JOURNAL OF AIRCRAFT
Vol. 29, No. 5, Sept.–Oct. 1992

Direct Simulation of Low-Density Flow over Airfoils

Tsze C. Tai*
Naval Surface Warfare Center, Bethesda, Maryland 20084

The low-density aerodynamics about airfoils in the transition region is considered using a direct simulation Monte Carlo method. Numerical results are presented for two airfoils (the NACA-0009 and a 9% thick, circular-arc airfoil) traveling at Mach 4 and 5, at an angle of attack of 1.25 deg and altitudes 56 and 62 miles above sea level. The flow, having a Knudsen number range between 0.47–1.15 and a Reynolds number range between 15–114, departs considerably from the continuum theory. Results indicate that both lift and drag are very much penalized in the transition region. In contrast with previous work on slightly rarefied flow with smaller Knudsen numbers (slip flow regime), the effect of rarefaction becomes much more dominant than that due to viscosity.

Introduction

CURRENT interest in high-altitude flight has prompted new exploration of low-density aerodynamics. Unfortunately, the theoretical models and computational capabilities developed to date have proven insufficient to fully characterize these flows. The difficulty has to do with the high degree of rarefaction of the flow. The low-density flows of interest occur in a region between the continuum flow region and the molecular flow region. In this "transition region," the concept of transport coefficients (which are the basis for the Navier-Stokes equations) becomes invalid. Thus, the traditional Navier-Stokes analysis ceases to give accurate results. The objective of this research is to investigate the flow physics of high-altitude flight and to develop a new computational scheme for assessing vehicle performance in this important flow region.

In order to achieve sufficient lift for level flight, a high-altitude vehicle must fly at high speed. A typical flight envelope would include freestream Mach numbers between 3 and 15, at an altitude of 30–60 miles above sea level. This flight regime, although considerably beyond the capability of existing aircraft, lies within the flight envelope of the National Aerospace Plane (X-30). The aerodynamic challenge is to achieve sufficiently high-lift coefficients, at the prescribed altitudes but at a relatively low Mach number, so that certain hypersonic flow problems (such as heat transfer, chemical reactions, and surface catalytic activities) can be circumvented.

In contrast with continuum flow, transitional flow requires a higher-order approximation to the Boltzmann equation. Numerical solution to the Boltzmann equation consists of 1) evaluation of the collision integral; and 2) integration of the differential equation. Although the integral form of the collision term causes much of the mathematical difficulty in solving the Boltzmann equation, the use of the velocity space coordinates as independent variables in partial differential equations requires large amounts of computer time and storage. The advert of high-speed computers has spurred the development of numerical solutions to the Boltzmann equation for several basic, one-dimensional cases.[1]

An alternate approach to solving the Boltzmann equation is the direct simulation of the physics of the gas flow.[2] If the simulation of the collision phenomenon is consistent with the Boltzmann formulation, it can also yield the Boltzmann solution. First applied by Bird to the problem of translational flow,[3] the intermolecular collisions in the simulation technique are treated on a probabilistic basis with the aid of the Monte Carlo method. The technique, referred to as the direct simulation Monte Carlo (DSMC) method, requires the assumption of molecular chaos and is restricted to dilute gas flows in which the mean spacing between the molecules is large in comparison to the molecular diameter. Because the DSMC method avoids the use of velocity space coordinates, more complex problems can be treated.[4–6]

In the present work, a hypothetical, infinite-span wing traveling at Mach numbers between 4 and 5 at altitudes 56 and 62 miles above sea level and at an angle of attack of 1.25 deg is considered. The flow has a Knudsen number range of 0.47–1.15, which falls into the regular transitional flow region. These conditions correspond to a Reynolds number range between 15 and 114. The infinite span allows the problem to be treated two-dimensionally. Two airfoils, namely an NACA-0009 airfoil and a 9%-thick, circular-arc airfoil, are considered as test cases. These conditions and airfoils are selected to maintain some continuation of our previous work with the extended Navier-Stokes approach.[7] The extended Navier-Stokes method, however, failed to converge for these flow conditions. The flow regions in both approaches are shown in Fig. 1.

