Article citation info:
Dopieralski M, Polak F. Cold flow modeling of the combustion chamber for a multi-fuel turbine engine. Combustion Engines. 2026;204(1):3-12. https://doi.org/10.19206/CE-208516

Marcin DOPIERALSKI
Filip POLAK

Combustion Engines
Polish Scientific Society of Combustion Engines

Cold flow modeling of the combustion chamber for a multi-fuel turbine engine

ARTICLE INFO

Received: 22 May 2025
Revised: 30 June 2025
Accepted: 22 July 2025
Available online: 22 September 2025

This article analyses the possibility of using a multi-fuel turbine engine as a primary source in hybrid systems. Turbine engines, thanks to their high efficiency and flexibility in using different types of fuels, can play a key role in integration with hybrid systems that combine electric drive with traditional energy sources. Additionally, different fuels may influence combustion temperatures and soot formation, affecting thermal loads and material degradation. The proposed methodology will consist of conduct a comprehensive review of current models of combustion chambers in turbine engines, focusing on multi-fuel capability and hybrid applications. As a result, it will allow to identify key performance parameters (efficiency, emissions, stability, etc.) and define system requirements for hybrid optimization. The article presents a comparative analysis of which geometry performs best for multi-fuel combustion. The model results were then compared with literature data. The conducted modelling of a multi-fuel combustion chamber intended for use in hybrid turbine systems has shown that the choice of fuel significantly influences combustion behaviour, temperature distribution, and emission profiles. Nevertheless, the developed model provides a solid foundation for future integration into hybrid propulsion architectures, offering adaptability to various fuels and operating regimes.

Key words: *turbine engine, multi-fuel engine, hydrogen, natural gas, hybrid drive*

This is an open access article under the CC BY license (http://creativecommons.org/licenses/by/4.0/)

1. Introduction

The global demand for sustainable and low-emission propulsion systems has intensified the search for alternative fuels and advanced combustion technologies. Gas turbine engines, widely used in power generation and aviation, face increasing pressure to adapt to cleaner energy sources while maintaining high performance and operational reliability. In this context, the combustion of alternative fuels, such as hydrogen, synthetic gases, and biofuels, presents both a promising solution and a complex engineering challenge. The paper [4] describes the challenges of adding hydrogen power to gas turbines, including high production costs, limited infrastructure, and an underdeveloped value chain. All major gas turbine companies are aiming for full hydrogen combustion capability by 2030, and some manufacturers, such as Siemens Energy, are already introducing selected hydrogen-capable models. Technical challenges include the combustion characteristics of hydrogen, in particular, lower ignition energy and higher flame speed [7].

Sample tests have shown that biofuels can offer similar or even better performance compared to conventional fuels, although some biofuels may require modifications to the injectors or combustion chamber. In terms of cost-effectiveness, biofuels and synthetic fuels may be more expensive to produce, but the environmental benefits and potential savings over the long term may make them cost-effective. The use of biofuels, such as biodiesel, in turbine engines can lead to comparable or even better performance compared to conventional fuels. However, some biofuels, especially those with high viscosity, may require modifications to the injectors or combustion chamber to ensure optimal combustion and avoid operational problems [25]. In some cases, especially in aviation, multi-fuel operation allows the engine to be adapted to different weather conditions or missions, which can be important for military operations or specialist missions.

In recent years, research on combustion chambers in turbine engines has focused on optimizing the combustion process to ensure better fuel and air mixing, leading to higher combustion efficiency and lower emissions of harmful chemicals [20]. In the work [20] an analysis of the combustion chamber of a micro gas turbine engine was presented, which can operate on various fuels such as methane, natural gas, and ethanol. The research showed that the appropriate configuration of the outlet holes and the angles of the rotor blades significantly affect the performance of the combustion chamber. High combustion efficiency (over 98% for methane and natural gas) and low emissions of pollutants, such as nitrogen oxides, were key outcomes of this research. In other work, Pasalkar [15] focused on reducing nitrogen oxide (NO_x) emissions in jet engine combustion chambers. By changing design parameters and using computational fluid dynamics (CFD) simulations, researchers significantly reduced NO_x emissions, which is crucial for the sustainable development of fuel technologies [15]. Another article [1] describes the effects of hydrogen-enriched biogas on combustion and emission of a dual-fuel diesel engine. The conclusions of the analysis showed that the addition of hydrogen to biogas significantly reduced carbon dioxide emissions. At 20% H_2 content in biogas, a reduction in CO_2 emissions of up to 56% was observed compared to running the engine on biogas alone.

In the context of hybrid systems, modelling a turbine engine will allow for mapping its operating conditions. Such systems operate in diesel-electric generators or gas-electric generators. Such an application is easier to implement because the turbine operates at a constant speed under different loads, which may suggest that certain operating parameters are much easier to predict. An example of using a turbine as a power generator is shown in Fig. 1. Such an application works as a serial hybrid propulsion called Rex [8]. Modelling a multi-fuel combustion chamber allows for flexible adaptation to different energy sources, increasing practical application possibilities. With advanced simulation tools, such as Ansys Fluent and Creo-6.0, researchers can accurately analyse the flow of fuel and air, leading to the optimization of the combustion process and the improvement of hybrid systems' efficiency.

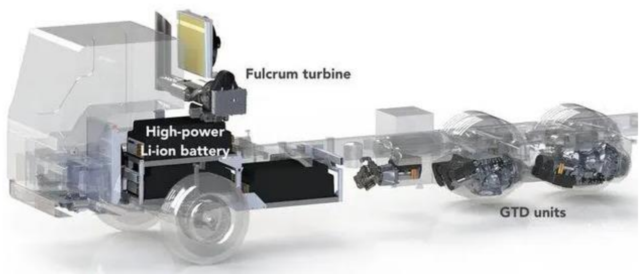

Fig. 1. Application of a turbine as a generator in heavy goods vehicles [14]

2. Research problem

The article describes combustion processes occurring in the chamber of turbine engines. By selecting, among others, the appropriate shape of the combustion chamber, the amount of fuel in the injector and chemical additives, the combustion process can be controlled. The types of combustion are presented: diffusion and laminar, together with examples of their occurrence. Then, the formation of pollutants and their types are described. The emissions of NO_x, CO_2, and particulate matter by gas turbines and reciprocating engines were compared. Ways to reduce gas emissions were proposed, among others, by rearranging the aerodynamics of the combustion chamber, increasing the volume of the primary flame zone, or improving fuel atomization.

