Numerical Investigation of the Impact of the Compressor Operation Mode on Working Process of the Combustion Chamber

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 method of integrated compressor/combustor simulation was used to investigate the impact of flow distortion, appeared due to compressor blades, during the combustion chamber workflow. The method was improved in terms of generating a common grid and of principles of the boundary conditions settings. The geometric model includes four geometric volume bodies: guide vanes of the penultimate stage of high-pressure compressor, the impeller and guide vanes of the last stage and the flow path of combustion chamber. The calculation was carried out for some operation mode of the engine (nominal, 0.7 of nominal and 0.5 of nominal regimes) with and without compressor. The results were compared with the results of combustion chamber simulation without the compressor. Simulations showed that blade wakes extend up to the flame tube head. These wakes influence on the flame tongue, pressure field, temperature and velocity in the recirculation-mixing zone. It can influence on combustion efficiency, ecological performance and on temperature field at the combustor outlet. Thus, the simulations, which take into account combustion chamber and compressor, are more fully represent the characteristics of the working process of the combustion chamber and increase the efficiency of the design of new products.

1. Introduction
Currently the performance improving of modern gas turbine engines is related to improvement of workflow in a combustion chamber. Operation of combustion chamber is a part of engine workflow and associated with working process of a compressor and a turbine. Previously, a design and further development of compressor, combustion chamber and turbine were carried out separately. It does not allow reaching required characteristics of their joint operation at the engine. Problems identified at the stage of the test of engine-demonstrator are quite complicated to solve. Therefore, it is necessary to know the relationships between compressor workflow and parameters variations inside the combustion chamber to improve workflow of the engine more effective. It will reduce the financial and time costs for its creation and further development at the phase of design and improve product competitiveness.

The use of the CFD-methods at the design and further development phases can help to solve this problem. In this case, simulations can determine not only the integrated, but also the local values of the flow parameters. It significantly increases the opportunities in the case of optimization of working process.
Airflow through the axial compressor stage may be regarded as the flow through the diffuser channel system. Also, it is important to establish the flow distortion, appeared due to compressor blades, and its impact on the combustion chamber workflow.

Currently, development of the method of integrated simulation is very urgent. There are many papers related to this study [1-9]. The aim of these papers is development of integrated simulation of compressor and diffuser of combustion chamber. In addition, it includes the investigation of the compressor influence on the workflow at the diffuser. All these studies are the steps of achievement the goal – create virtual gas turbine engines (GTE). The aim of this paper is the improvement of the method of integrated compressor/combustor simulation and investigation of the compressor influence on the workflow in the flame tube of the combustion chamber.

2. Models and methods

The method of integrated compressor/combustor simulation was developed by creating a common mesh model of compressor and combustion chamber (CC). The object of research is combustion chamber of aero-derivative power plant working on gaseous fuel.

The geometrical model, which was the basis of the grid model, is shown in figure 1. It includes four different geometric volume bodies: the flow path of combustion chamber, the impeller and the guide vanes of the last stage and guide vanes of the penultimate stage of high-pressure compressor [10-11].

The sector of the geometrical model was used instead of full model to decrease time calculation. The angle of the combustion chamber and blade rows sectors was the same. Fuel rate depends on the thrust rating and for the even and odd burners it may differ. For example, the decrease of operation mode leads to decrease of fuel rate through even burners but fuel rate through the odd keeps constant. Hence, the sector includes two burners. The number of blades might differ from the actual number by no more than 10% to ensure the periodicity condition, agreement of compressor and combustion chamber sectors, and the accuracy of calculation. The periodical sector is the geometrical volume of compressor and CC. The whole circumferential model can be obtained by repeating the sector.

Finite-volume model was generated for each geometric model by advanced size function Proximity and Curvature in Ansys Meshing (see figure 2).