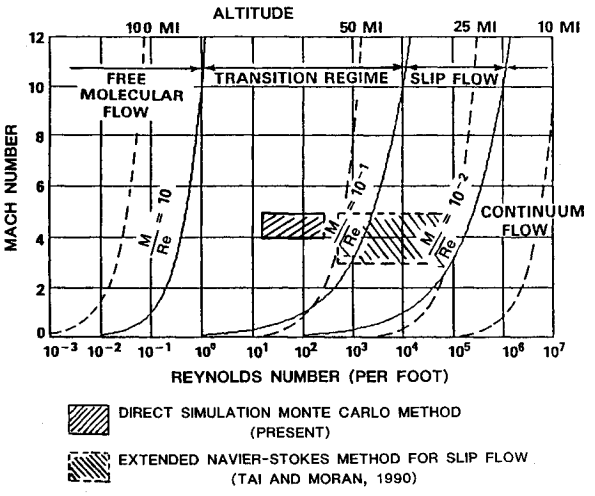

Fig. 1 Regimes of gas dynamics.

Presented as Paper 90-1539 at the AIAA 21st Fluid Dynamics, Plasmadynamics and Lasers Conference, Seattle, WA, June 18–20, 1990; received Oct. 9, 1990; revision received June 10, 1991; accepted for publication Aug. 14, 1991.

*Senior Technical Advisor, Systems Department, Carderock Division. Associate Fellow AIAA.

DSMC Method

The DSMC method is a technique for the computer modeling of a real gas by some thousands of simulated molecules. The method contrasts with the general philosophy of computational fluid dynamics which is to obtain solutions of the mathematical equations that model the processes. The computational task associated with the direct physical simulation becomes feasible when the gas density is sufficiently low. The low-density flow effects for a moderate Knudsen number ($Kn = M/\sqrt{Re} > 0.2$) include the development of an anisotropic pressure tensor, and thermal and pressure diffusion. These effects are not generally included in the Navier-Stokes formulations. Once the density becomes sufficiently low for the DSMC solution to be computationally feasible, it is a more valid method than the Navier-Stokes formulation or its extension.[7]

In the present work, the DSMC method originally developed by Bird[2,3] is applied. Discussion of the method is therefore confined to the Bird method known as the "G2" program.[8] In the Bird DSMC method, the molecules are selected for collisions on a probabilistic basis. The velocity components and position coordinates of these molecules are stored in the computer and modified with time as the molecules are concurrently followed through representative collisions and boundary interactions in simulated physical space. The G2 program system uses a new technique to explicitly calculate the appropriate number of possible pair samplings at each time step. Generally, the time step should be less than the mean collision time, and a typical cell dimension should be less than the local mean free path. The cell dimension should also be small compared to the distance over which there is a significant change in the flow properties. This latter condition will dictate the cell size in high Knudsen number flows and, in practice, the cell size in low Knudsen number flows is set to about one-third or one-half the mean free path. The time step is then set such that a typical molecule moves about one-third of the cell dimension at each time step.

The DSMC method uses the cell system only for the sampling of the macroscopic properties and for the selection of possible collision partners. The sampled density is used in the procedures for establishing the collision rate. It is desirable to have the number of molecules as large as possible, generally around 10–20. On the other hand, in selecting possible collision partners, it is desirable that this number be as small as possible to reduce the mean spacing of collision pairs and thereby minimize the smearing of gradients. This is important in regions of high vorticity. These conflicting requirements are reconciled by dividing the sampling cell structure into a set of subcells for the selection of collision pairs. The subcells may be chosen to contain an average of two or three molecules, so that all collisions approach the "nearest-neighbor" collisions. Should there be only one molecule in a subcell, the potential collision partner is chosen from a neighboring subcell within the same cell. Because the G2 program is for arbitrary geometry, there is a computing time penalty associated with small subcells. Tests have shown that the use of subcells is equally effective as a similar reduction in cell size; the latter can remain at about one-third the mean free path in moderate Knudsen number flows.