The next step is to describe the fuels used to power gas turbines. The main problems of the combustion process are described, including maintaining the initiation of combustion in a very stable flow, or a lack of stability [22]. For example, blends of biofuels with traditional aviation fuels can reduce particulate and CO_2 emissions without significant loss of efficiency. Specific blends, such as bioethanol with jet fuel, have shown a 15% reduction in CO_2 emissions and improved lubricity. The use of hydrogen as a fuel in gas turbines reduces CO_2 emissions by 100% and NO_x by 90% compared to conventional fuels [21]. Additionally, hydrogen as a fuel improves the efficiency of turbines at high temperatures. Other studies have shown that biogas can be effectively used in gas turbines, which allows for a 20–30% reduction in CO_2 emissions. Mixing biogas with natural gas does not significantly affect engine efficiency [16]. The use of gas turbines, for example, as generators or in industry, is discussed. Turbine engines can be used in hybrid motor vehicles as a generator to charge batteries, which allows for increased range and energy efficiency of vehicles.

The following section describes the research stand used for the simulation tests and specifies the operation of the stand, specifying the individual components of the system.

In the research part of the work, the combustion chamber of the tested engine was first modelled in the Ansys Fluent simulation environment. It detailed the calculation of boundary conditions and preparation of the model for the needs of the analysis. The next part included the results of simulation studies of the flow of the working medium through the combustion chamber, taking into account the selection of an appropriate volumetric mesh for the chamber and the simulation of cold air flow. Finally, the results of the simulation were compared with their analysis. The obtained assessment of the numerical analysis allowed for undertaking design changes in the future for the combustion chamber and highlighted further development possibilities of the turbine engine for further research and simulation. The numerical calculations performed using the models and mechanisms available in the Ansys Fluent program can be a starting point for the modernization of the existing research engine. The author will focus on the analysis of the flow through the above-mentioned combustion chamber in order to verify the design solution selected by the team building the turbine engine.

In diffusion combustion, fuel and air mix immediately prior to ignition, and the mixing process continues even after combustion has begun. This type of combustion occurs within the boundary layer where the fuel gas stream meets the still oxidizing environment. A classic example is a candle flame, where a visible glowing zone forms at the point where the amount of fuel and oxidizer is in stoichiometric balance. Depending on the nature of the gas flow, diffusion combustion can be categorized as either laminar or turbulent.

Laminar combustion is represented as molecular. Laminar flame velocity is defined as the propagation rate of the normal flame front relative to the unburned mixture (homogeneous combustible mixture). This is an important property for a mixed flame because it contains basic information about the diffusivity and exothermicity of the combustible hydrocarbon mixture. At a practical level, laminar flame velocity is related to the combustion rate in the chamber, which can affect combustion efficiency and exhaust emissions. Laminar flame speed values can be applied directly in turbulent combustion modelling or used indirectly to validate chemical kinetic models.

The theory shows that the creation of swirl and an internal recirculation zone significantly improves the flame stability in the combustion chamber. On the other hand, the increase in the number of inert gases causes a decrease in the laminar velocity, which causes a deterioration in the flame stability. There are many other models for describing turbulent combustion. Currently, the most commonly used model is the Launder and Spalding model, i.e., the model k-ε, where k denotes the kinetic energy, while the rate of dissipation of kinetic energy. They are described by a two-equation model - the first equation of kinetic energy transport, the second of dissipation. Turbulent viscosity in this model is expressed by the formula ε–ε:

$$u_t = C_\mu * \rho * \frac{k^2}{\varepsilon} \tag{1}$$

where C_μ in the formula means constant, while ρ – density.

The dissipation energy is equal to the energy of large-scale motion, which later transforms into motion with energy of smaller and smaller scale. For the level of fibbers of the smallest scale (Kolmogorov), the energy eventually dissipates (spreads). Therefore, the dissipation energy is described by the ratio: ρ

$$\varepsilon \sim \frac{u'^3}{l} = \frac{k^{3/2}}{l} \quad (2)$$

where, l – macroscale of turbulence, u' – speed pulsation.

Stability is a key characteristic of the combustion process in turbine engine chambers. Flame stability depends on two main factors: resistance to flame blow-off and resistance to flashback. In combustion chambers, flame blow-off is the primary cause of instability. This occurs when the flow velocity exceeds the flame propagation speed and there is no mechanism in place to stabilize the flame. Stabilization can be achieved by placing an obstruction in the airflow or by generating a swirling motion in the air. This swirl can be produced either by adjusting the air nozzles or by incorporating a swirler into the airstream.

In the exhaust gases, we can find unwanted pollutants that are created in the combustion chamber. There are four main pollutants that occur in the largest quantities [5]:

– unburned hydrocarbons (unburned fuel)
– smoke (carbon particles)
– carbon monoxide
– nitrogen oxides.

Gas turbines emit significantly lower levels of NO_x, CO_2, particulates, and other pollutants compared to reciprocating engines. This difference is due to the distinct combustion methods used: internal combustion engines produce power through thousands of high-temperature explosions within the cylinders, whereas gas turbines operate with a continuous combustion process that maintains a lower and more consistent temperature profile.

To substantially lower CO_2 emissions, achieving maximum net efficiency is essential, as greater efficiency leads to lower specific CO_2 emissions measured in grams per kilowatt-hour (kWh) of energy produced. For illustration, Fig. 2 presents data on the concentration of harmful compounds measured from a turboprop engine.

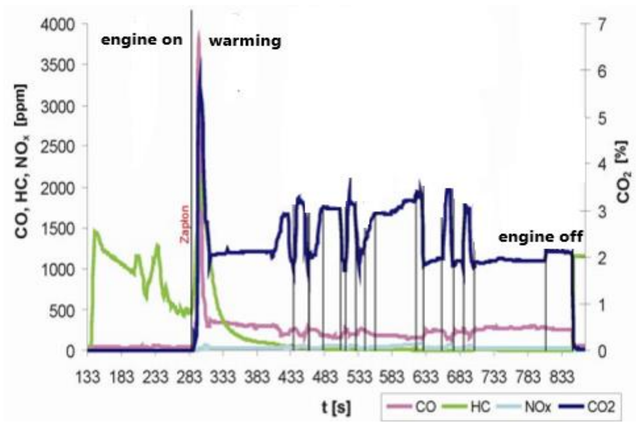

Fig. 2. Content of chemical compounds during measurement of exhaust gas composition concentration in the time domain [13]

When the engine is started, a sharp increase in the concentration of hydrocarbons can be seen, associated with the delivery of the mixture to the combustion chamber. With the initiation of ignition, there is an increase in the amount of carbon monoxide compounds, carbon dioxide, as well as hydrocarbons. The concentrations of the aforementioned compounds decrease in the further stage of engine warm-up [13].

