

REPORT DOCUMENTATION PAGE Dist: A

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank) 2. REPORT DATE 28 Nov 94 3. REPORT TYPE AND DATES COVERED Final 1 Apr 93 - 30 Sep 94

4. TITLE AND SUBTITLE A Study of fluid Problems Requiring Direct Particle Simulation 5. FUNDING NUMBERS F49620-93-1-0143 2307/AS

6. AUTHOR(S) Donald Baguff

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Stanford University Stanford, CA 94305 8. PERFORMING ORGANIZATION REPORT NUMBER AFOSR-TR-95 0055

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) AIR FORCE OFFICE OF SCIENTIFIC RESEARCH DIRECTORATE OF AEROSPACE SCIENCES BOLLING AFB, DC 20332-6448 10. SPONSORING / MONITORING AGENCY REPORT NUMBER F49620-93-1-0143 2307/AS

11. SUPPLEMENTARY NOTES 19950130 059

12a. DISTRIBUTION / AVAILABILITY STATEMENT APPROVED FOR PUBLIC RELEASE DISTRIBUTION IS UNLIMITED 12b. DISTRIBUTION CODE A

13. ABSTRACT (Maximum 200 words)
This final report is for work carried out under Grant No. F49620-93-1-0143 during the 18-month period from 1 April, 1993 to 30 September, 1994.
The principal objective of the research was to make use of a parallel computer code recently developed by our group, for Bird's direct simulation Monte Carlo (DSMC) method, and to gain experience in simulating various rarefied three-dimensional flows about bodies having reasonably complex geometries, for example, bodies based on geometries associated with current practical applications.
DTIC QUALITY INSPECTED 3

14. SUBJECT TERMS Particle Simulation, Hypersonic 15. NUMBER OF PAGES 57 16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT UNCLASSIFIED 18. SECURITY CLASSIFICATION OF THIS PAGE UNCLASSIFIED 19. SECURITY CLASSIFICATION OF ABSTRACT UNCLASSIFIED 20. LIMITATION OF ABSTRACT

REPORT DOCUMENTATION PAGE *Dist. A*

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank) 2. REPORT DATE *28 Nov 94* 3. REPORT TYPE AND DATES COVERED *Final 1 Apr 93 - 30 Sep 94*

4. TITLE AND SUBTITLE
A Study of Fluid Problems Encountering a Direct Particle Simulator

5. FUNDING NUMBERS
F49620 - 93-1-0143 2307/AS

6. AUTHOR(S)
Donald Baganoff

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
*Stanford University
Stanford, CA 94305*

8. PERFORMING ORGANIZATION REPORT NUMBER
APOSR-TR-95 0055

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
AIR FORCE OFFICE OF SCIENTIFIC RESEARCH
DIRECTORATE OF AEROSPACE SCIENCES
BOLLING AFB, DC 20332-6448

10. SPONSORING / MONITORING AGENCY REPORT NUMBER
F49620 - 93-1-0143 2307/AS

11. SUPPLEMENTARY NOTES
19950130 059

12a. DISTRIBUTION / AVAILABILITY STATEMENT
APPROVED FOR PUBLIC RELEASE
DISTRIBUTION IS UNLIMITED
DTIC ELECTE JAN 31 1995 S G D

12b. DISTRIBUTION CODE
A

13. ABSTRACT (Maximum 200 words)

This final report is for work carried out under Grant No. F49620-93-1-0143 during the 18-month period from 1 April, 1993 to 30 September, 1994.

The principal objective of the research was to make use of a parallel computer code recently developed by our group, for Bird's direct simulation Monte Carlo (DSMC) method, and to gain experience in simulating various rarefied three-dimensional flows about bodies having reasonably complex geometries, for example, bodies based on geometries associated with current practical applications.

