Numerical Simulation Application for the Design and Fine-Tuning of Small-Sized Gas Turbine Engine Combustor

Mikhail Y. Orlov* and Sergey S. Matveev

Samara State Aerospace University (SSAU), Moskovskoye shosse, 34, Samara, 443086, Russia

Abstract: The small-sized GTE are used more and more both as the basis of power plants and power machines for various purposes. The technical and operational characteristics of these engines are largely determined by the design quality of their combustion chambers (CC). One of the most advanced methods of CC scientific investigation is the bench testing. However, such CC operation study is complicated by labour-intensiveness of an experiment and its significant costs. Therefore, in recent years GTE designers started to use widely the numerical simulation for these purposes. However, the level of such calculation development does not allow often to solve the issue of CC fine-tuning in order to achieve the required parameters. It is also difficult to determine the reliability degree of such calculations. The preliminary calculation of the combustion chamber was performed first of all. Its purpose is the CC shape development and its determination of its geometric dimensions. At that, the main requirement was to ensure the possibilities of one CC placement between the previously-designed compressor and the turbine and the performance of a whole set of technical specifications by the engine. At the second stage using The CC fine-tuning was performed in order to optimize the performance by using CFD-calculation. At that the calculated method was used for which the verification has been performed by the bench tests. Such a CC fine-tuning allowed to reduce the product development period and improve its characteristics.

Keywords: Combustion chamber, design calculation, numerical simulation, small-sized gas turbine engine.

INTRODUCTION

The development of small gas turbine engines (SGTE) occurs in the direction of their effectiveness increase. The successful solution of this problem depends on the combustion chamber (CC) perfection to a large extent. This unit must ensure a stable and non-impulse combustion in a wide range of mode parameters with high combustion efficiency and a uniform das temperature field on a turbine inlet [1, 2].

Currently the numerical simulation is widely used in the combustion chamber design process [3,4]. The modern CFD calculation methods may describe well the qualitative and quantitative changes of the flow parameters, show of the tear-off zones presence and location and also show the dependences of the flow parameters change on the geometric, regime and other parameters [5,6]. That's why the numerical methods of gas dynamics shall be used also for the final design of the combustion chambers as the means of legality check concerning the design solutions and the evaluation of various constructive measures impact on the product and its components effectiveness [7-9].

This paper presents the example of the advanced CFD-technologies use for the design and fine-tuning of small-sized GTE CC.

METHODOLOGY

Two calculation methods were used during the study. The first method was used for the preliminary calculation of the combustion chamber in order to determine its basic dimensions and structural shape. The main source for the projected CC calculation are:

G_a-air flow, P^*_k-the pressure at the combustion chamber inlet, T^*_k-the air temperature at the combustion chamber inlet, 1-σ_k-the coefficient of total pressure restoration, η_m-fuel combustion efficiency, α_k-excess air coefficient of a combustion chamber.

Standard geometric dimensions of the CC annular flow part are shown in Fig. (**1**).

The technique developed on the basis of available statistical data concerning the CC design and operational parameters of various circuit solutions [1] and recommendations [2] was used to calculate the structural and integral parameters of the designed combustion chamber.

Basic geometrical CC parameters calculation method includes the following stages:

1. According to the average speed of the midlength section of the prototype flame tube the area and the diameter of the flame tube midlength section is determined;
2. The flame tube length $L_{жт} = k_1 * D_{ж}^{mid}$ is determined according to the statistical data or the prototype from the following ratio $\frac{L_{жт}}{D_{ж}^{mid}} = k_1$
3. The flame tube volume is determined.

