Symkom 2014 Open Access

Tomasz Suchocki*, Piotr Lampart, and Piotr Klonowicz

Numerical investigation of a GTM-140 turbojet engine

DOI 10.1515/eng-2015-0053
Received Mar 18, 2015; accepted Nov 26, 2015

Abstract: The paper presents three-dimensional numerical simulations of combustion in the GTM-140 miniature turbine engine. The main aim of the work is to understand the processes occurring in the combustion chamber. The coupling of chemical kinetics, thermochemistry, transport of mass, energy and momentum, and fluid mechanics is a challenge for the engineers. The knowledge of these issues is essential to achieve a high performance product. The k-ϵ (RANS) Turbulence Model and Non-Premixed Model for the combustion was used. The particles of fluid droplets were described by the Discrete Phase Model.

Keywords: GTM-140; turbine engine; CFD; PDF; Discrete Phase Model; combustion

1 Introduction, purpose of the work

The following paper presents numerical simulations of the combustion process in the turbine engine - GTM-140 produced in the JETPOL company from Poznań. This kind of engines are used for aircraft propulsion modeling, but may also be used for military purposes include UAV. The engines of this type are important to achieve high energy conversion efficiency and shape miniaturization. In order to understand the processes occurring in the engine the numerical calculations were carried out. Analysis of their results should give the knowledge necessary to implement the optimization to achieve high efficiency product. A number of small turbojet design examples are available that develop less than 200 N of static thrust (*e.g.* [2–5, 7, 11–14]). These designs have been derived from large turbojet scale-down procedures to yield micro scale engines. However, a deep understanding of the behavior of these jet engines is far from being ascertained [7].

2 Principle of operation

The inlet nozzle is used to deliver the required amount of air to the engine and to create the appropriate velocity profile upstream of the compressor (Figure 1). Then the air is compressed by the single-stage radial centrifugal compressor. Another element is the diffuser where due to the increased cross-section the kinetic energy of the stream (velocity drop) is converted into pressure energy. Behind the diffuser the air is directed into the reverse-flow annular combustion chamber. This is the place where all of the processes necessary for the proper preparation are progressing - the evaporation and the combustion of the fuel to obtain the maximum power.

The combustion chamber is divided into three major zones presented in Figure 2. Primary zone is designed to provide a stable and intense burning. Second (intermediate) zone - is where the radicals recombination occurs along with afterburning of the intermediate products of the combustion, such as CO, hydrocarbons and soot. The last section which is the mixing (diluting) zone. It is designed to cool the exhaust gas to the required allowable temperature before the turbine blade.

The final element is the single-stage axial turbine where the enthalpy of the exhaust gas flow is converted into the mechanical energy of the shaft (driving the compressor) and eventually produces the thrust.

The rated conditions for the engine as given by Jetpol can be seen in Table 1.

3 Geometry and mesh

The flow domain design takes into account the diffuser and the combustion chamber with six vaporizers. Odd number of blades of the compressor causes the inability to evenly

***Corresponding Author: Tomasz Suchocki:** The Szewalski Institute of Fluid-Flow Machinery, Fiszera 14 st., Gdansk 80-231 Poland, Email: tsuchocki@imp.gda.pl
Piotr Lampart, Piotr Klonowicz: The Szewalski Institute of Fluid-Flow Machinery, Fiszera 14 st., Gdansk 80-231 Poland

