Method of the numerical modelling of unstationary processes in the combustion chamber of a gas turbine engine

A T Savchuk and A A Yakovlev

Department of Aero Engines, Moscow Aviation institute (National Research University), 4 Volokolamskoe Highway, 125993, Moscow, Russian Federation

E-mail: yakovlevaa@mai.ru

Abstract. Estimation of the stability margins of combustion processes in the combustion chamber of a gas turbine engine is a vital task of current subsonic and supersonic propulsion engineering and this article describes the approach to its Computational Fluid Dynamics simulation. The aim of the work is to create and verify a methodology for assessing the stability regimes of combustion processes in the combustion chamber of a gas turbine engine based on an artificial simulation of the excitation of non-stationary processes in the temperature-pressure parameters using “ANSYS Fluent”. The further comparison of the computed results and the real data of the physical experiment in the model device will take place to verify the method. During the research this methodology and a software product will be developed for computing the characteristics of the combustion processes before and after the excitation of the artificial unstable process. This technique, when verified, will allow to clarify the instability limits for specific operating modes (engine stall, flame suppression, lean flameout) of existing and projected gas turbine engines. The obtained numerical results allow us to make an assumption about the fundamental possibility of obtaining convergent solutions in close accord with the future experimental data.

1. Introduction

Modern aircraft combustion chambers are three-dimensional devices with complex geometry of flow paths and a branched fuel feed system. Predicting the characteristics of the combustion chamber by mathematical modelling is hampered by design constraints. This problem is especially acute when describing the processes of ignition, flame stabilization, and combustion recovery in the envelope typical for the operating modes of high-speed combustion chambers (CC) and critical modes of conventional air-jet engines under the conditions of using simulation models of low fuel concentrations in the fuel-air mixture (less than 10%). General issues related to aviation combustion chambers and it characteristics are considered in [1] and [2], the analysis of the influence of the computational mesh on various processes is carried out in [3].

The overwhelming majority of works available for study use unique open-source software and Computational Fluid Dynamics (CFD) tools, simplified quasi-gasdynamic models [4-6] that significantly limit the versatility of the solution, due to the need for the author's implementation of chemical kinetics problems, the applied turbulence model, which cannot but affect the solution accuracy. Extremely significant difficulties in the author's implementation of the problem of network computing (grid computing) without which will require significant computing resources for problems scaling.

There are a number of studies devoted to related topics, in a specific formulation of the problem for the stability of the combustion process [7], quasi-one-dimensional set up of the problem [8],
investigation of the low-order dynamic model [9]. Problems of modelling flows with combustion are considered in a number of works from the standpoint of thermodynamic conditions of heat and mass transfer processes [10] and upstream aerodynamic effects of the combustion process [11,12].

No works were found using the commercial ANSYS Fluent code for the integrated solution of the above problems without involving software products from other manufacturers, which does not allow us to consider them industrially solvable in the present.

The essence of the proposed method is the following two-stage CFD code:

1. Obtaining a stationary Linearized Instability Sheet Atomization (LISA) - solution (steady-pressure based) of fuel assembly combustion in the model zone and artificial ignition with a given accuracy in the whole computing area.

2. Obtaining the transient solution (pressure base, adaptive time steps) process with the required accuracy parameters, the minimum recommended time step is $10^{-6}$ sec.

The actual problem of engine design is the estimation of the stability range of a combustion processes in the combustion chamber of the gas turbine engine (GTE) based on artificial simulation of the excitation during non-stationary processes (NP) in the CC simulation using CFD ANSYS Fluent software. This article describes the basic approach of its modelling. The validation experiment for comparison of the calculation results and the data of a physical experiment will be carried out in the next steps of the research. This technique while being verified will represent a software product for computing the basic characteristics of the burning process before and after the excitation of the NP, which allows to estimate the stability margin of operating process for CC structures of a certain geometry and operating parameters without using the significant computing power and a long calculation time (hereinafter computing resources). It will provide an opportunity for quick clarification of the boundaries of instability (engine stall, flame-out, lean flameout) for specific aircraft flight cases such as: the crossing of the jet backwash of the ahead flying aircraft, external shock wave in the air intake and other cases which are difficult to simulate in laboratory conditions and expensive in the process of the flight tests, both for existing and perspective gas turbine engines. Both at the initial design stages and during the experimental fine-tuning of the CC the proposed method will allow to determine the CC tendency towards the combustion instability with the least number of rig and flight tests, as well as to assess to make the decisions to use of combustion-stabilization devices.

