IOP Conf. Series: Materials Science and Engineering **302** (2018) 012021 doi:10.1088/1757-899X/302/1/012021

Investigation of the Effect of the Non-uniform Flow Distribution After Compressor of Gas Turbine Engine on Inlet Parameters of the Turbine

M Yu Orlov, S V Lukachev and V M Anisimov

Samara National Research University, Scientific Educational Center of Fluid Dynamics Research, 34, Moskovskoye Shosse, Samara, Russian Federation, 443086

Email: adler65@mai.ru, teplotex_ssau@bk.ru, vradik@mail.ru

Abstract. The position of combustion chamber between compressor and turbine and combined action of these elements imply that the working processes of all these elements are interconnected. One of the main requirements of the combustion chamber is the formation of the desirable temperature field at the turbine inlet, which can realize necessary durability of nozzle assembly and blade wheel of the first stage of high-pressure turbine. The method of integrated simulation of combustion chamber and neighboring nodes (compressor and turbine) was developed. On the first stage of the study, this method was used to investigate the influence of non-uniformity of flow distribution, occurred after compressor blades on combustion chamber workflow. The goal of the study is to assess the impact of non-uniformity of flow distribution after the compressor on the parameters before the turbine. The calculation was carried out in a transient case for some operation mode of the engine. The simulation showed that the inclusion of compressor has an effect on combustion chamber workflow and allows us to determine temperature field at the turbine inlet and assesses its durability more accurately. In addition, the simulation with turbine showed the changes in flow velocity distribution and pressure in combustion chamber.

1. Introduction

The performance improvement of modern gas turbine engine and power plants involves not only the optimization of parameters averaged over the cross section, but also the formation of the cross section distribution. Combustion chamber is one of the main elements of gas turbine engines and power plants, which directly produce an effect on almost all characteristics of GTE. For example, formation of the desirable temperature field at the turbine inlet determines the necessary durability of nozzle assembly and blade wheel of the first stage of high-pressure turbine.

Joint operation of compressor, combustion chamber and turbine implies that workflows of these elements are linked between each other. Compressor parameters change may affect the workflow in the combustion chamber, and then in the turbine. Thus, during the design of the combustion chamber it is important to determine not only the temperature distribution, but also to obtain an understanding of how it is influenced by various factors, including the change of the rotor revolutions and the corresponding change in the compressor.

Existing studies related to integrated simulation of combustion chamber and neighboring nodes [1-7] consider mainly the interaction of the compressor and the combustion chamber and in a less degree consider processes throughout the whole engine. Meanwhile computer simulation can make the engine design processes more effective. Therefore, the aim of this study is to investigate the influence of the compressor on the processes in the combustion chamber and through them on the parameters of the gas prior to the turbine.

2. Models and methods

The object of the investigation is core engine of power plant. Ansys Fluent three-dimensional simulation package was used as a calculation tool. Earlier, using this package, the method of combustion process simulation was tested and verified with experiments [8-15]. Geometrical model includes: the impeller and guide vanes of the last stage and guide vanes of the penultimate stage of high-pressure compressor, combustion chamber, nozzle assembly and blade wheel of high-pressure turbine [16-18].

Periodical sector model of engine core was used instead of full-sized model to decrease necessary computer memory and calculation time. The sector of combustion chamber includes two burners, because fuel rate depends on the thrust rating and for the even and odd burners it may differ. The blade number at sectors of compressor and turbine might differ from the real, by no more than 10%, to ensure the sector periodicity. Unstructured finite element model is presented in figure 1.

