УДК 621.45.034
DOI: 10.26467/2079-0619-2025-28-1-53-66

Grid influence issues in the methodology of numerical modelling of non-stationary combustion processes

A.T. Savchuk[1], L.V. Moskalenko[1]
[1]Moscow State Technical University of Civil Aviation, Moscow, Russia

Abstract: Estimation of reserves of combustion process stability in gas turbine engine (GTE CC) based on artificial modeling of non-stationary process (NP) excitation in the combustion chambers in temperature-pressure parameters is an actual problem in engine engineering. An increasing number of aircraft require the use of engines with high gas dynamic stability (GDS) up to 30% and more, for example, when creating power plants for vertical and short take-off and landing aircrafts, ekranoplans (ground-effect vehicles) and etc. The use of computational fluid dynamics (CFD) tools for calculating combustion flows in the combustion chamber of a gas turbine engine is currently an integral part of the design process, since a numerical study, in contrast to a full-scale experiment, requires significantly fewer material resources providing the ability to model expensive and unsafe cases of aircraft flight operation that are difficult to implement at the stage of bench tests, such as: crossing a jet distrail or a shock wave front (e.g., when an ammunition detonates) in front of the air intake of an air-jet engine, critical crosswind during takeoff leading to flow separation on the air intake cowl, vertical gusts and atmospheric turbulence, flight at high angles of attack, aircraft evolution (slip, etc.). The results of numerical simulation are decisively determined by the limitations of the applied models and simplifying assumptions for the simulated flow. There are many sources of errors in any calculation using computational gas dynamics methods: accumulated calculation errors, sensitivity to grid size, discretisation, flow extrapolation in grid interfaces of the used solver (ANSYS.Fluent), errors of turbulence models, assumptions and simplifications applied to the design, etc. This paper considers the grid effect on the problem of proving the random nature of gas oscillations in the combustion chamber of a gas turbine engine, which is essential for determining the gas dynamic stability of the engine as a whole.

Key words: computational fluid dynamics, non-stationary process, combustion chamber, grid effect.

For citation: Savchuk, A.T., Moskalenko, L.V. (2025). Grid influence issues in the methodology of numerical modelling of nonstationary combustion processes. Civil Aviation High Technologies, vol. 28, no. 1, pp. 53–66. DOI: 10.26467/2079-0619-2025-28-1-53-66

К вопросам сеточного влияния в методике численного моделирования нестационарных процессов горения

А.Т. Савчук[1], Л.В. Москаленко[1]
[1]Московский государственный технический университет гражданской авиации, г. Москва, Россия

Аннотация: Оценка запасов устойчивости процессов горения в камерах сгорания газотурбинных двигателей (КС ГТД), основанная на искусственном моделировании возбуждения нестационарных процессов (НП) в КС, в параметрах температура – давление представляет собой актуальную задачу двигателестроения. Все большее количество летательных аппаратов (ЛА) требуют применения двигателей с высокой газодинамической устойчивостью (ГДУ) вплоть до 30 % и более, например при создании силовых установок для самолетов вертикального и укороченного взлета и посадки, экранопланов и др. Применение инструментария вычислительной гидрогазодинамики (англ. CFD – Computational Fluid Dynamics) для расчета горящих течений в КС ГТД в настоящее время является неотъемлемым этапом процесса проектирования, так как проведение численного исследования, в отличие от натурного эксперимента, требует значительно меньших материальных ресурсов, предоставляющих возможности моделирования трудно реализуемых на этапе стендовых испытаний дорогостоящих и небезопасных случаев летной эксплуатации ЛА, таких как пересечение реактивной струи впереди летящего ЛА либо фронта ударной волны (например, при подрыве боеприпаса) перед воздухозаборником воздушно-реактивного двигателя (ВРД), критический боковой ветер при взлете, приводящий к срыву потока на обечайке воздухозаборника, вертикальные порывы и турбулентность атмосферы, полет на больших углах атаки, эволюции ЛА (скольжение и др.). Результаты численного моделирования решающим образом определяются учетом ограничений применяемых моделей и упрощающих предположений для моделируемого течения. Существует множество источников ошибок в любых расчетах с использованием методов вычислительной газовой динамики: аккумулированные ошибки вычислений, чувствительность к размеру сетки, дискретизации, экстраполяции потоков в сеточных интерфейсах используемого солвера (ANSYS.Fluent), ошибки моделей турбулентности, допущения и упрощения, применяемые к конструкции, и т. д. В данной работе рассмотрено сеточное влияние на задачу доказательства случайной природы колебаний газа в КС ГТД, имеющей существенное значение для определения газодинамической устойчивости двигателя в целом.

Ключевые слова: вычислительная гидрогазодинамика, нестационарный процесс, камера сгорания, сеточное влияние.

Для цитирования: Савчук А.Т., Москаленко Л.В. К вопросам сеточного влияния в методике численного моделирования нестационарных процессов горения // Научный вестник МГТУ ГА. 2025. Т. 28, № 1. С. 53–66. DOI: 10.26467/2079-0619-2025-28-1-53-66

Introduction

This article covers the issues of problem formulation and assumptions made on the basis of the computational model of the combustion chamber on stationary calculation modes (low throttle, cruising, maximum thrust mode), in its non-diverging burning transient solution with the required levels of basic temperature/pressure/velocity mismatches in the outlet section [1], followed by artificial excitation of a non-stationary process (NP) on the overall accuracy, as well as the required computational resources and stability of the resulting solution. Based on the results of the work, recommendations will be made to the main investigated parameters of the computational model, namely, the productive maximum size of the computational grid.

The created computational grid should satisfy the minimum criteria of solvers simulating high-gradient processes occurring in the combustion chamber, and it should be borne in mind that with an increase in the number of grid elements, the computation time increases accordingly. At the same time, as studies [2, 3] show, there is an upper limit of the grid size, when the obtained result does not depend on a further reduction in the size of computational cells.

When selecting the parameters of the mathematical model, the fundamental physical nature of the flow in the combustion chamber should be taken into account. When calculating the combustion chamber with the resulting NP, trans- and supersonic flow zones inevitably appear, the joint consideration of which with the subsonic domain will determine the choice of parameters of the entire computational model of the transition process. A number of publications [4–9] provide recommendations and an assessment of the influence of the specified flow parameters and the calculation grid with verification of the obtained results [10–14].

