Journal of Engineering Physics and Thermophysics, Vol. 79, No. 4, 2006

NUMERICAL ANALYSIS OF AERODYNAMICS AND HYDRAULIC LOSS IN THE GTÉ-150 COMBUSTION CHAMBER OF GAS-TURBINE POWER PLANT WITH THE USE OF THE FLUENT SUITE

D. A. Lysenko[a] and A. A. Solomatnikov[b]

UDC 621.438

The aerodynamic structure of the flow in the combustion chamber of the GTÉ-150 gas-turbine power plant has been simulated numerically by means of the FLUENT suite CFD. The influence of the processes of turbulent heat and mass transfer (and combustion) on the hydraulic loss in the combustion chamber has been investigated. Detailed comparison of the calculated and experimental data on the total pressure loss both n the combustion chamber proper and in its individual units has been carried out.

At the present time the development of modern computer technologies in the field of both computer engineering and software (including CAD/CAE technologies and CFD methods of computer gas dynamics) permits solving increasingly difficult and complex engineering problems in the field of gas-turbine building. One of such problems is the analysis of the aerodynamic and thermal state of the combustion chambers of gas-turbine engines. It is customary to assume that the characteristic features of modern combustion chambers are the complex geometry of the design and the complex mechanism of the physicochemical processes proceeding in the flame zone. The presence of such features leads to the appearance in the combustion chamber of a complex dynamic (nonstationary) interaction of vortex structures having a wide spectrum of various time and geometric scales that arise in the process of turbulent heat and mass transfer.

The processes of turbulent heat and mass transfer proceeding in the active volume of the combustion chamber strongly influence the efficiency and reliability of its operation. Both the radiative and convective transfer of thermal energy are largely determined by the aerodynamic structure of the primary and secondary air flows, the combustion products, and the cooling system. The physical mechanisms of convection and thermal radiation are extremely important under the conditions of velocity, temperature, and concentration inhomogeneities observed in the flame space of the combustion chamber. As was shown in [1], the higher the inhomogeneity of the temperature and concentration fields of the flame, the higher, as a rule, its effective temperature determining the radiant heat flux incident on the "flame" surfaces and, consequently, the higher (all other things being equal) their temperature level. Therefore, the processes of fuel-oxidizer mixing play the key role in the aerodynamic flow structure. Intensification mixing in the flame zone can be attained by enhancing turbulence and generating certain scales of turbulent flows that strongly depends on their aerodynamic structure.

In the present work, the aerodynamic processes in the GTÉ-150 combustion chamber of the diffusion type have been investigated numerically. It should be noted that this gas-turbine power plant was designed in the early 1990s on the basis of the elementary thermal scheme with a one-shaft turbogroup and is now operating in the peaking and semipeaking regimes at the hydroelectric power plant GRÉS-3 of the company Mosenergo RAO "EÉS Rossii". The combustion chamber is built-in, and it is of the tubular-circular type. Figure 1 shows the general view of the solid-state model of the GTÉ-150 combustion chamber. Fourteen identical flame tubes are located in separate sections parallel to the axis of the turbogroup. Each section consists of a case, a burner, and a reducer. Leaving the compressor, the air flows into the coupling cylinder and is disturbed between the sections of the combustion chamber. In each chamber part of the air (~25%) immediately enters the mixer holes and the remaining part enters the circular channel

---

[a]St. Petersburg State University of Civil Aviation, 38 Pilotov Str., St. Petersburg, 196210, Russia; [b]Branch of the open stock company "Silovye mashiny — ZTL, LMZ, Elektrosila, Énergomashéksport," St. Petersburg. Translated from Inzhenerno-Fizicheskii Zhurnal, Vol. 79, No. 4, pp. 45–49, July–August, 2006. Original article submitted November 8, 2004; revision submitted July 22, 2005.