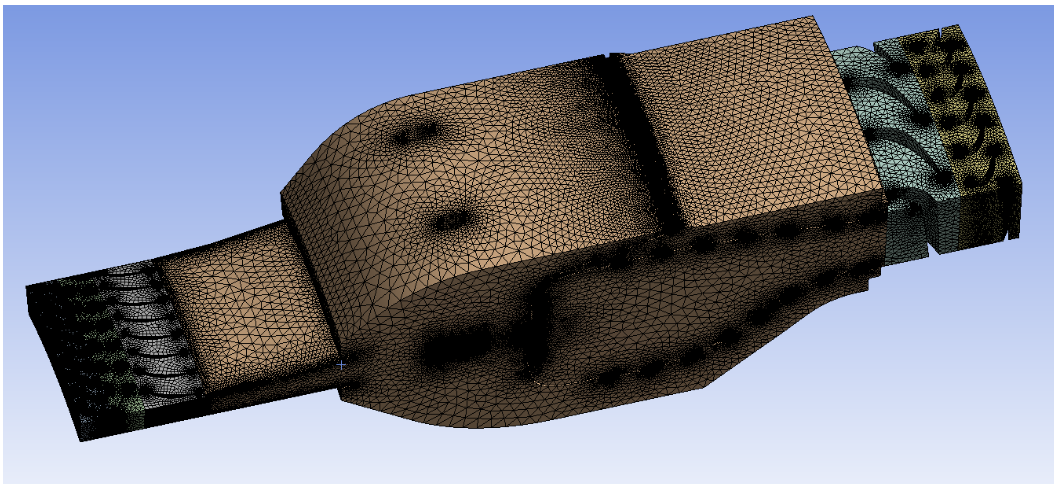


Figure 1. Finite element model of engine core.

Number and size of finite elements on contact surfaces of the compressor, combustion chamber and turbine should be approximately equal. Otherwise, such a discrepancy may cause an error during data transfer from one body to another. The mesh tools Match Control was used to set the periodicity on the lateral faces of the model. The number of finite elements is about 14 million; the skewness is less than 0.96. Inlet and outlet boundaries were set for air and fuel, also paired "interfaces" on contiguous faces of blade rows and the combustion chamber were set. After mesh generation, boundary conditions were set and mathematical model was specified. Mathematical model includes computational models, which are necessary for simulation workflow in combustion chamber [19-24]. The interaction between certain pre-established “interfaces” was set for transferring parameter values from compressor to combustor and from combustor to turbine. Turbine rotation was set equal to compressor rotation, because they are on the same engine shaft [25]. The calculation was carried out in transient case for two operation modes: nominal and 0.5 of nominal.

3. Results and discussion

Velocity field shown on figure 2 was calculated at nominal operation mode. Temperature fields at the cross-section of recirculation mixing zone (see figures 3 and 4) and at the outlet (see figure 5 and 6) were calculated on nominal and on 0.5 of nominal operation mode. In addition, radial diagram of

IOP Conf. Series: Materials Science and Engineering **302** (2018) 012021 doi:10.1088/1757-899X/302/1/012021

temperature distortion at the outlet of CC was obtained by simulation (see figure 7). The radial distortion was calculated by:

$$\Theta_{rad} = \frac{T_{Gi} - T_C}{T_G - T_C}. \tag{1}$$

The fields and diagram are presented by the comparison with similar fields obtained by CC simulation (without compressor and turbine) and by integrated compressor/combustor simulation.

