Figure 1 compares the predictions of the present theory and that of the k–ϵ model with the Wilcox compressibility correction[5] with case 2 of Ref. 7. For this case,

$$U_2/U_1 = 0.25, \quad \rho_2/\rho_1 = 0.58, \quad T_{0_1} = T_{0_2} = 276\ \mathrm{K}$$
$$M_1 = 1.96, \quad M_2 = 0.37, \quad M_c = 0.64 \tag{7}$$

where M_c is the convective Mach number. It is seen that the present model outperforms the k–ϵ model. Because the present model makes use of Morkovin's hypothesis, it is expected that its predictions will deviate from measurements at high convective Mach numbers and high stagnation temperature ratios.

To illustrate the ability of the model to compute wall bounded flows, the work of Kussoy and Horstman[8] for a cold-wall boundary at a freestream Mach number $M_\infty = 8.18$ is selected. Calculations were obtained using a modification of the boundary-layer code by Harris and Blanchard.[9] Solutions provided by this code are second-order accurate. The freestream boundary conditions imposed on the model are $k_\infty/U_\infty^2 = \zeta_\infty/(U_\infty/\ell)^2 \sim 10^{-7}$, where ℓ is a characteristic length scale and was set equal to 1. The wall boundary conditions are $k_w = 0$, and ζ_w was set such that $k \sim k_0 y^2$ with molecular diffusion balancing dissipation as required in the k equation. A grid of 175 points in the normal direction with geometric stretching of 7% was used in all boundary-layer calculations. A grid study was conducted, and it was determined that the results were grid independent. Comparisons with experiment and the k–ω model are shown in Fig. 2, where subscript e refers to edge conditions, u_τ is the friction velocity, and ν_w is the kinematic viscosity evaluated at the wall temperature. It is seen that the k–ζ model agrees well with experiment. Differences in measured skin-friction and heat transfer coefficients and predictions of the k–ζ model are 6.9 and 7.1%. Similar results for the k–ω model are 8.1 and 19.2%.

In conclusion, the present model outperforms both the k–ϵ and the k–ω models. Thus, we have at hand a model that is capable of calculating low-speed and high-speed flows with one set of model constants.

Acknowledgments

This work is supported in part by NASA Grant NAG-1-244. The authors would like to acknowledge many helpful discussions with Craig Dutton of the University of Illinois at Urbana–Champaign and Mo Samimy of Ohio State University. Part of the computations were carried out at the North Carolina Supercomputing Center.

References

[1]Robinson, D. F., Harris, J. E., and Hassan, H. A., "Unified Turbulence Closure Model for Axisymmetric and Planar Free Shear Flows," *AIAA Journal*, Vol. 33, No. 12, 1995, pp. 2325–2331.

[2]Speziale, C. G., Abid, R., and Mansour, N. N., "Evaluation of Reynolds Sress Turbulence Closures in Compressible Homogeneous Shear Flow," Inst. for Computer Applications in Science and Engineering, ICASE Rept. 94-17, Hampton, VA, March 1994.

[3]Morkovin, M., "Effects of Compressibility on Turbulent Flows," *Mecanique de la Turbulence*, edited by A. Favre, Gordon and Breach, 1964, pp. 367–380.

[4]Alexopoulos, G. A., and Hassan, H. A., "A k–ζ (Enstrophy) Compressible Turbulence Model for Mixing Layers and Wall Bounded Flows," AIAA Paper 96-2039, June 1996.

[5]Wilcox, D. C., *Turbulence Modeling for CFD*, DCW Industries, Inc., La Cañada, CA, 1993, pp. 163–165.

[6]Settles, G. S., and Dodson, L. J., "Hypersonic Turbulent Boundary-Layer and Free Shear Database," NASA CR 177610, April 1993.

[7]Samimy, M., and Elliott, G. S., "Effects of Compressibility on Characteristics of Free Shear Layers," *AIAA Journal*, Vol. 28, No. 3, 1990, pp. 439–445.

[8]Kussoy, M. I., and Horstman, K. C., "Documentation of Two- and Three-Dimensional Shock-Wave/Turbulent-Boundary-Layer Interaction Flows at Mach 8.2," NASA TM 103838, May 1991.

[9]Harris, J. E., and Blanchard, D. K., "Computer Program for Solving Laminar, Transitional, or Turbulent Compressible Boundary-Layer Equations for Two-Dimensional and Axisymmetric Flow," NASA TM 83207, Feb. 1982.