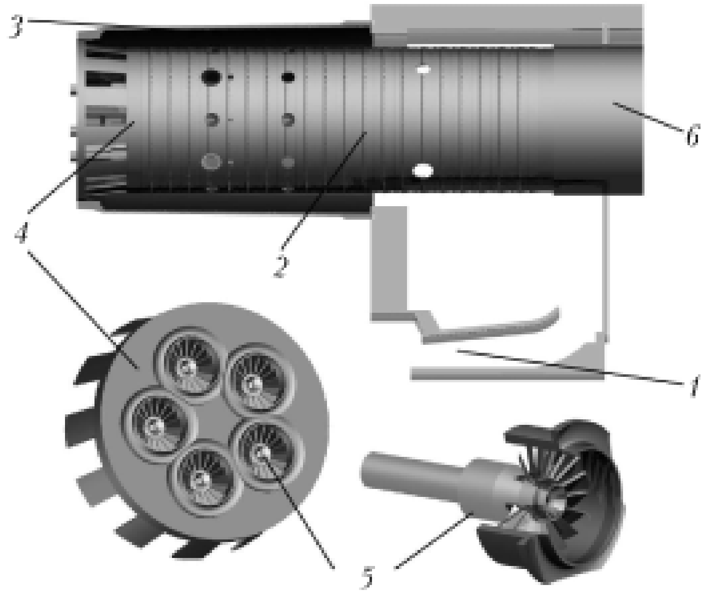

Fig. 1. General view of the solid-state model of the GTÉ-150 combustion chamber: 1) region of the exit from the compressor; 2) flame tube; 3) coupling cylinder; 4) front plate; 5) burner (vane swirler and fuel injector); 6) reducer.

located between the case and the flame tube. Here part of the air enters the burner and the cooling system. The jet-obstructing cooling system (see, e.g., [1]) creates an internal wall layer of cooling air near the flame tube shells. In the burner consisting of five vane swirlers and the same number of liquid fuel injectors, swirling of the flow occurs (to stabilize the flame and ensure stability of the combustion process). As a result of the turbulent processes of mixing and heat and mass transfer, combustion products go through the reducer to the turbine. The structure of the combustion chamber and the technical data are described in detail in [2].

The solid-state model of the combustion chamber (and its mathematical analog for CFD analysis) was developed in Unigraphics NX (see Fig. 1). The calculation region incorporates a region of exit from the compressor, a coupling cylinder, and a combustion chamber (a flame tube and a front plate with five burners). It should be noted that in developing the solid-state model no geometric assumptions in the cooling system of the flame tube and the front burner were made. The mathematical analog of the combustion chamber contains about 2000 punched holes of diameter from 4 to 5.5 mm. The CFD analysis was performed with the use of the parallel version of the commercial FLUENT suite [3]. The finite-element grid was constructed in a GAMBIT grid generator. For numerical simulation, we used a multizone tetrahedral mesh consisting of several blocks linked by interface boundary conditions [3]. The model has about 3.6 million elements (or 800 thousand computational nodes). The construction of the multizone finite-element grids is described in more detail in [4].

The system of stationary Reynolds-averaged Navier–Stokes equations (RANS) was solved by the finite-volume factorized method [5]. To close these equations, we used the k–ε turbulence model with allowance for the low Reynolds number effects [6] with standard wall functions [6]. Because of the low local flow velocities in the chamber compared to the velocity of sound ($M \sim 0.3$) the flow can be considered to be incompressible.

We used the following parameters of the operational conditions of the engine: the air pressure after the compressor was assumed to be equal to 12.26 MPa; the air temperature after the compressor was 630 K; the air consumption (per combustion chamber) was varied from 35 to 39 kg/sec; the turbulence intensity at the entrance was assumed to be equal to 5%, which corresponds to a completely developed turbulent flow. At the exit "soft" boundary conditions (in the case where the flow was considered to be incompressible) or static pressure (for compressible flows) were given. At the side boundaries periodic boundary conditions were given, since a region containing only one chamber was modeled. In the oscillations, we used schemes of the second order of approximation for convective terms of the equations and pressure, the SIMPLEC pressure correction algorithms for compressible flows [7], and SIMPLE for incompressible flows [3].