The statistical scatter of the simulation generally decreases as the square root of the sample size and, to attain a sufficiently small standard deviation, the G2 program employs either time averaging for steady flows or ensemble averaging for unsteady flows. Weighting factors can be specified in the code such that each simulated molecule near the axis represents fewer real molecules than the simulated molecules further away from the axis. This means that there may be equal samples in cells of the same size irrespective of their radius; however, it also means that a proportion of the molecules moving away from the axis must be removed from the flow, while some of those moving toward the axis must be duplicated. This duplication can cause undesirable statistical effects if identical molecules are in the same cell. In steady flow applications of the G2 programs, this can be avoided by putting a time delay on the duplication process.

The variable hard sphere (VHS) molecular model is used in the G2 program. The model is essentially a hard sphere with a diameter that varies with some inverse power of the relative velocity between the molecules in the collision. The viscosity coefficient is assumed to be proportional to a fixed power of the temperature. Because of a mathematical limitation of the model for the internal degrees of freedom, a single value must be chosen to cover all species in a gas mixture. The molecular diameter at a specified reference temperature is also set in the code. The VHS model appears to be adequate at normal temperatures.

For the vibrational modes, the Larsen-Borgnakke model[9] has been modified to cope with partially excited modes with the degree of excitation being given by the harmonic oscillator model. The essential feature of the model is that, while most of the collisions are regarded as elastic, a fraction of them are inelastic and, for these, new values of the translational and internal energies are chosen from the equilibrium distribution of these quantities. The Larsen-Borgnakke model is applied in its original form to the rotational degrees of freedom in the G2 program. At present, the collision number for rotational relaxation is set constant. It should be noted that the temperature of the equilibrium distribution is based on the translational energy in that particular collision, so that the method should not be regarded as an "equilibrium method."

The classical diffuse reflection model with complete accommodation of the gas to the surface temperature is appropriate for "engineering surfaces" that have not been exposed for a long period to ultrahigh vacuum. In the G2 program, these effects are simulated through the specification of some fraction of classical specular reflection. The code also includes the effects of chemical reaction, thermal radiation, and ionization. For the Mach range considered in the present work, these features are irrelevant and are therefore not presented.

Implementation of DSMC Method

The DSMC method, in the form of the G2 program system developed by Bird,[8] is applied in the present work. A special cell arrangement for airfoil flow computations is generated. The overall flowfield is divided into four regions with a total of 1512 cells (see Fig. 2). Both the upper and lower surfaces have cell segments of 30 by 18, followed by a wake region with a mesh of 12 by 18. Each cell is further divided into four

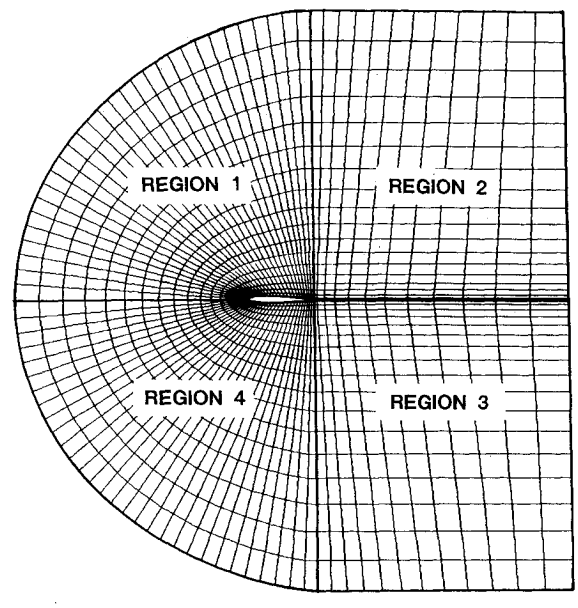

Fig. 2 Cell arrangement for moderately low-density flow over an airfoil using DSMC method.

subcells: two in the longitudinal direction and two in the normal direction. Converted from a "C" mesh used in the extended Navier-Stokes method[7] (but with fewer mesh points), this arrangement allows the cell size to be less than the mean free path of the flow in the inner region to capture the viscous effect. As noted in the preceding sections, the cells are used for sampling the flow information and establishing the collision rate; the subcells are used for the selection of collision pairs. Taking into account the subcells, there are 60 "panels" along both upper and lower surfaces of the airfoil. This number is nearly the same as that for typical "course grid" computations in the continuum flow.

