Journal of Applied Mechanics and Technical Physics, Vol. 37, No. 2, 1996

CALCULATION OF VORTEX GAS FLOWS IN COMPLEX-SHAPED DUCTS

O. B. Khairullina

UDC 533.6:519.6

Complex-shaped ducts are the main elements of propulsion in various technological facilities, in particular, combustion engines, in which complex unsteady hydro- and gasdynamic flows are formed. An important factor for a number of application problems is the knowledge of the structure of these flows and their parameters.

Let us adopt some simplifications [1–3] to calculate the gas flow in a combustion engine. We consider axisymmetric, simply connected ducts of complex geometry with a nozzle. Let us assume that the duct surfaces consist of porous surface segments through which the gas is injected, rigid walls, and gas efflux segments; the thickness of the boundary layers growing along the walls is small, compared with the crosswise duct size; the boundary layers do not interact with each other; the injected gas is homogeneous but not isoenergetic; and the gas flow is steady. Then the perfect gas model with flow satisfying the Euler equations may be used for numerical simulation of gasdynamic processes in the ducts.

Even in such a simplified formulation the development of effective numerical methods for computing vortex gas flows in complex-shaped ducts involves certain difficulties, especially if the gas velocity varies within the transonic range. The following problems arise: the gas flow parameters vary within wide ranges; closed streamlines are formed in the gas flow because of the complex duct geometries and counterflow interactions; the type of equation is different for subsonic and supersonic flows [4].

The absence of closed vortex flows or restrictions on the duct configurations are assumed in most papers where vortex flows in the ducts are calculated. A review is given in [5]. The present paper describes a constructive approach to the problem that allows effective calculations of steady gasdynamic flows with closed streamlines in the subsonic region of axisymmetric, simply connected, complex-shaped ducts.

Steady gas flows are often calculated by the time-dependent method. However, in the case of complex-shaped ducts and embedded nozzle the wave processes arising in the low-velocity region are slowly attenuated because of the gas counterflow interactions, and the time-dependent method converges very slowly or turns out to be a diverging one [4, 6]. The present paper employs a combined approach for calculating gas flows when the region of the entire gas tract is split into three overlapping subregions (subsonic, transonic, and supersonic) [4].

The subsonic flow computation requires three conditions specified at the entrance boundary and one condition at the duct exit [1, 4]. The exit condition is normally the gas flow rate. The mass flow in the normal direction and enthalpy are usually specified at the entrance boundaries in most papers [6]. The suggested technique is capable of calculating flows both with the above set of initial data and with the entrance conditions being density and velocity in the normal direction. In both cases the pressure is specified at a certain point. A SOKOL software system employing the suggested method can also use different sets of initial data but they are reduced to the two basic techniques. Subsonic flow is calculated by the finite difference iteration method [5] in stream function–vortex variables, which is a modification of the approach in [6].

One of the main differences between the methods of [5] and [6] is the use of nonorthogonal optimal curvilinear grids [7, 8], which allow one to remove the restrictions on the class of the considered duct configurations.

Institute of Mathematics and Mechanics, Ural Division, Russian Academy of Sciences, Ekaterinburg 620219. Translated from Prikladnaya Mekhanika i Tekhnicheskaya Fizika, Vol. 37, No. 2, pp. 103–108, March–April, 1996. Original article submitted March 1, 1995.

 0021-8944/96/3702-0236 $15.00

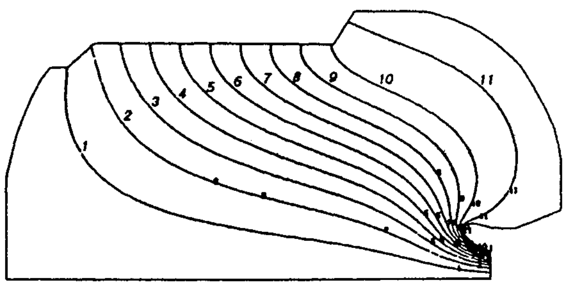

Fig. 1

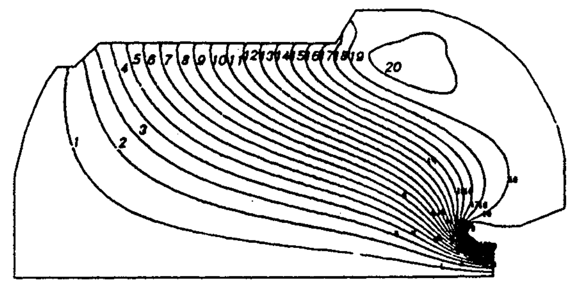

Fig. 2

For numerical simulation the Euler equations are written in integral form in the curvilinear coordinate system (q_1, q_2):