F. W. Chambers
Associate Editor

Motion of a Body Through Large-Scale Inhomogeneity in the Stratified Atmosphere

Vilen U. Nabiev* and Sergey V. Utyuzhnikov†
Moscow Institute of Physics and Technology,
Dolgoprudny Moscow Region 141700, Russia

Introduction

BODY motion in the atmosphere is essentially a coupled problem: the body motion depends on aerodynamic coefficients, which in turn are found by solving the problem of the flowfield around the body. The input data for the aerodynamic problem are parameters of the incident flow, body geometry, and spatial orientation of the body.

To solve this problem with a high degree of accuracy, one should integrate simultaneously both the equations of motion of the body and the equations describing the flowfield around the body. This approach requires considerable computer resources but can be improved in the case of free supersonic motion of a slender body of revolution. Such a motion is usually accompanied by small angles of attack.

We consider supersonic flight of a blunt slender body of revolution through a large-scale cloud of heated gas (thermal) floating in the stratified atmosphere. An efficient numerical method to determine the body motion is proposed. The dependence of the solution of the flow problem on the angle of attack is found analytically by using this method. The proposed method shows how the thermal environment alters the trajectory, spatial orientation, and flight stability of the body. How the location of the center of pressure depends on the Mach and Reynolds numbers of the incident flow has also been studied.

Statement of the Problem

We will assume that, at an instant of time in the stratified atmosphere, an axisymmetric cloud of heated gas is formed having the following parameters:

$$T(h,d) = T_a(h) + [T_{\max} - T_a(h)]\exp\left[-\left(R_T^{-1}R\right)^2\right]$$

where $R_T = 1.2$ km, $T_{\max} = 1600$ K, $T_a(h)$ is the temperature of the unperturbed atmosphere at the altitude h, d is the distance from the thermal's axis of symmetry, $R = [(h - H)^2 + d^2]^{1/2}$, and $H = 20$ km. The position of the temperature maximum $T_{\max}$ corresponds to the altitude H.

The cloud floats up and forms a vortex ring. The cloud begins to float up at the altitude of 22 km, and 15 s later the body enters it horizontally with a velocity $V_0 = 2000$ m/s (the vertical component of the velocity is equal to zero). The body has the form of a cone with a blunt spherical nose of radius $R_0 = 0.1$ m, semivertex angle of 15 deg, length $L = 2$ m, and mass of 1000 kg; the center of mass of the solid is situated a distance L_c from the spherical nose. Initially, while the thermal does not influence the body motion, the trajectory plane is 500 m from the axis of symmetry of the thermal. The relative position of the body and the thermal at the initial instant of time is shown in Fig. 1.

At the instant the body enters, the gas is twisted into a toroidal vortex ring, which floats up into the atmosphere. In view of this, the freestream flow has a varying space–time structure. We will assume that the body does not influence the gas motion in the thermal. The flow past the body is assumed to be laminar.

Received March 8, 1996; revision received March 25, 1997; accepted for publication March 26, 1997.

*Postgraduate Student, Department of Computational Mathematics, Institutskiy per. 9. E-mail: svu@svu.crec.mipt.ru.

†Associate Professor, Department of Computational Mathematics, Institutskiy per. 9. E-mail: svu@svu.crec.mipt.ru.

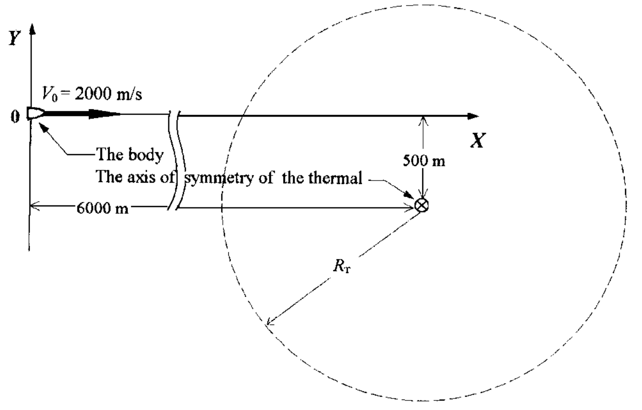

Fig. 1 Relative position of the body and the thermal at the initial instant of time (top view); the X axis is situated along the direction of velocity vector V_0 of the body at the initial instant of time, and velocity vector V_0 is parallel to the horizontal plane (X, Y).

Two cases have been considered: the centers of mass at 0.5 m and at 1.4 m from the body's spherical nose, which correspond to high and low stability of the body's flight, respectively.