A maximum of 20,000 simulated molecules are imposed for the initial stream. Each cell, therefore, has an average of 10–15 simulated molecules; each subcell has 3–4 simulated molecules. For flow at an altitude of 56 miles, each simulated molecule represents 2.5×10^{17} real molecules in each region (although the code allows different regional representation of the computational molecules). The number of real molecules representation decreased to 4.1×10^{16} for flow at $h = 62$ miles while maintaining the same population of the simulation. This total number of simulated molecules is chosen so that the requirements of the cell and subcell sizes, as discussed in the preceding section, are satisfied. Standard atmosphere air is used, with oxygen and nitrogen being the only gas components. No chemical reaction is considered for the subject low Mach number calculations.

Unlike the usual computational fluid dynamics, there are two unique areas in implementing the DSMC method. First, Monte Carlo programs need a random number generator. Most computers and workstations are equipped with good pseudorandom number generators that should be satisfactory. Virtually all of the pseudorandom number generators reset to the same starting point each time the program is run or started. The longer the sequence, the better for the method. Second, Monte Carlo programs are generally unsuitable for vector processing. Running the program in a supercomputer offers no distinct advantages. Time savings in using a supercomputer may not be worth the "CPU" charges. On the other hand, a powerful workstation with a 64-bit word length and several mega-bite memory should be the most cost-effective machine to run the program. A "Personal Iris" workstation is used for most of the computations. A typical run with approximately 15,000 samples takes about 60–70 hours.

Results and Discussion

Results are computed in terms of aerodynamic lift and drag coefficients for two airfoils (namely, a NACA-0009 airfoil and a 9% thick, circular-arc airfoil) at altitudes of 56 and 62 miles with Reynolds numbers (Re) of 114 and 15, and Knudsen numbers (Kn) of 0.47 and 1.15, respectively. The air density is still fairly high for these conditions from the molecular flow standpoint. The flowfield is divided into four regions which are computed at the same time. Aerodynamic force coefficients are determined by summing up the incident and reflected forces of molecular interactions on the airfoil surface and divided by the freestream dynamic pressure and unit chord length.

Figure 3 shows the lift coefficient vs Reynolds number for the NACA-0009 airfoil at $M_\infty = 4$ and 5, and $\alpha = 1.25$ deg. The lift coefficient increases slightly as the Reynolds number decreases or the Knudsen number increases. The local mean Knudsen numbers are indicated at various data points. The trend is similar to that given by the extended Navier-Stokes (ENS) method, that fails to yield any results for flow having a Reynolds number less than 300.

If the results of the Navier-Stokes code and the ENS method are extrapolated to a low Reynolds number range, a considerable gap exists between the extrapolated values and the present results. The gap due to the ENS method is smaller than that by the regular Navier-Stokes method, however. This implies that the extension of the Navier-Stokes method to the slip flow is qualitatively correct, but that the range of its applicability is fairly limited.

While it is reasonable to assert that there is considerable lift penalty in the transition region, the gap between the DSMC and ENS methods might be, in part, attributed to the different mesh sizes used in the two distinct approaches. As noted earlier, the cell arrangement for the present DSMC calculations is basically converted from the "C" mesh used in the ENS technique, but with fewer mesh points. This is especially true in the vicinity of the airfoil surface, in both longitudinal and normal directions. The use of subcells has improved this

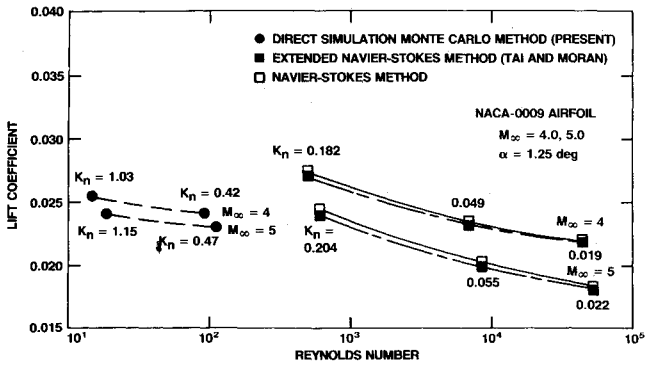

Fig. 3 Lift coefficient vs Reynolds number for NACA-0009 airfoil at $h = 56$ and 62 miles.