In conditions of optimal excess air coefficient (in the starting range) and high combustion temperature, the greatest amount of NO_x occurs. One way to reduce them is to use a recirculation zone in the combustion chamber (Fig. 3). Its task is to stabilize the flame, lower its temperature, and separate it from the flame tube, which is the internal part of the combustion chamber [5]. The fuel burns with high intensity in a stream of strongly swirled air. The use of lean kinetic flames reduces NO_x emissions compared to diffusion flames (flame temperature exceeds 1900°C). Thanks to the presence of turbulence zones in the combustion chamber, the combustion process can be maintained within wide limits of pressure changes, flow velocity, and mixture composition. Another concept aimed at reducing nitrogen oxides is the use of catalytic combustion. It involves the use of catalysts to accelerate the chemical reaction and initiate the combustion process at very low temperatures, which radically reduces NO_x emissions.

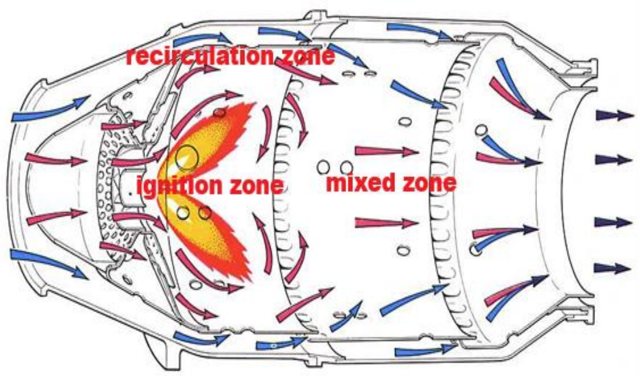

Fig. 3. Individually separated zones in the combustion chamber of a turbine engine [17]

All changes in the combustion chamber must be the result of a compromise between changes in combustion parameters and permissible emission of toxic components. It should be remembered that current standards require a reduction in the components of incomplete combustion – carbon monoxide and unburned hydrocarbons – both fuels ejected from the combustion chamber in the form of droplets or vapours, and products of partial decomposition of hydrocarbons into hydrocarbons with a lower molecular weight.

The most common way of classifying fuels is to divide them into gaseous and liquid fuels, and within gaseous fuels, they can be distinguished according to their calorific value. Table 1 presents the classification of fuels, which is divided into natural gas (mainly methane with small amounts of volatile hydrocarbons and inert gases), and high calorific value gases.

They consist of volatile hydrocarbons with small fractions of inert gases, which are usually very clean and work well in gas turbines. This can be propane, butane, or a mixture of them. They often contain some hydrogen and are usually available as by-products of refineries. Additionally, there are medium calorific value gases. These fuels contain methane additives with a high content of inert substances

(CO_2, N_2), or they are also processing gases or gasifier coal. The latter, i.e., low calorific value gases, contain carbon monoxide and hydrogen diluted with inert components, namely nitrogen and carbon dioxide. They come from the chemical, oil and gas, or steel sectors; many of these fuels cannot be transported or stored, and their main advantage is to reduce the fuel supply in industrial plants in a carbon-constrained environment [11].

Table 1. Classification of fuels for turbine engines [11]

Fuel category	Typical composition	Lower Heating Value [kJ/Nm³]	Typical specific fuels
Ultra/Low LHV gaseous fuels	$H_2 < 10\%$ $CH_4 < 10\%$ $N_2 + CO_2 > 40\%$	< 11,200 (< 300)	Blast furnace gas (BFG), air blown IGCC, biomass gasification
High hydrogen gaseous fuels	$H_2 > 50\%$ $CH_4 = 0–40\%$	5,500–11,200 (150–300)	Refinery gas, petrochemical gas, hydrogen power
Medium LHV gaseous fuels	$CH_4 = 10–50\%$ $N_2 + CO_2 = 30–50\%$ $H_2 = 10–50\%$	11,200–30,000	Weak natural gas, landfill gas, coke oven gas, corex gas
Natural gas	$CH_4 \approx 90\%$ $C_2H_6 \approx 5\%$ Inert ≈ 5%	30,000–45,000	Natural gas, liquefied natural gas
High LHV gaseous fuels	CH_4 and higher hydrocarbons $C_3H_8 > 10\%$	45,000–190,000	Liquid petroleum gas (butane, propane), refinery off-gas
Liquid fuels	C_nH_m with $n > 6$	32,000–45,000	Diesel oil, naphtha crude oils, residual oils, bio-liquids

Hydrogen's relatively low power density compared to other fuels like gasoline or natural gas underscores the difficulties in using it as an energy source. Moreover, hydrogen can cause embrittlement in certain materials, potentially compromising the safety and durability of storage and transport systems, making material compatibility a crucial consideration. When used in fuel cells to produce electricity, hydrogen also suffers energy losses during conversion, which further decreases the overall power density of hydrogen-based systems [2]. Another significant challenge is hydrogen’s broad flammability range spanning from 4% to 75% by volume in air, which makes it more prone to ignition than fuels with narrower flammability limits. Additionally, its low ignition energy means it can be ignited by minimal energy sources, such as sparks or hot surfaces. As a result, minimizing ignition sources is critical to safety when working with hydrogen. Finally, the shock wave created by a hydrogen detonation can propagate at very high speeds, causing significant damage to structures and equipment. Therefore, managing and mitigating the risk of detonation is essential in any hydrogen project [2]. The high temperatures and pressures associated with hydrogen combustion can cause wear and damage to engine components. In addition, hydrogen's reactivity poses a risk of corrosion, and its low density creates challenges in fuel injection systems. To address these challenges, special materials are used in engine components exposed to high temperatures and pressures. In addition, corrosion-resistant materials are used in components exposed to hydrogen to mitigate the potential effects of corrosion.

In article [22], a novel concept for a gas turbine engine utilizing pressure gain combustion (PGC) was introduced. This design effectively addressed challenges related to the precise timing of combustion chamber opening and closing. The proposed valve timing system optimized gas flow, enhancing the conversion of high-pressure gas into mechanical energy. The use of rotary combustion chambers allowed for an efficient sealing solution. Notably, the design stands out for its simplicity and potentially low power-to-weight ratio [22]. CFD simulations demonstrated high efficiency and low specific fuel consumption, highlighting the promise of this hybrid gas turbine engine. It achieved a notable energy efficiency of 37% and a specific fuel consumption of just 219.9 g/kWh, all while maintaining a potentially low power-to-weight ratio. The engine's straightforward construction could reduce manufacturing costs compared to traditional engines with isobaric combustion. It incorporates elements common in piston engines, such as fuel injection systems and turbochargers, but eliminates the need for a crankshaft. Additionally, the use of an advanced ceramic sealing system in the rotating combustion chambers further enhances performance [22].