DTIC QUALITY INSPECTED 3

14. SUBJECT TERMS
Particle Simulation, Hypersonic

15. NUMBER OF PAGES
27
16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT
UNCLASSIFIED

18. SECURITY CLASSIFICATION OF THIS PAGE
UNCLASSIFIED

19. SECURITY CLASSIFICATION OF ABSTRACT
UNCLASSIFIED

20. LIMITATION OF ABSTRACT



**Department of AERONAUTICS and ASTRONAUTICS
STANFORD UNIVERSITY**

Final Report

for the Period

1 April, 1993 to 30 September, 1994

on

**A STUDY OF FLUID PROBLEMS REQUIRING A DIRECT
PARTICLE SIMULATION**

Grant No. F49620-93-1-0143

Submitted to the

Air Force Office of Scientific Research
Building 410, Bolling AFB, D. C. 20332
L. Sakell - Technical Officer

by the

Department of Aeronautics and Astronautics
Stanford University, Stanford, California 94305

Donald Baganoff - Principal Investigator

28 November, 1994

Accession For	
NTIS CRA&I	<input checked="" type="checkbox"/>
DTIC TAB	<input type="checkbox"/>
Unannounced	<input type="checkbox"/>
Justification	
By	
Distribution /	
Availability Codes	
Dist	Avail and/or Special
A-1	

INTRODUCTION

This final report is for work carried out under Grant No. F49620-93-1-0143 during the 18-month period from 1 April, 1993 to 30 September, 1994.

The principal objective of the research was to make use of a parallel computer code recently developed by our group, for Bird's direct simulation Monte Carlo (DSMC) method, and to gain experience in simulating various rarefied three-dimensional flows about bodies having reasonably complex geometries, for example, bodies based on geometries associated with current practical applications.

The computational facilities needed to carry out the work were made available as a result of our close collaboration with several researchers at NASA-Ames Research Center, in particular W. J. Feiereisen (Computational Aerosciences Project Manager) and T. A. Edwards (Chief, Aerothermodynamics Branch). In the 18-month period of the grant this gave our students access to the use of supercomputers such as the Intel iPSC/860 Gamma, Intel iPSC/860 Delta, the Intel Paragon XP/S-15 and the Cray Y-MP C90. In early stages of their work, the students would either make use of an HP 9000/730 workstation in our laboratory or access the supercomputers from microcomputers in our laboratory. In later stages, they had the opportunity to conduct their research on workstations at NASA-Ames.

The work carried out in this period was closely tied to earlier contributions of several students who participated in developing the version of the code used in these studies. Before the start of this investigation, Jeffrey D. McDonald had finished a multiple species version of a particle code which he developed to take advantage of the vector capabilities of the Cray family of supercomputers (Cray-2, Cray Y-MP), as well as to take advantage of certain algorithmic improvements introduced by our group [1,2]. The most recent version of this serial code is called **PSim2**. McDonald then worked on a parallel version of the same code for use on the Intel family of parallel supercomputers. The most recent version of this code, with very important contributions from Michael A. Fallavollita [3], is called **PSim4**. Some additional development of the parallel code was then continued by Douglas C. Dahlby, to introduce changes that would make it easier to use in the study of problems having important practical aspects.

PARALLEL CODE APPLICATIONS

In an effort to gain experience in the application of our parallel computer code to problems that contain practical elements with moderately complex bodies, we turned our attention to the study of a body shape that corresponds to a wing-body, single-stage to orbit (SSTO) vehicle, where we have attempted to reproduce the general shape of a proposed design but have used our own values for the different parameters defining the vehicle geometry. Figure 1 shows the generic vehicle shape we have been considering. In this case the Mach number was set at 10, the angle of attack at 20 degrees, and the Knudsen number at 0.05, based on the length of the body. The display shows the density distribution in the central plane of the body and in a single cross section of the flow just downstream of the body. One of the many quantities that can be obtained from the simulation is the heat-flux to the body surface, which is shown in Fig. 2 for the vehicle at zero angle of attack. The flow field and body were resolved by use of cubic cells, with 120 in both the stream-wise and vertical directions, and with 75 cells along the semispan direction, for a total of 1.1 million cells. Because we did not feel it was appropriate to invest a large fraction of our time in the detailed design of a body, certain details such as the inclusion of wing-body fillets were ignored. After the file was created that defined the body, the generation of the three-dimensional displays shown in the two figures was made possible by use of a graphics software package called MATLAB (Math Works Inc.) which was run on our HP 9000/730 UNIX workstation; color hard-copy prints were made using an HP 1200C postscript color ink-jet printer. In both cases, the acquisition and maintenance costs proved to be nominal and ideal for a university environment.