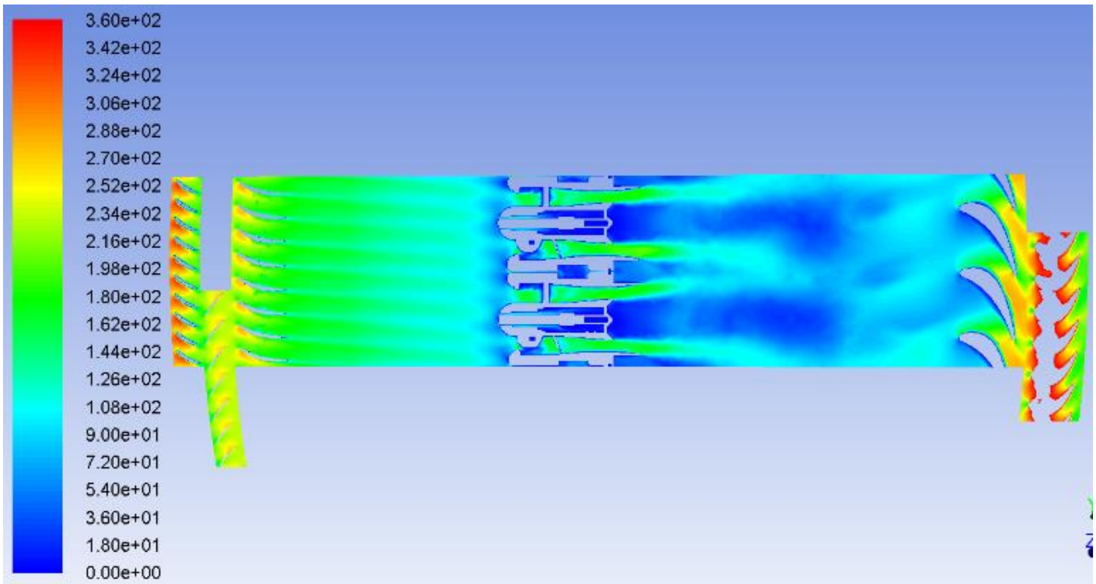
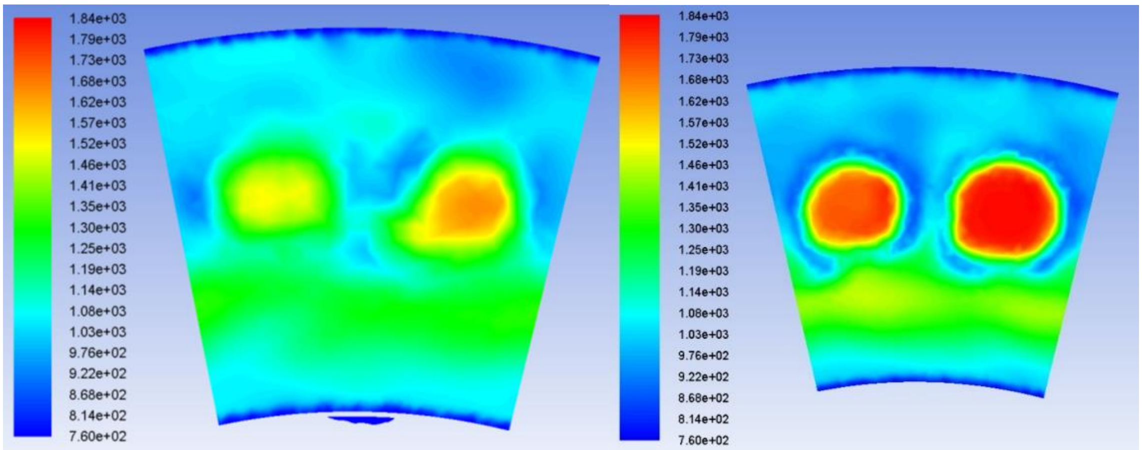
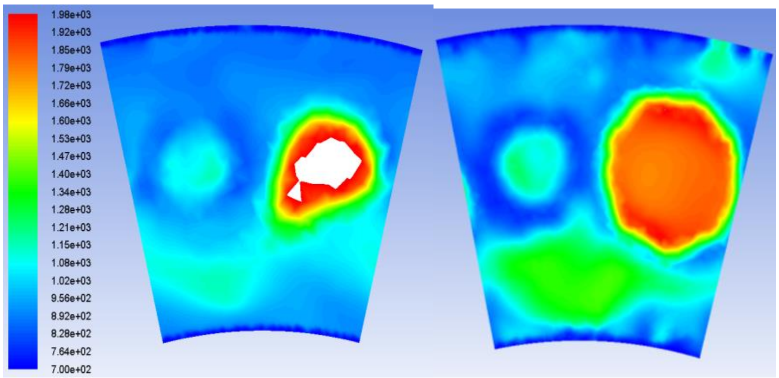


Figure 2. Velocity field at the radial section of engine core at nominal operation mode.



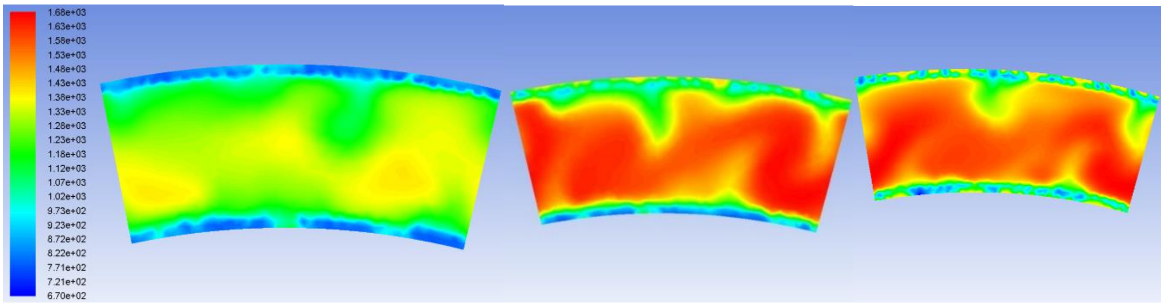
(a) (b)

Figure 3. Temperature field at the cross-section of recirculation mixing zone at nominal operation mode (a – cc with turbo compressor, b – combustor).