![Figure 1. Geometrical model of compressor and combustion chamber.](image1)

![Figure 2. Finite-volume model of compressor and CC.](image2)

One of the main conditions for the periodicity is the mesh identity on each side of the geometry. This condition can be achieved by using the grid function Match Control. Unstructured grid model contains less than 12 million elements. Skewness is an important quality parameter of the grid, which should not exceed 0.96. As a result, this parameter became equal 0.85, which proves that the grid is acceptable. Inlet and outlet of air and fuel were set as well as paired "interfaces" on contiguous faces blade rows and the combustion chamber.

Three-dimensional simulation package Ansys Fluent 16.2 was used for simulation of joint action of compressor and combustion chamber, because the simulation methods were well-mastered in previous
research works [12-17]. The methods of simulation of compressor and CC workflows were created and tested.

After mesh generation, mathematical model and boundary conditions were formulated. Mathematical model includes calculation models, which are necessary for workflow simulation in combustion chamber. Reynolds Stress Model (RSM), selected as a turbulence model, is one of the most complex turbulence models proposed by Fluent [18-20]. The calculations took into account the effect of the radiation, as the heat flux from the radiation is comparable to the heat flux from convection or conduction in combustion chamber. This is due to high level of temperature. Radiation model Discrete Ordinates (DO) was chosen for the simulation [21]. Adiabatic boundary conditions were used for walls. Model wsggm-domain-based (weighted-sum-of-gray-gases model) was used to calculate the absorption coefficient of the mixture [22]. Flamelet Generated Manifold was selected to simulate combustion processes in combustion chamber. Kinetic scheme GRI 3.0 was used to simulate mechanism of methane combustion [23]. Model of Turbulent flame speed was used with the FGM. The boundary conditions are specified by input parameters of pressure and temperature of the airflow coming from the impeller of the penultimate stage of the compressor, and fuel supplying to the combustion chamber.

The simulation on the grid model, which included only the combustion chamber without the compressor, were also performed for a more complete investigation of the compressor effect on the combustion chamber workflow. Averaged parameters were set on the inlet of the CC for the simulation without compressor. The parameters values were received by thermogasdynamic calculation. The compressor determined the parameters at the inlet of the diffuser for the case of CC with compressor.

Direction vectors were set in cylindrical coordinates, because the airflow runs onto the guide vanes at a certain angle: in the radial direction – 0, in the circumferential direction – 0.74, in the axial direction – 0.67.

Periodic conditions on the lateral faces of the combustion chamber and blade rings sectors were set by text commands. The interaction between certain previously created “interfaces” was defined to transfer values of the flow parameters from one body to another (see figure 3).

![Interfaces](image)

**Figure 3.** Interfaces on contact sides of the bodies.

The calculation was carried out in a transient case. It is necessary to specify certain time step and number of time steps required for the convergence. According to the recommendations proposed by software package ANSYS, the value of time step was calculated depending on the number of impeller blades. Number of time steps was set so that the impeller has three committed full rotations. [21].

### 3. Results and discussion

The integrated compressor/combustor simulation was carried out in a transient case for several operation modes of the engine (nominal, 0.7 of nominal and 0.5 of nominal regimes) with and without compressor. Three surfaces were used for results visualization (see figure 4). The URANS simulations showed that blade wakes extend up to the flame tube head (see figure 5). These wakes were obtained
by specific time step. These wakes influence on the flame contour (see figures 6-8) and on the temperature field at the exit of combustion chamber (see figures 9-11).

The figure 12 shows the radial diagram of outlet temperature obtained by simulation of three operation modes with and without compressor. The radial distortion was calculated by:

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

A comparison of radial diagrams shows that compressor rotation influences on the diagram view. As well as in the case of temperature circumferential distribution, the smallest distribution takes place at the nominal operation mode.

**Figure 4.** Surfaces mapping the calculation results.

**Figure 5.** Velocity distribution on cylindrical surface “a” of CC without (a) and with (b) compressor.
Figure 6. Temperature distribution at cross-section “b” without (a) and with (b) compressor at nominal operation mode.