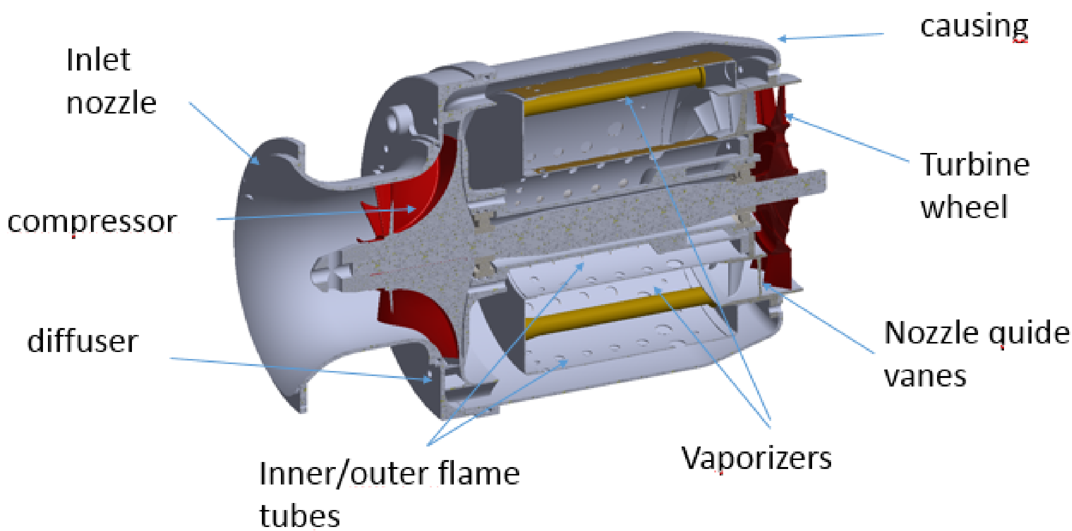

Figure 1: The interior of the GTM-140 turbojet engine.

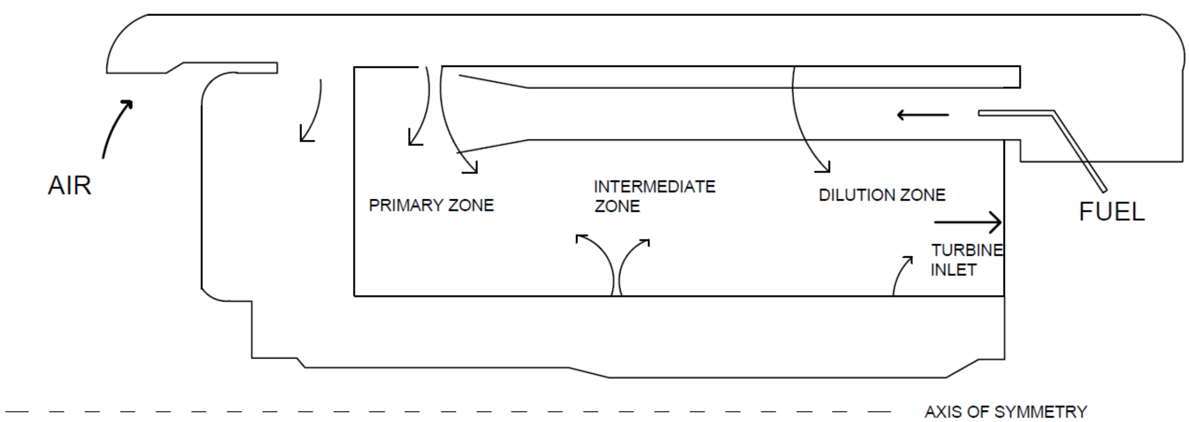

Figure 2: Annular combustor with reverse-flow vaporizer.

Table 1: The rated operation conditions for the GTM-140 Turbojet Engine specified by the manufacturer.

Engine Compression Ratio	2.8:1
Air flow rate	0.35 [kg/s]
Exhaust Gas Temperature	700°C
Engine Diameter	110 mm
Engine Length	265 mm
Mass Flow	0.35 kg/s
Design Maximum Thrust	140 N
Compressor Type	Single Stage Centrifugal
Turbine Type	Single Stage Axial Flow
Design Maximum RPM	120000
Fuel consumption	420 [g/min]

split engine geometry giving rise to the discretization of the whole area – Figure 3, 4.

The computational grid was created in ANSYS Meshing. It consists of 3 500 000 tetrahedral finite volume elements, Figure 5. According to the literature [2–7] the proposed grid should be sufficient to achieve a good convergence of the model and to obtain reliable results.

The boundary conditions for the calculation are shown in Table 2. The assumed values of the inlet pressure and the temperature must be confirmed on test bench. For combustion the Non-Premixed Model was used in which the fuel and oxidizer are fed separately into the combustion zone and the progress of the reaction determines the phenomenon of diffusion. Discrete phase model in a lagrangian fashion for representation of the fuel evapora-

Table 2: Boundary conditions setup for simulation.