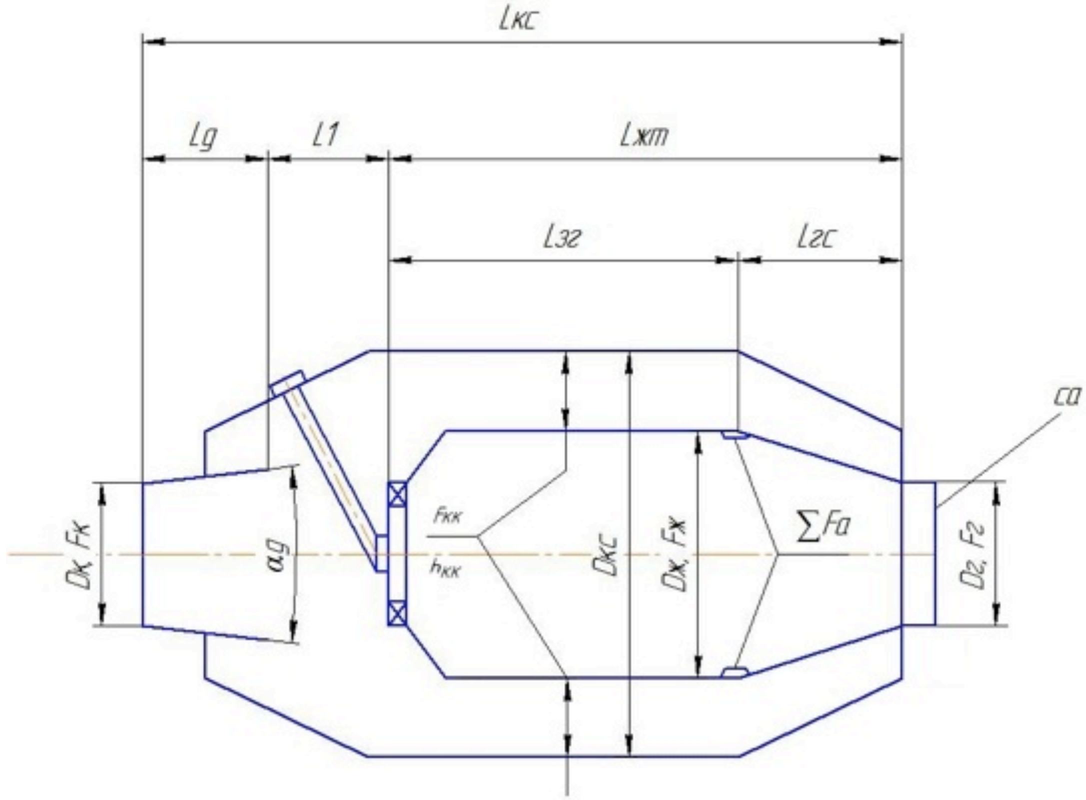

Fig. (1). Typical scheme and the main designations of external tubular CC.

4. The total effective area of the flame tube holes (including front device) is determined.

   $F_{эф} = G_в * \sqrt{2 * \rho_к^* * \Delta P_{ж}}$ where $\Delta P_{ж}$ is the pressure drop of the flame tube

   In the first approximation we take $\Delta P_{ж}$ is a constant one across the flame tube length

   $\Delta P_{ж} = \sigma_{ж} * P_к^*$, where $\sigma_{ж}$ is the pressure losses of the flame tube (for modern combustion chambers usually $\sigma_{ж}$ =3…5%, $\sigma_к = \sigma_д + \sigma_{ж}$).

   The geometric area of the holes makes $\Sigma F_{отв} = \frac{F_{эф}}{\bar{\mu}}$, where $\bar{\mu}$ is the average coefficient of air flow through the holes (including front device).

5. The air supply rule is selected either by the prototype or according to the literature recommendations concerning the combustion chamber design [1, 2, 9]. Usually, the air flow through the front-line units ($G_{фу}$)makes about 20% for the CC SGTE; The air flow through the primary zone $G_{взз} \leq 20\%$; through the mixer it makes $G_{см} \leq 40\%$; for the flame tube cooling it makes $G_{охл} \leq 20\%$.

6. The entry area into the combustion chamber $F_к$ and the diameter $D_к$ are determined by the reduced rate $\lambda_K = W_K / a_{KP}$ where $a_{кр}$ is the critical sound speed.

   $a_{кр} = \sqrt{2 * \frac{\kappa}{\kappa+1} * R * T_к^*}$, [m/s]. For the modern combustion chambers of small gas turbine engines $\lambda_K = 0,1...0,25$; the area of the diffuser entrance is determined according to the formula $F_к = \frac{G_в}{\rho_к * W_к} [м^2]$; the CC entrance diameter is determined according to the formula $D_к = \sqrt{\frac{4 * F_к}{\pi}}$

7. Let's determine the diffuser dimensions with a sudden expansion.

   According to the specified structure $\frac{L_g}{D_к} \leq 1$; $\alpha_g \approx 6 \dots 7°$; $L_1 = (0,5 \dots 2) D_к$;

8. The annular channel area $F_{кк}$ and the inside diameter of the combustion chamber case $D_{кс}$.

   - Let's determine the outer diameter of the flame tube

     $D_{ж}^{нар} = D_{ж}^{mid} + 2 * \delta$, where δ is the FT wall thickness considering the design;

   - Let's specify the annular channelheight $h_{кк}$, usually $h_{кк} \geq 0,01 \dots 0,015 m$;

   - Let's determine $D_{кс} = D_{ж}^{нар} + 2 * h_{кк}$; $-F_{кк} = \frac{\pi}{4}(D_{кс}{}^2 - D_{ж}^{нар2})$;

   - Let's define the maximum velocity value in the annular channel (assuming that all air is passed through the annular channel).

     $W_{кк}^{max} = \frac{G_в}{\rho_к^* * F_{кк}}$ and compare it with the recommended values.