As described in detail in [1, 3], the proposed methodology is multi-stage and the solution of the problem passes sequentially through the stages of stationary and non-stationary calculations. As a part of the current study, a series of non-stationary calculations of the model combustion chamber were carried out on grids of different resolutions (coarse, medium, fine) and equivalent topology and finite element quality (unstructured grid with tetrahedral finite elements generated by the standard ANSYS-Mesh toolbox using the Delano triangulation method with comparable quality of the grid elements) were preceded by a stationary calculation until a stable solution was achieved according to the criteria of achieving low root-mean-square residuals (about 1e-2 on primitive variables) and mass imbalance of incoming and outgoing flows (no more than 2%). The actual estimates and criteria of the state of the transient process of non-stationary combustion according to the formal calculation criteria used below are described in detail in [4].

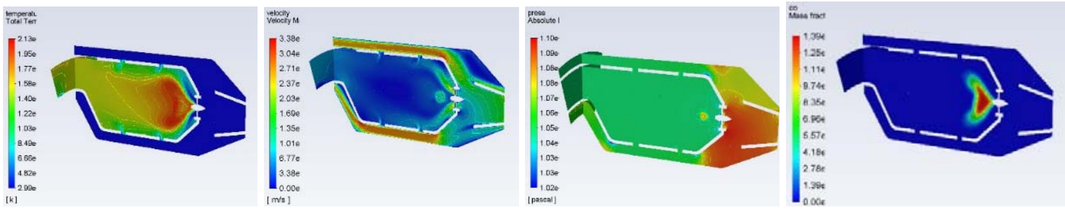

Fig. 1. From left to right: temperature, flow rate, absolute pressure, mass fraction of CO

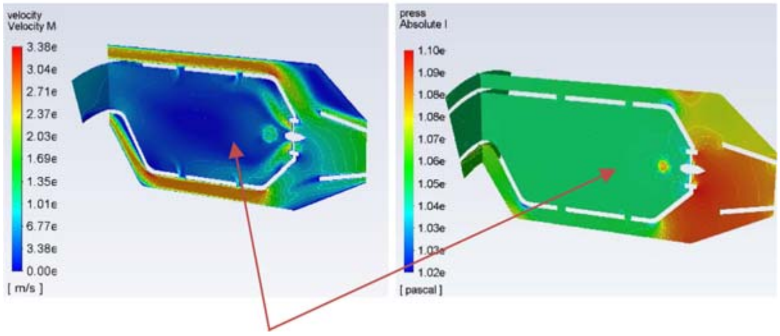

Fig. 2. Constant ignition region, velocity on the left, absolute pressure on the right

General parameters of the series of numerical experiments

The computational grid of the minimum CAD element of the combustion chamber sector is created adequate to the computational resources (the main limiting factor is the total computation time of no more than 48 hours) and to satisfy the minimum requirements and limitations of the applied ANSYS models:

- turbulence k–ε Standard;
- Discrete Phase Modeling (DPM) ANSYS Fluent;
- Linearized Instability Sheet Atomization (LISA);

Non-Premixed Combustion[1];

- mechanism of chemical kinetics of combustion of "kerosene + air" fuel pair (Jet-A) [15].

If a fixed time step is used, its maximum value should not be greater than the time of passage of both the flow and its disturbance, one layer of cells in any direction. The use of an adaptive time step, implemented in ANSYS. Fluent, allows automated calculation of the values of each time step, taking into account the above-mentioned limitations both in solutions for steady flow and in transient processes.

After achieving the stable result of the steady-state calculation of the combustion process of the fuel assemblies in the combustion chamber (fig. 1), under the following initial conditions, which are the same for all studied calculation grids, namely: pressure drop in the combustion chamber 0.5 atm, inlet air temperature 300 K, fuel supply 1.5 g/s (kerosene), turbulence model k–ε of the 2nd order of accuracy, number of inflationary wall layers 12, combustion chamber volume 0.266432 litres.

After switching the problem to an non-stationary solver with an adaptive time step at a point 10 mm downstream of the constant ignition region of the fuel assembly (a sphere with a diameter of 2.5 mm and a temperature of 4500 K) (fig. 2).

Using ANSYS software control, an artificial non-stationary region with a diameter of 5 mm was created with a temperature equal to the ignition temperature and an overpressure of 25 atm (fig. 3).

After the NP excitation (fig. 4), the calculation on grids of different resolution was continued at the same settings of the adaptive non-

[1] CFD EXPERTS Simulate the Future. (2021). Ansys Fluent Theory Guide, 1069 p. Available at: https://dl.cfdexperts.net/cfd_resources/Ansys_Documentation/Fluent/Ansys_Fluent_Theory_Guide.pdf (accessed: 25.08.2024).

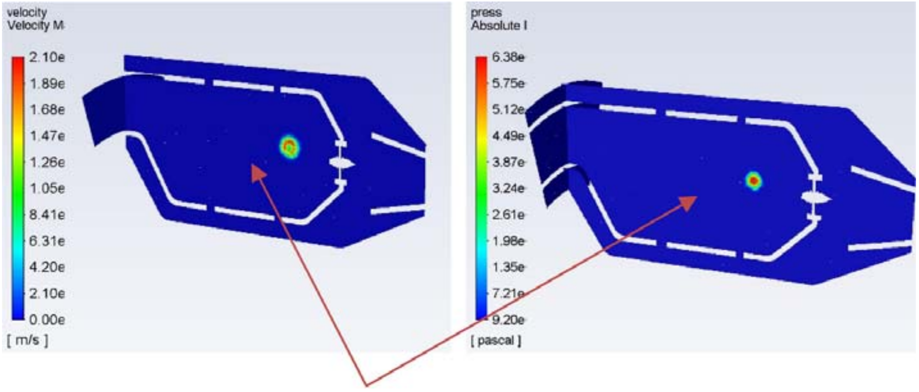

Fig. 3. Excitation region of the NP, flow velocity on the left, absolute pressure on the right

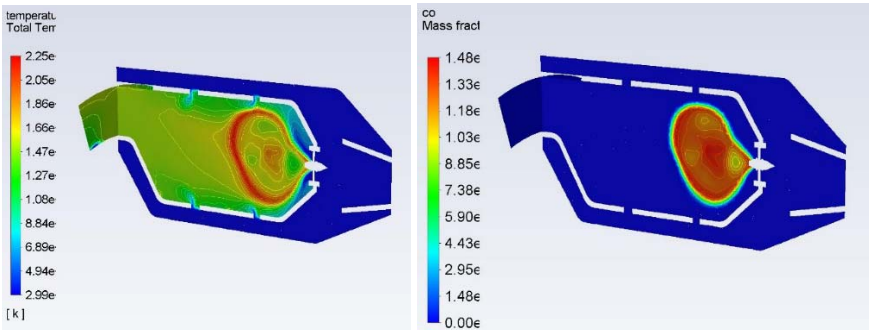

Fig. 4. Temperature after excitation of NP on the left, mass fraction of CO on the right, fine grid