It is assumed that the angular velocity of body rotation around its own symmetry axis is sufficiently small, so that one can neglect any viscous torques parallel to the axis of symmetry of the body. Such an assumption is equivalent to assuming the condition that the angular velocity of the body is small initially.

Method of Solution

A numerical method of calculating the convective-diffusion motion of air in the thermal domain has been described in Ref. 1.

The flowfield around the body is described by the system of equations of the full viscous shock layer (FVSL) (for example, see Ref. 2). The asymptotic small-parameter method is used to solve the equations of spatial FVSL, where the small parameter is the angle of attack α (Ref. 2). The method consists of expanding the required spatial solution in an asymptotic series in α. Only the first term of expansion with respect to α remains. If one takes into account higher terms of the expansion, the correction is 2–3% (Ref. 3). The coefficients of expansion in the angle of attack are determined by an efficient iterative numerical method.[2] The aerodynamic coefficients are obtained in the following form: the drag coefficient $C_d = C_d^{(0)}$, the lifting force coefficient $C_l = \alpha C_l^{(1)}$, and the pitching moment coefficient $C_m = \alpha C_m^{(1)}$. Here $C_d^{(0)}$, $C_l^{(1)}$, and $C_m^{(1)}$ do not depend on α, i.e., linear aerodynamics is assumed.

The whole system of the governing equations can be solved as follows.

Initially, the system of FVSL is solved for the specified parameters of the incident flow. Then, during a time Δt_g, the system of ballistics equations with six degrees of freedom (it governs three-dimensional motion of the body's center of mass and three-dimensional rotational motion of a solid around its center of mass) is integrated with fixed values of $C_d^{(0)}$, $C_l^{(1)}$, and $C_m^{(1)}$, which depend only on the parameters of the incident flow, but with variable values of the angle of attack. The time step Δt_g takes values of 0.2–3 s, depending on the rate of changes in the parameters of the incident flow during the motion of the body. The system of the ballistics equations is integrated with step $\Delta t_b \ll \Delta t_g$. This is because the characteristic time of variation of the body's motion parameters (about 0.1 s for the angle of attack) is much less than the characteristic time of variation of the parameters of the incident flow. After the time Δt_g, the sequence of the solution of the whole system is repeated with, possibly, other values of Δt_g and Δt_b. The system of ballistics equations is integrated on the base of the Runge–Kutta method of third-order accuracy with respect to Δt_b. In the calculations, the characteristic ratio $\Delta t_b / \Delta t_g$ was 10^{-2}–10^{-3}. The characteristic time to calculate the interval of physical time Δt_g amounted to 35–40 min on an IBM 386 computer.

Results

For the considered body's mass center locations, the pressure center in the unperturbed atmosphere turns out to be behind the center of mass, and a body flight is statically stable. If the body moves toward the thermal center, the density of the incident flow diminishes rapidly, while the temperature grows, which leads to a reduction in the Mach number M_∞ and the Reynolds number Re_∞. In Table 1 we give data on the gas temperature T_∞, the gas density ρ_∞, the altitude of the flight h, the gas flow velocity in the thermal W, and the angle ϑ of inclination of the vector $\hat{\boldsymbol{W}}$ to the axis of symmetry of the thermal at several points of the trajectory (the angle ϑ is measured from the pointing upward thermal's axis of symmetry). To calculate the Reynolds number, we take the radius of the nose R_0 as a characteristic scale.

A reduction in the mach number M_∞ at fixed Reynolds number Re_∞ causes the center of pressure to shift toward the nose of the cone (Fig. 2). At the same time, reduction in Reynolds number Re_∞ for fixed Mach number M_∞ shifts the center of pressure in the opposite direction (see Fig. 2). Nevertheless, the overall effect of changes in Mach number M_∞ and Reynolds number Re_∞ along the trajectory is to displace the center of pressure toward the nose of the cone, which reduces its flight stability.

While in motion, the body oscillates about the position corresponding to zero angle of attack. On passing through an unperturbed atmosphere, the amplitude of the oscillations of the angle of attack is small (about 0.03 deg for $L_c = 0.5$ m). When $L_c = 0.5$ m, the body has a considerable reserve of stability, and the oscillations of the angle of attack occur with relatively small amplitude. When $L_c = 1.4$ m, the angle of attack varies with period and amplitude, which are 5–10 times greater than for $L_c = 0.5$ m. In the neighborhood of the center of the thermal, where the velocity of the gas motion is a maximum, the angle of attack is maximum (about 0.3 deg for $L_c = 0.5$ m and about 6 deg for $L_c = 1.4$ m). In this region, the center of pressure turns out to be ahead of the center of mass for $L_c = 1.4$ m (Fig. 3), and the body flight becomes unstable.