9. Integralcharacteristics of the combustion chamber.

- Reduced air flow $G_{впр} = \frac{G_в\sqrt{T_к^*}}{P_к^*}$;
- Residence time$_{пр} = \frac{P_к^* * V_{жс}}{R * T_к^* * G_в}$

  where $P_к^* - Па$; $R - \frac{Дж}{кг*К}$; $G_в - кг/с$; $V_{жс} - м^3$; $T_к^*$ –К.
- Forcing coefficient = $K_V = \frac{G_в}{(P_к^*)^{1.25} * T_к^* * V_{жс}}$,

  $K_V = \frac{G_в}{(P_к^*)^{1.25} * T_к^* * V_{жс}}$, $(кг/с)/(кПа^{1,25} * К * м^3)$;
- The completeness of fuel combustion

  $1 - 0,8K_V^2$, $(кг/с)/((кг/см^2)^{1,25} * К * м^3)$;
- Volumetric combustion intensity $Q_V = \frac{H_U * G_m * \eta_г}{V_{жс} * P_к^*}$ $[\frac{кВт}{м^3 * Па}]$;

The article also used the numerical simulation technique of the operational process in SGTE using the ANSYS package. In terms of the numerical calculations one may define its following stages:

- geometric model development;
- network model generation;
- mathematical model development;
- calculation performance;
- visualization and the analysis of results.

The geometric model development was carried out on the basis of the appearance and geometric dimensions of the CC obtained by a preliminary calculation.

Then the boundary and initial conditions were stated. There are several following models fuel-air mixtures formation for the volume chemistry of the gaseous medium calculation in the ANSYS Fluent software package:

- Species transport (components transfer);
- Non-premixed Combustion (without preliminary mixing);
- Premixed Combustion (premixed mixtures);
- Partially Premixed Combustion (partially mixed mixtures);
- Composition PDF Transport;

We used the first model Species Transport. This model defines the components mixing process by solving the convection, diffusion and the component change equations in the chemical reaction process for each mixture component. This is a time-consuming model from the calculation point of view, but it has the required accuracy.

In general, the volume combustion processes calculation procedure is as follows:

- the calculation of cold non-reacting flows on a coarse grid model is performed using the simple models of turbulence. Thus, we obtain the component concentrations distribution of the concentrations among the basic flow parameters;
- the turbulence parameters are changed for more accurate ones, the solver settings are adjusted to obtain a more accurate picture of the flow;
- the mixture burning process is simulated, the temperature higher than required one for this composition ignition is set in some areas of the computational domain for this purpose;
- the adaptation of the computational domain grid in the areas with the largest gradients of the flow parameters (velocity and temperature) is performed;
- the calculation on the adapted grid is performed with the solver parameters that determine the increased accuracy of the calculation results.

A single-stage reaction of propane combustion is used as the chemical reaction design scheme:

$$C_3H_8 + 5O_2 = 3CO_2 + 4H_2O$$

This scheme and all the reaction parameters necessary for the correct calculation and the component properties are available in ANSYS Fluent standard database. The isobaric heat capacity $C_p$is set as a polynomial of the temperature to get the correct temperature field. The following parameters are used as the boundary conditions:

- Air flow *via* CC
- Gas temperature at the CC inlet
- The reduced flow rate at the inlet;
- Fuel consumption;
- Air excess ratio;
- Static pressure at the outlet.

RSM model (the model of Reynolds stresses transfer) is used as a turbulence model. This model is one of the most complex turbulence models proposed by Fluent. This model, in contrast to the models of k-ε family, does not use the assumption of turbulent viscosity isotropy, and solves the transfer equations for the Reynolds stresses, together with the equation for the turbulent dissipation rate ε.

Since the "RSM" model describes the effects of curvature, twist, rotation, an abrupt voltage change between the layers more strictly than two-parameter turbulence models, it has a greater potential for the accurate calculation of complex flows. However, "RSM" model has some simplifications that have been taken to set up the equations of Reynolds stresses transfer. The use of this turbulence model is recommended in cases when the turbulent flow anisotropy has a dominant influence on the turbulent flow nature (cyclones, strongly swirling flows in the combustion chambers, the rotating fields, the secondary flows in channels caused by large normal stresses, etc.).

There are several models that determine the chemical reaction rate with the flow turbulence in the ANSYS Fluent software package to calculate combustion transfer of chemical components: In this case "The Eddy-Dissipation-Concept Model" was used as the combustion model. It allows you to simulate the detailed chemical mechanisms in turbulent flow. It is assumed that the reactions within this model occur in small turbulent structures, called thin scales.

EDC model may include the detailed chemistry in a turbulent flow.

The considered technique is validated during the comparison of numerical simulation data with the carried out experimental studies results including the use of laser-optical measurements at the fire bench for small combustion chambers.

RESULTS

The constructive shape and the basic dimensions of the SGTE combustion chamber were obtained at the first stage using the calculation method.