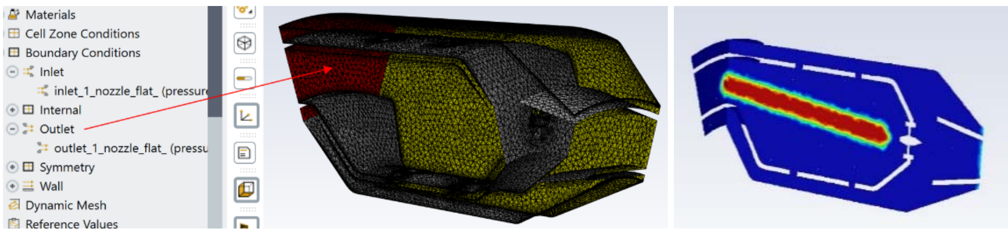

Fig. 5. Outlet section on the right, "fire path" on the left

stationary solver, either until the values close to stable by primitive variables (temperature/velocity/pressure) and mass flow rate were achieved in the outlet section of the combustion chamber Outlet, provided that there is a "fire path" (having the physical meaning of a cylindrical control volume) from the ignition zone to the geometric centre of the Outlet section (fig. 5), or until a critical (poor stall combustion termination) or non-physical result is obtained, the time step is assumed to be combined (fixed at the steady stage and adaptive at the transient stage).

According to the authors of this article, the criterion for stabilization of the burning solution should be as "inert" as possible, i.e. it should not be subject to instantaneous computational precessions in different solvers, inevitable when calculating transient processes in spatial grids, which leads to the assumption that attempts to find criteria for formal stabilization of burning solutions in the area of high gradients of primitive gas-dynamic variables (temperature, velocity, pressure) and concentration of unstable chemical components in the combustion front are unpromising. In view of the unstable nature

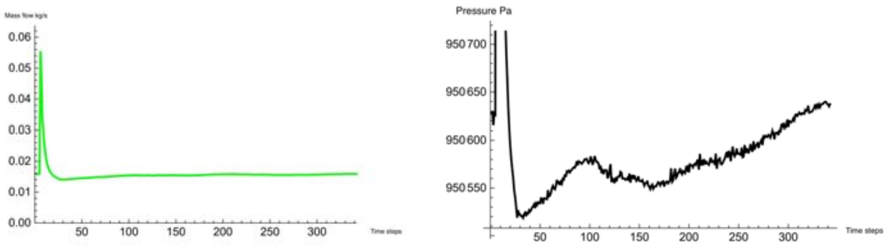

Fig. 6. Fine grid, mass flow on the left, pressure on the right

of the fuel assembly combustion process in the combustion chamber caused by the discreteness of the fuel supply modeling, which is reflected in the real combustion chamber by pressure pulsations in the fuel supply path to the injectors, the criteria for stabilization of the non-stationary calculation are proposed to be the occurrence of oscillations of the controlled parameters (primitive variables and mass flow rate) having a pronounced random nature from a specific time point of the calculation until its completion. It is proposed to determine the determination of the random nature of instability of measured variables as the values of the Pearson criterion for the average values of flow parameters (temperature, velocity, pressure, mass flow rate) in the Outlet section, under the conditions of existence of the combustion process (temperature in the "fire path" is not less than 1100 K) and restoration of engine thrust (not less than 98% of the initial value).

General parameters of generated grids

The study included a series of combustion chamber calculations on unstructured grids with tetrahedral elements of different resolutions (coarse, medium, fine) of equivalent topology and finite element quality. The grid model was built in the standard ANSYS. Fluent grid builder from the boundary layer (with the highest possible topological quality of the third layer of prismatic boundary layer elements, due to the greatest energy contribution of these layers to the formation of the turbulence pattern) with further moderate growth of finite element sizes towards the geometric centre of the computational domain. To determine the required grid characteristics in the near-wall regions, the Y+ criterion was determined according to the method [16] in a series of cold blowdowns of the combustion chamber for all computational grids; the number of prismatic layers in all experiments was assumed to be 12 with exponential growth of thickness.

Calculation results on a fine grid (1.16 million cells)

The linearly interpolated calculation results, namely the average integral values of mass flow rate, pressure, temperature, velocity and CO concentration in the outlet section are presented in Figures 6, 7. The solution on the fine grid did not require adaptation of the steady result for its subsequent use by the non-stationary solver, which allowed to excite the non-stationary process at the 5th iteration.

The red vertical line in Figure 8 indicates the iteration of NP excitation – iteration No. 5.

The correspondence between calculation time and time steps (iterations) is shown in Figure 9.

The calculation results show that at the time of 0.201965 s (iteration No. 325) from the start of non-stationary calculations until its completion, the calculation can be considered as having a tendency towards non-divergence, all fluctuations of primitive variables and mass flow rate are predominantly random in nature and the temperature on the entire "fire path" (fig. 10) is higher than the temperature of stable combustion of the fuel assembly.

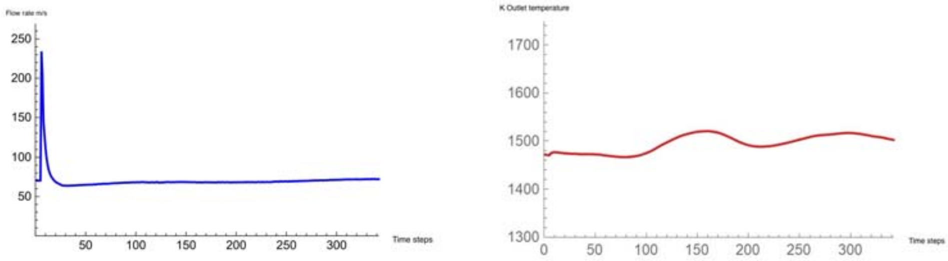

Fig. 7. Fine grid, flow rate on the left, temperature on the right

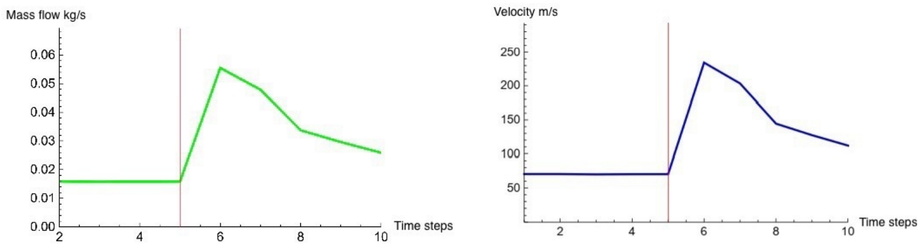

Fig. 8. Fine grid, NP excitation iteration, mass flow on the left, flow rate on the right

