UDC 621.694.2:629.3082.3 (045)

Kostjantin Kapitanchuk[1]
Pavlo Grekov[2]
Mykola Bohdanov[3]
Galina Nikitina[4]

FINITE DIFFERENCE METHOD FOR CALCULATING OF THE GAS FLOW IN A SUBSONIC GAS EJECTOR

National Aviation University
1, Kosmonavta Komarova avenue, Kiev, 03680, Ukraine
E-mails: [1]k.kapitanchuk@ukr.net; [2]grekov59@gmail.com; [3]BogdanovNY@gmail.com; [4]Galyna_Nikitina88@mail.ru

Abstract. *Describe analysis of eddy viscosity actual mathematical models for numerical simulation a reversal gas flow in subsonic gas ejector. Considered advantages and disadvantages each of it. Proposed use method of finite elements for provides viscous gas flow calculation of gas ejectors.*

Keywords: calculation; finite elements method; gas ejector; gas turbine engine; Navier-Stokes combined equations; subsonic gas flow.

1. Introduction

The development of the aviation industry, the creation of multifunctional aircraft, requiring a large complex of research, deep study of gas dynamics phenomena for use under development of gas turbine engines (GTE) power plants in general and its separate components.

2. Problem formation

Increasing requirements for power plants, the desire to get maximum power and provide the least resistance to external integration of aircraft glider have led to the necessity in-depth research of gas dynamics flow in the output devices, which are an essential element of power plants.

Using the ejector jet nozzle (output devices) of different purposes creates additional problems related to the issues of optimal aerodynamic design to improve the efficiency of their application with minimal loss of energy and reducing the life cycle cost of GTE.

3. Analysis of research and publications

The essential three-dimensional flow, the presence of turbulent flow and detachable zones, variety of gas dynamic and geometric parameters that define the type, scheme and low of regulations, don't allow at present time, solve theoretically all problems that arise in the development and creation nozzles for modern and future aircraft [1].

These effects in subsonic gas ejectors with flow rotation require providing research to improving the possibility of using some finite-difference methods of subsonic flow calculation in channels of complex shapes and models of turbulent viscosity.

The basic mathematical model in approaching the continuous medium is the system of Navier-Stokes equations. Creation of efficient algorithms for numerical simulation of flow, incompressible viscous fluid and subsonic currents of viscous gas in various technical devices by the Navier-Stokes equations with the broad changing of parameters is an actual scientific task.

Because of the complexity of determining the Navier-Stokes equations in the system with other transfer equations (heat conduction, energy conservation and so on.), simulation is possible only through the use of numerical methods, in particular - finite-difference. Basic approaches to integrating Navier-Stokes equations associated with the construction methodology of calculation of the pressure. There are methods for incompressible fluids without calculation pressure [2], where the function is excluded introduction of pressure stream function, which eliminates the problems associated with the calculation pressure, but leads to other problems with the construction of the boundary conditions for the function of the vortex. Simulation of turbulent flows of liquid and gas is also carried out on the basis of Navier-Stokes equations, of bringing them to Reynolds averaged equations and closure of by the semi-empirical theories that are suitable only for specific movements [3].

The most promising is the direct modeling of turbulent flows by the Navier-Stokes equations, which requires the presence of a large computer performance compared with those that are common today.

4. Formation of the research problem

For one classes of internal problems of fluid flow or gas modeling, can be attributed simulation of viscous flows in rotating system with the presence of Carioles force. In such flows Carioles force is equivalent to the value of external forces, they have a complex structure even for simple cases of straight boundaries. The most difficult in the process of gas flow modeling is calculation of unbalanced flows that are based on Navier-Stokes equations together with diffusion and mass concentrations of the reactants. At present great practical importance gained simulation of gas flow in combustion chamber of gas turbine engines. The most important theoretical and practical results at the process of flow calculation with flow separation were obtained with the help of viscous liquid asymptotic theory and semi-empirical methods that are based on the numerical and natural experiments [4].

Complementing each other, the calculation and experiment provide new opportunities for the study of complex interdependent processes. The main problem of obtaining solutions unsteady Navier-Stokes incompressible fluid is the difficulty of solving simultaneous equations of momentum and continuity equations.