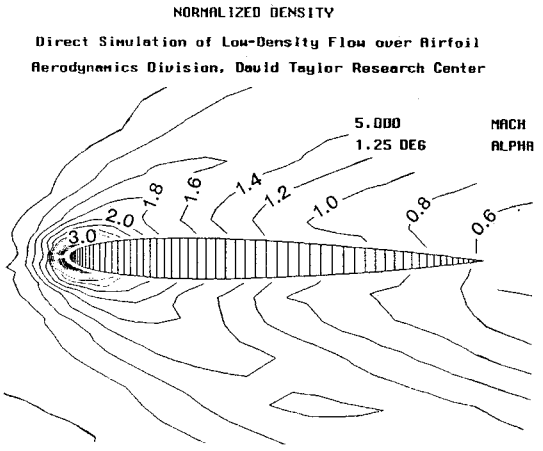

Fig. 4 Density contour over NACA-0009 airfoil at $M_\infty = 5$, $\alpha = 1.25$ deg, and $h = 56$ miles.

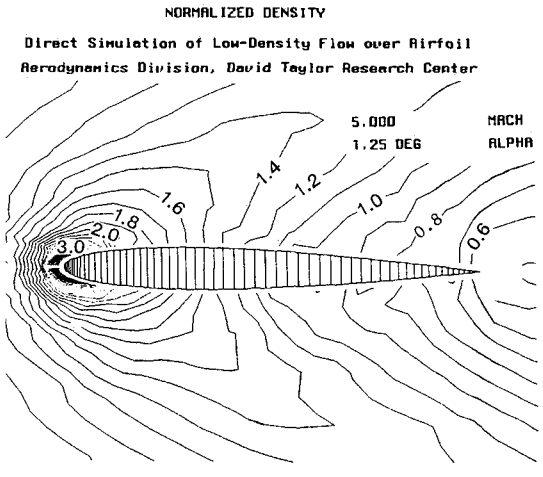

Fig. 5 Density contour over NACA-0009 airfoil at $M_\infty = 5$, $\alpha = 1.25$ deg, and $h = 62$ miles.

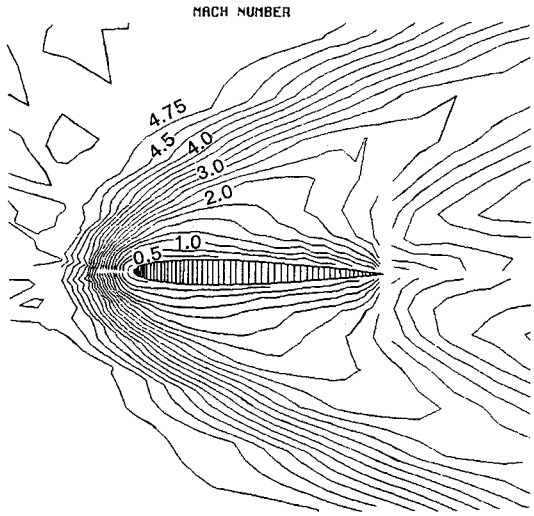

Fig. 6 Mach contour over NACA airfoil at $M_\infty = 5$, $\alpha = 1.25$ deg, and $h = 56$ miles.