Surface	Boundary Condition	Mass flow rate [kg/s]	Temperature [K]
Fuel inlet	Mass flow inlet	0,0052 kg/s	303
		Gauge pressure [Pa]	
Air inlet	Pressure inlet	280000 Pa	400
Outlet	Pressure outlet	190000 Pa	-
Walls	Radiation	-	-

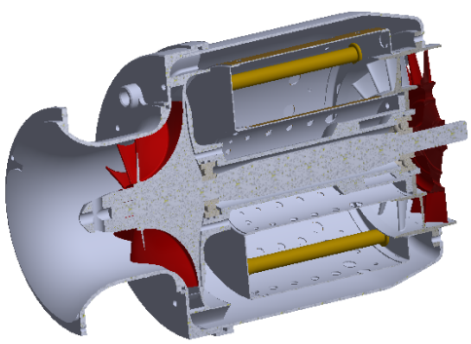

Figure 3: Geometry of the GTM-140.

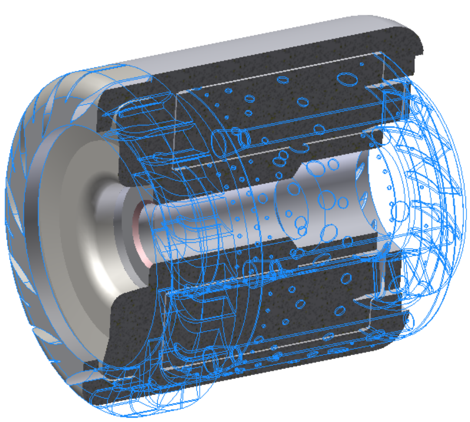

Figure 4: Fluid domain of the GTM-140.

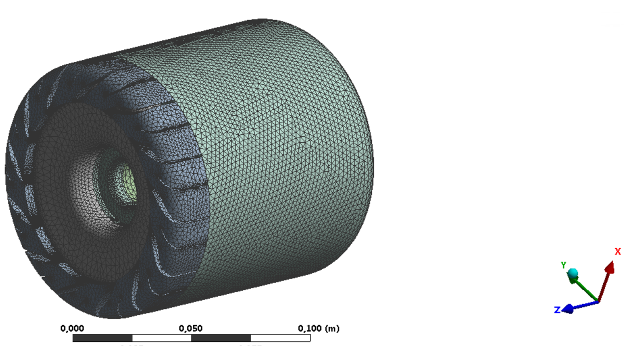

Figure 5: The discretized domain.

Table 3: The settings of the Fluent numerical task.

Setup parametr	Setting
Solver type	Pressure based
Turbulence model	k-ϵ realizable
Wall treatment	Standard wall function
Radiation model	Discrete Ordinates
Species model	Non-premixed combustion
Spatial discretization	Second Order

tion was used. Trajectory is calculated by integrating the particle force balance equation.

Dispersion of particles due to turbulent fluctuations in the flow can be modeled using stochastic tracking (discrete random walk). The continuous phase approach represented by Eulerian fashion is calculated using the Navier-Stokes equations.

The settings of the Fluent numerical task are specified as seen in Table 3.

4 CFD results

All of results are shown three sections of jet engine: compressor diffuser, combustion chamber and inlet guide vanes.

4.1 Distribution of the temperature field

Results corresponding to the temperature fields are presented in Figure 6. The distribution and values of the temperature show a good agreement with the operating data provided by the manufacturer and available literature data. The averaged total temperature at the outlet condition reaches 998 K. It is approximately 4% higher than the rated value. The probable cause of the difference is the assumption of the adiabatic engine walls.