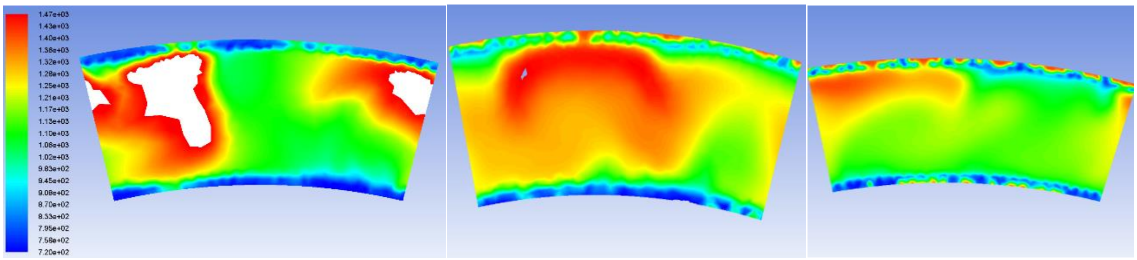
(a) (b)

Figure 4. Temperature field at the cross-section of recirculation mixing zone at 0.5 of nominal operation mode (a –cc with turbocompressor, b – combustor).



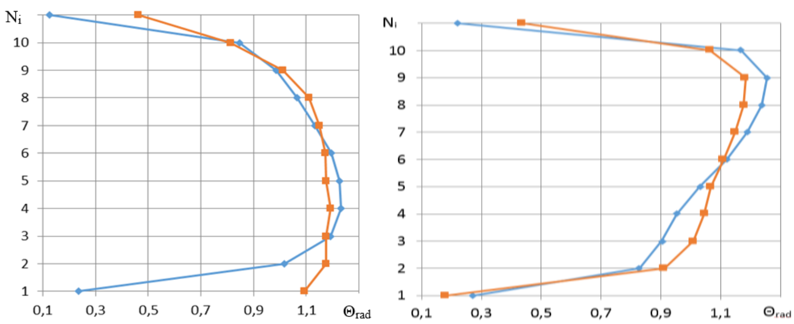
(a) (b) (c)

Figure 5. Temperature field at the cc outlet at nominal operation mode (a – cc with turbo compressor, b – combustor, c – cc with compressor).



(a) (b) (c)

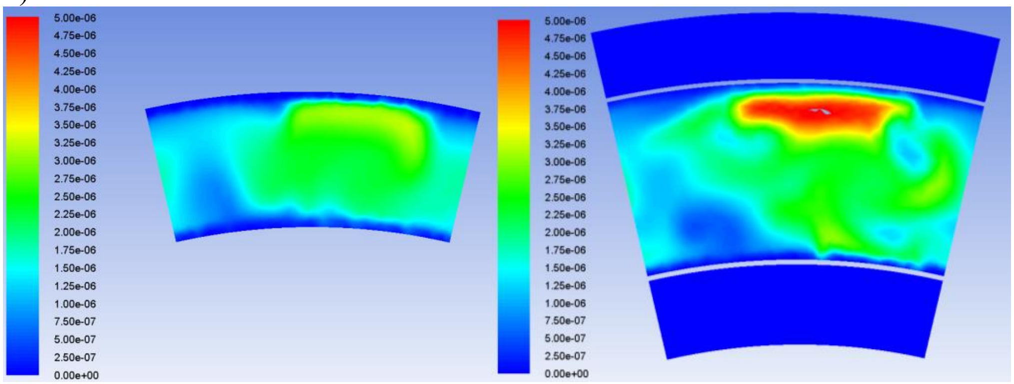
Figure 6. Temperature field at the outlet of combustion chamber at 0.5 of nominal operation mode (a – cc with turbo compressor, b – combustor, c – cc with compressor).