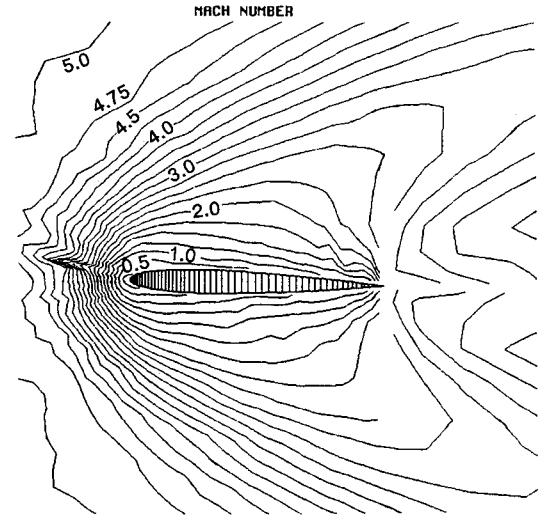

Fig. 7 Mach contour over NACA airfoil at $M_\infty = 5$, $\alpha = 1.25$ deg, and $h = 62$ miles.

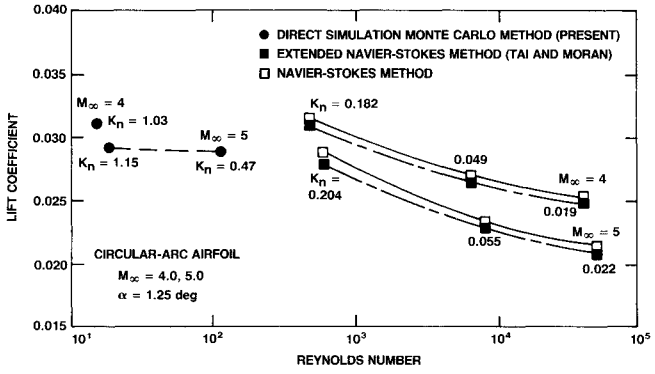

Fig. 8 Lift coefficient vs Reynolds number for 9% thick, circular-arc airfoil at $h = 56$ and 62 miles.

aspect to some extent, but it offers less clustering as usually employed in the CFD approach.

The density contours over a NACA-0009 airfoil at $M_\infty = 5$, $\alpha = 1.25$ deg, and altitude, $h = 56$ and 62 miles are shown in Figs. 4 and 5; the corresponding Mach contours are given in Figs. 6 and 7, respectively. The density contour levels represent the values of local density normalized by its freestream value. The shock wave is rather smeared where the velocity decreases gradually, and the shock "standoff distance" can

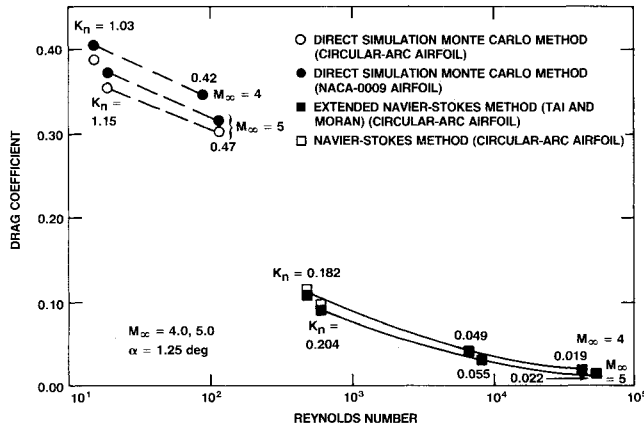

Fig. 9 Drag coefficient vs Reynolds number for NACA-0009 and 9% thick, circular-arc airfoil at $h = 56$ and 62 miles.