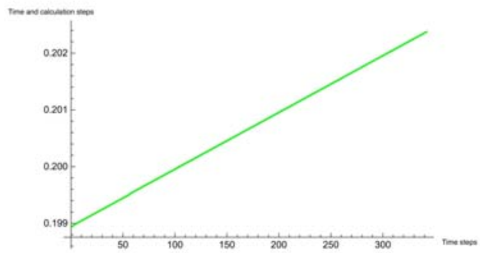

Fig. 9. Fine grid, time and calculation steps

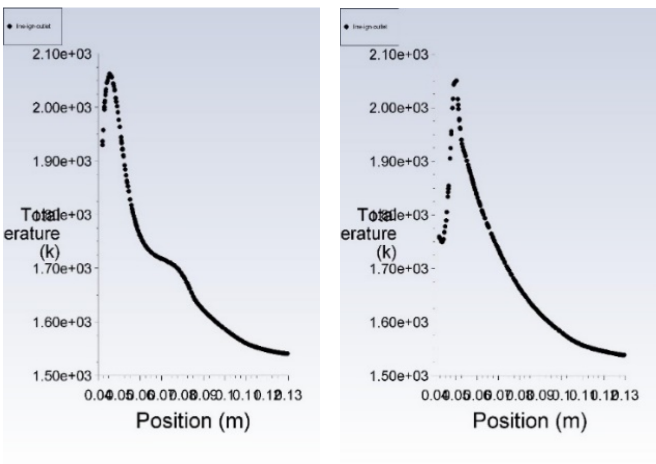

Fig. 10. Fine grid, temperature on the “fire path” iteration No. 325 on the left, iteration of the end of the calculation on the right

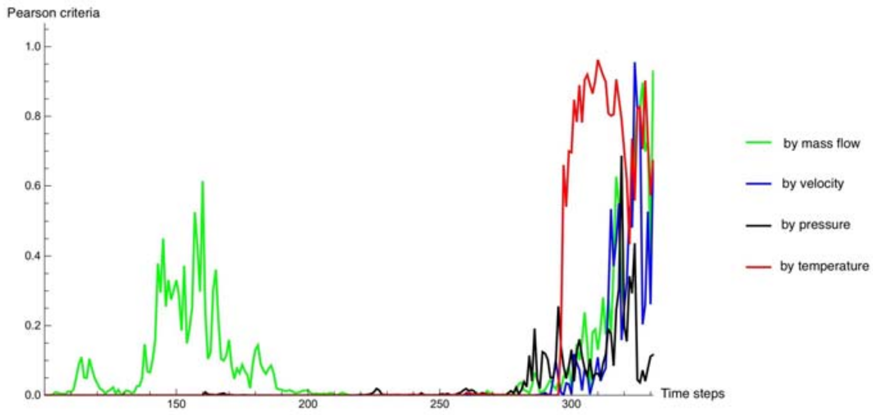

Fig. 11. Fine grid, Pearson criteria

The results of calculations on a fine grid, presented above, allow us to conclude that the process of combustion of fuel assembly combustion in the combustion chamber stabilizes within about 3e-3 s and thrust recovery at the level of 99.6% of the moment preceding the NP excitation. The formal criterion for determining the iteration from which the process should be considered non-divergent is proposed to be the iteration where the value of the Pearson criterion for all observed variables in the range up to the end of the calculation minus 10 iterations (for the possibility of fundamental use of the statistical mathematical apparatus) does not fall into the region of small positive values (not less than 0.2).

Pearson's criteria for achieving output flow stability are shown in Figure 11.

Calculation results on the medium grid (668 thousand cells)

The calculation results are linearly interpolated, the average integral values of mass flow rate, pressure, temperature, velocity and CO concentration in the outlet section are presented in Figures 12, 13. The solution on the medium grid required adaptation of the steady result for its use by the non-stationary solver (conducting a non-stationary calculation up to small fluctuations in the residuals of primitive variables at the level of 1^e-3), the conditions for NP excitation were achieved at the 78^{th} iteration.

The red vertical line in Figure 14 indicates the iteration of NP excitation, iteration No. 78.

The calculation shows that during the whole non-stationary calculation (715 iterations), the calculation has no formal sign of predominantly random temperature fluctuations according to the Pearson criterion (fig. 15 on the right) and mass flow rate (fig. 17), the surge in the Pearson criterion value temperature around the 700^{th} iteration is due to the small volume of statistical sample and is not an indicator of stabilization. The temperature on the “fire path” (fig. 16) is higher than the temperature of stable combustion of the fuel assembly from the 450^{th} iteration, but the observed intermediate data of the flow velocity abandonment (≈ 1.5 km/s) and mass flow rate after NP excitation do not allow us to consider this result acceptable.