5. Using the finite-difference methods for calculating flow in channels of complex shape

In the first phase of development of numerical algorithms for solving the Navier-Stokes incompressible flows, frequently used variables such vorticity - function flow.

The use of physical variables allows solving two-dimensional and three-dimensional problem of a single algorithm. Basic mathematical problems in solving Navier-Stokes equations associated with different types of differential equations for the laws of conservation of mass and momentum. Different ways of overcoming these difficulties that are arise during pressure determination of Poisson equation for amendments [5], different penalty functions [6], add the continuity equation transient member [6], regularization matrix factors in determining the time derivative.

An actual problem in the process of numerical calculations by solving the Navier-Stokes equations is the construction and development of new approaches that would reduce the required amount of time in modeling at computer. During modeling the subsonic flows with decreasing of Mach down to zero finite-difference schemes essentially lose stability.

To improve stability possible to use smoothing oscillations without reducing order of approximations and introduce artificial diffusion in areas with large gradients to provide additional smoothing numerical result. However, the presence of flow turns, detachable and reverse currents, paired vortices leads to significant differences of calculation and experiment, and also in most cases, prevent the calculation of the finite-difference method.

As you know, during the movement of fluid along the curved channel velocity of fluid particles decreases with the increasing of curvature radius. That is why the value of pressure at the inner wall is less than the value of the pressure at the other wall. So, at the entrance of the curved knees in the inner wall is formed confuser type area, and at external - diffuser shape of area. However, at the exit from curved knees directly into the pipe, vice versa we have diffuser form at internal wall and infuser at external wall.

From the theory of boundary layer known that the diffuser boundary layer region is growing very intensively becomes unstable and can easily break away from the wall. This phenomenon is observed in regions of curved diffuser knees. Photos of process flow stream in the knee area clearly show location of local separation near external and internal walls.

Schematic view of a flow in knee (fig.)

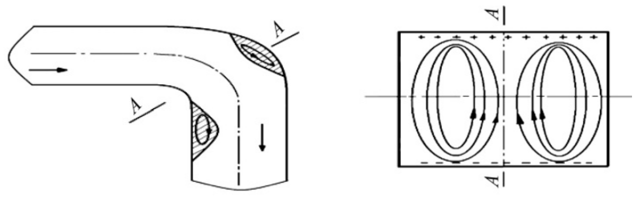

Fig. Scheme of gas flow in turning knee of GTE exhaust device

Because at the external wall the value of a pressure is greater than at the inside, than along the side wall where velocity is equal zero, will be the overflow of liquid or gas from external wall to internal, that’s leads to pair vortex.

This transverse overflow of fluid, usually called secondary flows. The secondary flow can be observed by measuring the velocity components that form vortex doubles.

In section A-A shown a tangential velocity field. Installation of ejector devices at the turning section of a gas flow at exhaust device GTE leads to additional energy losses which are caused by the presence of paired vortex and reverse flows in the second circuit of the gas nozzle ejector.

Therefore, the structure of flow in curved channel is determined by the pressure increase and velocity decrease in the direction from the external to the internal wall of knees, the presence of local boundary layer separation, and finally the formation of vortex paired in the knee. These phenomena determine the nature of the losses in the knee and qualitative changes uniformity of flow that occur in the presence of the pipeline knee.

From the analysis of the physical picture of the flow in turn can be concluded that the loss of the knee consists of friction losses, losses on the formation of paired vortices and losses through the presence of local separated flow. The latter are the most relative value, and friction losses are the smallest share of the total losses.

Therefore, to reduce losses in the knee, first of all, we are necessary to avoid of local aerodynamic diffusers, which often lead to local separated flow. It is necessary to reduce the intensity of secondary currents, which form paired vortices, and only after this provide reduction of friction. Besides, the loss depends on the geometrical characteristics of the knee, namely the radii of curvature of external and internal walls, forms cross-sectional area ratio of the input and output sections and others.

Calculation of such complex flows require detailed studies in the direction of definition suitability of various finite-difference methods that could be considered: the presence of separation areas and reverse flow, pressure changing at transverse direction, the presence of perturbations up by flow, friction on the wall.