In gas turbine power systems, including hybrid fuel cell plants, transitioning from conventional fuels to pure hydrogen or hydrogen-natural gas blends is a critical development. However, this shift presents several challenges, such as the risk of flashback, acoustic combustion instabilities, higher temperatures on smoke tube walls, and, in some instances, increased nitrogen oxide emissions. The study in [20] focuses on enhancing the efficiency of gas turbine power systems by utilizing pure hydrogen and hydrogen-natural gas mixtures as fuels. The paper examines the operational setup of both premixed combustion chambers and chambers with sequential injection of eco-friendly, energy-efficient steam, specifically for Aquarius-type power plants. The research evaluates the key aerodynamic and energy characteristics of combustion chambers fuelled with hydrogen-containing gases, using conservation and transport equations within a multicomponent reactive system. A four-step chemical reaction model for burning the hydrogen-natural gas mixture was applied, enabling the calculation of optimal parameters for environmentally sustainable combustion systems. The premixed combustion chamber can be recommended only for working with natural gas-hydrogen mixtures with a hydrogen content not exceeding 20% (by volume). An increase in the hydrogen content leads to the formation of flashback zones and fuel combustion inside the swirler channels (Fig. 4). In the case of the combustion chamber of the Vodoley combined-cycle power plant, when operating on pure hydrogen, there are no flashback zones.

In study [17], the impact of hydrogen co-combustion on aircraft engine performance and emissions was examined. Researchers utilized zero-dimensional models of the JetCat P140 RXI and DGEN 380 engines, developed using the GSP (Gas Turbine Simulation Program). Combustion simulations in GSP rely on a real gas model and the NASA Chemical Equilibrium Applications (CEA) equations. The study evaluated engine performance using Jet A-1 fuel as well as blends containing hydrogen or methane. Simulations were conducted both at ground-level design conditions and during flight at selected altitudes and speeds. Results showed that as the proportion of gas in the fuel mixture increased, there was a slight rise in both thrust and turbine outlet temperature, while specific fuel consumption decreased due to the higher energy content of hydrogen and methane [17]. The performance of JetCat and DGEN 380 engines was calculated for kerosene mixtures with methane or hydrogen. This knowledge will be used to convert these engines to gas fuels. When it comes to fuels and emissions, GSP has limitations related to the set of available chemicals and the zero-dimensional combustion chamber model.

a) b)

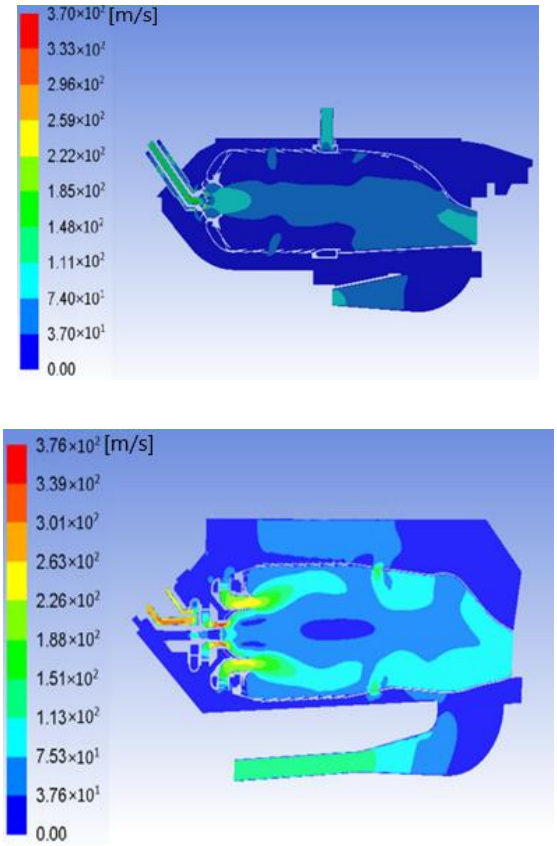

Fig. 4. Contours of velocity values [m/s] inside the combustion chamber with premixing of components (a) and with steam injection (b) [19]

Generalized Spray Combustion (GSC) modelling was also used to predict the performance of JetCat and DGEN engines fuelled with kerosene-methane or hydrogen blends. As the gas content in the fuel increased, both thrust and exhaust gas temperature rose slightly, while specific fuel consumption decreased due to the higher energy content of hydrogen and methane [17]. These results demonstrate the impact of hydrogen co-combustion on engine performance, which is crucial for planning experiments and redesigning fuel systems. Basic aircraft missions were simulated to compare fuel consumption under various conditions. However, accurately modelling off-design operating conditions requires detailed compressor and turbine characterization along with model validation. Additionally, emission predictions necessitate a multireactor approach with the combustion chamber divided into separate zones.

This paper [12] introduces a new mathematical model that incorporates the effects of fuel chemistry on the combustion process in turbine engines. The model was initially validated using tests on the Minijet Rig bench. It can be applied to analyse how various fuel components impact combustion, specifically mixtures of Jet A1 with synthetic paraffinic hydrocarbons C15 and C17 blends added at 10% concentration, and C8 and C11 blends added at 10%. The findings indicated that the variation in the reactivity coefficient (α88/39) between the two fuel types aligns with the experimentally observed differences in combustion behaviour.

3. Description of the tested system and research methodology

The work began with a thorough analysis of the existing literature on combustion chambers in multi-fuel engines and hybrid systems. We focused on research related to combustion mechanisms, fuel efficiency, and emission levels under various operating conditions. The key problems identified were: optimizing the engine's fuel efficiency, reducing harmful emissions, and improving combustion stability when using different fuels (e.g., natural gas, diesel, biofuels).

The first stage was to develop a preliminary design of a test stand for turbine engines based on a radial compressor. The design was created so that in the future it would be possible to conduct analyses of flows in the combustion chamber and to study the concentration of gaseous exhaust components when powered by different fuels. This will allow for the selection of the best fuel supply on the dynamometer, as well as the possibility of powering full-size structures with it. When building a test stand for a radial engine based on a car turbocharger, it was necessary to initially determine the requirements on which the design would be based: selection of an appropriate compressor, design of the combustion chamber, testing of designed elements in numerical analysis programs, design of systems cooperating with the engine, assembly of finished elements and systems on the base. Fig. 5 shows what the stand consists of: housing, wiring, control elements, pressure sensors, oil, control buttons, turbine, combustion chamber, injector, intake duct, ignition system, and fuel (gas).