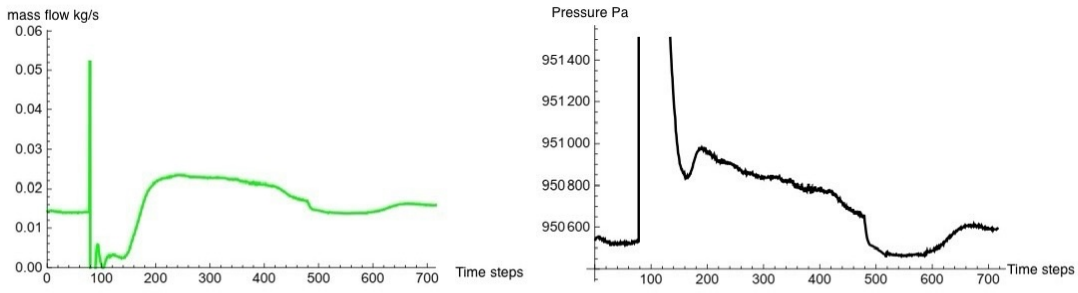

Fig. 12. Medium grid, mass flow on the left, pressure on the right

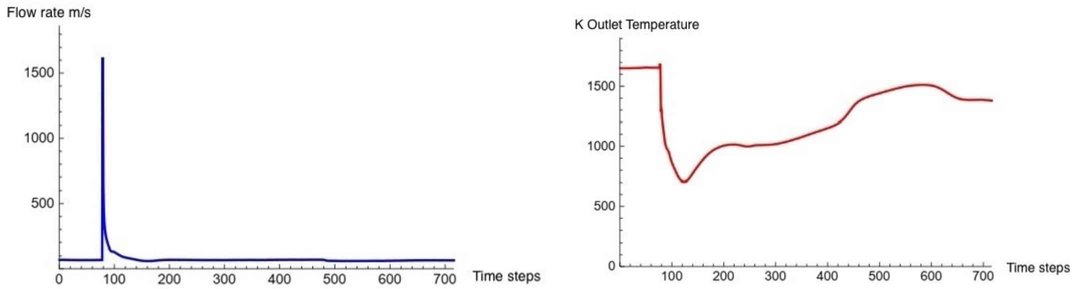

Fig. 13. Medium grid, flow rate on the left, temperature on the right

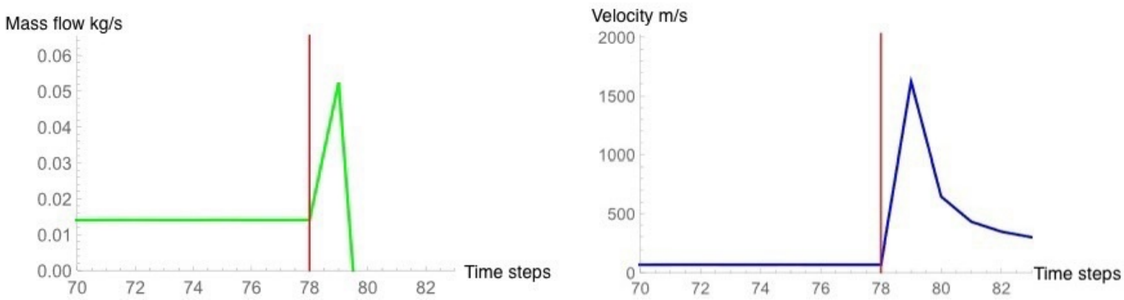

Fig. 14. Medium grid, NP excitation iteration, mass flow on the left, flow velocity on the right

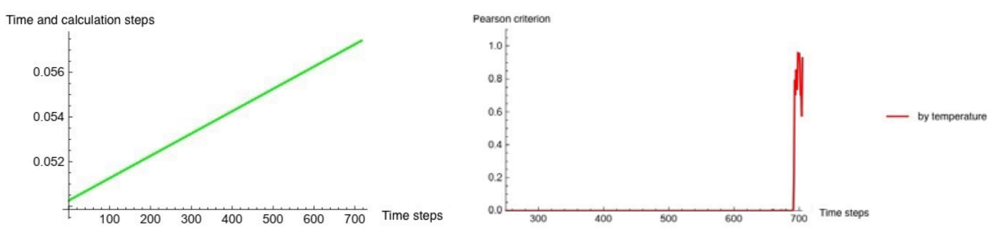

Fig. 15. Grid medium, time and calculation steps on the left, Pearson criterion on the right

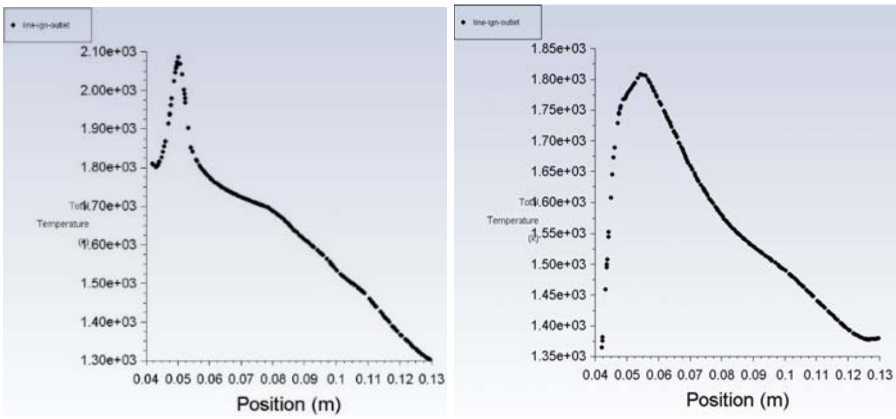

Fig. 16. Medium grid, temperature on the "fire path" iteration No. 450 on the left, iteration of the end of the calculation on the right

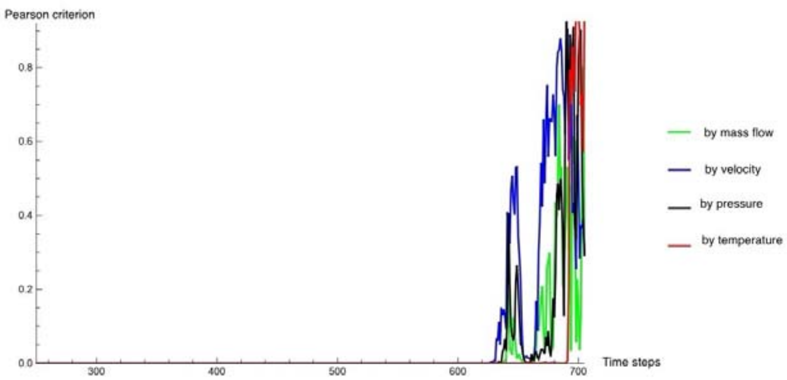

Fig. 17. Medium grid, Pearson criteria

Calculation results on coarse grid (333 thousand cells)

The calculation results are linearly interpolated, the average integral values of mass flow rate, pressure, temperature, velocity and CO concentration in the outlet section are presented in Figures 18, 19. The solution on the coarse grid required a significant adaptation of the steady result for its use by the non-stationary solver; the conditions for NP excitation were achieved at the 878^{th} iteration.

The red vertical line in Figure 20 indicates the NP excitation iteration, iteration No. 878, the correspondence between the calculation time and calculation iterations in Figure 21.

The calculation shows that the temperature on the "fire path" (fig. 22) is lower than the combustion temperature of the fuel assembly throughout the entire non-stationary calculation, i.e. the engine does not provide thrust. The intermediate data of the flow velocity ≈ 6 km/s and mass flow rate ≈ 0.14 kg/s, with the mass of air in the combustion chamber volume = 3.261e-4 kg, after NP excitation, as well as a significant difference in the transition point to the range of significant value of the Pearson criterion for the most inert and, as a consequence, the most delayed fixed physical quantity (temperature) relative to other variables (fig. 23) do not allow us to consider the result obtained on the coarse grid correct and physical.