(a) (b)

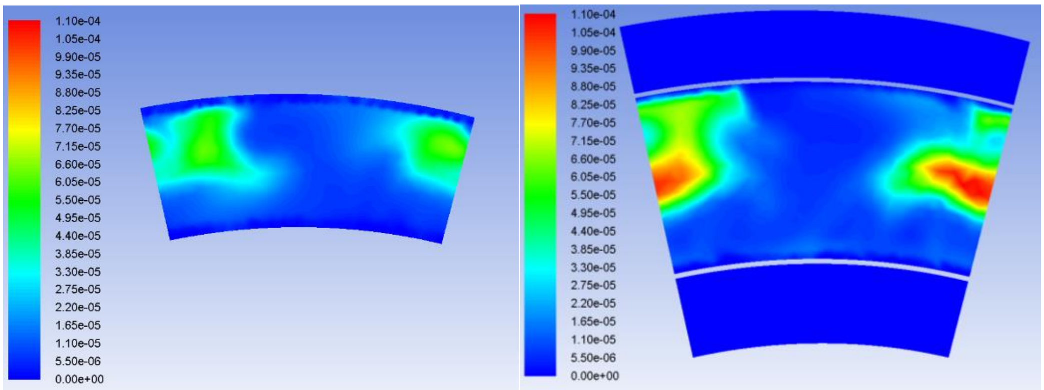
Figure 7. Radial diagram of temperature distortion at the outlet of cc at nominal (a) and 0.5 (b) of nominal operation mode (orange – combustor, blue – cc with turbo compressor).

Mass fraction of NOx was obtained to estimate the influence of different simulation cases (CC with or without turbo compressor) on performance characteristics at 0.5 of nominal operation modes (see figure 8-9).



(a) (b)

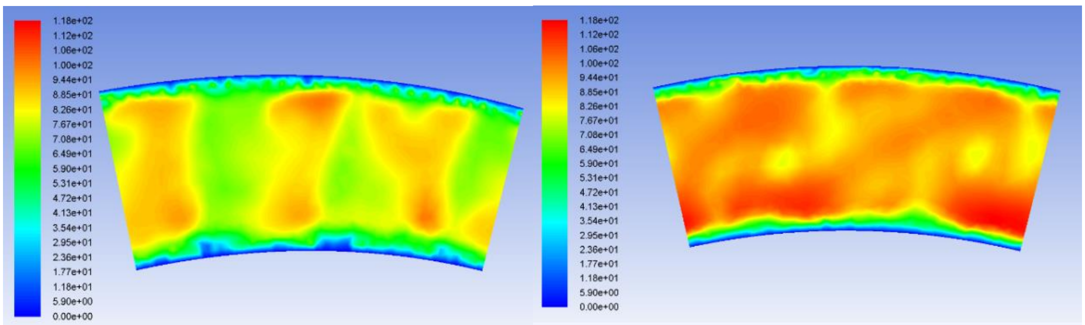
Figure 8. Mass fraction of NOx at the outlet of cc (a) and at the cross-section after dilution holes (b) for the cc simulation without turbo compressor at 0.5 of nominal operation mode.



(a) (b)

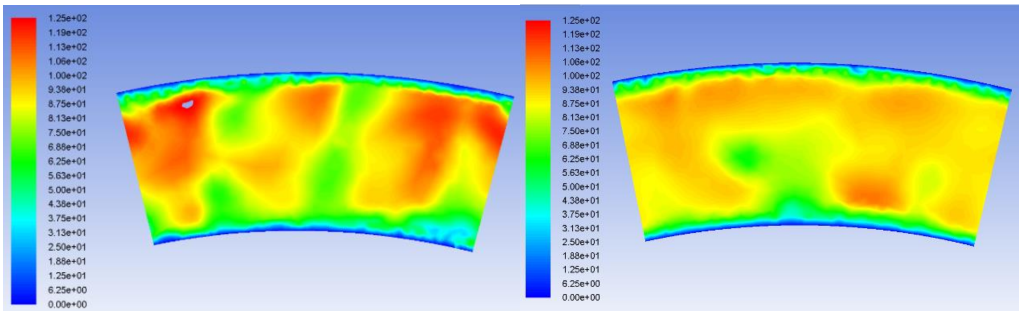
Figure 9. Mass fraction of NOx at the outlet of cc (a) and at the cross-section after dilution holes (b) for the cc simulation with turbo compressor at 0.5 of nominal operation mode.