The comparison of the integral parameters calculated values with the statistical data of modern combustion chambers with various dimensions was performed. This comparison showed that most parameters of the designed camera are in the statistical intervals for existing combustion chambers. This indicates that the developed CC will have sufficiently high hydraulic performance and a wide range of stable operation.

Based on these data a simplified geometric model of the combustion chamber was developed. Also, the verification of the combustion chamber geometry conformity with the projected overall SGTE layout was performed. Then, the estimated area-(Fig. **2**)-was marked on the geometric CC model.

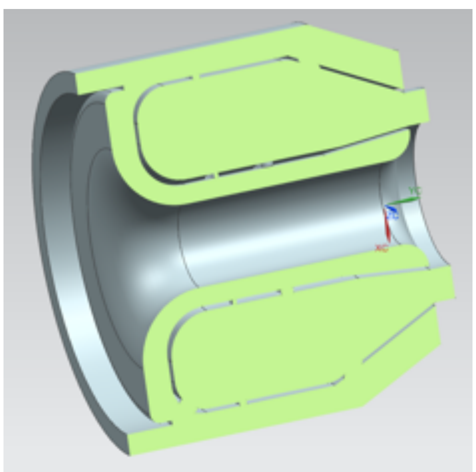

Fig. (2). The combustion chamber estimated area.

The numerical simulation of the operational process of the projected combustion chamber was performed using the ANSYS package according to the method of the operational process numerical simulation to obtain the estimated area.

Figs. (**3**, **4**) show the estimated areas of full pressure and speed.

The local values of the velocity evaluation and the total pressure losses allowed to make an assumption that on the basis of the selected CC image one may obtain the required characteristics.

During the next step, based on the data analysis of the preliminary calculation the design of modified flame tube was performed. According to the described above method the following parameters were calculated: the flame tube

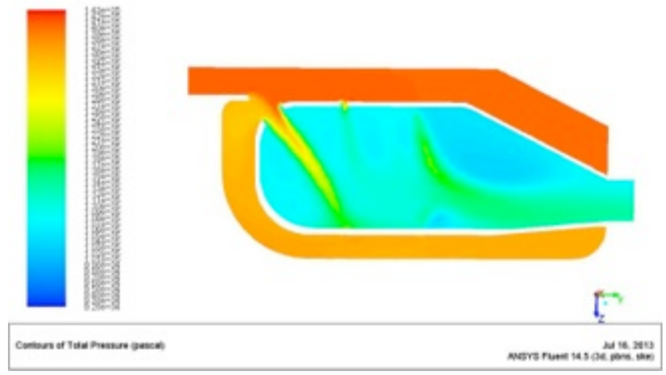

Fig. (3). Full pressure field.

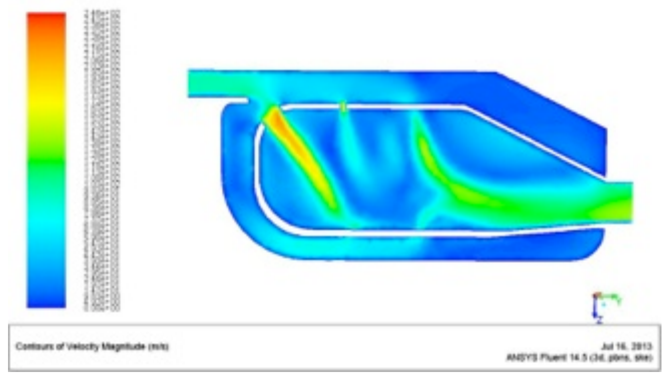

Fig. (4). Velocity field.

volume, its height, the total area of the holes in the FT, as well as the air distribution regularity. The CC swirler was designed. According to the calculation results the corresponding changes were introduced into the original geometric CC model. For example five rows of holes for the outer shell of the flame tube and four for the internal one were developed. They differed from the original version by size and location. The Figs. (**5-7**) present the fields of the full pressure, velocity and temperature distribution for the combustion chamber new version.

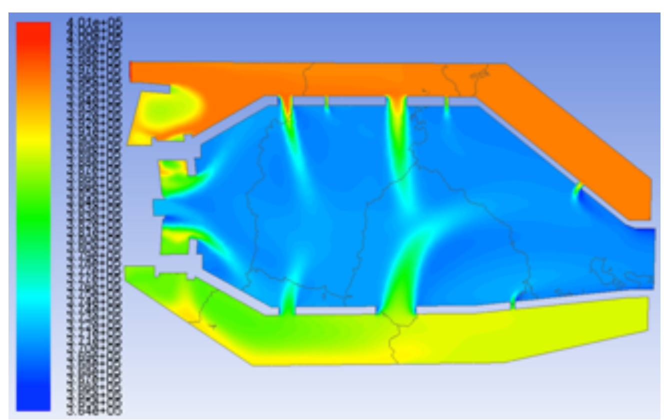

Fig. (5). Full pressure field.