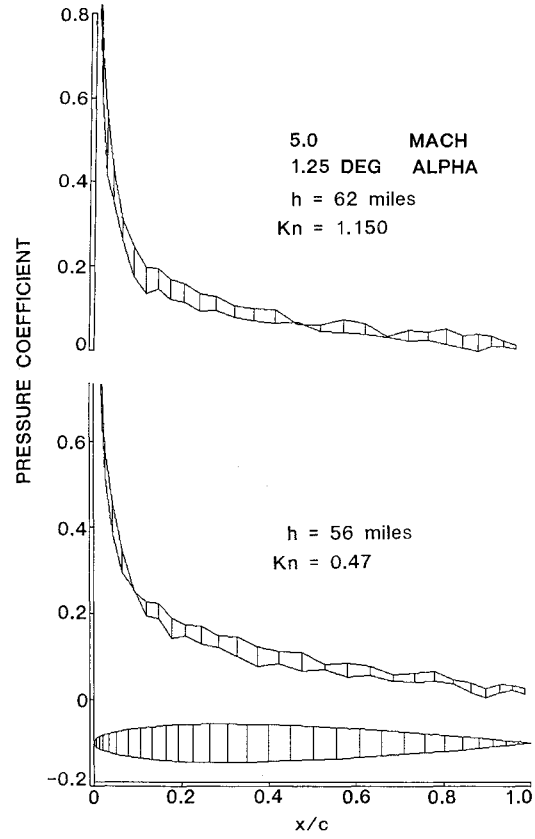

Fig. 10 Pressure distributions over NACA-0009 airfoil.

hardly be defined. The region is characterized by the Knudsen layer. It differs from the shock layer in that the flow is thermally nonequilibrium, and its relaxation takes place in a few mean free paths. The layer, that is much thicker than the shock layer given in the ENS method,[7] grows even thicker at $h = 62$ miles (see Fig. 7). In contrast with slightly rarefied flows of smaller Knudsen numbers, the effect of rarefaction here becomes much more dominant than that due to viscosity.

Circular-arc airfoils, or their derivatives such as parabolic-arc airfoils or conical-section airfoils, are considered to be efficient for supersonic flight.[10] In the present work, a 9% thick, circular-arc airfoil is used as another test case. For the same flow conditions and airfoil thickness, the lift coefficient of the circular-arc airfoil with various Reynolds numbers is generally higher than that of the NACA-0009 airfoil (see Fig. 8). The result of the $M_\infty = 4$ and $h = 56$ miles is discarded because of large errors discussed below.

The airfoil drag coefficient rises sharply with increasing altitude. Figure 9 shows the drag coefficient vs Reynolds num-

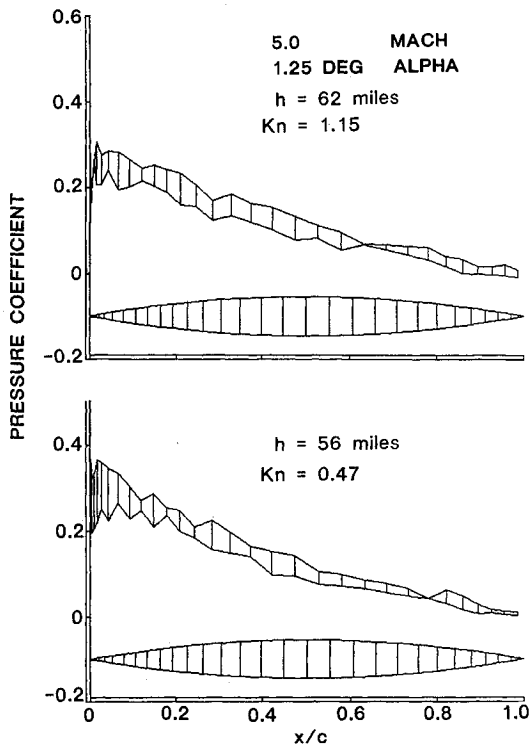

Fig. 11 Pressure distributions over 9% thick circular-arc airfoil.

ber for the NACA-0009 airfoil and the 9% thick, circular-arc airfoil traveling at $M_\infty = 4$ and 5, and $\alpha = 1.25$ deg. The viscous effect mixed with the rarefaction effect is mainly responsible for the drag rise. Similar effects in flow with very low Reynolds numbers have been found by Dogra et al.[5] on blunt bodies of revolution using the same DSMC method.

The computed pressure distributions over the two airfoils at $M_\infty = 5$ and $\alpha = 1.25$ deg are shown in Figs. 10 and 11. The values of the pressure coefficient exhibit some scattering despite the large amount of computer time—60 h in a Personal Iris workstation. This is due to the probabilistic nature of the approach.