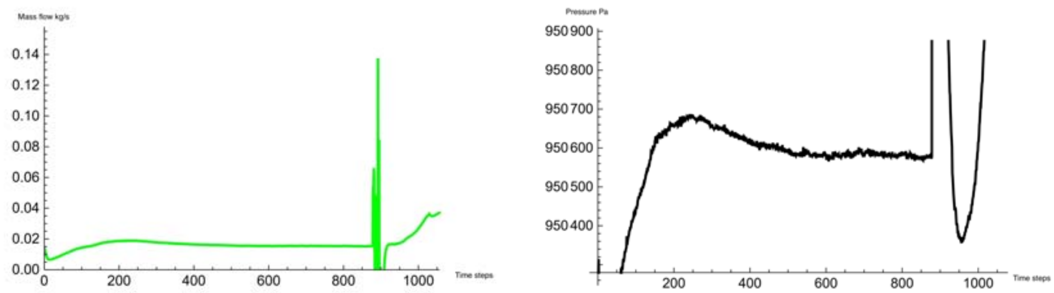

Fig. 18. Coarse grid, mass flow on the left, pressure on the right

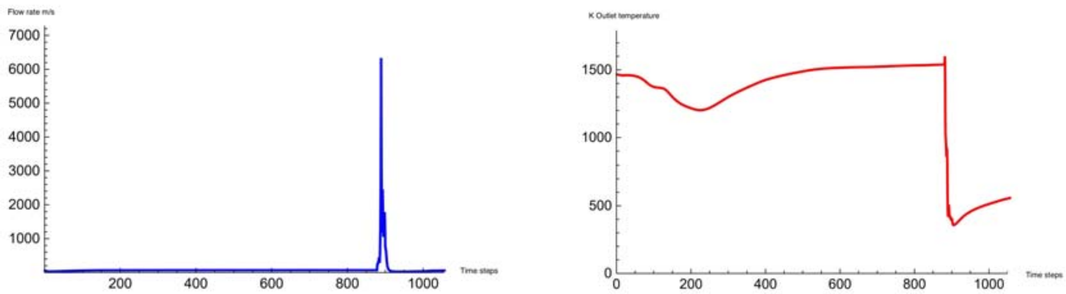

Fig. 19. Coarse grid, flow velocity on the left, temperature on the right

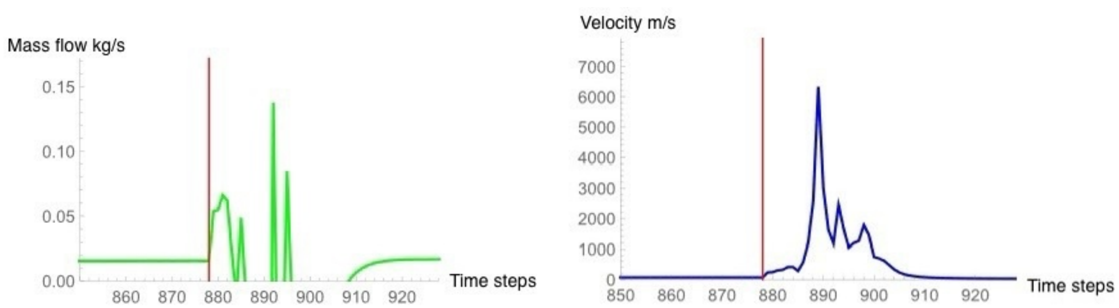

Fig. 20. Coarse grid, NP excitation iteration, mass flow on the left, flow velocity on the right

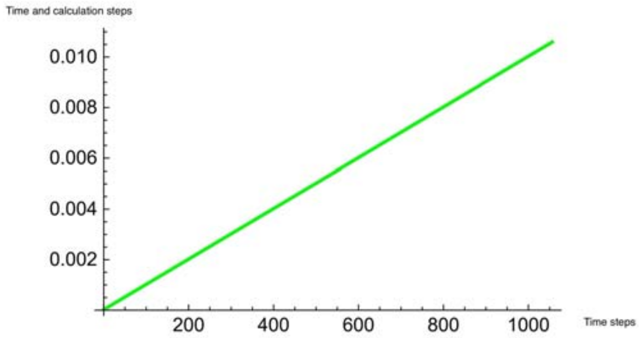

Fig. 21. Coarse grid, time and calculation steps

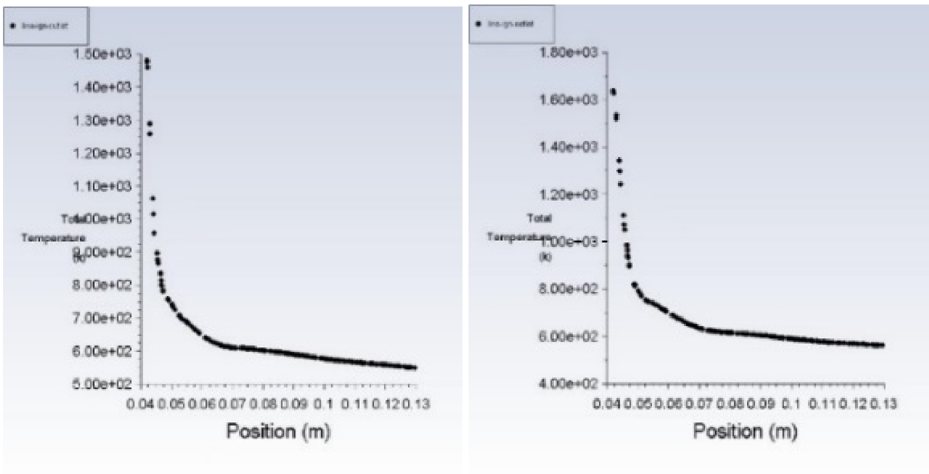

Fig. 22. Coarse grid, temperature on the "fire path" iteration No. 1000 on the left, iteration of the end of the calculation on the right

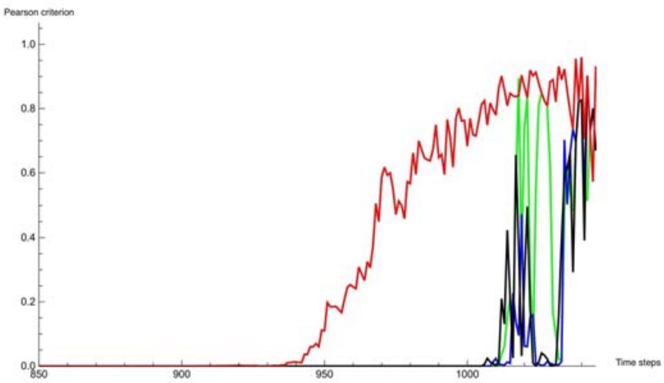

Fig. 23. Coarse grid, Pearson criteria

Discussion of results

The results of calculations on grids of different resolutions:

- fine – 1.16 million cells;
- medium – 668 thousand cells;
- coarse – 333 thousand cells

have shown the possibility of fundamentally using medium-resolution grids with obtaining significant results. Coarse-resolution grids produce non-physical results. High-resolution grids, as expected, produce results with significant computational resource costs.