The aim of the study is to develop a technique that allows solving typical problems (flameout, shock wave propagation inside combustion chamber) of low-concentrated (up to 10% by volume fraction of fuel in the fuel-air mixture) lowspeed and highspeed unsteady burning flows without involving significant computing resources.

2. Materials and methods
The principle of the proposed technique is the following. The GTE CC is segmented to the minimum CAD element which allows to build a 3D model of the CC as a circular array as shown at figure 1. On its basis, the mesh generated adequate to computational resources and meets the minimum requirements of the ANSYS Fluent DPM (Discrete Phase Modelling) multiphase model. The multi-stage CFD calculation is performed as described below.

The training model with a centrifugal nozzle and the simplest profiled swirler atomizer was chosen as a CC model, the mesh has the volume as 128 thousand tetra elements, the maximum cell size is 2.5 mm, 7 inflationary wall layers, the maximum linear chamber size is 130 mm, a combustion zone has length 96 mm, the turbulence model was chosen as k-epsilon, as the lowest demanding mesh quality. ANSYS Fluent was chosen because of its greater stability in simulating flaming flows comparable to the ANSYS CFX Flamelet approach. The DPM multiphase model was chosen as the least resource-intensive, the fuel is kerosene with supply $10^{-3}$ kg/s, the pressure gradient inlet/outlet is 0.5 atm, the inlet turbulentization is about 10%, second-order upwind calculation scheme with the pressure-based solver.

To obtain a burning CFD solution with artificially excited nonstationarity (AENP) with the required accuracy under subsonic conditions without taking into account the effects of FSI (Fluid Structures Interaction), it is proposed to consider the sequence of the following stages.
1. Obtaining a non-burning initial (overclocked) solution, up to $10^{-1} \sim 10^{-2}$ residual’s level and the realistic physical picture of the flow based on initial condition as shown at figure 2.

2. Obtaining a converging non-burning steady solution with the required accuracy as shown at figure 3.

3. Obtaining a non-diverging burning transient solution with the required levels of the basic residuals of temperature / pressure / velocity in the outlet section as shown at figure 4.

4. Artificial excitation of a non-stationary process in an arbitrary region of the computational mesh as shown at figure 5.

The first two steps of the calculation are the tasks to obtain the non-combustible solution as shown at the figure 6 below which is necessary for field of the primitive variables (temperature, pressure, velocity) formation, the physical reflection of this process is the CC launch at the cruise regime.

Obtaining of a steady non-combustible solution with the minimum time spent is possible by using max-min limiters set based on the inlet-Outlet pressure and evaluating the maximum achievable flow rate (so-called overclocking solution). This step prevents the appearance of long-term false oscillations and also significantly reduces the required computing resources. It is enough to form the overclocked solution with the time step size recommended by the ANSYS Fluent 19R3 to the basic residuals value (pressure, velocity, temperature, continuity, energy) at the level of $10^{-1} \sim 10^{-2}$ and obtaining a fundamental physical picture of the flow. The overclocking solution can be extrapolated to a finer mesh if a highly accurate solution is needed and this will be a productive resource-saving step for calculating meshes of complex topologies.
Figure 4. Result of non-divergent burning solution: residuals at the 1100 and 2750 time steps, $t_s = 3 \cdot 10^{-6}$ sec, temperature, flow velocity, pressure, qualitative pictures.

Figure 5. AENP, absolute pressure, “strip 150 atm” before and in the combustion zone at successive times, qualitative pictures.

Figure 6. Results of the unburning solution: residuals, flow velocity, pressure, qualitative pictures at the 120 time steps, $t_s \approx 3 \cdot 10^{-3}$ sec.

The resulting overclocking solution is the basis for the next stage for the non-burning steady solution with the required accuracy determined by the level of the basic residuals of conservative variables by one or two orders of magnitude higher than the required accuracy of the following burning transient solution.

After obtaining a converging non-burning solution with the required accuracy, it is possible to obtain a non-diverging burning transient solution. Due to the discrete nature of the fuel supply function
implemented in the ANSYS-Fluent-DPM approach, the solution cannot be stable by itself and should be estimated only by the average level of the basic residuals of conservative variables and the frequency of the absolute pressure pulsations, as the most sensitive conservative variable at the centreline of the CC’s outlet. The specified DPM method is the only possibility to use the ANSYS-Fluent without usage the significant computing resources. The ANSYS’s tools also include the phenomenological model Volume of Fluid (VOF). The VOF model application will force to require a high-resolution computing mesh with the size of the mesh equal to the smallest atomized fuel drop, which will dramatically increase the required computing resources in simulating multilayer processes of chemical kinetics, which is necessary condition for ANSYS simulation of combustion processes and, as a consequence, will seriously reduce the overall efficiency of the mesh. The specific recommendations on the parameters of this stage of the calculation will be given later after the physical validation experiment.