Calculation of gas flow in channels of different shapes dedicated a significant amount of work performed at the Institute of Mechanical Engineering University them. Podgorny and Dnipropetrovsk National University them. Oles Gonchar.

For circuiting the system of equations, that are describing turbulent flow, turbulent viscosity models are used. Selecting the model of turbulence for calculating separated flow is an individual issue.

Although the substantial progress at modeling Large Eddy Simulation (LES) and Detach Eddy Simulation (DES) during solving practical problems there are widely used models based on the average Reynolds number Navier-Stokes equation (RANS).

With the closure of RANS used algebraic, one- and two-parameter model of turbulent viscosity. There is a widespread one-parameter model of Spalart-Allmaras, which demonstrates a reasonable compromise between computational expenses and accuracy of the results.

One-parameter model of turbulence Spalart-Allmaras (SA) [7] was developed in 1992 y. It is designed for solving problems of to meet the challenges of the external body flow around of a gas flow at small angles of attack with existing loose areas in boundary layer. The generation of turbulence is determined rotor field of velocity.

The presence of ambiguity and instability of solving the problem of motion of a viscous incompressible fluid needs to improve the accuracy and efficiency of numerical methods for calculating heat transfer and hydrodynamics in domains of arbitrary shape. It is particularly important to ensure properties conservatism and monotony numerical method in any coordinate system.

This makes it possible to minimize the disturbance introduced by the solution due to features of difference approximations and curvature of the grid lines.

To achieve this aim the most suitable family of SIMPLE algorithm based on the method of controlling the volume and type of scheme TVD, which are modified to calculate incompressible flow.

The decision on the use of models of turbulent viscosity depends on the presence of certain theoretical and experimental research.

Model of incompressible fluid, typically used to describe flows with Mach number is much less than one because direct application of methods based on the model of the compressed gas becomes ineffective due to hardness of the original system of equations [8]. Historically it so happened that the development of methods for calculating the flow of viscous incompressible fluid going in two directions.

Methods that use transformed variables of vorticity - flow function (ω–ψ), competed with methods based on recording original equations for variable velocity - pressure (u, v, p) [9].

Methods for solving the Navier-Stokes equations, written in the transformed variable (ω-ψ), make it possible to reduce the number of unknowns in the two-dimensional problems, but to calculate the number of unknown three-dimensional movements significantly increased compared to methods which use variable velocity - pressure.

In the methods (ω-ψ) there are difficulties with the production of physically reasonable boundary conditions for vorticity consideration on solid walls as well as for function in connections areas.

Methodology for determining the subsonic flows in variable velocity - pressure that are based on the model of incompressible fluid, and the idea of splitting by physical processes also different. They are divided by SIMPLE similar algorithm [9], method of artificial compressibility [8] and mixed Lagrangian-Euler methods such method of particles in cells [10].

One of the main problems for both methods (ω-ψ), and methods for (u, v, p), is the task of calculating the pressure field. The source of problems is the system of equations of incompressible fluid in which no equation for pressure.

In the transition to variable vorticity - function of pressure flow is excluded from the calculation, because the determination of influence on the body as part of such methods is problematic.

In such algorithms SIMPLE, SIMP-LEC [6], PISO [10] and the method of artificial compressibility, actually put an evolutionary task for pressure defines.

From the analysis of the equations of incompressible fluid it is stated that the field of pressure instantly responds to changes in velocity vector fields and massive force and doesn't depend from pressure distribution in the previous moment of a time.

In SIMPLER algorithm realized just such a scheme of interaction fields. This caused to noticeable increasing of efficiency of SIMPLER algorithm compare with other methods [4], especially for tasks where a decisive influence with massive force, particularly for temperature and concentration of free convection and also flows with twisting movements.

Disadvantage of SIMPLER algorithm can assume that it is implemented only distanced chess grid, while both methods of artificial compressibility [8, 9] and algorithms SIMPLE, SIM-PLEC, PISO could be realized on combined and part combined meshes.

At present, the most popular among ta specialist of calculating fluid dynamic is a family of SIMPLE algorithms.

Real geometry of industrial plants separate components also gas ejector requires, first of all, the possibility of using adaptive grids and secondly, independence of calculation methods parameters of selected coordinate system. On this point of view the greatest flexibility has a finite-element method.