The size of the computational grid for a particular problem should be determined by a series of numerical experiments with equivalent initial conditions and gradual refinement of the grid until obtaining slightly different time points of the beginning of stabilization of the solution by the values of throws of the controlled parameters during the simulation of the entire process under the conditions specified in paragraph "General parameters of the series of numerical experiments".

Conclusion

The currently used numerical methods, widely used for solving gas dynamics problems, allow modeling complex multiphysics processes occurring in burning gas flows. At the same time, all numerical methods have limitations related to grid sensitivity, i.e. the dependence of calculation results on the size and topology of the grid on which they are carried out. This can lead to errors and inaccuracies in the results, especially if the grid is coarse or its topology does not correspond to the features of the problem being solved.

The Pearson criterion is one of the most common methods of testing hypotheses about the correspondence of the empirical distribution to the theoretical one. It allows assessing the degree of discrepancy between the observed and expected in accordance with the theoretical distribution.

The proposed computational technique is critically sensitive to the volume and quality of the grids used and shows results of sufficient accuracy when using fine and medium grids with acceptable time cost and using average power computing resources (19.2 GFlops).

References

1. **Savchuk, A.T., Yakovlev, A.A.** (2021). Method of the numerical modelling of unstationary processes in the combustion chamber of a gas turbine engine. *Journal of Physics Conference Series*, vol. 1925. ID: 012009. DOI: 10.1088/1742-6596/1925/1/012009 (accessed: 25.08.2024).

2. **Le Sausse, P., Fabrie, P., Arnou, D., Clunet, F.** (2013). CFD comparison with centrifugal compressor measurements on a wide operating range. *In: EPJ Web of Conferences*, vol. 45. ID: 01059. DOI: 10.1051/epjconf/2013 4501059 (accessed: 25.08.2024).

3. **Savchuk, A.T., Yakovlev, A.A.** (2020). Numerical modelling of unstationary processes in the combustion chamber of a gas turbine engine. *In: XIII Mezhdunarodnaya konferentsiya po prikladnoy matematike i mekhanike v aerokosmicheskoy otrasli (AMMAI'2020): materialy konferentsii*. Moscow: MAI, pp. 101–103. (in Russian)

4. **Savchuk, A.T., Yakovlev, A.A.** (2021). On the issues of numerical modeling of unsteady processes in the combustion chamber of a gas turbine engine. *In: XXII Mezhdunarodnaya konferentsiya po vychislitel'noy mekhanike i sovremennym prikladnym programmnym sistemam (VMSPPS'2021): materialy konferentsii*. Moscow: MAI, pp. 624–626. (in Russian)

5. **Mangani, L., Casartelli, E., Mauri, S.** (2012). Assessment of various turbulence models in a high pressure ratio centrifugal compressor with an object oriented CFD code. *Journal of Turbomachinery*, vol. 134, no. 6. ID: 061033. DOI: 10.1115/1.4006310 (accessed: 25.08.2024).

6. **Pecnik, P., Pieringer, P., Sanz, W.** (2005). Numerical investigation of the secondary flow of a transonic turbine stage using various turbulence closures. *In: Proceedings of the ASME Turbo Expo 2005*, vol. 6, pp. 1185–1193. DOI: 10.1115/GT2005-68754.

7. **Trébinjac, I., Kulisa, P., Bulot, N., Rochuon, N.** (2009). Effect of unsteadiness on the performance of a transonic centrifugal compressor stage. *Journal of Turbomachinery*, vol. 131, no. 4. ID: 041011. DOI: 10.1115/1.30 70575 (accessed: 25.08.2024).

8. **Ibaraki, S., Matsuo, T., Kuma, H., Sumida, K., Suita, T.** (2002). Aerodynamics of a transonic centrifugal compressor impeller. *ASME Turbo Expo 2002: Power for Land, Sea, and Air*, vol. 5, pp. 473–480. DOI: 10.1115/gt 2002-30374

9. **Danilishin, A.M., Kozhukhov, Y.V., Neverov, V.V., Malev, K.G., Mironov, Y.R.** (2017). The task of validation of gas-dynamic characteristics of a multistage centrifugal compressor for a natural gas booster compressor station. *In: AIP Conference Proceedings*, vol. 1876. ID: 020046. DOI: 10.1063/1.4998866 (accessed: 25.08.2024).

10. **Elfert, M., Weber, A., Wittrock, D., Peters, A., Voss, C., Nicke, E.** (2017). Experimental and numerical verification of an optimization of a fast rotating high-performance radial compressor impeller. *Journal of Turbomachinery*, vol. 139, no. 10. ID: 101007 DOI: 10.1115/1.4036357 (accessed: 25.08.2024).

11. **Zandsalimy, M., Ollivier Gooch, C.F.** (2024). Mesh optimization for improved computational fluid dynamics numerical stability and convergence rate. *In: AIAA Aviation Forum and Ascend 2024*. DOI: 10.2514/6.2024-3615 (accessed: 25.08.2024).

12. **Zandsalimy, M., Ollivier-Gooch, C.** (2022). A novel approach to mesh optimization to stabilize unstructured finite volume simulations. *Journal of Computational Physics*,

vol. 453. ID: 110959. DOI: 10.1016/j.jcp.2022.110959 (accessed: 25.08.2024).

13. Sharbatdar, M., Ollivier Gooch, C.F. (2013). Eigenanalysis of truncation and discretization error on unstructured meshes. *In: 21st AIAA Computational Fluid Dynamics Conference*, ID: 3089, 25 p. DOI: 10.2514/6.2013-3089 (accessed: 25.08.2024).

14. Chen, L. (2016). Stability analysis and stabilization of unstructured finite volume method. *A thesis Master of Applied Science*. University of British Columbia, 122 p. DOI: 10.14288/1.0300002 (accessed: 25.08.2024).

15. Lee, C.-M., Kundu, K. (2013). Simplified Jet-A kinetic mechanism for combustor application. *NASA Technical Memorandum 105940 AIAA-93-0021*, 13 p. Available at: https://ntrs.nasa.gov/api/citations/19930006315/downloads/19930006315.pdf (accessed: 25.08.2024).