Table 1 Values of the flow parameters[a] in the thermal and the altitude of the body at several points along the flight trajectory

Flight time, s	Flight altitude, m $L_c = 0.5$ m	Flight altitude, m $L_c = 1.4$ m	T_∞, K	$\rho_\infty \times 10^2$, kg/m³	W, m/s	υ, deg
1	21,996	21,996	221	5.9	18	117
2	21,982	21,983	391	3.6	59	24
3	21,961	21,966	637	2.4	224	1
4	21,933	21,948	448	3.2	92	10
5	21,896	21,926	231	6.3	23	136
6	21,852	21,896	217	6.6	6	154

[a]Difference between flow parameters in the thermal for $L_c = 0.5$ and 1.4 m is insignificant.

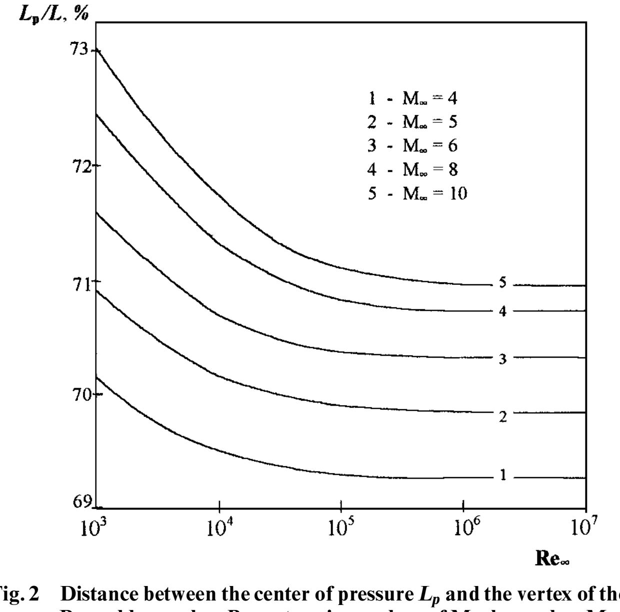

Fig. 2 Distance between the center of pressure L_p and the vertex of the cone vs Reynolds number Re_∞ at various values of Mach number M_∞. The flow past the body is assumed to be laminar.

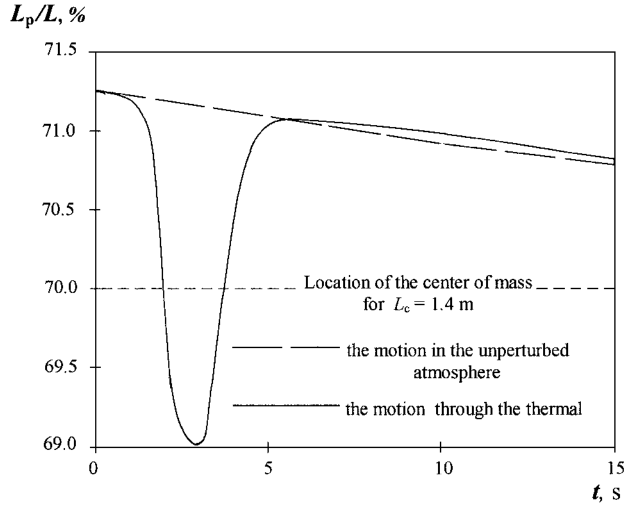

Fig. 3 Distance between the center of pressure L_p and the vertex of the cone vs flight time *t*.

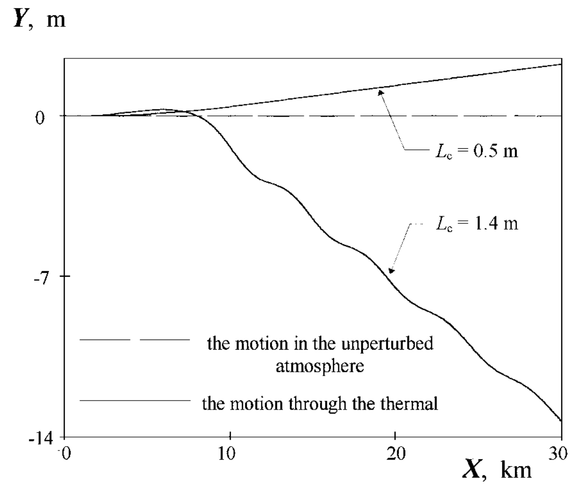

Fig. 4 Projections of body trajectories onto the horizontal plane (lateral drift *Y* vs *X* coordinate); *X* and *Y* axes are the same as in Fig. 1.