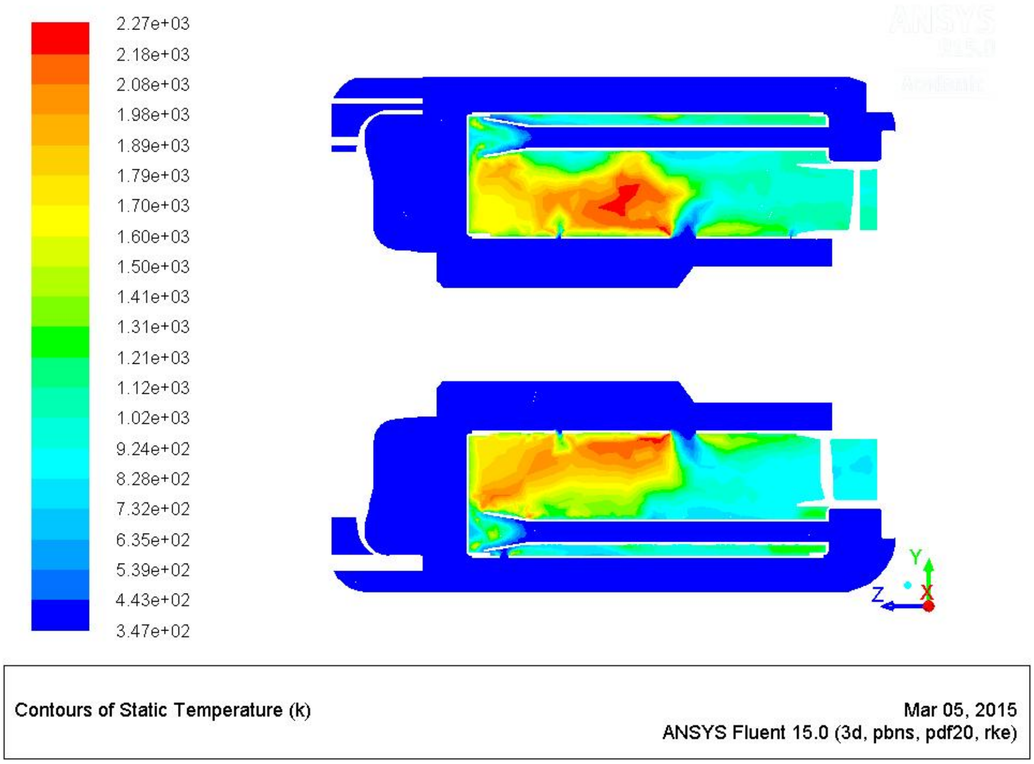

Figure 6: Static temperature contours – axial section.

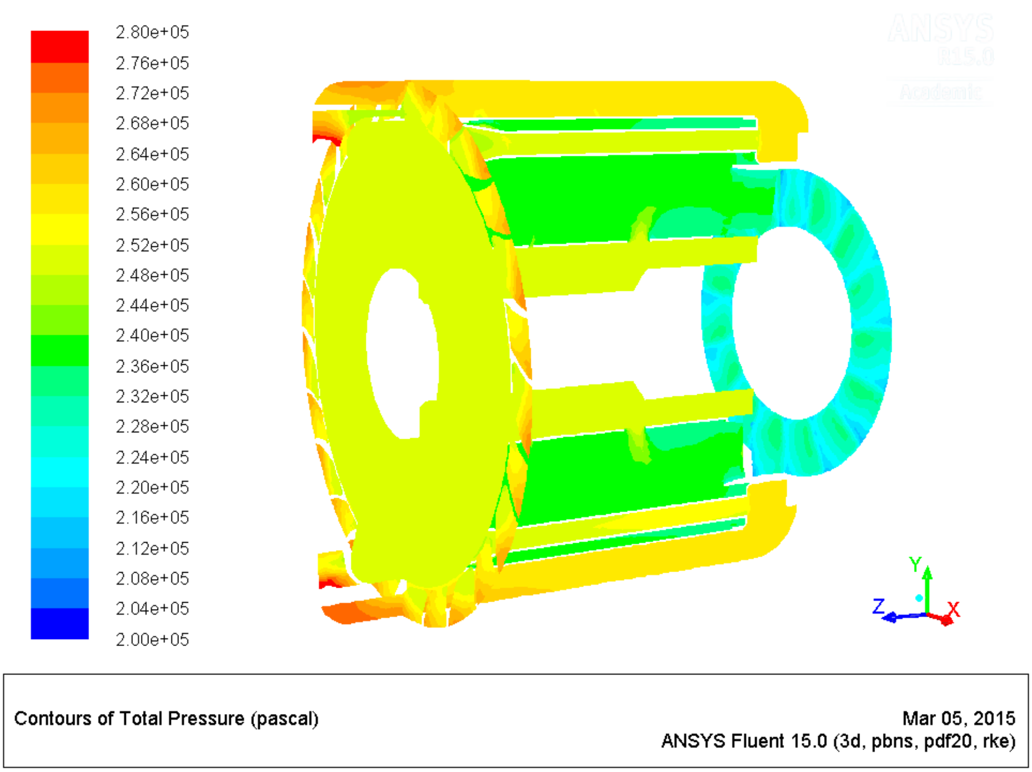

Figure 7: Total pressure contours - axial section; diffuser outlet; nozzle guide vanes outlet.