Fig. (**5**) shows that the pressure in the lower annular channel is considerably lower than in the top one. This means that it is necessary to reduce the total area of the openings in the inner shell. Figs. (**6**, **7**) show that the air streams coming through the holes into the flame tube have a greater penetration depth. Therefore, the flame tube amount

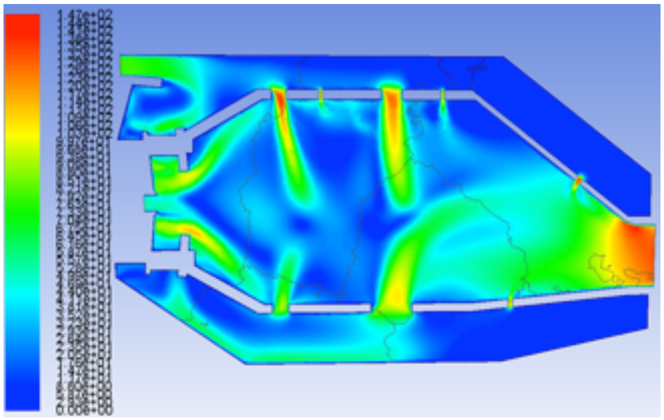

Fig. (6). Velocity field.

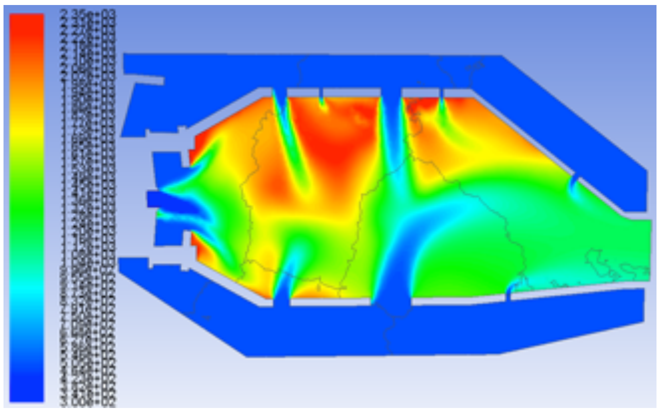

Fig. (7). Temperature field.

is well distributed into the combustion and mixing zones. However, the high temperature area for a given FT configuration is located near its upper wall, which may be the cause of burnout and the CC service life reduction.

To solve this problem the caps were added into the CC design of the COP were added. These caps are located on the swirler outlet (Fig. **8**) and aimed to push the high temperature zone from the wall.

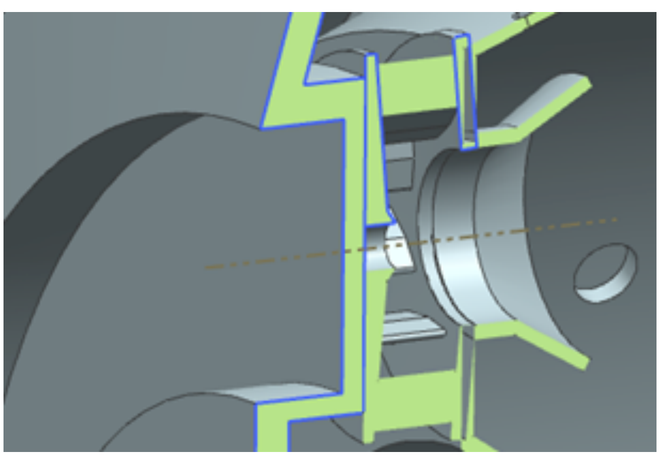

Fig. (8). The swirler modification with a cap.

CC geometric and grid models were redeveloped. The full pressure, velocity and temperature fields calculation results for this CC type are presented by Figs. (**9-11**).

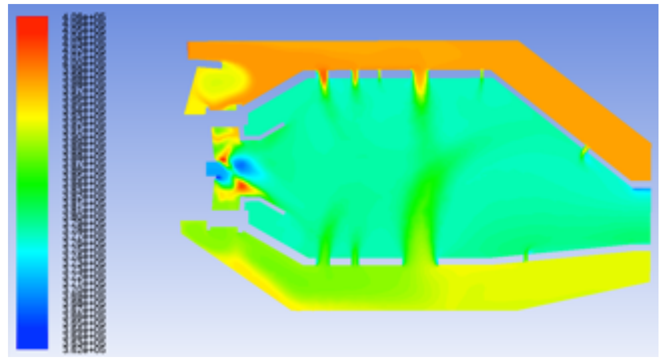

Fig. (9). Full pressure field.