In order to correctly model the chamber, it was necessary to first determine the values needed to assign boundary conditions. In order to correctly design the combustion chamber, it was necessary to calculate and determine the parameters of the working medium at its inlet. The input data for thermo-gas-dynamic calculations were selected based on the compressor characteristics available on its manufacturer's website.

The next step was to utilize Computational Fluid Dynamics (CFD) software to simulate the combustion process within the chamber. A three-dimensional geometric model of the chamber was created, taking into account all significant physical and chemical parameters. The simulations allowed us to analyse cold flow and heat transfer under various operating conditions.

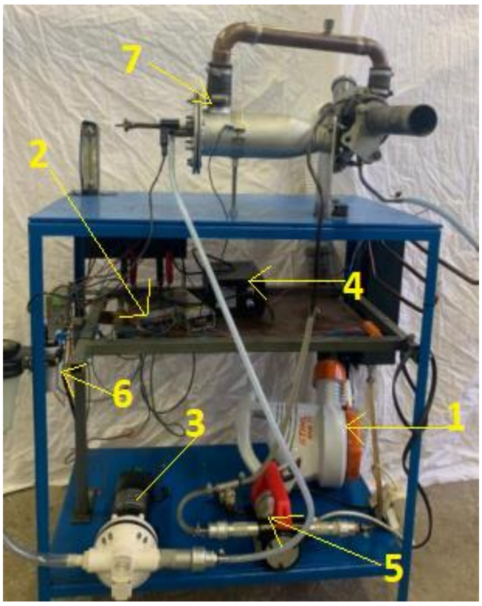

Fig. 5. Miniature jet engine station; 1 – air blower, 2 – control system, 3 – fuel pump, 4 – control display, 5 – oil pump, 6 – ignition system, 7 – turbine engine

One of the most important points in the preparation of a combustion simulation is the selection of a reaction kinetics model. It depends primarily on the fuel used and the available computing power. The article [18] presents simulations of propane combustion based on a one-equation combustion model. Another paper using propane as a fuel uses a five-equation combustion model that includes the following reactions: oxidation of propane, oxidation of carbon monoxide, oxidation of hydrogen, and conversion of carbon monoxide with water vapor [26]. In the articles [9,26] simulations were made using the one-equation Westbrook-Dryer model. The Ansys Fluent program allows simulation of kinetic and diffusion combustion. Depending on the selected model, the approach differs from the point of view of the „solver" according to the division in the user manual [3].

4. Assumptions for numerical modelling

After the first engine start-up, it turned out that the compressor was working at a point other than the one originally selected for calculations. The compression ratio was 2.5, while the rotational speed was about 168,000 rpm. The mass air flow rate measured at the engine inlet was 0.11 kg/s. The actual operating point was plotted on the compressor characteristic, which is shown in Fig. 6.

The photos of the stand shown in Fig. 5 show that the air inlet to the combustion chamber is connected to the air outlet from the compressor diffuser by a copper connector. At first glance, it seems to cause total pressure losses. Consequently, assuming the air parameters at the inlet to the combustion chamber as equal to those at the compressor outlet seems to be too much of an oversimplification. According to the author of this paper, first performing simple simulations of the flow through the aforementioned connector will allow us to estimate, with an acceptable degree of uncertainty, the thermodynamic parameters of the air flowing into the combustion chamber.

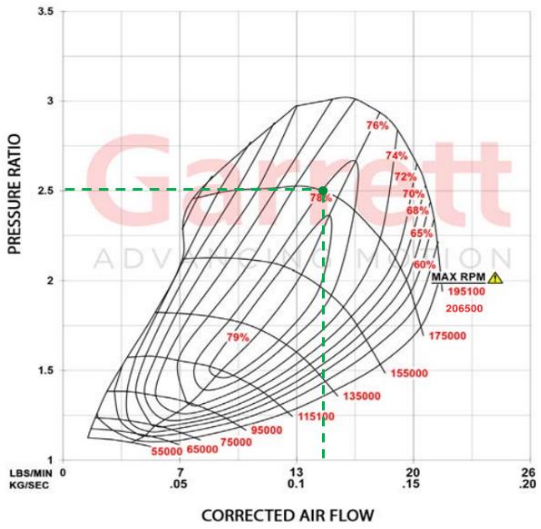

Fig. 6. Selected point of the turbocharger operating characteristic [24]

The study began with the discretization of the computational domain of the copper connector, which links the air inlet to the combustion chamber with the outlet from the compressor diffuser. The size and quality of the computational mesh significantly influence the effectiveness of numerical methods. Simulation results are highly sensitive to the number of mesh elements – generally, the more elements used, the higher the accuracy of the results. However, excessively refined meshes drastically increase computational requirements and do not always yield significantly improved precision. Therefore, a mesh independence study is conducted to assess how variations in mesh density affect the results. The goal of this approach is to determine the coarsest mesh that still captures the key physical phenomena within the studied system.

Two polyhedral meshes were prepared for the analysis. The first consisted of 19,570 elements with an orthogonal quality value of 0.52. The second, more refined mesh contained 30,302 elements and had an orthogonal quality of 0.35. An example of one of the generated meshes is shown in Fig. 7.

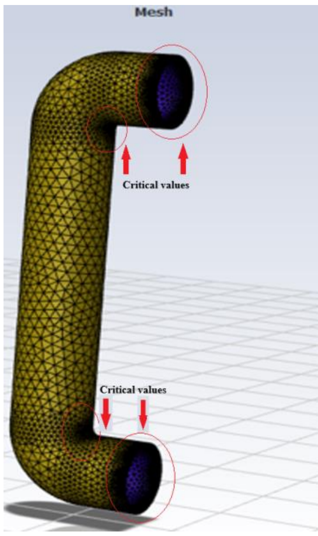

Fig. 7. Example of a connector mesh created in Ansys Fluent

Due to the lack of information about pressure and velocity in the outlet cross-section of the connector, an "outflow" boundary condition was applied at the domain's exit. A drawback of this condition is the necessity to define a "velocity inlet" boundary condition at the domain entrance.

The input data for the calculations are summarized in Table 2.

Table 2. Input data of the model being tested

Parameter	Value [unit]
ṁ (compressor air mass flow rate)	0.11 $[\frac{kg}{s}]$
T_2(measured air temperature at pipe inlet/compressor outlet)	448.15 [K]
p_2(air pressure at pipe inlet/compressor outlet)	202600 [Pa]
R (gas constant for air)	287 $[\frac{J}{kg*K}]$
D (diameter of the connecting pipe cross-section)	0.034 [m]

Using the Clapeyron equation, the air density in the cross-section behind the compressor can be calculated:

$$\rho_{pow} = \frac{p_2}{R*T_2} = [\frac{Pa}{\frac{J}{kg \cdot K} \cdot K} = \frac{\frac{N}{m^2}}{\frac{N \cdot m}{kg}} = \frac{kg}{m^3}] \quad (3)$$

$$\rho_{pow} = \frac{202600}{287 \cdot 448.15} = 1.575\ \frac{kg}{m^3}$$

Having calculated the air density, the velocity can be determined from the flow continuity equation:

$$\dot{m} = \rho_{pow} \cdot v \cdot A \quad (4)$$

where: A – cross-sectional area perpendicular to the direction of movement, v – flow velocity.