Since the SSTO body was the first wing-body we studied, we were interested in determining whether a vortical flow would be seen in the DSMC method owing to lift on the wings. Figures 3 and 4 show the velocity fields generated at cross sections located at approximately 60% and 90% of the wing root chord, respectively. The simulation was carried out for the case of a subsonic vehicle Mach number, where vortical structures would be expected to be clearly present. The case shown corresponds to an angle of attack of 20 degrees and a Mach number of 0.8; all other parameters were the same as those employed for Fig. 1. As can be seen, vortex structures are not only observed for the entire flow but one can also see in the second downstream cross-section that certain portions of the vortex flow can be associated

with the main wing section and certain portions with the end winglet. Because of the necessity of using finite lengths in plotting the velocity vectors, visual correction is often needed for vectors located near solid boundaries, if the proper boundary conditions are to be seen. In view of the rarefied conditions of the run, the results are very fascinating and add greatly to ones intuitive understanding of fluid mechanics.

For problems involving steady flow, time averaging is used in the DSMC method to improve the simulated results, but time averaging can only be initiated after the steady-state condition has been reached. For some problems, the transient period can be a significant fraction of the corresponding averaging time. If one first wishes to explore a particular problem without investing a large amount of time in averaging, and later wishes to refine the results by increasing the averaging time, or to use the results as a starting state for a similar problem, then the ability to restart the simulation is clearly needed. In the DSMC method, the amount of data that must be saved and then loaded for a restart on a large parallel machine is truly huge and quite impractical to consider. However, there is reason to believe that the normal output from a typical run can be used to re-create an initial run state that is close enough to the final steady state that the corresponding transient period would be negligibly small. This approach has been implemented and found to be quite practical and very effective.

Figure 5 shows the result of a test of the restart capability for a problem where the Knudsen layer is the critical element in the flow. When the mean free path length is relatively large in a Prandtl-Meyer flow, a Knudsen layer develops on the down-stream wall bounding the expanding two-dimensional supersonic flow. The Knudsen layer is fully developed only for the steady state. One highly resolved run was conducted to serve as the reference solution and then the *rms* temperature difference was found for two other runs; one starting from a conventional initial free-streaming state and the other starting from a state created from the reference solution, where a Maxwellian velocity distribution function is initially employed for all cells. The figure shows that the reconstructed initial state approaches the final steady state much more quickly than the start-up used in the conventional method. A second restart test was conducted for the case where a shock wave became the critical element in the flow. In this test, a flow past a circular cylinder was studied. Figure 6 shows the results

obtained and it is clear that in this case a considerable number of time steps are saved by the restart procedure in reducing the transient period.

Because the restart procedure used constructs an Euler flow field, the greatest error is clearly introduced in shock layers. This can be seen in Fig. 7, where the average *rms* density error between the restart solution and the reference solution is displayed for the period 50 to 100 time steps after restart. The greatest difference is clearly seen in the shock wave as it attempts to adjust itself to the correct form, while the corresponding error in the boundary layer on the cylinder is not seen as clearly on the scale displayed in the figure.