Figure 7. Temperature distribution at cross-section “b” without (a) and with (b) compressor at 0.7 of nominal operation mode.

Figure 8. Temperature distribution at cross-section “b” without (a) and with (b) compressor at 0.5 of nominal operation mode.
Figure 9. Temperature distribution at the outlet “c” of the CC without (a) and with (b) compressor at nominal operation mode.

Figure 10. Temperature distribution at the outlet “c” of the CC without (a) and with (b) compressor at 0.7 of nominal operation mode.

Figure 11. Temperature distribution at the outlet “c” of the CC without (a) and with (b) compressor at 0.5 of nominal operation mode.
Figure 12. Radial diagram of temperature distortion at the outlet of the CC at nominal (a), 0.7 (b) and 0.5 (c) of nominal operation modes (orange with and blue without compressor).

Also, the fields of the rate of thermal NO distribution were carried out at the cross section behind the burner (see figure 13) and at the cross section on a distance of 70 mm behind the burner (see figure 14). It can be seen that the inclusion of the compressor leads to changes in the shape and size of the area of NOx formation zone and, consequently, to changes in its concentration in the combustion products.

The influence of the compressor on the concentration of NOx in the combustion products can be explained as follows. Taking the compressor into account in the simulation changes the velocity field in front of the burner swirler (see figure 15). These flow disturbances impact on the fuel distribution in the burner and behind it (see figure 16).

Figure 13. Rate of thermal no at the cross-section behind the burner of the CC without (a) and with (b) compressor at nominal operation mode.

Figure 14. Rate of thermal no at the cross-section on a distance of 70 mm from the burner of the CC without (a) and with (b) compressor at nominal operation mode.
At the nominal operation mode of the engine for even and odd burners, the equivalence ratio is equal. However, as shown in figure 16, local fuel distribution is different for the CC simulation with and without compressor. Hence, the disposition of fuel rich local zones is varies, as well as high temperature zones. It leads to differences in concentrations of NOx at the outlet of the combustion chamber. Thus, the calculation data of NOx shows significant differences in the level of its formation for the both cases.

4. Conclusion
Summarizing all the above it can be noted that:

- The simulation taking the compressor into account allows obtaining the qualitatively and quantitatively different data from the results of simulation without compressor.
- The influence of the compressor in the simulation depends on the design of flame tube head, burner and engine operation mode. The results showed that the least impact of the compressor is at the nominal operation mode and the biggest – at the 0.7 of nominal regime.
• Blade wakes extend up to the flame tube head and influence on the flame contour, pressure field, temperature field and velocity at the recirculation-mixing zone. These changes can effect on combustion efficiency, ecological characteristics and outlet temperature.

• The simulation of combustion chamber with compressor can better reflect the special features of combustion chamber workflow in a greater extent. It increases the efficiency of design new products.

Developed method can be used for optimization of combustion chamber workflow with interaction of other GTE components, and during its development. In addition, it can be used for analyzing the effect of various factors on the combustion chamber workflow. For example, the location of aircraft in space, changes of GTE operation mode, foreign objective in the engine flow path etc.

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
[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 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] Popov G, Kolmakova D, Shklovets A and Ermakov A 2015 Optimization of the axial compressor flow passage to reduce the circumferential distortion IOP Conference Series: Materials Science and Engineering 9th International Conference on Compressors and Their Systems London United Kingdom, Article number 012053
[11] Matveev V N, Baturin O V, Popov G M and Egorov I N 2014 Seven-stage axial compressor optimization Engineering Optimization IV - Proceedings of the 4th International Conference on Engineering Optimization ENGOPT 2014 Lisbon Portugal pp 821-826
[12] 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

[13] 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 GT2016-57479

[14] 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 pp 450-456

[15] 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

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

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

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

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

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

[21] ANSYS 16.2 User’s Guide.

[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 L1994 An Implicit Upwind Algorithm for Computing Turbulent Flows on Unstructured Grids”. Computers Fluids. 23(1) Pp. 1–21.