We determine the cross-sectional area of the pipe from the equation:

$$A = \pi \cdot (\frac{D}{2})^2 \quad (5)$$

$$A = 3.14 \cdot \left(\frac{0.034}{2}\right)^2 = 0.00091\ m^2$$

Therefore:

$$v = \frac{\dot{m}}{\rho_{pow} \cdot A} = [\frac{\frac{kg}{s}}{\frac{kg}{m^3} \cdot m^2} = \frac{kg}{s} * \frac{m}{kg} = \frac{m}{s}] \quad (6)$$

$$v = \frac{0.11}{1.575 \cdot 0.00091} = 76.75 \frac{m}{s}$$

In the Ansys Fluent Solution program, a boundary condition was given at the pipe inlet – the calculated velocity was 76.75 m/s. Additionally, the static pressure value was entered as 250,000 Pa. The k – omega SST turbulence model was adopted for the calculations.

As a result of the analysis, pressure losses due to air flow through the connector were estimated. For this purpose, the relationship was used:

$$\frac{p_{inl} - p_{out}}{p_{inl}} \cdot 100\ \% \quad (7)$$

where: p_{inl} – air pressure at the chamber inlet, p_{out} – air pressure at the outlet to the chamber.

The combustion chamber was modelled based on the technical data available in the research engine documentation. The CAD model of the chamber was made using the SolidWorks program (Fig. 8).

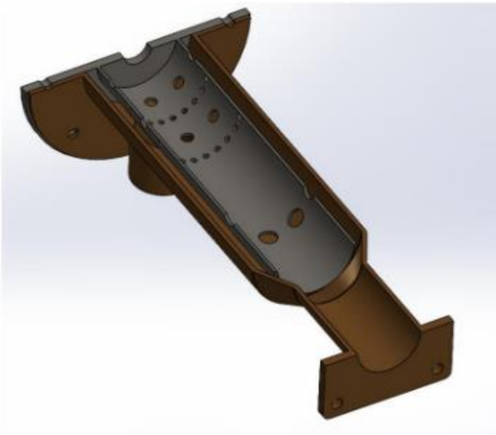

Fig. 8. Combustion chamber cross-section model

When selecting an appropriate combustion model, flame stability should be taken into account. On the one hand, an increase in the number of inert gases causes a decrease in the laminar combustion velocity, which means a deterioration in flame stability. However, the creation of a swirl and an internal recirculation zone significantly improves this stability. The selection of boundary conditions and the turbulence or combustion model strictly depends on the expected simulation results and how complex the simulation will be. It was decided to select the non-premixed combustion diffusion model due to the possibility of selecting the equation used to determine the temperature, defining the parameters of the unburned and burned medium, determining the reaction progress equation and the progress variable at the inlets and outlets, and the option of selecting the flame propagation speed model. When generating the mesh of the tested combustion chamber, it was decided to compare both types of meshes (unstructured and hybrid) and decide which one would be used for CFD analysis. It was decided to discretize the "fluid" domain, which contains the internal volume of the combustion chamber. For this purpose, a chamber volume was created in ANSYS Space Claim (Fig. 9) by replacing the chamber body with a fluid-filled structure of a given volume.

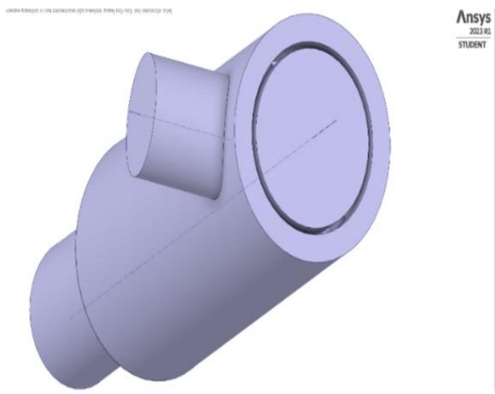

Fig. 9. Ready domain in the form of combustion chamber volume

For comparison purposes with the polyhedra mesh, a poly-hexcore mesh was generated in the next step. For this mesh, a min. Orthogonal Quality = 0.4. In the vicinity of the walls of the fire tube chamber, it was noted that there was a step change in volume of 1:8. This can cause numerical errors during solution. The hexcore mesh also obtained a larger number of elements, which would increase the calculation time of the program. A summary of both meshes is shown in Table 3. The most advantageous mesh parameters were demonstrated by the Polyhedral mesh. It was decided to select this mesh for further numerical tests of the combustion chamber. Also, compared to the Poly-hexcore mesh, it has a smaller number of elements, which will allow for shorter calculation times. The generated mesh meets the requirements for a good quality mesh - all parameters are within the ranges of permissible coefficient values, and the use of an increase in the mesh perpendicular value to 0.4 allowed for a reduction in skewness while maintaining good mesh quality.

Table 3. Input data of the model being tested

Grid type	Skewness	Orthogonal Quality	Number of elements
Polyhedra	0.5984901	0.4	254009
Poly-hexcore	0.79976186	0.4	398248

After the first attempt to simulate cold flow through the combustion chamber, it was checked that the Y+ parameter (dimensionless distance factor from the wall) reached maximum values of about 250 in the lower parts of the body, so it does not exceed acceptable values (about 500) and there is no need to compact the grid.

5. Simulation results and analysis

This part of the paper presents the results of CFD analyses for the previously selected settings in the Ansys Fluent program. Due to the significant computational cost and model complexity, it was decided to start the numerical studies by modelling the flow of the working medium through the chamber, without the combustion process (so-called cold flow). After completing the analysis, the simulation results were developed in the CFD Post post-processor. The first result of the analysis shows a map of the streamline distribution inside the combustion chamber (Fig. 10). It is clearly visible that some of the air flows into the interior of the glow tube through the holes and is swirled. The rest of the air flows around the outer part of the glow tube. Initially, it can be stated that this flow is close to the theoretical one. For a more accurate visualization of the flow through the chamber, the velocity vectors were projected onto the plane of symmetry of the chamber (Fig. 10).