The smallest volume of physical space used in the simulation is a cell, usually cubic in shape. Of course these should be made as small as possible for good resolution of both the body and the flow field. However, in certain regions where the density is low, there may be far too few particles per cell to correctly model the collision mechanics, and thus get accurate results, even when using lengthy time averaging. In those regions it would be useful to collect several adjacent cells into a single macrocell while carrying out the simulation. A test study was conducted where the flow ahead of and within the bow shock wave generated by a circular cylinder was intentionally set at a level where too few particles per cell were present to produce accurate results in those regions when using single cells. A second test study was then carried out at exactly the same conditions except macrocells were introduced in those regions where the particle count was too low. Both of these runs were compared with a reference run carried out at a much higher particle number density.

The first test study conducted used a particle density of 1 particle per cell in the upstream flow; and the reference run was carried out at a density of 16 particle per cell in the upstream flow. These results are shown in Fig. 8a where the reference solution is given by the solid curves and the test study is given by the dashed curves. One can see that having too few particles per cell shifts the position of the shock wave toward the high density side and causes the shock wave to become too thin, leading to error in the prediction of the flow field. The second test study was carried out with the same particle density of 1 particle per cell in the upstream flow, but macrocells were introduced in the low-density regions to correct the collision mechanics in the simulation. These results are shown by the dashed curves in Fig. 8b and they are seen

to compare very well with the reference solution, establishing the fact that macrocells can be used in an effective way to improve the predictions of a DSMC simulation.

In this comparison, the total computation time was held fixed by increasing the averaging time for the runs having the low particle number density. The important conclusion drawn is that when computer memory limits one to the use of 1 particle per cell in a given simulation then the introduction of macrocells allows one to obtain results that are equivalent in quality to a run employing 16 times as many particles per cell. That is, a factor of 16 reduction in the size of computer memory required clearly has a large impact on the selection of problems that can be run for a given computer.

Before one can confidently predict the optimum operating conditions for a given simulation, one must thoroughly explore parameter space for a wide variety of conditions. One question that can be raised is how important is the trade-off between the improved spatial resolution gotten by reducing the size of each cell versus the effect of the resultant reduction in the number of particles per cell, assuming the total number of particles and thus the computational effort is held fixed? Figure 9 gives a summary of such a study, where the problem represents the simulation of three-dimensional supersonic flow past a sphere, and the quantity studied was the *rms* temperature difference with respect to a reference solution. The size of the sphere relative to the size of the wind tunnel was held fixed, while the number of cells making up the wind tunnel was increased by factors of two, from a resolution of 8^3 cells to 64^3 cells. When a total of 1.05 million particles are used, Fig. 9a shows that the solution continually improves as the cell size is decreased, indicating that improved spatial resolution is the controlling factor, even though an average as low as approximately 4 particles per cell is reached. For a lesser total amount of particles, the trade-off can be clearly seen, with an optimum occurring at 32^3 cells. When the same data are displayed as in Fig. 9b, one is able to study the effect of increasing the total number of particles, assuming a fixed spatial resolution or cell size. This study was conducted to explore a concept, while working within the bounds of simulation capability and acceptable run-time cost for a particular problem, rather than to produce results that can be used to draw general conclusions. A full report on the above material together with further refinements and applications of our parallel code will appear in a Ph.D. thesis by Douglas Dahlby within the next year.

HYBRID MODEL

The DSMC method becomes computationally inefficient when the conditions of the flow approach the near continuum, i.e., when the Navier-Stokes equations become valid. In this limit the required cell size becomes small and the number of particles required by the simulation becomes large. For flow conditions where the stagnation enthalpy is very high and the body temperature is low, the boundary layer may become a high-density, near-continuum region that is best handled by the Navier-Stokes equations. Therefore, it is most desirable to develop a procedure whereby the DSMC method and a continuum method can be employed in their respective regions of validity and then joined in a suitable fashion at some interface, i.e., the creation of a hybrid scheme. This need has been previously considered for certain special problems, especially in the study of plume flows in space where the gas density in a nozzle is high and the density in the outer plume is very low, clearly suggesting that the two approaches be joined by some suitable scheme.