After obtaining a burning transient solution of the required accuracy it is possible to excite an artificially process in an arbitrary CC region with restrictions on the size of the non-stationary source region (at least 3 layers of mesh cells covering the non-stationary region in all directions) and without any constraints of connectivity. The time step in this case should be not more than the time for the shock wave passage of the one layer of the mesh in all directions. The shock wave in the general case has the speed greater than the speed of sound and significant pressure/temperature, which is observed when using conventional explosives. In finite element modelling of this process, one-two-step temporal oscillations of conservative variables in the range of non-physical values (ex. negative pressure) are inevitable, that do not have the significant effect for general non-divergence character of the solution.

Further study of the non-stationary process with shock wave propagation in the burning flow will be limited only by the computing resources, the essence of the problem (clarification of the engine stall boundaries, lean flameout) and the capabilities of the ANSYS Fluent software. In the presence of the significant computing resources, the limit of the calculation accuracy will be boundary of the physical simplifying assumptions of the ANSYS-Fluent-DPM method.

The parameters of artificial nonstationarity – geometry, time of occurrence, conservative parameters of the initiation zone of the nonstationary process, are transferred to the ANSYS-Fluent in the form of a formatted text (ini) file, available for editing by any text editor. The accuracy of the obtained solution depends on the greatest extent on three parameters: the pressure of the excited nonstationarity, the quality of the mesh, and the associated minimum possible time step.

3. Results and discussion
In the numerical experiment results on the model combustion chamber at the figure 7 below, the only artificial non-stationarity of the "sphere" type, specified in the region combustion’s stable front formation, was modelled.

![Figure 7. Numerical experiment with artificial non-stationarity volume 1 “sphere” (left) and outlet observation volume 2 (right).](image)

The observation volume 1 in the form of the sphere with the diameter of 2.5 mm was set at the epicentre of the "sphere" type AENP with the same diameter and position with the pressure of 1500 atm and temperature of 4500 K. The duration of the AENP is equal to one fixed time step of $3 \times 10^{-4}$ sec. The observation volume 2 is realized in the outlet section of the combustion chamber in the form of the 10x10 mm parallelepiped with three layers of cells upstream as shown at the figure 7. The duration of
the numerical experiment on the computing grid with the already obtained data of the converging non-burning solution is set at 2000 iterations as shown at the figures 8 and 9.

**Figure 8.** Numerical experiment results, temperatures in the observation volumes 1 and 2.

**Figure 9.** Numerical experiment result, average droplets diam in the observation volume 1.

Basic initial conditions were the following:
- For non-burning steady solution: inlet pressure = 10 atm (air), with turbulent intensity = 10%, inlet temperature = 300 K, outlet pressure = 9.5 atm, outlet temperature = 300 K.
- For non-diverging burning transient solution: fuel mixture (kerosine + air), initial fuel temperature = 350 K, fuel flow rate = 0.001 kg/sec, fuel injection pressure = 20 atm, fuel spray half angle = 22.5°, atomizer dispersion angle = 6°, ignition time = 0.001 sec from the beginning of experiment.

For the AENP problem simulation, the ini-file, containing process parameters such as: the type of nonstationarity (strip or sphere) with its geometric dimensions and position, the AENP excitation time, its duration, temperature and pressure, observation zones with its type, geometry and position (since the number of ini-parameters are significant amount, more than 50, only some of them are indicated here), was created as txt-file and imported into ANSYS Fluent using UDF.

The average droplet diameter is measured as the statistical mean of all droplets in the observation volume 1 during whole numerical experiment. Temperatures in both observation volumes are calculated using standard ANSYS Fluent postprocessing tools. The obtained results of numerical experiments show the measurability of significant quantities affecting the stability of the process, the possibility of modelling difficult-to-reproduce and expensive of operation regimes for critical and close to critical modes, as well as the availability of fundamental possibilities of using this method for both rapid and
final calculations of the flow paths, including rotating impeller machines and units with using all available ANSYS tools and models.