16. Pugachev, P.V., Svoboda, D.G., Zharkovsky, A.A. (2016). Calculation and design of blade hydraulic machines. Calculation of viscous flow in blade hydraulic machines using the ANSYS CFX: tutorial. St. Petersburg: SPbPU, 120 p. (in Russian)

Список литературы

1. Savchuk A.T., Yakovlev A.A. Method of the numerical modelling of unstationary processes in the combustion chamber of a gas turbine engine [Электронный ресурс] // Journal of Physics Conference Series. 2021. Vol. 1925. ID: 012009. DOI: 10.1088/1742-6596/1925/1/012009 (дата обращения: 25.08.2024).

2. Le Sausse P. CFD comparison with centrifugal compressor measurements on a wide operating range / P. Le Sausse, P. Fabrie, D. Arnou, F. Clunet [Электронный ресурс] // EPJ Web of Conferences, 2013. Vol. 45. ID: 01059. DOI: 10.1051/epjconf/20134501059 (дата обращения: 25.08.2024).

3. Савчук А.Т., Яковлев А.А. Численное моделирование нестационарных процессов камеры сгорания газотурбинного двигателя // XIII Международная конференция по прикладной математике и механике в аэрокосмической отрасли (AMMAI'2020): материалы конференции. Алушта, 06–13 сентября 2020 г. М.: МАИ, 2020. С. 101–103.

4. Савчук А.Т., Яковлев А.А. К вопросам численного моделирования нестационарных процессов камеры сгорания газотурбинного двигателя // XXII Международная конференция по вычислительной механике и современным прикладным программным системам (ВМСППС'2021): материалы конференции. Алушта, 04–13 сентября 2021 г. М.: МАИ, 2021. С. 624–626.

5. Mangani L., Casartelli E., Mauri S. Assessment of various turbulence models in a high pressure ratio centrifugal compressor with an object oriented CFD code [Электронный ресурс] // Journal of Turbomachinery. 2012. Vol. 134, no. 6. ID: 061033. DOI: 10.1115/1.4006310 (дата обращения: 25.08.2024).

6. Pecnik P., Pieringer P., Sanz W. Numerical investigation of the secondary flow of a transonic turbine stage using various turbulence closures // Proceedings of the ASME Turbo Expo 2005, 2005. Vol. 6. Pp. 1185–1193. DOI: 10.1115/GT2005-68754.

7. Trébinjac I. Effect of unsteadiness on the performance of a transonic centrifugal compressor stage / I. Trébinjac, P. Kulisa, N. Bulot, N. Rochuon [Электронный ресурс] // Journal of Turbomachinery. 2009. Vol. 131, no. 4. ID: 041011. DOI: 10.1115/1.3070575 (дата обращения: 25.08.2024).

8. Ibaraki S. Aerodynamics of a transonic centrifugal compressor impeller / S. Ibaraki, T. Matsuo, H. Kuma, K. Sumida, T. Suita // ASME Turbo Expo 2002: Power for Land, Sea, and Air. 2002. Vol. 5. Pp. 473–480. DOI: 10.1115/gt2002-30374

9. Danilishin A.M. The task of validation of gas-dynamic characteristics of a multistage centrifugal compressor for a natural gas booster compressor station / A.M. Danilishin, Y.V. Kozhukhov, V.V. Neverov, K.G. Malev, Y.R. Mironov [Электронный ресурс] // AIP Conference Proceedings, 2017. Vol. 1876. ID: 020046. DOI: 10.1063/1.4998866 (дата обращения: 25.08.2024).

10. Elfert M. Experimental and numerical verification of an optimization of a fast rotating high-performance radial compressor impeller /

M. Elfert, A. Weber, D. Wittrock, A. Peters, C. Voss, E. Nicke [Электронный ресурс] // Journal of Turbomachinery. 2017. Vol. 139, no. 10. ID: 101007 DOI: 10.1115/1.4036357 (дата обращения: 25.08.2024).

11. Zandsalimy M., Ollivier Gooch C.F. Mesh optimization for improved computational fluid dynamics numerical stability and convergence rate [Электронный ресурс] // AIAA Aviation Forum and Ascend 2024. DOI: 10.2514/6. 2024-3615 (дата обращения: 25.08.2024).

12. Zandsalimy M., Ollivier-Gooch C. A novel approach to mesh optimization to stabilize unstructured finite volume simulations [Электронный ресурс] // Journal of Computational Physics. 2022. Vol. 453. ID: 110959. DOI: 10.1016/j.jcp.2022.110959 (дата обращения: 25.08.2024).

13. Sharbatdar M., Ollivier Gooch C.F. Eigenanalysis of truncation and discretization error on unstructured meshes [Электронный ресурс] // 21st AIAA Computational Fluid Dynamics Conference, 2013. ID: 3089. 25 p. DOI: 10.2514/6.2013-3089 (дата обращения: 25.08.2024).

14. Chen L. Stability analysis and stabilization of unstructured finite volume method [Электронный ресурс] // A thesis Master of Applied Science. University of British Columbia, 2016. 122 p. DOI: 10.14288/1.0300002 (дата обращения: 25.08.2024).

15. Lee C.-M., Kundu K. Simplified Jet-A kinetic mechanism for combustor application [Электронный ресурс] // NASA Technical Memorandum 105940 AIAA-93-0021, 1993. 13 p. URL: https://ntrs.nasa.gov/api/citations/19930006315/downloads/19930006315.pdf (дата обращения: 25.08.2024).

16. Пугачев П.В., Свобода Д.Г., Жарковский А.А. Расчет и проектирование лопастных гидромашин. Расчет вязкого течения в лопастных гидромашинах с использованием пакета ANSYS CFX: учеб. пособие. СПб.: СПбПУ, 2016. 120 с.

Information about the authors

Andrey T. Savchuk, Candidate for a Scientific Degree of the Aircraft Engines Chair, Moscow State Technical University of Civil Aviation, andrey.sav4uk@yandex.ru.

Lada V. Moskalenko, Candidate of Technical Sciences, Associate Professor, Aircraft Engines Chair, Moscow State Technical University of Civil Aviation, l.moskalenko@mstuca.ru.

Сведения об авторах

Савчук Андрей Тодорович, соискатель МГТУ ГА, andrey.sav4uk@yandex.ru.

Москаленко Лада Вячеславовна, кандидат технических наук, доцент, доцент кафедры двигателей летательных аппаратов МГТУ ГА, l.moskalenko@mstuca.ru.

Поступила в редакцию	25.11.2024	Received	25.11.2024
Одобрена после рецензирования	16.12.2024	Approved after reviewing	16.12.2024
Принята в печать	23.01.2025	Accepted for publication	23.01.2025