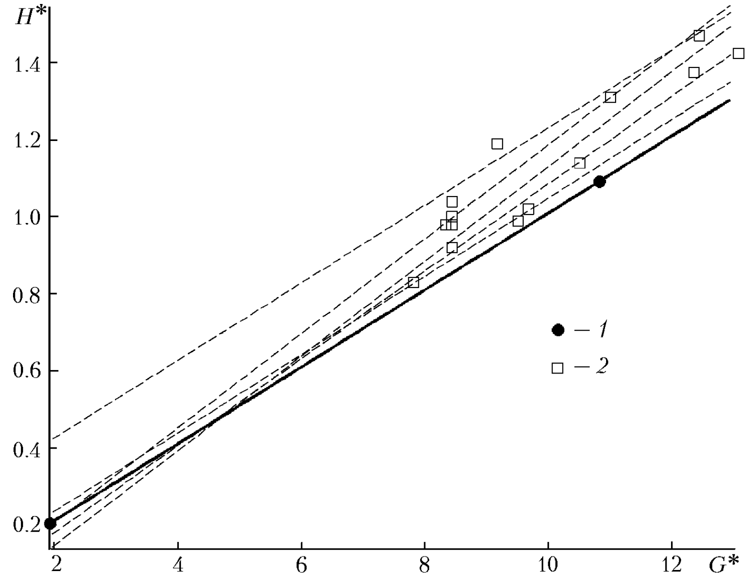

Fig. 2. Relative hydraulic resistance of the register H^* versus reduced air consumption G^*: 1) results of the numerical simulation (FLUENT); 2) experimental data.

In simulating the combustion of diesel fuel (n-heptanol–(nC_5H_{17})), the passive scalar model with the assumption about the existence of chemical equilibrium at the molecular level was used [8]. In calculating the discreet phase (liquid fuel drops), the following assumptions were made:

1) particle motion occurs in the Lagrangian coordinate system: the basic equations are given in detail in [9];

2) numerical simulation is carried out for the combined interaction between the continuum and the fuel drops, i.e., the Navier–Stokes equations and the equations of particle paths were solved simultaneously;

3) the influence of turbulence on the trajectories of fuel drops is taken into account by means of the cloud-particle model [10] based on the application of statistical methods for calculating the deviation of particles from the main (rms) trajectory because of the influence of turbulent pulsations;

4) the input parameters for this model are the minimum and maximum diameters of the cloud which can be determined from physical considerations (e.g., from the geometrical dimensions of the combustion chamber).

The simulation of the process of liquid fuel breakup into drops in air-mechanical injectors [2], whose principle of operation consists of the action on liquid of the centrifugal forces and the atomized air energy, is based on the swirl breakup of liquid fuel. The mathematical model for such injectors used in the gas-turbine industry was specially developed and evaluated in [3, 11]. It should be noted that the mechanism of liquid breakup into separate drops under the action of a swirling air flow is still to be studied physically. Therefore, in simulating atomization processes (both from the viewpoint of basic research and in engineering calculations), selection of constants for mathematical models proceeding from comparison of the results of numerical and physical experiments if required. One more assumption (because of the resource-intensiveness of the problem) is neglect of the influence of the interaction between fuel drops (e.g., under their collision). From physical considerations this assumption should not have a pronounced effect on the final pattern of the flow, since it can be shown that practically all drops completely burn up at a distance of ~0.1 m from the burner. In describing the interaction of a fuel drop with a continuum, we took into account the following processes that have a dominant role int he heat exchange: 1) heating-cooling [12, 13]; 2) evaporation of a particle [12, 13]; 3) boiling, gas formation [8]. Moreover, in simulating atomization, it is essential to determine the dynamic resistance (change in the shape of the fuel droplet in the process of turbulent combustion). To this end, the law of spherical particles was used in the numerical simulation [14].