The central issue in introducing a hybrid scheme is to identify the proper conditions to be used at the interface. For the nozzle problem, where the flow in the nozzle is nearly an Euler flow, the matching conditions are not critical and probably can be handled by assuming a Maxwellian velocity distribution function, to model the one-sided fluxes emanating from the Euler flow. However, in flows where viscous and heat conduction effects dominate, such as in a boundary layer, the matching conditions are far more critical, not presently known, and have not been adequately studied. One of the principal problems that arises in this case is that the physics associated with the DSMC method leads to a clear definition for the one-sided fluxes (mass, momentum, and energy) passing through a given interface. On the other hand, the Navier-Stokes equations provide a clear definition for the total fluxes (the sum of the left- and right-directional values). Although the differences between the two properly define the fluxes entering the DSMC zone, one does not have access to the proper velocity distribution function for these returning particles, and this is needed if the DSMC zone is to blend smoothly with the Navier-Stokes zone. Additionally, if one were to make use of one of the flux vector splitting schemes in computational fluid dynamics (CFD), such as the Steger-Warming scheme, the split fluxes do not have the same physical meaning as the one-sided fluxes in the DSMC method, and therefore, lead to new complications.

Our objective in studying the problem was to identify the proper general conditions to be used at the interface between a DSMC simulation and a suitable CFD calculation. The initial approach taken was to study several simple problems where viscous and heat conduction effects are clearly important and then explore different matching conditions at the interface, starting with the simplest concepts and then evolving towards the use of more complex concepts, as required.

The first model problem considered was a strict one-dimensional nonsteady flow in a finite tube with closed ends. One half of the tube was initially filled with an equilibrium gas and the other half remained empty. This corresponds to the shock-tube problem when no test gas is present, only a driver gas. When the driver gas is suddenly released by the bursting diaphragm, an expansion fan forms and the accelerated gas rushes towards its corresponding closed end. On striking the close end, a reflected shock wave forms and propagates back into the onrushing flow created by the expansion process. The hybrid concept is studied by using a CFD scheme to handle the zone represented by the initial driver gas and the DSMC method to handle the zone represented by the initial vacuum. A comparison is made by employing a reference solution obtained by using the DSMC method to simulate the entire flow.

Results of such a study are presented in Fig. 10, where the vertical dash-dot line identifies the location of the fixed interface station used, with the CFD zone on the left side and the DSMC zone on the right side of each plot. Because a CFD solution is not able to handle regions of zero density, the interface station had to be placed slightly to the left of the diaphragm station (located at the midpoint of the tube). The dashed curve gives the reference DSMC solution and the solid curve gives the hybrid solution, with the CFD result to the left of the vertical dash-dot line and the DSMC result to the right of the vertical line. The two simulations were advanced until the reflected shock wave just reached the interface station (580 time steps). This is the time for which the comparisons shown in the figure are made. For these initial studies, for which we set $Kn = 0.5$ (cell Knudsen number), $CFL = 0.2$ and tube length = 100 cells, we assumed that the conditions at the interface could be approximated by using a Maxwellian velocity distribution function. Until the reflected shock wave arrives at the interface station, the assumption proves to be a good one since the expansion fan is essentially an isentropic process. This is borne

out by the data in the figure as the hybrid solution matches the reference solution quite well for all variables displayed.

The same comparison is shown in Fig. 11 for a slightly later time (620 time steps), a time when the high density side of the reflected shock wave reaches the interface station. At this point the simple matching conditions used are clearly in error because of the differences seen in the temperature and density variables at the interface station. Similar differences are seen at later times, such as when the shock wave reaches the closed end at the left-hand side, as well as when it passes the interface station the second time. This test proves to be quite useful because of the otherwise good agreement found between the two simulation schemes; it is only at specific locations like the interface station that the error is highlighted.

