels problems obtained based on biomass pyrolysis using the COMSOL Multiphysics software package. When creating the finite element mesh, the maximum element size was 25 mm, the minimum size was 3 mm, the maximum element size growth factor was 1.35 and the curvature coefficient was 0.3, and the degree of definition of narrow geometric domains was taken as 2. Air and alternative fuels obtained based on biomass pyrolysis are transmitted through channels installed on the lower sides of the combustion chamber. As a result, the fuel-air mixture formed inside the combustion chamber is discharged through three channels installed in the upper part.

The geometric parameters for a two-dimensional model of the combustion chamber, built using the COMSOL Multiphysics software package, are listed in Table 1 below.

To numerically model the aerodynamic and hydrodynamic behavior of the flow inside the combustion chamber, we use the law of conservation of mass and momentum and the k-ε turbulent flow model. We perform the numerical modeling of the process in the time-dependent mode in the following order.

Table 1.

***Geometric parameters of the combustion chamber***

| No | Parameter name | Designation | Unit | Value |
|---|---|---|---|---|
| 1. | Combustion chamber width | A | 500 | mm |
| 2. | Combustion chamber height | H | 500 | mm |
| 3. | Inlet channel width | $A_{in}$ | 150 | mm |
| 4. | Inlet channel height | $H_{out}$ | 10 | mm |
| 5. | Outlet channel width | $A_{in}$ | 50 | mm |
| 6. | Outlet channel height | $H_{out}$ | 120 | mm |

To determine the velocity field, pressure distribution, flow acceleration and direction of the gas flow in the combustion chamber, we use the following equation.

$$\rho\frac{\partial u}{\partial t}+\rho(u\cdot\nabla)u=\nabla\cdot[-pI+K]+F \qquad (1)$$

where, $\rho -$ gas density, $\frac{kg}{m^3}$; $\frac{\partial u}{\partial t} -$ -rate of change of gas flow velocity over time, $\frac{m}{s^2}$ ; $u -$ gas flow velocity vector, *m/s*; $\nabla -$ spatial differential operator, *1/m*; $pI -$ represents the effect of pressure in all coordinate directions, Pa; $K -$ molecular and turbulent viscosity stress tensor $K=(\mu+\mu_T)\cdot[\nabla u+\nabla u^T]$, $\frac{kg}{(m\cdot s^2)}$; $F$ - volumetric force vector acting on the gas, $\frac{kg}{(m^2\cdot s^2)}$; $\mu$ - dynamic viscosity coefficient, $Pa\cdot s$; $\mu_T -$ turbulent dynamic viscosity coefficient $\mu_T=\rho C_\mu\frac{k^2}{\varepsilon}, Pa\cdot s$.

We use the continuity equation to ensure that the substance in the combustion chamber is conserved.

$$\rho\nabla\cdot u=0 \qquad (2)$$

To determine in which parts of the combustion chamber turbulent mixing is strong or weak, we use the following turbulent kinetic energy equation.

$$\rho\frac{\partial k}{\partial t}+\rho(u\cdot\nabla)k=\nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_k}\right)\nabla k\right]+Q_k-\rho\varepsilon \qquad (3)$$

where, k- turbulent kinetic energy, $m^2/s^2$; $\sigma_k -$ diffusion coefficient of turbulent kinetic energy; $\nabla k -$ turbulent kinetic energy gradient, $m/s^2$; $P_k -$ turbulent kinetic energy generation capacity $Q_k=\mu_T:[\nabla u:(\nabla u+(\nabla u)^T)], W/m^3$;$\varepsilon -$ turbulent kinetic energy dissipation rate, $m^2/s^3$.

We can determine how quickly turbulent vortices in a combustion chamber decay and how much turbulent energy is lost using the turbulent dissipation equation. This equation allows us to determine the rate ε at which turbulent kinetic energy is converted into heat due to molecular friction.

$$\rho\frac{\partial\varepsilon}{\partial t}+\rho(u\cdot\nabla)\varepsilon=\nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_\varepsilon}\right)\nabla_\varepsilon\right]+C_{\varepsilon 1}\frac{\varepsilon}{k}Q_k-C_{\varepsilon 2}\rho\left(\frac{\varepsilon^2}{k}\right) \qquad (4)$$

To solve differential equations 1-4 by numerical modeling, we determine the special solutions and boundary conditions at the inlet, outlet and wall boundaries of the combustion chamber. We perform the calculations based on the finite element method of the time-dependent behavior COMSOL Multiphysics software package.