Despite the latest advances in increasing the accuracy of approximation in finite element method, finite difference algorithms based on the method of control volume [4], exceed the finite element method for efficiency and accuracy.

In the finite-difference algorithms we have easier implemented opportunities to increase the accuracy of calculations by the use of high-order TVD schemes.

The first problem that arose during the application SIMPLE procedures in nonorthogonal coordinate systems associated with the inconsistency fields of velocity and pressure with the formation of so-called chess fields of pressure and velocity [4]. The main idea of SIMPLE algorithms is that, for pressure calculation used difference equation derived from discrete analogues of momentum and continuity equations.

In the output algorithm [4], recorded for the Cartesian coordinate system on a chess grid, a pressure gradient is calculated using the pressure values in the two neighboring nodes. In this case, there is no inconsistency between the fields of speed and pressure and grid model for pressure includes 2D+1 node, where D – task dimension. As a result on regular grid we have a matrix of equations system with 2D+1 diagonal. For solving such systems developed effective iterative methods.

If the momentum equation written in projection on the basis vectors of the Cartesian coordinate system, then going to nonorthogonal coordinate systems on the grid close to the orthogonal grid system of discrete equations for velocity and pressure falls apart into two independent systems. A large number of papers devoted to the elimination of this effect.

So, at research [9] proposed partially combined grid, where components are stored in the corners of volume and pressure and also authors noted the difficulty in obtaining the solution of which coincides through the wall pressure oscillations.

At research [10] M.P. Lobachev equation of motion shows in conservative form on partial combined grid of different velocity vectors. However, at difference equation of pressure, part of members moved to the right side, which has led to need of introduced an internal interactive cycle for solving Poisson equation during each global iteration. Consequently, using combined grid impossible to use the algorithm SIMPLER, that according to the efficiency superior to other SIMPLE methods.

The second problem that arises during the transition to curvilinear coordinates is the impact of curvature grid lines on the accuracy of numerical solution, including the conversion of Galileo for discrete equations of motion.

During recording of a discrete analogue for differential equations form of output equations and using as desired variables or counter covariant velocity component homogeneous flow, in general, does not satisfy the discrete counterparts.

Implementation of a discrete level conversion Galileo does not guarantee even for such simple currents as linear shear flow Kuetta, accurate calculation to not arbitrary orthogonal grid.

6. Conclusions

Nowadays there is almost no research that would clearly show the benefits of such scheme of TVD compared with different scheme for solving the problem of incompressible fluid. So, there are the tasks to develop a methodology to compare different algorithms for that type of task.

All the mentioned events and factors that occur in complex technical devices, one of which is subsonic gas ejectors with flow turns of active gas, in most cases need to research the possibility of using some finite-difference calculation methods of subsonic flow in channels complex forms and models of turbulent viscosity.

References

[1] *Lavruhin G. N.* Aerohydrodynamics of a jet nozzles // External characteristics of a nozzles – M. : Fizmatlit, 2003. – 376 P. (In Russian).

[2] *Rouch P.* Calculation hydrodynamics / P. Rouch. – M. : Mir, 1980. – 616 P. (In Russian).

[3] *Gosmen A. M.* Calculation methods of researching of viscous fluid flow / A. M. Gosmen, V. M. Pan, A. K. Ranchel; pod red. A. M. Gosmena. – M. : Mir, 1972. – 323 P. (In Russian).

[4] *Gogish A. V.* Theory of two-dimensional flows / A. V. Gogish, V. Ja. Nejland, G. Ju. Stepanov // Itogi nauki i tehniki. Gidromehanika. – M. : Nauka, 1975. – P. 5–73. (In Russian).

[5] *Chorin A. J.* A numerical method for solving in compressible viscous flow problems: Journal of Compu-tation Physics / A. J. Chorin. – 1967. – № 2. – P. 12–26.

[6] *Spalart P. R.* A one-equation turbulence model for a aerodynamic flow / P. R. Spalart, S. R. Allmaras // AIAA Paper. – 1992. – № 12.1. – P. 439–478.