The highest values are obtained in the holes from the glow tube and at the narrowing of the outlet from the chamber. The obtained velocities reach maximum values of approximately. 100 m/s, which is a realistic value. In the primary zone of the chamber, the formation of small recirculation vortices can be seen on the velocity vector maps near the areas of the tube holes. This increases the turbulence of the micro flow scale needed for an effective mixing process, however, in the author's opinion, the vortices created as a result of the simulation are insufficient to fulfil their function. The increase in velocity and the reflection of the airflow from the small openings lead to the formation of backflow, which causes air to recirculate into the mixing zone. Distributed large holes lead to the formation of cold areas in the flow, which are a result of cold streams, which can be a result of generating additional thermal stresses [12].

Comparing the results obtained with those presented in [10] for the GTD 350 turboshaft engine, which has a combustion chamber of a similar type, it can be seen that large symmetrical pairs of air vortices, which are recirculation zones, are formed in it. In the case under review, such vortices are absent. There is a lack of air vortices that would keep the flame "in place". The increase in velocity and the reflection of the air stream from small holes lead to the formation of a reverse flow, which causes air recirculation to the mixing zone.

a)

b)

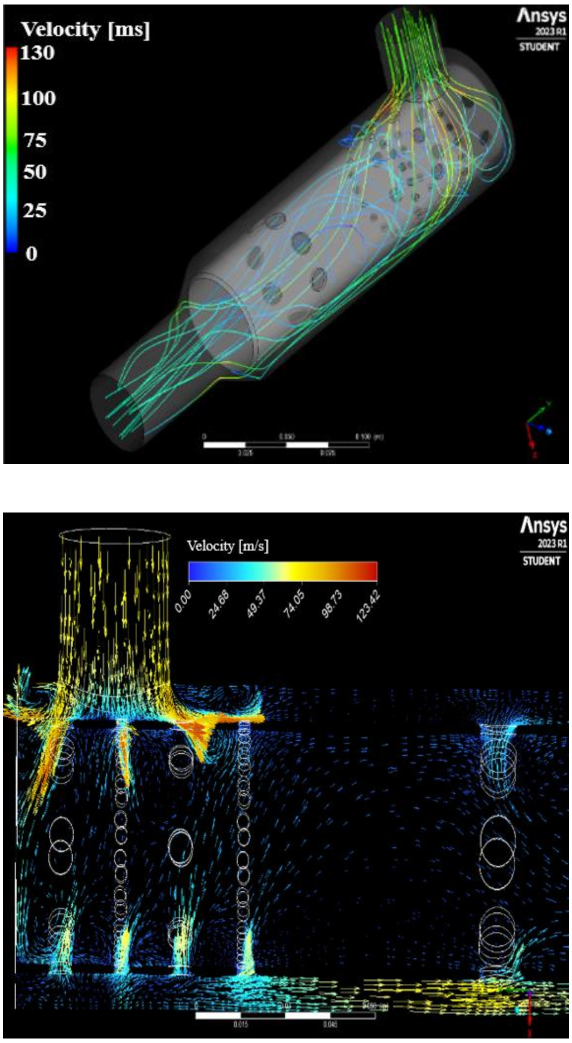

Fig. 10. Maps: a) view of the velocity line layout inside the pipe, b) velocity vector map for the tested chamber

In the further dilution zone, the formation of undesirable vortex zones can be observed. The flow is disturbed there. The velocity in the upper part of the combustion chamber outside the glow tube has significantly lower values than in the lower part.

In the case of velocity distribution, a lack of symmetry can be seen on both sides of the glow tube. The lack of symmetry on both sides of the flame tube in a gas turbine engine combustion chamber can lead to uneven temperature distribution, affecting efficiency and component durability. In article [6], the design of the flame tube in micro gas turbines was analyzed. The authors emphasized key dimensions, such as the total length of the flame tube, which can influence flow symmetry and the combustion process. This thesis [23] demonstrated that combustion primarily occurs within the combustion chamber. When a shorter combustion chamber was used, flames were present in the initial phase, suggesting that asymmetry may influence the combustion process. In the upper part of the combustion chamber, the air is accelerated through small holes, while at the bottom, the increase in velocity has much smaller values. However, before this, the flow velocity drops to zero, which does not favour changes in pressure at the chamber inlet. A compromise should be sought between a large number of small holes and a smaller number of larger holes. A large number of small holes allows for a uniform circumferential temperature distribution; on the other hand, it clearly inhibits the main flow (the flow velocity decreases). On the other hand, too sparsely distributed large holes lead to the formation of cold areas in the flow, which are a result of cold streams, which can be a result of generating additional thermal stresses, e.g., on the turbine blades. In the analysed chamber, the temperature distribution is omitted due to the fact that hot exhaust gases reach the diffuser of the centripetal turbine, which is resistant to this type of disturbance. It can be concluded that the flow through the holes of the glow tube not only depends on their size and pressure drop, but also on the geometry of the holes and the flow conditions in their vicinity.

6. Summary and remarks

1) In the context of research into new fuel technologies, multi-fuel engines can be tested for their performance and combustion efficiency of different fuel types, which can lead to innovations in the field of power supply.
2) It is worth noting that the use of multi-fuel engines may require special design and technological solutions to enable efficient combustion of various substances, as well as maintain the necessary performance parameters.
3) The use of a multi-fuel turbine engine in hybrid systems brings a number of benefits due to its dimensions and the process of preparing it for the combustion process. By using different fuels, it is possible to adapt to the types of fuel available in a given region, which can reduce operating costs and dependence on a single fuel source (e.g. diesel oil, gas, biofuel).
4) The conclusions indicate that multi-fuel turbine engines have the potential to greatly enhance the energy efficiency and environmental impact of hybrid systems, presenting new possibilities for future mobility solutions. In research focused on emerging fuel technologies, these engines can be evaluated for their efficiency and combustion performance with various fuel types, potentially driving innovations in power generation. It is important to highlight that employing multi-fuel engines may demand specialized design and technological approaches to ensure efficient combustion of diverse fuels while maintaining the required performance standards.
5) Analysing the flow velocity distribution, in the author's opinion, the geometry of the glow tube should be changed by using a less frequent arrangement of smaller holes, the diameters should be selected so that the air flow velocities are equal in the upper and lower parts of the chamber, so as to obtain flow symmetry. Additionally, it may be necessary to slightly enlarge the holes in the primary zone, which would result in the possibility of creating larger air turbulence zones.
6) The usefulness of numerical simulations in identifying and shaping mixing and combustion zones has been demonstrated. As a result of the conducted research, the conditions and methods for forming a combustible mixture enabling efficient operation of a pulsating combustion chamber (particularly suitable for heating applications) were determined. The considerations presented in this study may serve as a starting point for the aforementioned extensive research program involving modern measurement techniques.