We enter the values of the gas acquisition program $u=u_{in}$, turbulent kinetic energy $k=k_{in}$ and dissipation process $\varepsilon=\varepsilon_{in}$ in the inlet channels into the initial parameters, assume the sliding condition with the combustion chamber and take the gas to be equal to zero (u=0).

We conduct a numerical experiment to verify the alternative processing and forward operation of the combustion chamber based on biomass pyrolysis and analyze the aerodynamic behavior of the chamber only for air capacity. In the numerical process, we assume the air flow in the inlet channels to be 1 m/s. The numerical modeling process determined the air velocity module and the relative pressure distribution in the combustion chamber and is presented in Figure 2.

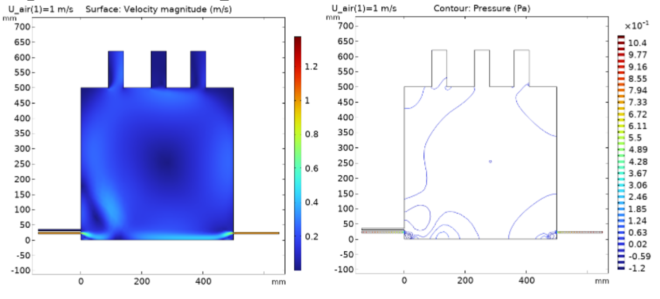


***Fig.2. Aerodynamic behavior in the combustion chamber when the inlet air flow velocity is 1 m/s.***

Figure 2 shows the velocity and pressure distribution of the air flow along the combustion chamber. We can see that the largest values of the air flow were observed in the inlet channels and in their adjacent parts with the combustion chamber. As the air flow entered the combustion chamber, its velocity decreased and was distributed throughout the combustion chamber. In most of the main volume of the combustion chamber, the flow velocity moved in the range of 0.1-0.5 m/s.

(E) ISSN: 3030-3214
Volume 4, № 3
2026

Relatively high velocity zones were observed in the lower part of the chamber, especially near the inlet channels. The velocity values of the high combustion products in the outlet channels were different, which affected the uneven distribution of the air flow along the three channels. Figure 2 also shows the distribution of relative pressure inside the combustion chamber, and we can see that the relative pressure varied in the range of 0.12-1.04 Pa. The pressure values in the internal volume of the combustion chamber were relatively evenly distributed.

The preliminary results presented in Figures 1 and 2 show that the velocity and pressure fields of the air flow in the combustion chamber can be numerically determined based on the adopted geometric model, computational grid, initial and boundary conditions. To assess the numerical reliability of the model, the convergence of the solution, the balance of mass flows at the inlet and outlet, and the independence of the results from the computational grid density were additionally checked.

When liquid pyrolysis fuel obtained from biomass pyrolysis is sprayed into the combustion chamber in separate channels with air, the air flow velocity, fuel droplet motion trajectories, and relative pressure distribution in the combustion chamber were numerically modeled using the COMSOL Multiphysics software package, and the results are presented in Figures 3 and 4.

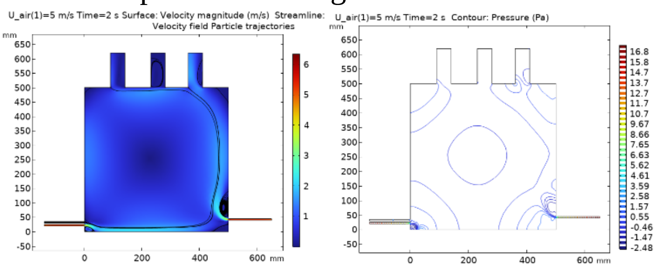


***Fig.3. Aero-hydrodynamic motion in the combustion chamber when the inlet air flow and pyrolysis fluid velocity are 5 m/s.***

Figures 3-4 show the air flow and pyrolysis fluid velocity modulus, flow lines, and relative pressure distribution inside the combustion chamber. The results of numerical modeling using the COMSOL Multiphysics software package show that a circulation flow is formed that completely occupies the working volume of the combustion chamber. The formation of a closed contour of the flow lines leads to the recirculation of the air supplied from both sides of the combustion chamber inside the chamber and an increase in its hydrodynamic residence time. However, this state of air circulation creates favorable aerodynamic conditions for increasing the duration of mixing of pyrolysis fluid droplets obtained from biomass pyrolysis with the air flow, improving the distribution of pyrolysis fluid droplets throughout the entire volume of the combustion chamber, and accelerating the subsequent evaporation and combustion processes.