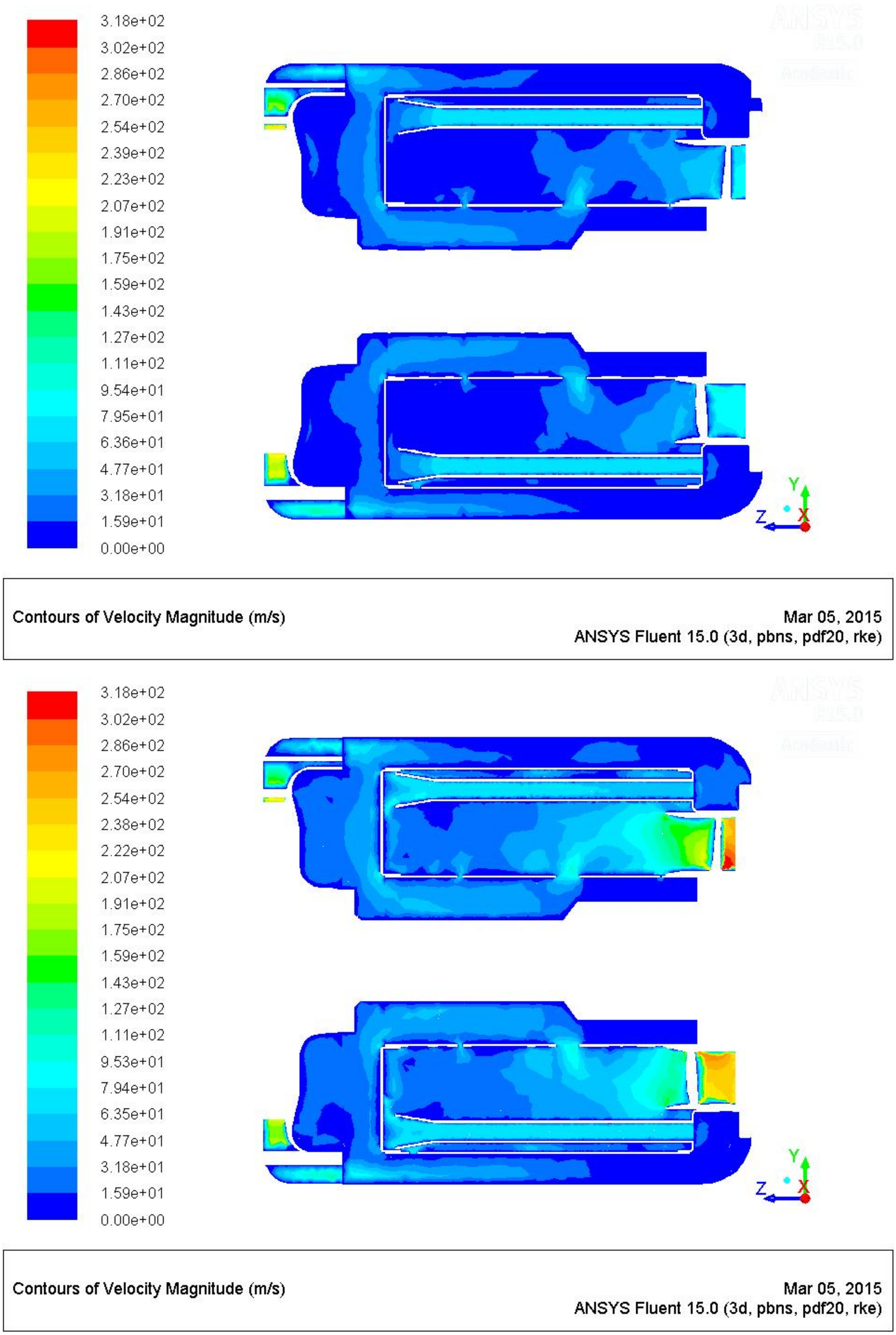

Figure 8: Contours of Velocity Magnitude: - top–cold flow, - bottom–reaction flow.

The obtained values must be verified on the real object. The prepared test stand is equipped with temperature sensors before and after the diffuser, the outlet of the combustion chamber and the turbine. In each of these sections the three sensors are mounted circumferentially. So the configured measurement system will be sufficient to validate the numerical model and thereof engine modifications.

4.2 The distribution of the pressure field

Figure 7 presents the contours of the total pressure in the axial section to illustrate the flow losses. Pressure losses are determined for the inlet to the combustion chamber and the outlet of the combustion chamber. The average total pressure at the outlet of the diffuser is 2.49 bar, while at the outlet section it is 2.25 bar having regard 1 bar reference pressure. This corresponds the total pressure loss of approximately 9% and it is consistent with the expectations to the specified flow system. The impact on the value of the losses is the uneven velocity streamlines at the outlet of the diffuser and in the vaporizer.