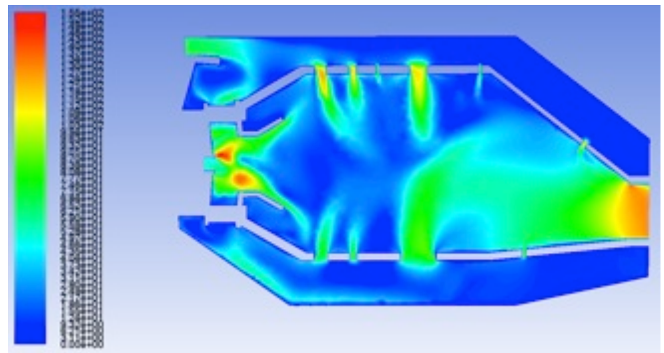

Fig. (10). Velocity field.

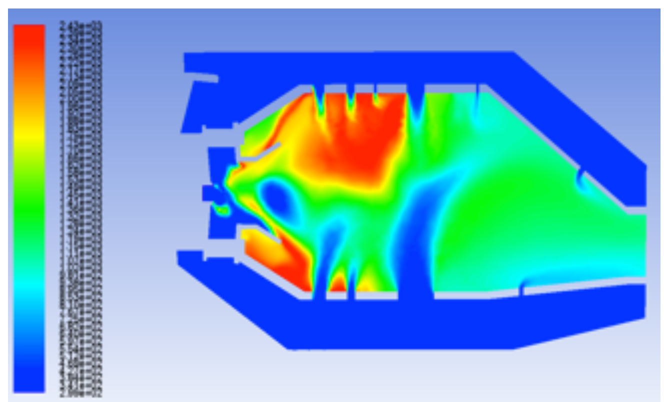

Fig. (11). Temperature field.

The temperature estimated field shows that this design solution remained almost unchanged concerning the high temperature zone location. Therefore, it was decided to design the slots in the flame tube (Fig. **12**) for the film cooling of the walls.

Re-calculations were performed, the results of which are shown by Figs. (**13-15**).

The figures show that through these activities managed to push the high temperature zone from the outer FT wall. However, the temperature field has a large non-uniformity at the CC outlet. This non-uniformity creates problems ensuring the turbine resource.

A series of calculations for different hole sizes and their amounts in the flame tube walls was performed. The calculation purpose was to find the best variant ensuring the series of requirements implementation concerning:

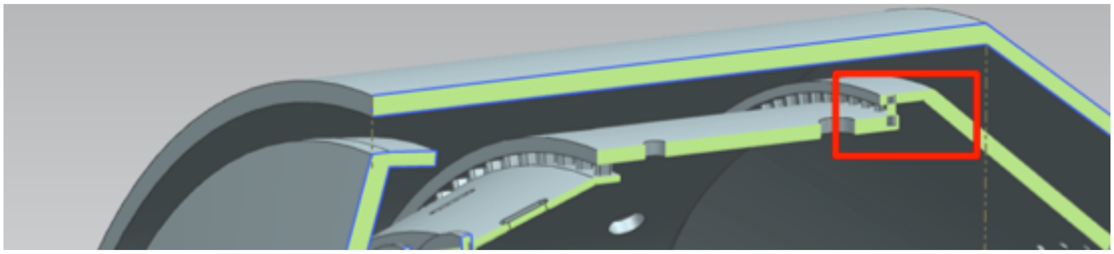

Fig. (12). FT case cracks.

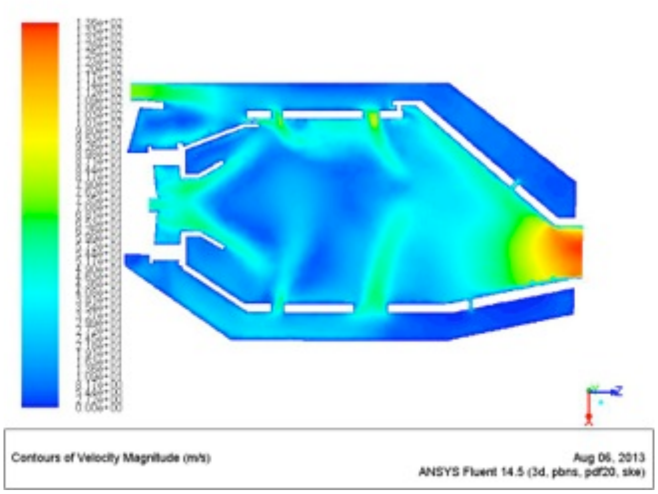

Fig. (13). Velocity field.