The attained numerical results are physically correct and are in the range of permissible values of the operating parameters of existing technical devices, which allows us to make an optimistic conclusion about the fundamental applicability of the proposed technique in future calculations of burning flows with zones of strong discontinuities. As shown at the figures 8 and 9 at the 20th iteration, the flame is blown off due to a combustion is restored at the 850th step.

Taking into account FSI effects will require a multidisciplinary set up of the problem at the stage of excitation of artificial nonstationarity and goes beyond the scope of the problem posed in the study. If it is necessary to solve the problem in a supersonic burning flow, the most promising resource-saving technique is proposed to consider the use of space-marching techniques and schemes that do not transmit upstream information downstream.

4. Conclusion
In the proposed study, the authors described a technique for modeling unsteady combustion processes of a fuel-air mixture in the combustion chambers of gas turbine engines. The advantage and distinguishing feature of this methodology in relation to the existing ones is its low demand to the availability of significant computing resources for the engineering predicting.

In the process of solving the AENP problem, first a stationary overclocked solution was obtained, then high precision stationary converging non-burning solution, then the non-diverging burning solution, on which an AENP process was imposed. This approach made it possible to control and evaluate the results at each stage of solving the problem.

The results achieved to date show the promising industrial use of this approach for modelling non-stationary processes at existing production facilities and design bureaus after experimental confirmation of the proposed method.

References
[1] Agulnik A B, Bakulev V I, Golubev V A, Kravchenko I V, Krilov B A. 2002 Thermogasdynamic Calculations and Calculation of the Characteristics of Aviation Gas Turbine Engines (MAI Publishing House) p 257
[2] Onishchik I I, Krylov B A, Yun A A. 2009 Modeling the Processes of Heat and Mass Transfer in Model Combustion Chambers. Bulletin of the Moscow Aviation Institute 16(1) 27
[3] Yanyshev D S, Bykov L V, Molchanov A M. 2018 Mesh Models for Solving Engineering Thermophysical Problems in the ANSYS Environment (LENAND) p 264
[4] Molchanov A, Yanyshev D, Bykov L. 2017 Simulation of High-Speed Nonequilibrium Heterogeneous Turbulent Flows with Phase Transition. J. of Phys.: Conf. Series 891(1) 012051 DOI:10.1088/1742-6596/891/1/012051
[5] Molchanov A, Yanyshev D, Bykov L. 2017 Influence of Channel Geometrical Properties and Turbulence on Propellant Ignition In Hypersonic Ramjet Combustion Chamber. J. of Phys.: Conf. Series 891 012107 https://doi.org/10.1088/1742-6596/891/1/012107
[6] Platonov I, Molchanov A, Yanyshev D, Bykov L. 2019 On the V2-Based Turbulence Model for Free-Stream and Wall-Bounded High-Speed Compressible Flows. Int. J. of Fluid Mech. Res. 46(6) 565 https://doi.org/10.1615/IntJFluidMechRes.2018025734
[7] Rashkovskiy S A, Yakush S E, Baranov A A. 2018 Combustion Stability In A Solid-Fuel Ramjet Engine. J. of Physics: Conf. Ser. 1009 012032 https://doi.org/10.1088/1742-6596/1009/1/012032
[8] Wadwankar N, Kandasamy G, Ananthkrishnan N. 2018 Dual combustor ramjet engine dynamics modeling and simulation for design analysis. J. of Aerospace Eng. 233(4) 1307 https://doi.org/10.1177/0954410117749867
[9] Liu K, Cui T. 2020 Combustor-Inlet Interactions in a Low-Order Dynamic Model of Ramjet Engines. The Aeronautical J. 124(1282) 2001 https://doi.org/10.1017/aer.2020.64
[10] Li W, Zhao D, Chen X S, Zhu L, Ni S 2021 Numerical Investigation of Inlet Thermodynamic Conditions on Solid Fuel Ramjet Performances. *Int. J. of Aer. Eng.* (3-4) 20 https://doi.org/10.1155/2021/8868288

[11] Ro R M, Prasad A R, Charyulu B V N, Singh H 2020 Numerical Studies and Validation of Combustor and Annular Isolator Interactions of Hydrocarbon Based Axisymmetric Dual Combustion Ramjet. *Aer. Sci. and Techn.* 106(6) 106185 https://doi.org/10.1016/j.ast.2020.106185

[12] Ma K, Zhang Z, Liu Y, Jiang Z 2020 Aerodynamic Principles of Shock-Induced Combustion Ramjet Engines. *Aer. Sci. and Techn.* 103 105901 https://doi.org/10.1016/j.ast.2020.105901