As boundary conditions for the liquid fuel (n-heptane), the following parameters were given: the fuel temperature was 323 K, the fuel consumption per injector was 0.194 kg/sec, the pressure drop of the air jet was 4 MPa, and the atomization angle of the fuel injector was 60$^{\text{o}}$,

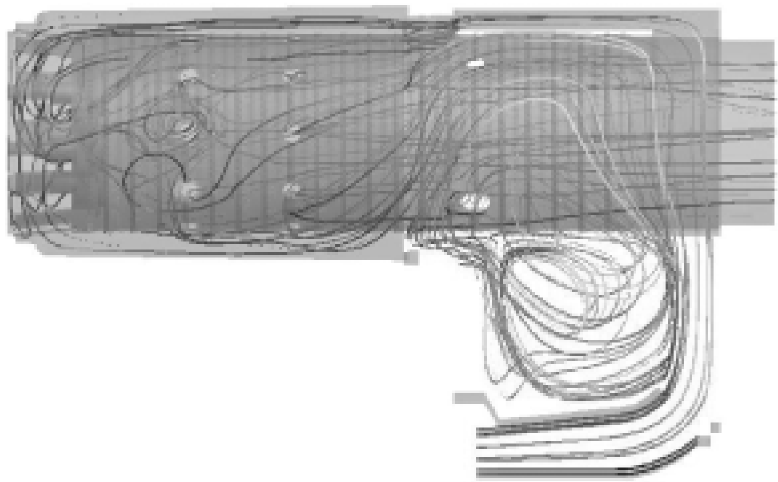

Fig. 3. Vertex structure of the flow in the GTÉ-150 combustion chamber.

TABLE 1. Total Pressure Loss in Different Parts of the GTÉ-150 Combustion Chamber

Parts of combustion chamber	H^*, %	
	Calculation	Experiment
Exit from compressor, coupling cylinder	~3.1	~3
Flame tube	~2.3	
Front plate, five burners	~3.0	~3
Entire combustion chamber	~8.4	~6

At the first stage of the numerical simulation the FLUENT suit was evaluated on a simpler model of a vane swirler separately from the burner and without an air-mechanical fuel injector. To compare the calculated and experimental data, a series of calculations on operating and nonoperating parameters of the gas-turbine plant. Figure 2 presents the results of the numerical simulation 1 and a number of experimental curves 2 incorporating the generalized data on hydraulic tests [2]. It is seen that they are in good agreement, which attests to the correctness of the chosen mathematical models and finite-elements grids for subsequent simulation of the whole combustion chamber.

The field of air particle paths in the combustion chamber having a complex vortex character of the flow with a wide spectrum of various vortex scales and structures is given in Fig. 3.

In numerical analysis of the gas-dynamic flow, we performed a series of calculations with variation of different parameters (air discharge, static exit pressures, account of the flow compressibility, various numerical schemes and algorithms). On the basis of the computation data we can give, with a certain accuracy, the results of hydraulic losses in the section between the exit from the compressor and the turbine coupler.

As a result of the CFD analysis performed, the values o hydraulic losses in different sections of the GTÉ-150 combustion chamber have been obtained (see Table 1). The calculated value of the total pressure loss in the section incorporating the compressor exit, the coupling cylinder, and the combustion chamber is ~8.4% (the minimum and maximum values are 7.9 and 8.9%, respectively), the experimental value (in the same section) is ~6% [5]. Such a discrepancy between the calculated and experimental values can be explained by two groups factors (note that they are equivalent). The first group includes losses directly connected with computing techniques such as errors in interpolating on interface boundary conditions, etc. A marked effect can be produced by the value of the static pressure at the exit from the calculation region of the model, since the reducer connecting the flame tube and the suction nozzle apparatus of the turbine was nit included i the model. This group also includes the influence of geometric factors, i.e., the discrepancy between the structural arrangements of the combustion chamber, on which experimental measurements were taken, and its solid-state analog (mathematical analog) used in the numerical experiment. The second group of factors includes mistakes made in the course of experimental measurements.

CONCLUSIONS

1. The aerodynamic pattern of the flow in the combustion chamber of the GTÉ-150 gas-turbine plant has been analyzed numerically.