$$\oint\limits_C \left[\frac{1}{\rho r\Delta}\Big(A_1\frac{\partial\psi}{\partial q_2} - A_3\frac{\partial\psi}{\partial q_1}\Big)dq_1 - \frac{1}{\rho r\Delta}\Big(A_2\frac{\partial\psi}{\partial q_1} - A_3\frac{\partial\psi}{\partial q_2}\Big)dq_2\right] = \iint\limits_{D_C} \omega\Delta\, dq_1\, dq_2,$$
$$\oint\limits_C \frac{\omega}{r}\Big(\frac{\partial\psi}{\partial q_1}dq_1 + \frac{\partial\psi}{\partial q_2}dq_2\Big) = -\oint\limits_C \frac{\rho}{2}\Big(\frac{\partial V^2}{\partial q_1}dq_1 + \frac{\partial V^2}{\partial q_2}dq_2\Big), \qquad \oint\limits_C H\Big(\frac{\partial\psi}{\partial q_1}dq_1 + \frac{\partial\psi}{\partial q_2}dq_2\Big) = 0. \qquad (1)$$

Here

$$A_1 = x_1^2 + r_1^2,\ A_2 = x_2^2 + r_2^2,\ A_3 = x_1x_2 + r_1r_2,\ \Delta = x_1r_2 - x_2r_1,\ x_i = \partial x/\partial q_i,\ r_i = \partial r/\partial q_i\ \ (i = 1, 2),$$

D_c is an arbitrary domain with a smooth boundary C of the given flow region Ω. The velocity vector components V_1, V_2, stream function ψ, vortex function ω, enthalpy H, pressure P, and density ρ satisfy the relations

$$V_1 = -\frac{1}{\rho r\Delta}V_1 = -\frac{1}{\rho r\Delta}\Big(\frac{\partial\psi}{\partial q_1}x_2 - \frac{\partial\psi}{\partial q_2}x_1\Big), \qquad V_2 = -\frac{1}{\rho r\Delta}\Big(\frac{\partial\psi}{\partial q_1}r_2 - \frac{\partial\psi}{\partial q_2}r_1\Big),$$
$$\omega = \Big(\frac{\partial V_1}{\partial q_1}x_2 - \frac{\partial V_1}{\partial q_2}x_1 + \frac{\partial V_2}{\partial q_1}r_2 - \frac{\partial V_2}{\partial q_2}r_1\Big)\frac{1}{\Delta},$$
$$P = P_0 - \int\limits_{L(M_0,M)} \Big(\frac{\rho}{2}\frac{\partial V^2}{\partial q_1} + \frac{\omega}{r}\frac{\partial\psi}{\partial q_1}\Big)dq_1 - \int\limits_{L(M_0,M)} \Big(\frac{\rho}{2}\frac{\partial V^2}{\partial q_2} + \frac{\omega}{r}\frac{\partial\psi}{\partial q_2}\Big)dq_2$$

and the boundary conditions

$$\psi = \psi_0 + \int\limits_{q_{10}}^{q_1} r(\xi, q_2)\rho(\xi, q_2)\,[V_1(\xi, q_2)r_1(\xi, q_2) - V_2(\xi, q_2)x_1(\xi, q_2)]\,d\xi$$
$$+ \int\limits_{q_{20}}^{q_1} r(q_1, \eta)\rho(q_1, \eta)[V_1(q_1, \eta)r_2(q_1, \eta) - V_2(q_1, \eta)x_2(q_1, \eta)]d\eta, \qquad \rho|_\Gamma = \rho_0, \quad \mathbf{Vn}|_\Gamma = V_0,$$

where x and r are the Cartesian coordinates; Γ is the gas permeability boundary; $L(M_0, M)$ is an arbitrary curve connecting the point $M \in \Omega$ with the point M_0 at which pressure is specified; q_{10}, q_{20} is the starting point of the boundary traversal in the stream function calculations.