Velocity and pressure fields were obtained by simulation for both operation modes. Figures 10 and 11 show the velocity changing at the CC outlet.



(a) (b)

Figure 10. Velocity field at the cc outlet for the cc simulation with turbo compressor (a) and without turbo compressor (b) at nominal operation mode.



(a) (b)

Figure 11. Velocity field at the cc outlet for the cc simulation with turbo compressor (a) and without turbo compressor (b) at 0.5 of nominal operation mode.

The comparison of simulation results show significant differences in the cases of combustion chamber simulation, integrated compressor/combustor simulation and CC with turbo compressor. Besides, differences occur in quantitative (see Fig. 3) and qualitative ratio (see Fig. 7). At both operation modes in the case of CC simulation with turbo compressor the flame contour, its size and temperature are different from similar parameters in the case of combustion chamber simulation. For example, the temperature difference is about 300 K. Since these parameters influence on organization of the CC workflow, it can be assumed that the received changes will affect the mixing zone. For example, mass fraction of NOx at the outlet of combustion chamber differs by two orders of magnitude, and velocity differs by 20 m/s. This is confirmed by the results presented in Figures 5,6–11.

Differences in obtained results can not be explained only by compressor influence. In this case calculation results of combustor simulation with the turbo compressor and combustor with only compressor will be equal. As a comparison shows the differences (see Fig. 5, 6) it is evident that the presence of turbine in the calculation affects the workflow in the combustion chamber. Thus, the influence of such elements as the compressor, the combustor and the turbine on each other, takes place in forward and reverse direction.

Calculation of total pressure loss for the combustion chamber with turbo compressor and without showed that the obtained values may differ from each other by up to 30%.

4. Conclusion

Summarizing all the above it can be noted that:

- The flow distortion after the compressor influences on the processes occurring in the combustion area of the combustion chamber.
- Changes in processes occurring in the combustion zone of the combustion chamber, lead to gas flow parameters change at the turbine inlet.
- Calculation of NOx emission values and a total pressure loss in the simulations of combustion chamber with and without turbo compressor gives different results.
- Operation of the turbine influences on the formation of the temperature field at the turbine inlet, particularly making it more distorted in circumferential direction.

The research of this field is ongoing in order to inspect the chemical processes more clearly. Reactor models are used. Course of chemical processes of the harmful substances formation after combustion chamber in a turbine is under estimation.

Acknowledgments

This work was supported by the Ministry of education and science of the Russian Federation in the framework of the implementation of the Program "Research and development on priority directions of scientific-technological complex of Russia for 2014-2020" (RFMEFI58716X0033).

References

[1] Kannan K V and Page G J 2016 Automated multi-code URANS simulation of compressor-combustor components. Proceedings of the ASME Turbo Expo 2016, GT2016-56904

[2] Kannan K V and Page G J 2014 Coupling of Compressible Turbomachinery and Incompressible Combustor Flow Solvers for Aerothermal Applications. ASME Turbo Expo 2014: Turbine Technical Conference and Exposition. ASME.

[3] Walker A D, Carrotte J F, Peacock G L, Spencer A and McGuirk J J 2013 Experimental Study of the Unsteady Aerodynamics of the Compressor-Combustor Interface of a Lean Burn Combustion System San Jose, 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference.

[4] Schlüter J U, Wu S K, Alonso J J and Pitsch H 2003 "Integrated RANS-LES Computations Of Turbomachinery Components: Generic Compressor/Diffuser", Center for Turbulence Research Annual Research Briefs ADA420749.

[5] Boudier G, Gicquel L Y M, Poinsot T, Bissières D and Bérat C 2007 "Comparison of LES, RANS and experiments in an aeronautical gas turbine combustion chamber," Proceedings of the Combustion Institute, 31(2), 3075-3082.