The impulsively started flat plate represents another problem for which useful testing can be carried out. To properly handle the Knudsen layer that always develops close to a material surface, we must place the DSMC zone next to the plate surface, while the CFD zone is set up to handle the remaining outer flow. Selection of the interface position may seem to be arbitrary, but in actual fact it is not entirely arbitrary. A CFD solution depends on the Navier-Stokes equations and they are valid only when the value of the ratio of the shearing stress to the local pressure τ/p is small. Because this quantity starts as a very large number next to the plate surface and then decays as it diffuses out from the plate, one must place the interface station at a location where this ratio is always small. If not, the conditions used for interfacing become critical. Such a test is shown in Fig. 12, where the interface station is located at a vertical distance of 0.2 mean free path lengths above the plate. The same type of comparison with a hybrid solution is made as that considered in developing Figs. 10 and 11, and the same notation for each curve is used, except in this case we are also able to display the known solution for the incompressible viscous fluid, shown as a dotted curve. The observation made in this case is that as long as the ratio τ/p is smaller than 0.1 beyond the interface location, the hybrid solution compares favorably with the reference DSMC result. However, when the ratio becomes equal to 0.1 at the interface position, the hybrid solution and the reference solution show a very significant difference, as seen in the velocity profile.

The two cases studied clearly show that for a flow that is dominated by viscous stress and heat conduction the interfacing problem requires the development of an approach that goes beyond the simple use of a Maxwellian velocity distribution function to model the transfer of particles from a CFD solution to a DSMC solution. What is needed is a numerically simple scheme that can account for the translational nonequilibrium of the gas by predicting the proper number of particles for each velocity interval, as particles are transferred to a DSMC solution. This work was carried out by Tawei Lou and a complete report will be given in his Ph.D. thesis, which should appear in the next year.

COUETTE FLOW: A REFERENCE SOLUTION

In an effort to better understand the limitations on the DSMC method when the mean free path length becomes small, and to gain the necessary information for interfacing a DSMC solution with a continuum solution, we turned our attention to the study of Couette flow. This work is fully described in a Ph.D. thesis by T. Denery [4], so only a very brief description of the results obtained will be given here. The interest in this particular problem was based on several factors: 1) Following the recent work of Baganoff [5], it has become possible to extend the original analytical work of Lees and Liu, which was limited to the case of Maxwell molecules, to the more general case of inverse-power molecules, including the hard-sphere limit; 2) because of our previous work in developing a DSMC code for use on a parallel computer [3], we were in a good position to carry out a large number of simulations of the flow for comparison purposes; and 3) The DSMC method is most securely founded for the case of hard-sphere molecules, a case for which a theoretical solution became available through our recent work [5], and therefore it has become possible for the first time to carry out a careful and proper comparison between DSMC and theory.

The analytical study has also shed light on a very important question in the application of the moment method in kinetic theory. It is well known that in forming moments of the Boltzmann equation, when the order of the moment is higher than the orders of the collisional invariants, that the corresponding integration required to treat the collision integral cannot be carried out in closed form, unless one assumes an interaction potential known as the Maxwell molecule. A common practice

employed when considering a general inverse-power interaction potential has been to assume that one can use the closed form results found for the Maxwell molecule and simply replace the linear temperature dependence of the transport coefficients for the Maxwell molecule by the particular temperature dependence associated with the molecular law of interest, for example, square root for the case of the hard-sphere molecule. No sound theoretical justification exists for this replacement, but it has been frequently used as a reasonable approximation, especially in the analysis of shock-wave structure and its comparison with experiment.

Once we were able to develop an analytic solution for Couette flow, for a general inverse-power molecule, we were in a strong position to test the above procedure. Figure 13 shows a comparison for a plate Mach number of 0.5 and for two ratios of the plate temperature, one near unity and the other fairly extreme. Both stress and heat flux are shown, normalized by their respective values for free-molecule flow, with the Navier-Stokes result in each case given by the dotted straight line. The comparison of interest is between the dashed curve and the solid curve in each case, which are seen to nearly overlap; and because the hard-sphere model is furthest removed from the Maxwell molecule, it was chosen for the comparison. The solid curve represents the theoretical prediction of our analytical solution for hard spheres, while the dashed curve gives the prediction based on the replacement procedure discussed above. In Fig. 14 the same comparison is made except the plate Mach number was raised to 2.0. Because of the nearly perfect agreement and because of the fact that the computational effort required by the full theory is over 10 times greater than that required for the replacement procedure, it is clear that the common practice is fully justified for Couette flow; and it lends strong support to the belief that it is also a very reasonable approximation for more general flows.