While Khakimzyanov and Yaushevbsence [6] assumed the absence of closed vortex flows, the suggested method uses a special approximation of Eqs. (1) that takes into account the peculiar features of curvilinear grids and reflects exactly continuous flow, and a direct economical method of matrix inversion in each iteration for the solution of linear algebraic systems of equations with regularization and allowance for the block-tridiagonal matrix structure (the matrices are asymmetric because of curvilinear grids). All of this

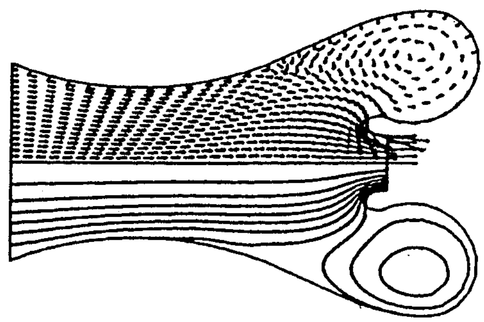

Fig. 3

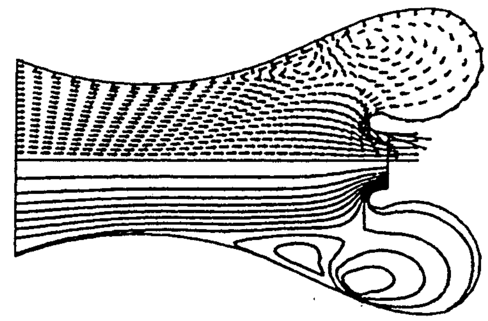

Fig. 4

allowed computation of steady gas flows with a large number of closed vortex flows at low Mach numbers [5]. The modified formulation of the boundary conditions required another organization of the iteration process compared with [6]. The pressure is calculated by the method of consistent approximation developed in [9].

To check the feasibility of the algorithm for subsonic flow computations [10] and for qualitative and quantitative comparison in a number of regions, subsonic flow was calculated by the proposed method, by the finite-element method for incompressible flow, and by another iteration finite-difference method within the framework of a viscous compressible flow model [11]. These three methods revealed closed vortex flows. A quantitative comparison showed that the calculated velocity modulus and pressure had maximum differences at the exit boundary from the subsonic region (the disagreement was less than 10%).

Flow calculations with specified mass inflow, enthalpy, and with specified density and gas injection rate showed tnat these initial data provided different flow structures in a number of regions. Although the density and gas injection rate in the first set of data were little different from the constant parameters specified in the second set of data, in the first case isoenergetic flow was, as a rule, calculated (Fig. 1), whereas in the second case, a nonisoenergetic flow (Fig. 2).

The second set of initial data provided a circulation zone occupying an appreciable area in the flow region upstream of the nozzle (the figures show the corresponding streamlines).

Comparisons of similar calculations support the statement in [1, p. 78] that the definition of boundary conditions for a system of equations describing the gas flow in combustion engines is a separate problem of high complexity and in every particular case these conditions should reflect the specific construction of the studied device. Therefore, in actual calculations one should carefully define the boundary conditions.

If the duct has a nozzle part, the specified entrance boundary conditions do not always ensure the needed gasdynamic regime in the subsonic region: either the Mach number in the throat is different from unity or flow choking occurs in the chamber. The initial data changed, and the phenomena vanished. Thus, an algorithm for correcting the boundary conditions was required. The approach of [4] was used for this purpose. Its essence lies in organization of subsonic–transonic iterations, i.e., in alternate flow calculations with the proper methods for the flow types individually in the subsonic ($x_0 \leqslant x \leqslant x_d$) [10] and transonic ($x_t \leqslant x \leqslant x_s$) regions with some overlapping of the regions ($x_t < x_d < x_s$).

After the subsonic flow calculation in the cross section $x = x_t$, which is located to the left of the right boundary $x = x_d$ of the subsonic region, the total enthalpy, entropy, and the slope of the streamlines are computed. They are specified as the boundary conditions [4] for calculating flows in the transonic region. Transonic flow is calculated so that the Mach number in the throat was about unity; after that the gas flow rate is computed in this cross section. Since the gas flow rate should be equal to the gas inflow, the obtained mass flow in the throat is used to correct the nondimensional density ρ_0 at the entrance. The entire procedure is repeated until the mass flow rate differs from the gas inflow by less than 1%. Normally, 2–4 iterations are needed. Passing to dimensional values, the specified gas density ρ_0 and velocity V_0 at the entrance boundary and the stagnation pressure P_0 are used to calculate a new value of stagnation pressure

$$P_{0k} = \rho_0 P_0 / \rho_k,$$

which matches the calculated flow with density ρ_k at the entrance.

Transonic flow is calculated by the time-dependent method using the MacCormack scheme following the code by A. D. Rychkov, which is included as a TRANS module into the SOKOL software system. We consider as an example the gas flow in the duct shown in Figs. 3 and 4 (the upper half presents the velocity fields and the lower one shows the streamlines).