Error Estimate

Scattering of data, which is the nature of statistical approach, makes the interpretation of results rather difficult. In principle, the larger the number of samples, the more reliable the result of a statistical method. Thus, the number of molecular samples used in most cases ranges from 15,000 to 20,000. It corresponds to a total number of entering molecules of approximately one million. A typical case takes about 60–65 h of computer time in a Personal Iris workstation.

Nevertheless, the results are subject to errors. The large number of samples used has kept the error to the minimum. After about 15,000 samples, the results of aerodynamic forces tend to stabilize, fluctuating around ± 5–10%.

Numerical results having errors of more than 10% were discarded. These are the calculations on the 9% thick circular-arc airfoil at Mach 4 at 56 miles above sea level, and all Mach 3 cases on both airfoils at both 56 and 62 miles altitudes. The number of collisions in the computation regions in all these cases was several times higher than the total number of entering molecules. It is of interest to note that the number of molecular collisions depends not only on the Knudsen number of the flow, but also on the shape of airfoil. The circular-arc airfoil at Mach 4 and 56 miles has considerably more collisions than the case of NACA-0009 airfoil. In addition, both airfoils at Mach 3 encounter twice as many collisions as in Mach 4 or 5 cases. Since the DSMC method is valid only for dilute gas flows, the large number of collisions may have in fact violated the dilute gas requirement.

Concluding Remarks

The low-density aerodynamics about airfoils in the transition region is considered using a direct simulation Monte Carlo method. At altitudes of 56 and 62 miles above sea level, the flow departs considerably from the continuum theory. The use of a direct simulation method is feasible and seems to be more adequate than the Navier-Stokes approach or its slip-flow correction type extension. The present DSMC results indicate that both lift and drag are very much penalized in the transition region. The shock layer is replaced by the Knudsen layer where the effect of rarefaction becomes much more dominant. Regardless of its large computer time requirement, the direct simulation method may be the only feasible approach presently capable of handling the rarefied aerodynamics in the Knudsen number range of interest.

Acknowledgments

This work was supported by the Independent Research Program at the Carderock Division, Naval Surface Warfare Center (WU 1660-115 and 1270-004). The author would like to thank Jim Moss of NASA Langley Research Center for his suggestions and useful discussions.

References

[1]Yen, S. M., "Numerical Solution of the Nonlinear Boltzmann Equation for Nonequilibrium Gas Flow Problems," *Annual Review of Fluid Mechanics*, Vol. 16, 1984, pp. 67–97.

[2]Bird, G. A., "Monte Carlo Simulation of Gas Flows," *Annual Review of Fluid Mechanics*, Vol. 10, 1978, pp. 11–31.

[3]Bird, G. A., "Approach to Translational Equilibrium in a Rigid Sphere Gas," *Physics of Fluids*, Vol. 6, Oct. 1963, pp. 1518–1519.

[4]Moss, J. N., Cuda, V., Jr., and Simmonds, A. L., "Nonequilibrium Effects for Hypersonic Transitional Flows," AIAA Paper 87-0404, Reno, NV, Jan. 1987.

[5]Dogra, V. K., Moss, J. N., and Simmonds, A. L., "Direct Simulation of Stagnation Streamline Flow for Hypersonic Reentry," AIAA Paper 87-0405, Reno, NV, Jan. 1987.

[6]Hermina, W. L., "Monte Carlo Simulation of Rarefied Flow Along a Flat Plate," AIAA Paper 87-1547, Honolulu, HI, June 1987.

[7]Tai, T. C., and Moran, M. S., "Supersonic Low-Density Flow Over Airfoils," *Journal of Aircraft*, Vol. 26, No. 12, 1989, pp. 1118–1122.

[8]Bird, G. A., "General Programs for Numerical Simulation of Rarefied Gas Flows," G.A.B. Consulting, Killara, Australia, Jan. 1988.

[9]Borgnakke, C., and Larsen, P. S., "Statistical Collision Model for Monte Carlo Simulation of Polyatomic Gas Mixture," *Journal of Computational Physics*, Vol. 18, No. 4, 1975, pp. 405–420.

[10]Pittman, J. L., "Supersonic Aircraft Optimization," *Journal of Aircraft*, Vol. 24, No. 12, 1987, pp. 873–880.