Numerical Simulation of Turbulent Heat Transfer in Gas Turbine Combustion Chambers with Application of Parallel Techniques in CFD package Fluent

Dmitry A. Lysenko[1*], Alexander A. Solomatnikov[1],
Nikolai N. Shabrov[2], Mikhail V. Lopatin[3]

[1] Joint Stock Company Leningradsky Metallichesky Zavod, St.-Petersburg, Russia
[2] St.-Petersburg State Polytichnical University, St.-Petersburg, Russia
[3] Process Flow Ltd. Oy, Turku, Finland

The three-dimensional combustion's processes in gas turbine combustion chambers using multi-purpose CFD package Fluent with application of different high- performance multiprocessors systems are numerically modeled.

At present the development of high-performance parallel systems allows to decide complicated tasks of aerodynamics and heat transfer in three-dimensional formulation very efficiently, using CAE (Computer Aided Engineering) and CFD (Computational Fluid Dynamics) – techniques. In this paper the three-dimensional simulations of combustion processes in gas turbines chambers using CFD package Fluent is considered. The numerical modeling was kept in different classes of parallel computer systems, with using both MPP and SMP systems.

The gas turbine combustion chamber is a complex technical system in many respects specifying operation of a gas turbine as a whole. Therefore integrated computational simulation of physicochemical processes, occurring in a combustion chamber, allows to improve of operation reliability considerably, to remove different imperfections of a design at the development stage and to optimize basic performances of a gas turbine.

The parallel version of the CDF package Fluent was used in the numerical simulations. The Reynolds-averaged Navier-Stokes (RANS) equations (steady and unsteady), based on control-volume method, were solved. The modified k-ε turbulence model both with the standard and non-equilibrium wall functions was used. The following combustion models were used in the simulations: the eddy dissipation model of Magnussen and Hjertager, based on the solution of transport equations for species mass fractions, and the non-premixed combustion model, in witch individual species transport equations are not solved, instead, transport equations for one or two conserved scalars (the mixture fractions) are solved and individual component concentrations are derived from the predicted mixture fraction distribution. Radiative heat transfer was simulated using the P-1 model. This radiation model is the simplest case of the more general P-N model, which is based on the expansion of the radiation intensity into an orthogonal series of spherical harmonics. There are several significant products of combustion in gas turbine emissions: oxides of nitrogen (NO and NO_2, collectively called NO_x) and carbon monoxide (CO). There are two sources of NO_x emissions in the exhaust of a gas turbine. Most of the NO_x is generated by the fixation of atmospheric nitrogen in the flame, which is called thermal NO_x. Nitrogen oxides are also generated by the version of a fraction of any nitrogen chemically bound in the fuel (called fuel-bound nitrogen). Thermal NO_x is generated by a chemical reaction sequences called the Zeldovich Mechanism. This set of well-verified chemical reactions postulates that the generation of thermal NO_x is an exponential function of the temperature of the flame and a linear function of the time witch the hot gases exposed at the flame temperature. The

[*]E-mail: dmitrylysenko@mail.ru

formation of carbon monoxide CO was considered in the combustion chemical reaction mechanism.

While computational modeling the unstructured meshes with tetrahedral control-volumes (elements) were used. An amount of nodes was varied from 0.2 to 1.1 millions that was equivalent to the amount of control-volumes (elements) from 0.9 to 5.5 millions. The high-performance algorithm Metis was used for partition of the grids.

The special attention was given to verification of the package. The series of calculations were carried with the purpose of a methodology development for the solution of three-dimensional combustion problems using CFD package Fluent. The satisfactory confirming of numerical and experimental data was obtained (in case of engineering calculations). The analysis of numerical and experimental data was carried out based on the experiments of VTI (Moscow, Russia) and CKTI (St.-Petersburg, Russia). The numerical results were carried out using CFD package Fluent.

The results of aerodynamics and thermal modeling both of the some modules and complex simulations of new design of low toxic combustion chamber are presented in the paper. Also the unsteady effects, taking place in low toxic chambers, are described.

While numerical simulation the following parallel systems were used: the symmetrical multiprocessor system HP9000, based on the Unix HP OS (LMZ), 8-processor computational cluster, based on Windows 2000 OS (LMZ) and 16-processor computational cluster (MMF SPbSPU) based on Linux Red Hat 7.2 OS. The detailed comparison of process scaling efficiently as function of processor number, graphs of operational effectiveness of this or that parallel system for this or that class of problems are considered.