The gas with constant density and velocity is supplied along the side surface of the duct, and its velocity at the left end face follows a cosine law distribution and, hence, on the symmetry plane it is 15 times as high as on the side surface. Gas flow calculations solely in the subsonic region provided a closed vortex flow which occupied the entire region upstream of the nozzle (Fig. 3); the Mach number at the entrance to the transonic region was about 0.35. After subsonic-transonic iterations in this model variant, the preset stagnation pressure decreased by nearly 45%, the velocity at the entrance to the transonic region increased up to 0.5, the vortex vacated a considerable area, and the flow pattern in the region upstream of the nozzle changed (Fig. 4). While in the first computation, the gas went up the rigid wall, in the second case, the vortex center shifted to the left and the gas flowed down the rigid wall.

Multiple computations show that allowance for transonic parameters affects the vortex formation in the subsonic region, the stagnation pressure either increasing or decreasing. Comparison of numerical and experimental results with subsonic-transonic iterations showed that the stagnation pressure recovers with an accuracy of up to 0.1%.

The SOKOL software system is developed for personal computers and has a varied graphical service for visualizing the calculated flow. SOKOL allows effective computations of steady potential and vortex flows of gas and fluid with a large amount of closed vortex flows in simply connected axisymmetric ducts of arbitrary configurations. The use of the SOKOL and SPECTR (computation of natural frequencies and functions of acoustic oscillations in combustion engines [12]) codes for investigating the interaction of acoustic waves with vortex flow, which is accompanied by pressure fluctuations, allowed computations in the gas tract in designing gasdynamic equipment with reliable predicted characteristics.

REFERENCES

1. B. A. Raizberg, B. T. Erokhin, and K. P. Samsonov, *Theory of Working Processes in Solid Fuel Reactive Systems* [in Russian], Mashinostroenie, Moscow (1972).
2. A. M. Lipanov, V. P. Bobryshev, A. V. Aliev, et al., *Numerical Experiment in Solid Fuel Rocket Engine Theory* [in Russian], Nauka, Ekaterinburg (1994).
3. V. F. Akhmadeev, G. N. Guseva, L. N. Kozlov, et al., *Hydrodynamic Sources of Acoustic Oscillations in Combustion Engines* [in Russian], TsNIINTIKPK, Moscow (1990).
4. A. D. Rychkov, *Mathematical Modeling of Gasdynamic Processes in Ducts and Nozzles* [in Russian], Nauka, Novosibirsk (1988).
5. O. B. Khairullina, "Calculation of steady subsonic vortex flows of a perfect gas in axisymmetric ducts of complex geometries," in: *Vopr. Atom. Nauki Tekh., Mat. Model. Fiz. Protses.*, No. 3 (1990), pp. 32–39.
6. G. S. Khakimzyanov and I. K. Yaushev, "Iteration scheme for computing two-dimensional subsonic steady internal flows of a perfect compressible fluid," Preprint No. 4-87, Inst. Theor. and Appl. Mech., Sib. Div., Russian Acad. of Sciences, Novosibirsk (1987).
7. O. B. Khairullina, "A method for computing optimal block grids in two-dimensional multiply connected regions," in: *Vopr. Atom. Nauki Tekh., Mat. Model. Fiz. Protses.*, No. 1 (1992), pp. 62–66.
8. O. B. Khairullina, "Construction of optimal block-regular grids," *Vopr. Atom. Nauki Tekh., Mat. Model. Fiz. Protses.*, No. 1, 19–25 (1994).

9. G. S. Khakimzyanov and I. K. Yaushev, "On pressure calculation in two-dimensional steady hydrodynamic problems," in: *Problems of Dynamics of a Viscous Fluid*, Inst. Theor. and Appl. Mech., Novosibirsk (1985), pp. 280–284.
10. O. B. Khairullina, "RDT IMM software system for steady subsonic flows in complex-shaped ducts," in: *Computing Technologies*, **1**, No. 2: Proc. of School–Seminar on Mathematical Physics software, Rostov-on-Don, 1990, Novosibirsk, (1992), pp. 327–333.
11. V. F. Akhmadeev, A. F. Sidorov, F. F. Spiridonov, and O. B. Khairullina, "On three methods of numerical simulation of subsonic flows in axisymmetric complex-shaped ducts," *Model. Mekh.*, **4**(21), No. 5, 15–25 (1990).
12. O. V. Kokovikhina, "On the propagation of acoustic vibrations in vortex flows," *Model. Mekh.*, **7**(24), No. 1, 89–97 (1993).