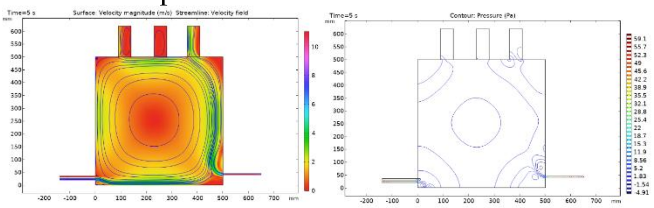


***Fig.4. Aero-hydrodynamic motion in the combustion chamber when the inlet air flow and pyrolysis fluid velocity are 10 m/s.***

High flow velocity values were observed near the inlet channels, the bottom of the combustion chamber, the right-side wall, and the outlet channels. In the center of the combustion chamber, the flow velocity is relatively small, forming a circulation core, and the air flow on the right side is directed upwards along the chamber wall, the combustion chamber turns in the upper part and returns to the central area. As a result, a vortex is formed in the combustion chamber, which enhances the mixing of the pyrolysis liquid and air.

As can be seen in Figure 4, the flow intensity in the three outlet channels located at the top of the combustion chamber is different. The main part of the flow lines is directed towards the outlet channel on the right, which affects the uneven distribution of the temperature field in the combustion chamber, therefore, it is necessary to determine the optimal location of the outlet channels.

When the pyrolysis liquid is sprayed from the inlet channel located on the left side of the combustion chamber, the initial hydrodynamic motion of the droplets is determined by the spray velocity and angle. The aerodynamic drag force of the air flow affects the motion of the droplets in the combustion chamber. As a result, small-diameter liquid droplets adapt to the air flow faster and spread throughout the entire volume of the combustion chamber along the circulating flow. Large droplets, on the other hand, retain their initial direction longer due to their large inertia and are more likely to hit the combustion chamber wall. Therefore, the effective dispersion of droplets when the pyrolysis liquid is transferred to the combustion chamber depends not only on the flow area, but also on the droplet diameter, spray velocity, spray angle, pyrolysis liquid density, temperature and viscosity.

**Conclusions.** The results obtained in the research work show that a stable circulation motion is formed in the combustion chamber, which increases the residence time and mixing intensity of the pyrolysis liquid and air mixture obtained from biomass pyrolysis. However, the shift of the flow in the combustion chamber to the right and its uneven distribution along the outlet channels indicate the need for additional optimization of the combustion chamber design and inlet flow parameters. The optimal mode should be selected based on the fact that liquid droplets and air flow occupy the entire volume of the combustion chamber, reducing the proportion of droplets hitting the side walls, equalizing the mass flow in the outlet channels, and limiting pressure loss.

## REFERENCES

[1] Узаков, Г. Н., Давланов, Х. А., & Тошмаматов, Б. М. (2021). Энергоэффективные системы и технологии с использованием альтернативных источников энергии. Muqobil energetika, 1, 7-20.
[2] Toshmamatov, B. M., Uzboev, M. D., & Mamatova, M. (2025). Analysis of thermal and technical parameters of alternative fuels obtained from biomass pyrolysis. American Journal of Applied Science and Technology, 5(12), 161-166.
[3] Kuznetsov, G. V., Malyshev, D. Yu., Kostoreva, Zh. A., Syrodoy, S. V., & Gutareva, N. Yu. (2020). The ignition of bio-water-coal fuel particles based on coals of different degrees of metamorphism. Energy, 201, Article 117701.
[4] Polyakovs, A., et al. (2020). Preparation of coal-water slurry using a high-speed mixer-disperser. Environment and Environmental Effects, 8, 77-81.
[5] Toshmamatov, B. M., Uzboev, M. D., & Toshboev, A. R. (2025). FTIR analysis of coal-water alternative fuel. Muqobil energetika, 4(21), 45-48.
[6] Toshmamatov, B. M., Uzboev, M. D., & Mamatova, M. (2025). Development of a method for obtaining energy-efficient water-coal fuel based on biomass pyrolysis products. Muqobil energetika, 4(21), 38-44.

**Maqolaga iqtibos keltirish | Как цитировать статью | How to cite this article**

Davlonov, K. A., Uzboev, M. D., & Toshmamatov, B. M. (2026). CFD modeling of the aerohydrodynamic behavior of biomass pyrolysis liquid and airflow in a combustion chamber. Digital Technologies in Industry, 4(3). https://doi.org/10.70769/3030-3214.SRT.4.3.2026.11