[6] Schluter J U, Wu X, Kim S, Alonso J J and Pitsch H 2004 "Coupled RANS-LES computation of a compressor and combustor in a gas turbine engine." In 40 th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit.

[7] Sehra A K 2006 "The numerical propulsion system simulation: A vision for virtual engine testing". In Proceedings of the ASME Turbo Expo, 2006: Power for Land, Sea, and Air, ASME, New York, USA.

[8] Matveev S S, Zubrilin I A, Orlov M Y and Matveev S G 2015 Numerical investigation of the influence of flow parameters nonuniformity at the diffuser inlet on characteristics of the GTE annular combustion chamber. Source of the Document Proceedings of the ASME Turbo Expo 4A, GT2015-42676

[9] Orlov M Y and Matveev S S 2014 Numerical simulation of an influence of a compressor and a turbine on characteristics of a combustion chamber of a small-sizes gas turbine engine. Life Science Journal 11 (11), 119, pp. 650-654

[10] Orlov M Y, Matveev S S, Makarov N S and Zubrilin I A 2014 Numerical modeling problems of operating process of combustion chambers of GTE and solution approaches. Source of the Document ARPN Journal of Engineering and Applied Sciences 9 (12), pp. 2894-2899

[11] Anisimov V, Zubrilin I and Orlov M 2016 Investigation of thermal and stress states of the annular combustion chamber flame tube walls. Proceedings of the ASME Turbo Expo, 2016. GT2016-57479.

[12] Orlov M Y and Matveev S S 2014 Numerical simulation application for the design and fine-tuning of small-sized gas turbine engine combustor. Source of the Document Open Mechanical Engineering Journal 8, A450, pp. 450-456

[13] Zubrilin I A, Dmitriev D N, Matveev S S and Matveev S G 2015 Numerical investigation of the nonreacting swirling flow structure downstream of industrial gas turbine burner with the central body. Proceedings of the ASME Turbo Expo 4A, GT2015-42181

[14] Matveev S S, Zubrilin I A, Orlov M Y, Matveev S G and Chechet I V 2016 Investigation of fuel distribution in partially premixed swirled burner with pilot flame. Proceedings of the ASME Turbo Expo 4B-2016, GT2016-57478

[15] Zubrilin R A, Matveev S S, Zubrilin I A and Matveev S G 2106 Gaseous fuel flame stabilization in a modular swirled burner. Proceedings of the ASME Turbo Expo 4B-2016, GT2016-57441

[16] Popov G M, Baturin O V, Kolmakova D A and Krivcov A V 2014. «Improvement results of TK-32 turbocompressor turbine with gas-dynamics and strength CAE-systems». International Journal of Engineering and Technology, 6(5), pp. 2297-2303.

[17] Kolmakova D A, Baturin O V and Popov G M 2014. «Knowledge lack impact assessment of the source date on numerical simulation results of operational process in axial flow turbine blade row». ARPN Journal of Engineering and Applied Sciences 9(12), pp. 2880-2889.

[18] Matveev V N, Popov G M, Baturin O V, Goryachkin E S and Kolmakova D A 2015. «Workflow optimization of multistage axial turbine». 51st AIAA/SAE/ASEE Joint Propulsion Conference 2015, Orlando, United States, p. 13

[19] Gibson M M and Launder B. E 1978, "Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer," J. Fluid Mech., 86 491–511.

[20] Launder B E 1989 "Second-Moment Closure: Present... and Future?", Inter. J. Heat Fluid Flow, vol. 10, no. 4. pp. 282–300.

[21] Launder B E and Reece G J 1975 “Progress in the Development of a Reynolds-Stress Turbulence Closure,” J. Fluid Mechanics, vol. 68, no. 3, pp. 537–566.

[22] Coppalle A and Vervisch P 1983 "The Total Emissivities of High-Temperature Flames". Combustion and Flame. 49. 101–108.

[23] Anderson W and Bonhus D L 1994 "An Implicit Upwind Algorithm for Computing Turbulent Flows on Unstructured Grids". Computers Fluids. 23(1). Pp. 1–21.

[24] Oijen A and Goey L P H 2000 "Modelling of Premixed Laminar Flames Using Flamelet-Generated Manifolds". Combust. Sci. Tech. 161. Pp. 113–137.

[25] ANSYS 15.0 User’s Guide.