2. Comparison of the calculated and experimental data on the total pressure losses both throughout the combustion chamber and in its individual units has been made. We have obtained good (within the framework of the engineering calculations) agreement between the results of the numerical and natural experiments for individual units of the combustion chamber, which confirms the correctness of using the FLUENT suite CFD for conducting engineering aerodynamic processes in GTU combustion chambers.

3. The discrepancy between the results of the calculations and the experimental data on the total pressure loss for the whole GTÉ-150 combustion chamber can be explained by both errors made in the course of the physical experiment and various assumptions made in the numerical simulation.

NOTATION

G, air consumption, kg/sec; $G^* = \dfrac{G^2 T}{P^2}$, reduced air consumption, $m^2 \cdot sec^2 \cdot K$; $H^* = \dfrac{\Delta P^*}{P^*}$, total pressure loss, %; k, energy of turbulent pulsations, m^2/sec^2; M, Mach number; P^* and P, total and static pressures, Pa; T, temperature, K; ε, turbulent energy dissipation rate, m^2/sec^3.

REFERENCES

1. A. V. Sudarev and V. I. Antonovskii, *Combustion Chambers of Gas-Turbine Plants* [in Russian], Mashinostroenie, Leningrad (1985).
2. V. I. Antonovskii, V. A. Asoskov, A. P. Pekov, S. M. Khairullin, and S. V. Burtasov, Discharge chamber GTE-150. Testing on the bench of the Central Boiler and Turbine Instutute and implementation at regional hydroelectric plant GPÉS-3 of MOSÉNERGO of RAO "UES of Russia," *Tr. TsKTI*, Issue 284, 54–71 (2002).
3. Fluent Inc. Fluent 6.1 users guide, Lebanon, 2003.
4. D. A. Lysenko, 3D numerical simulation of gas-dynamic processes in the combustion chamber of gas-turbine plants, in: *Proc. XIV School-Seminar of Young Scientists and Specialists Guided by Academician A. I. Leontiev* [in Russian], Vol. 2, MÉI, Moscow (2003), pp. 114–118.
5. J. M. Weiss, J. P. Maruszewski, and W. A. Smith, Implicit solution of preconditioned Navier–Stokes equations using algebraic multigrid, *AIAA J.*, **37**, No. 1, 29–36 (1999).
6. T.-H. Shih, W. W. Liou, A. Shabbir, and J. Zhu, A new k–ε eddy-viscosity model for high Reynolds number turbulent flows — model development and validation, *Comput. Fluids*, **24**, No. 3, 227–238 (1995).
7. J. P. Vandoormaal and G. D. Raithby, Enhancements of the SIMPLE method for predicting incompressible fluid flows, *Numer. Heat Transfer*, **7**, 147–163 (1984).
8. K. K. Kuo, *Principles of Combustion*, John Wiley and Sons, Inc., New York (1986).
9. Z. Han, S. Perrish, P. V. Farrell, and R. D. Reitz, Modeling atomization processes of pressure-swirl hollow-cone fuel sprays, *Atomization Sprays*, **7**, No. 6, 663–684 (1997).
10. L. L. Baxter and P. J. Smith, Turbulent dispersion of particles: the STP model, *Energy Fuels*, **7**, 852–859 (1993).
11. V. Yakhot and S. A. Orszag, Renormalization group analysis of turbulence: I. Basic theory, *J. Sci. Comput.*, **1**, No. 1, 1–51 (1986).
12. W. E. Ranz and W. R. Marshall, Jr., Evaporation from drops. Pt. I, *Chem. Eng. Prog.*, **48**, No. 3, 141–146 (1952).
13. W. E. Ranz and W. R. Marshall, Jr., Evaporation from drops. Pt. II, *Chem. Eng. Prog.*, **48**, No. 4, 173–180 (1952).
14. A. B. Liu, D. Mather, and R. D. Reitz, Modeling the effects of drop drag and breakup on fuel sprays, SAE Tech. Paper, 930072, SAE (1993).