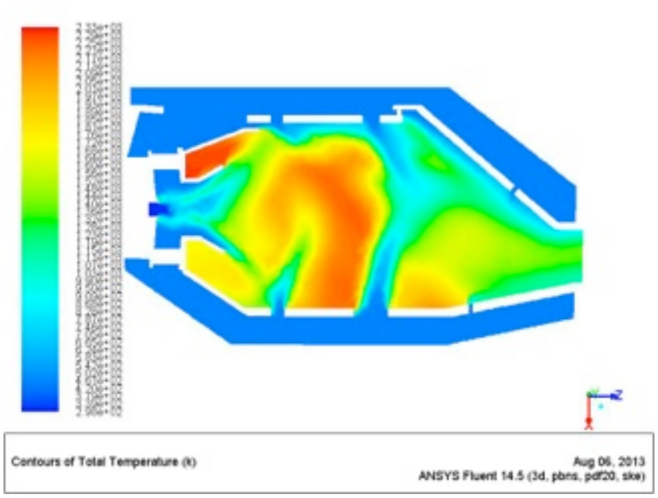

Fig. (14). Temperature field.

- the normal high temperature zone distribution relative to the flame tube walls,
- the level of total pressure loss in the chamber,
- the formation of radial temperature field non-uniformity diagram at CC output, which is within the tolerance parameters of the SGTE turbine.

CC model configuration was obtained as the result of the performed operation. The Figs. (**16**, **17**) demonstrate the temperature field in the longitudinal section of the chamber and the temperature field at the output for this model.

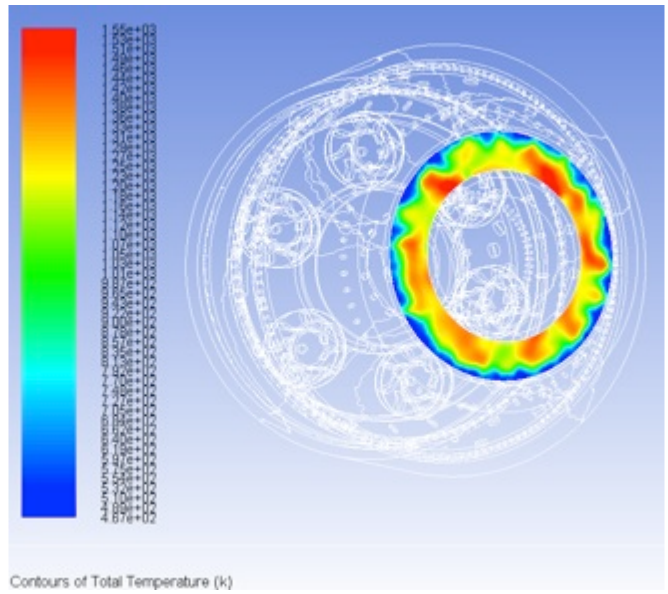

Fig. (15). Temperature field at CC outlet.

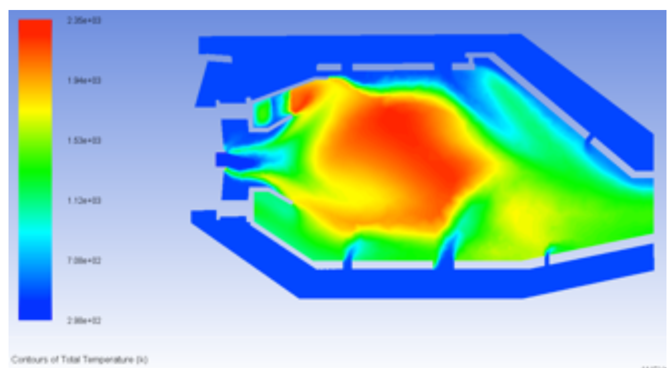

Fig. (16). Temperature field.

The obtained CC variant fully satisfied its consumer demands and was used for the manufacture of a full-scale product prototype.

DISCUSSION

In this paper we used an original method of a SGTE CC preliminary calculation based on the years of its designing and fine tuning experience [1]. There are also no evidence of a similar study in Russian sources, where numerical simulation is carried out for the whole small combustion chamber and not for the part of it. The accuracy of calculation parameters obtained by numerical simulation is confirmed by the fire tests.

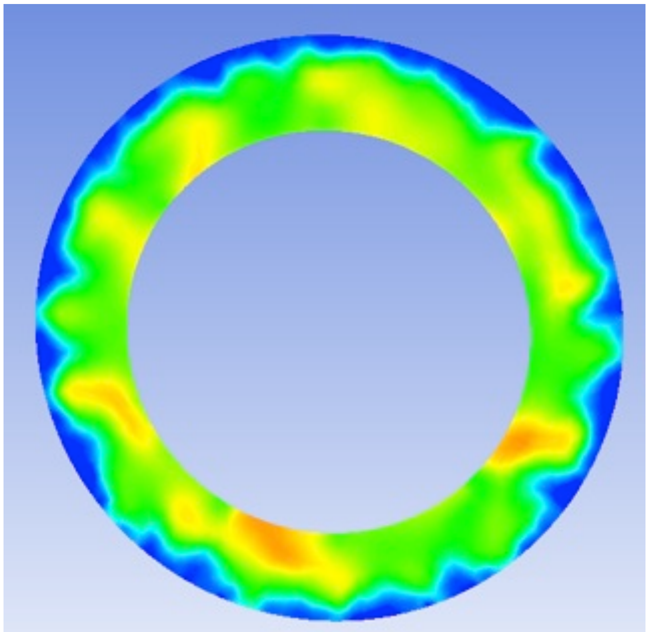

Fig. (17). Temperature field at CC output.