When $L_c = 0.5$ m, the presence of the thermal along the path of the body has practically no effect on the trajectory; when $L_c = 1.4$ m, the change of the trajectory is much more significant than in the first case due to less stability of the flight (or its instability in the thermal domain).

The great difference between flight stability in cases $L_c = 0.5$ and 1.4 m causes an interesting effect obtained from the computations. Projections of the body trajectory onto the horizontal plane are shown in Fig. 4. At each point along the flight trajectory into the thermal domain, the velocity $\hat{W}$ has a positive Y component. On entering the thermal domain, the body's center of mass begins to drift to the left under the influence of an incident flow, and the body's axis of symmetry begins to turn in the direction of the incident flow (clockwise in Fig. 4). When $L_c = 0.5$ m, the body lines up rapidly with an incident flow and continues to shift to the left. When $L_c = 1.4$ m, it turns out that clockwise rotating body gets into the region where it loses stability (see Fig. 3). In this situation, the clockwise rotation of the body is enhanced even more. As a result, on leaving the thermal the body goes to the right, i.e., in the direction in which it has been turned. The trajectory oscillations in Fig. 4 are due to the yaw angle oscillations.

References

[1] Muzafarov, I. F., Tirskiy, G. A., Utyuzhnikov, S. V., and Yamaleev, N. K., "Numerical Simulation of the Flow over a Body Flying Through a Thermal in a Stratified Atmosphere," *Computers and Fluids*, Vol. 23, No. 2, 1994, pp. 295–304.

[2] Tirskiy, G. A., Utyuzhnikov, S. V., and Yamaleev, N. K., "Efficient Numerical Method for Simulation of Supersonic Viscous Flow Past a Blunted Body at Small Angle of Attack," *Computers and Fluids*, Vol. 23, No. 1, 1994, pp. 103–114.

[3] Karyakin, V. E., "Computation of the Supersonic Viscous Flow over the Body of Rotation at Small Angle of Attack," *Jurnal Vychislitel'noi Matematiki i Matematicheskoi Fiziki*, Vol. 21, No. 1, 1981, pp. 150–156 (in Russian).

W. Oberkampf
Associate Editor

New Conservative Formulations of Full-Potential Equation in Streamline-Aligned Coordinates

Azat M. Latypov*
University of Windsor,
Windsor, Ontario N9B 3P4, Canada

I. Introduction

DURING the past 20 years, a number of formulations, based on the streamfunction as a coordinate (SFC) concept, i.e., writing the governing equations in a system of independent coordinates aligned with streamlines, have been used in computational fluid dynamics (CFD).[1–6] One attractive advantage is that SFC formulations permit computation of the parameters of the flow without prior grid generation in the computational domain. The governing equations play a double role of both the equations describing the motion of the media and the grid generation equations. As a result of this, both computational time and memory requirements can be reduced. Furthermore, the resulting streamline-aligned computational grid naturally conforms to the boundaries of the physical domain. These features are utilized in most of the works dealing with the SFC method in CFD especially for the solution of inverse or optimal design problems in aerodynamics.[1,3]

An essential feature of the SFC technique is the choice of the independent coordinate, which compliments the streamfunction to produce the nondegenerate system of independent coordinates. It is well known that one of the major limitations of the original von Mises approach[7] is that the resulting system of coordinates degenerates at locations where the velocity vector is normal to the axis of the transverse Cartesian coordinate. This situation is typical for the flow in the vicinity of the leading edge of an airfoil. This limits applicability of the von Mises approach to cases where one can expect in advance that the flow direction will not change significantly over the flow domain.

In the current work, a streamline-based transformation that generates an orthogonal streamwise coordinate system is used. Unlike similar formulations used in Refs. 4 and 5, the formulations developed and used in this work have an advantage in that they have a conservative form, ensuring both global and local conservation of mass and irrotationality of the velocity vector field.

II. Reformulation of Governing Equations

The assumptions that the velocity vector field is potential (irrotational) and that the flow is isentropic and isenthalpic are well established in modeling of compressible flows. Under these assumptions, the equations of momentum and energy follow from the continuity and irrotationality equations,[8] and the governing system may be

Received Jan. 4, 1996; accepted for publication March 21, 1997.

*Research Assistant, Ph.D. Candidate, Fluid Dynamics Research Institute and Department of Mathematics and Statistics, 401 Sunset Avenue. E-mail: latypov@na-net.ornl.gov.