Changing the diffuser geometry for additional deceleration of velocity (to obtain more equal distribution of velocity contours) and higher pressure at the intake to the combustion chamber should be considered.

The measurements of the static pressure will be performed analogously to the temperature measurement.

4.3 The distribution of the velocity field

For the assumed values of the pressure and mass flow rate of the air obtained with the cold-flow, the velocity at the inlet was 90 m/s, while the reaction-flow 318 m/s – Figure 8. Such a high velocity is due to the increase of specific volume of the fluid in the combustion chamber caused by evaporation and density decrease.

5 Conclusions

The gas turbine and jet engines had a wide variety of flow regimes from mainly high Reynolds number fully turbulent flows to transitional flows in some areas. The combination of such a wide range of flow phenomena with complex geometry makes numerical modeling of gas turbines very difficult [15]. The results show the correctness of the relation with the literature data [3–5, 12] and the performance data given by the manufacturer. The proposed solution can be used for modeling the combustion process including the tracking trajectory of the fuel particles. Acquired knowledge of the processes occurring in the combustion chamber can be used to optimize the operating conditions, to increase the efficiency of the entire flow system and to influence the local parameter values.

Acknowledgement: The authors appreciate the financial support of the National Centre for Research and Development project "Developing integrated technologies of fuel and energy production from biomass, agricultural wastes and other resources".

STRATEGIC PROGRAMME OF THE NATIONAL CENTRE FOR RESEARCH AND DEVELOPMENT
The research task financed within the framework of the Strategic Research and Development Programme entitled 'Advanced Technologies for Energy Generation' carried by The National Centre for Research and Development and electric power holding company ENERGA SA in Poland
Research Task No. 4

DEVELOPMENT OF INTEGRATED TECHNOLOGY OF FUELS AND ENERGY FROM BIOMASS, AGRICULTURAL WASTE AND OTHER

References

[1] T. Kamps, Model Jet Engines, Traplet Publications Ltd 2005.
[2] M. Gieras, Obliczenia parametrów użytkowych lotniczych silników turbinowych, OWPW 2013.
[3] M. Badami, P. Nuccio, A. Signoretto, Experimental and numerical analysis of a small-scale turbojet engine, Energy Convers Manag. 2013.
[4] M. Gieras, T. Stańkowski, Computational study of an aerodynamic flow through a micro-turbine engine combustor, J Power Technol. 2012.
[5] C.A. Gonzalez, K.C. Wong, S. Armfield, A computational study of the influence of the injection characteristics on micro-turbine combustion, 16th Austral. Fluid Mechanics Conf, Australia, 2007.
[6] T. Suchocki, J. Żabski, P. Lampart, Badania numeryczne spalania paliwa lotniczego w silniku turbinowym GTM-140, XXI Warsztaty Naukowe PTSK, 2014.
[7] E. Benini, S. Giacometti, Design, manufacturing and operation of a small turbojet-engine for research purposes, Appl Energy. 2007.
[8] R. M. Bering, Numerical Investigation of the Soot Initiated Formation of Ultra Fine Particles in a Jet Turbine Engine Using Conventional Jet Fuel, Master Thesis 2012.
[9] R. Tuccillo, M.C. Cameretti, Comparing different solutions for the micro-gas turbine combustor. Proceedings of ASME Turbo Expo. 14–17 June, Viena, Austria, 2004.
[10] Fluent Inc, 2015. Fluent User's Guide.
[11] M. Badami, P. Nuccio, D. Pastrone, A. Signoretto, Performance of a small-scale turbojet engine fed with traditional and alternative fuels, Energy Convers Manag. 2014.
[12] T. Trebunskikh, A. Ivanov, G. Dumnov, FloEFD simulation of micro-turbine engine.

[13] J. Ling, K.C. Wong, S. Armfield, Numerical Investigation of a Small Gas Turbine Compressor, 16th Austral. Fluid Mechanics Conf, Australia, 2007.
[14] C.A. Gonzalez, K.C. Wong, S. Armfield, Computational study of a micro-turbine engine combustor using large eddy simulation and Reynolds average turbulence models, Austral Mathematical Soc, Australia, 2008.
[15] Y. Zhiyin, Large-eddy simulation: Past, present and the future, Chinese Journal of Aeronautics 2015.