Acknowledgements

This work was financed/co-financed by the Military University of Technology under the research project UGBWIM__22012025_15.

Nomenclature

CEA	chemical equilibrium applications	GSC	generalized spray combustion
CFD	computational fluid dynamics	GSP	gas turbine simulation program

Bibliography

[1] Ahmed SA, Zhou S, Tsegay S, Ahmad N, Zhu Y. Effects of hydrogen-enriched biogas on combustion and emission of a dual-fuel diesel engine. Energy Sources Part A. 2020:46(1): 5644-5659. https://doi.org/10.1080/15567036.2020.1736694

[2] Algayyim SJM, Saleh K, Wandel AP, Fattah IMR, Yusaf T, Alrazen HA. Influence of natural gas and hydrogen properties on internal combustion engine performance, combustion, and emissions: a review. Fuel 2024;362:130844. https://doi.org/10.1016/j.fuel.2023.130844

[3] ANSYS Fluent Tutorial Guide. 2016. www.ansys.com

[4] Bhuiyan MMH, Siddique Z. Hydrogen as an alternative fuel: a comprehensive review of challenges and opportunities in production, storage, and transportation. Int J Hydrogen Energy. 2025;102:1026-1044. https://doi.org/10.1016/j.ijhydene.2025.01.033

[5] Bulewicz E, Dyjakon A. Spalanie i Paliwa (in Polish). Wroclaw University of Technology Publishing House. Wroclaw 2008.

[6] Czarnecki M. Small scale gas turbine combustor sizing. Autobusy – Technika, Eksploatacja, Systemy Transportowe 2019;20(1-2):180-184. https://doi.org/10.24136/atest.2019.032

[7] Decarbonisation pathways for gas turbines – Modern Power Systems. https://www.modernpowersystems.com/analysis/decarbonisation-pathways-for-gas-turbines/?utm_source=chatgpt.com&cf-view&cf-closed&cf-view&cf-closed (accessed on 8 May 2025).

[8] Electric & Hybrid Marine Technology International. 2024. https://ehm.mydigitalpublication.com/september-2024-issue-/page-56 (accessed on 8 May 2025).

[9] Geng T, Schoen MA, Kuznetsov AV, Roberts WL. Combined numerical and experimental investigation of a 15-cm valveless pulsejet. Flow Turbulence Combust. 2006;78:17-33. https://doi.org/10.1007/s10494-006-9032-8

[10] Gieras M. Komory spalania silników turbinowych (in Polish). Warsaw University of Technology Publishing House. Warsaw 2010.

[11] Jones R, Goldmeer J, Monetti B. Adressing Gas Turbine Fuel Flexibility. GE Energy 2011.

[12] Lefebvre AH, Ballal DR. Gas Turbine Combustion. CRC Press 2010. https://doi.org/10.1201/9781420086058

[13] Merkisz J, Markowski J, Pielecha J, Mikutel T, Kozłowski R. Emisja spalin z silników lotniczych [Exhaust emissions from aircraft engines] (in Polish). Zeszyty Naukowe Akademii Marynarki Wojennej. 2013;54(1):89-104.

[14] New Zealand Adopts Wrightspeed Jet And Battery Power For Buses. https://www.forbes.com/sites/samabuelsamid/2016/04/22/new-zealand-adopts-wrightspeed-jet-and-battery-power-for-buses/ (accessed on 13 May 2025).

[15] Pasalkar SM, Shaikh MS, Pipulyawala MM, Jasdanwalla H. Design and CFD analysis of combustion chamber of jet engine to reduce formation of NOx. International Research Journal of Engineering and Technology. 2020;7(6):5172-5180. www.irjet.net

[16] Pukalskas S, Kriaučiūnas D, Rimkus A, Przybyła G, Droździel P, Barta D. Effect of hydrogen addition on the energetic and ecologic parameters of an SI engine fueled by biogas. Appl Sci. 2021;11:742. https://doi.org/10.3390/app11020742

[17] Rolls-Royce Ltd. The jet engine. Rolls-Royce 2005.

[18] Sayres JR, Scott J. Computational fluid dynamics for pulsejets and pulsejet related technologies. Master’s thesis, North Carolina State University 2011.

[19] Serbin S, Radchenko M, Pavlenko A, Burunsuz K, Radchenko A, Chen D. Improving ecological efficiency of gas turbine power system by combusting hydrogen and hydrogen-natural gas mixtures. Energies. 2023;16(9):3618. https://doi.org/10.3390/en16093618

[20] Sher AA, Ahmad N, Sattar M, Phelan P, Lin A. Computational analysis of multi-fuel micro-gas turbine annular combustion chamber. J Therm Anal Calorim. 2024;149:3317-3329. https://doi.org/10.1007/s10973-024-12924-z

[21] Skobiej K. A review of hydrogen combustion and its impact on engine performance and emissions. Combustion Engines 2025;200(1):64-70. https://doi.org/10.19206/CE-195470

[22] Suchocki T. Performance and emission characteristics of a small gas turbine engine using hexanol as a biomass-derived fuel. Materials. 2024;17:6011. https://doi.org/10.3390/ma17236011

[23] Trzeciak A. Badania procesu pulsacyjnego spalania. Doctoral Thesis. Warsaw University of Technology. Warsaw 2023.

[24] Turbosprężarka Garrett 880693-5002S G30-770 (super core) | TOMSON Motorsport. https://tomson.com.pl/product-pol-7305-Turbosprezarka-Garrett-880693-5002S-G30-770-super-core.html (accessed on 14 May 2025).

[25] Wtryskiwacz Swirl Burst umożliwia ultra czyste spalanie biopaliw (in Polish). https://karlobag.eu/pl/ekologia/wtryskiwacz-swirl-burst-umozliwia-ultraczyste-spalanie-biopaliw-rewolucyjne-rozwiazanie-w-zakresie-redukcji-emisji-i-zrownowazonego-rozwoju-sektora-energetycznego-vxdjo?utm_source=chatgpt.com (accessed on 8 May 2025).

[26] Zheng F. Computational investigation of high speed pulsejets. Doctoral Thesis. NC State 2009. http://www.lib.ncsu.edu/resolver/1840.16/5968

Marcin Dopieralski, MSc. – Faculty of Mechanical Engineering, Military University of Technology, Warsaw, Poland.

e-mail: *marcin.dopieralski@student.wat.edu.pl*

Filip Polak, DEng. – Faculty of Mechanical Engineering, Military University of Technology, Warsaw, Poland.

e-mail: *filip.polak@wat.edu.pl*