CONCLUSION

The design of the SGTE combustion chamber was performed during the operation. The total pressure losses reduced from 6% to 3%. The temperature non-uniformity field at the chamber outlet is tuned finely in terms of turbine resource provision. The developed area of reverse currents was obtained by the flame tube, which indirectly indicates the possibility of a good flame stabilization and sufficiently complete fuel combustion in the primary zone;

In scientific terms:

- The technique of design calculations for the annular small combustion chamber is developed;
- The technique of the working process numerical simulation in the combustion chamber of a small-size GTE is developed.

CONFLICT OF INTEREST

The authors confirm that this article content has no conflict of interest.

ACKNOWLEDGEMENTS

This work was financially supported by the Russian Federation Government (Ministry of Education) on the basis of the resolution №218 issued on 09.04.2010 (theme code 2013-218-04-4777).

This work was supported by the Ministry of education and science of the Russian Federation in the framework of the implementation of the Program of increasing the competitiveness of SSAU among the world's leading scientific and educational centers for 2013-2020 years.

REFERENCES

[1] Kovylov, Yu. L., Krasheninnikov, S.V., Lukachev, S.V., Tsyganov, A.M. The generalized characteristic of the combustion chamber for a gas-turbine engine(1999) Thermal Engineering (English translation of Teploenergetika)46(1)PP. 35-41.

[2] Lansky A.M. Small turbine engine combustors operation process / A.M. Lansky, S.V. Lukachev, S.G. Matveev.-Samara: SNTS Academy of Sciences, 2009-335 pages: illustrations.

[3] Matveev S.G. Joint calculation of small-sized turbine engine gas generator in ANSYS CFX // M. Y. Orlov, S.S. Matveev, V.S.Zinkovsky, A.V.Kryvtsov / Bulletin of the Samara State Aerospace University named after the academician S.P. Korolev (the National Research University).-2012. № 3-3 (34).-pages 293-298.

[4] Matveev S.G. Propane combustion processes modeling during the GTE internal combustion chamber transfer into gaseous fuel // A.M. Lansky, M.Y. Orlov, V.Y.Abrashkin, D.N.Dmitriev, I.A.Zubrilin, A.V. Semenov / Bulletin of the Samara State Aerospace University named after the academician S.P. Korolev (the National Research University).-2011. № 5.-pages 168-178.

[5] Matveev, S.G. Flow aerodynamic structure modeling in the combustion chamber of small size GTE using CAE-systems // M.Y. Orlov, V.Y.Abrashkin, I.A. Zubrilin, S.S. Matveev / Bulletin of the Samara State Aerospace University named after the academician S.P. Korolev (the National Research University).-2011. № 5.-pages 179-187.

[6] Bantikov, D.Y.Multi nozzle low emission combustor of modern TFE-creative heritage of N.D.Kuznetsov // V.I. Vasiliev, V.N.Lavrov, Y.I.Tsybizov, S.G. Matveev, M.Y. Orlov, I.A.Zubrilin / Bulletin of the Samara State Aerospace University named after the academician S.P. Korolev (the National Research University).-2012. № 2 (33).-pages 17-24.

[7] Mingazov, B.G. The applicability analysis of combustion models for the calculation of multi nozzle GTE combustion chamber // B.V.Yavkin, A.N.Sabirzyanov, A.V.Baklanov / Bulletin of the Samara State Aerospace University named after the academician S.P. Korolev (the National Research University).-2011. № 5 (29).-pages 208-214.

[8] Kostyuk V.E. Effect of flow non-uniformity on the hydraulic resistance of the aircraft GTE combustion chamber tear-off diffuser // E.I.Kirilash, I.F.Kravchenko, T.V.Stepanova, E.V.Kalinichenko / Aerospace engineering equipment and technology.-2008. №7 (54).-pages 99-104.

[9] Piralishvili, Sh.A. The influence of the compressor flow output parameters on the hydraulic resistance of a separated combustion chamber diffuser // M.M.Guryanova / Bulletin of the Rybinsk State Aviation Technological Academy named after P.A.Soloviev.-2012.-№2 (23).-pages 203-211.

Received: September 10, 2014 Revised: November 5, 2014 Accepted: November 5, 2014