One of the most remarkable results found from our theoretical work for Couette flow is exhibited in Fig. 15. In the Navier-Stokes limit where the viscous stress and velocity gradient are related by $\tau = \mu du/dy$, one may define a new spatial variable $ds = dy/\mu$ and write $\tau = du/ds$. Because τ is a constant across the layer in Couette flow, one concludes that the velocity profile must be linear in the new spatial variable s . However, one does not know *a priori* whether the result should also apply to rarefied flow. The figure shows that straight lines are in fact found for the velocity

profile for all degrees of rarefaction, ranging from Maxwell molecules to the hard-sphere limit. In addition, it shows that all the straight lines intersect at a common point. Because the Navier-Stokes limit is represented by a diagonal line connecting the corners of the square box enclosing the plot and the free-molecule limit is merely a vertical line, and its position can be predicted theoretically, the intersection point can therefore be established uniquely. However, a most remarkable result clearly seen in the figure is the fact that all the lines pass through this one point. The temperature variable is also shown plotted in the same way as for the velocity, and again all the curves pass through a common point, but a slight curvature is seen in the temperature data indicating a small quadratic term is present. At this point we have an exciting result but we do not have a suitable theory that can be used to predict it. The figure suggests strongly that the velocity slip and temperature slip should also be given by a simple theory. Some work has been carried out along these lines by Denery but nothing has been found that is as elegant as the figure suggests.

Figure 16 shows a comparison between the extended Lees-Liu solution for hard spheres and the predictions given by the DSMC method. As can be seen, a small difference appears in the transition regime, which seems to increase as the temperature ratio between the two plates is increased. Otherwise, the two give the same predictions for the free-molecule limit and for the Navier-Stokes limit. Relatively large values of the temperature ratio and the Mach number were chosen for display in the figure in order to strain the comparison, otherwise, for small values of the parameters the two agree very well. The cause of the difference for large values of the parameters is most likely associated with the assumed two-sided Maxwellian distribution function used in the Lees-Liu method.

Figure 17 shows a comparison of the velocity distribution functions given by the two methods and it is clear that the two-sided character of the assumed theoretical distribution is not fully supported by the DSMC simulated results. The general trend is certainly present but the details are significantly different. On the other hand, the Lees-Liu velocity distribution function was originally introduced for modest degrees of translational nonequilibrium; and for that case, the discontinuity seen in the figure would be much smaller and the assumed distribution would then more nearly match the DSMC result. A complete account of all of the above work on Couette flow can be found in Denery's thesis [4].

PUBLICATIONS

Three student theses were completed during the period of the grant. These are the ones listed in the references as Fallavollita [3], Denery [4] and Goswami [6]. Likewise, three archive publications appeared in print in the period, and these are the ones by Woronowicz et al. [7], Baganoff [8], and Fallavollita et al. [9]. In addition, an AIAA paper by Goswami et al. [10] also appeared in the same period. It is fully expected that two or more journal papers will soon be submitted as an outgrowth of Denery's work and likewise several should also result from the work of both Goswami and Lou.

STUDENTS SUPPORTED

A total of three graduate students were supported, at various levels, in the 18-month period of the grant. Terry Denery and Tawei Lou received their principal support from the grant and Craig Dutweiller, who just recently joined the research group to pursue a Ph.D. program, received partial summer support. Douglas Dahlby has carried the principal responsibility for continuing the development of our parallel computer code, and has contributed greatly in making it possible to carry out simulations having many practical elements, but because he has been supported by an NSF fellowship he does not appear in this accounting.