[7] *Hartvich P. M.* One-way scheme of high precision for calculation of incompressible three-dimensional flows according to Navier-Stokes equations / P. M. Hartvich, Ch.-H. Su // Ajerokosmicheskaja tehnika. – 1990. – № 7. – P. 95–105. (In Russian).

[8] *Belocerkovskij O. M.* Numerical calculation at continuum mechanics / O. M. Belocerkovskij. – M. : Fizmatlit, 1994. – 448 P. (In Russian).

[9] *Hirt C. W.* An Arbitrary Lagrangian Eulerian Com-puting Method for all Flow Speeds / C. W. Hirt, A. A. Amsden, J. L. Cook // Journal of Computational Physics. - 1974. – Vol. 14. – № 3. – P. 227-253.

[10] *Lobachev M. P.* Developing the algorithm of pressure field calculation at viscous fluid flow with the help of method of finite elements: Tehn. otchet CNII im. akad. A. N. Krylova / M. P. Lobachev. – Vyp. 355. – 1993. – 21 P. (In Russian).

Received 15 September 2015.

К.І. Капітанчук[1], П.І. Греков[2], М.Ю. Богданов[3], Г.М. Нікітіна[4]. Кінцево-різницевий метод розрахунку течії газу у дозвуковому газовому ежекторі
Національний авіаційний університет, просп. Космонавта Комарова, 1, Київ, Україна, 03680
E-mails: [1]k.kapitanchuk@ukr.net; [2]grekov59@gmail.com; [3]BogdanovNY@gmail.com; [4]Galyna_Nikitina88@mail.ru

Проведено аналіз сучасних математичних моделей турбулентної в'язкості для численного моделювання течії в'язкого газу в дозвукових газових ежекторах з поворотом потоку. Розглянуті переваги та недоліки кожного з них. Запропоновано до використання кінцево-різницевий метод для проведення розрахунку течії в'язкого газу в дозвукових газових ежекторах.

Ключові слова: газотурбінний двигун; газовий ежектор; дозвукова течія газу; кінцево-різницевий метод; розрахунок; система рівнянь Нав'є-Стокса.

К.И. Капитанчук[1], П.И. Греков[2], Н.Ю. Богданов[3], Г.Н. Никитина[4]. Конечно-разностный метод расчета течения газа в дозвуковом газовом эжекторе
Национальный авиационный университет, просп. Космонавта Комарова, 1, Киев, Украина, 03680
E-mails: [1]k.kapitanchuk@ukr.net; [2]grekov59@gmail.com; [3]BogdanovNY@gmail.com; [4]Galyna_Nikitina88@mail.ru

Проведен анализ современных математических моделей турбулентной вязкости для численного моделирования течения вязкого газа в дозвуковых газовых эжекторах с поворотом потока. Рассмотрены достоинства и недостатки каждого из них. Предложено использовать конечно-разностный метод для проведения расчета течения вязкого газа в дозвуковых газовых эжекторах.

Ключевые слова: газотурбинный двигатель; газовый эжектор; дозвуковое течение газа; конечно-разностный метод; расчет; система уравнений Навье-Стокса.

Kapitanchuk Kostjantin. PhD, Associate Professor.
Aviation Engines Department, National Aviation University, Kyiv, Ukraine.
Research area studying of heat transfer and hydrodynamics in energy machines channels.
Education: Vasilkov Military-Aviation Technical School (1976).
E-mail: k.kapitanchuk@ukr.net

Grekov Pavlo. PhD, Associate Professor.
Aviation Engines Department, National Aviation University, Kyiv, Ukraine.
Education: Vasilkov Military-Aviation Technical School (1979).
Research area: studying of heat transfer and hydrodynamics in energy machines channels.
E-mail: grekov59@gmail.com

Bohdanov Mykola. Senior Lecturer.
Aviation Engines Department, National Aviation University, Kyiv, Ukraine.
Education: National Aviation University, Kyiv, Ukraine (2008).
Research area: studying of gas dynamic stability problems in axial fans and compressors.
Publications: 8.
E-mail: BogdanovNY@gmail.com

Nikitina Galina.
Aviation Engines Department, National Aviation University, Kyiv, Ukraine.
Research area: studying of purification and control quality of jet engine fuel and working fluid
E-mail: Galyna_Nikitina88@mail.ru