REFERENCES

1. McDonald, J. D. "A Computationally Efficient Particle Simulation Method Suited to vector Computer Architectures," Ph.D. thesis, Stanford University, Stanford, CA, December 1989.
2. Baganoff, D. and McDonald, J. D. "A Collision Selection Rule for a Particles Simulation Method Suited to Vector Computers," *Phys. of Fluids A*, Vol. 2, No. 7, 1248 (1990).
3. Fallavollita, M. A. "Implementation and Performance of a Particle Simulation Method Suited to MIMD Parallel Computer Architectures," Ph.D. thesis, Stanford University, Stanford, CA, May 1994.
4. Denery, T. "Development and Application of a Solution to Couette Flow for General Inverse-Power Molecules," Ph.D. thesis, Stanford University, Stanford, CA, December 1994.
5. Baganoff, D. "Maxwell's Second- and Third-Order Equations of Transfer for non-Maxwellian Gases," *Phys. of Fluids A*, Vol 4, No. 1, 141 (1992).
6. Goswami, A. "A Comparative Study of Turbulence Decay using Navier-Stokes and a Discrete Particle Simulation," Ph.D. thesis, Stanford University, Stanford, CA, December 1994.
7. Woronowicz, M. S. and Baganoff, D. "Drag and Heat Transfer Correlations for Rarefied Hypersonic Flow Past Flat Plates", *Jour. of Thermophysics and Heat Transfer*, Vol 7, No. 1, 63 (1993).
8. Baganoff, D. "Kinetic Relaxation of a non-Maxwellian Monatomic Gas in a State of Gross Rest," *Phys. of Fluids A*, Vol 5, No. 5, 1260 (1993).
9. Fallavollita, M. A., Baganoff, D. and McDonald, J. D. "Reduction of Simulation Cost and Error for Particle Simulations of Rarefied Flows," *Jour. of Comp. Physics*, Vol 109, No. 1, 30 (1993).
10. Goswami, A., Baganoff, D., Lele, S. and Feiereisen, W. "A Comparative Study of Turbulence Decay Using Navier-Stokes and a Discrete Particle Simulation", AIAA Paper No. 93-3093, (1993).

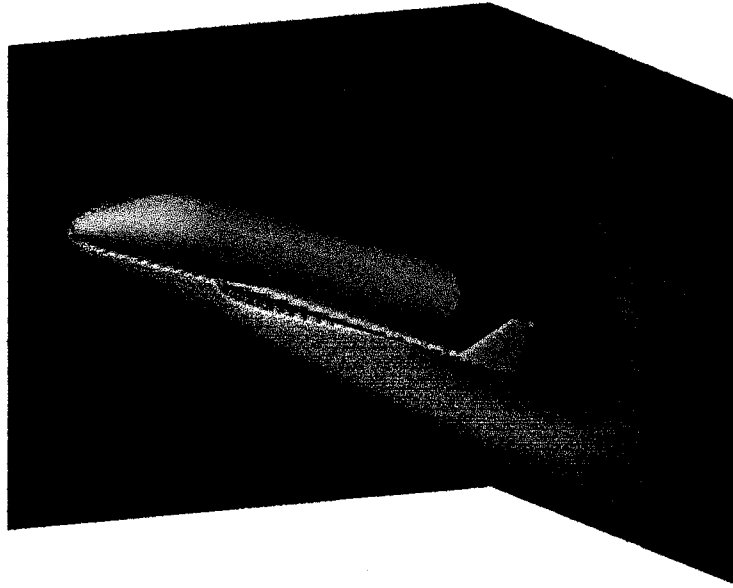


Fig. 1. Density distribution about a generic wing-body SSTO vehicle in a rarefied flow, with $M = 10$, $Kn = 0.02$ and $\alpha = 20$ degrees.



Fig. 2. Distribution of heating flux for the generic SSTO body geometry and conditions considered in figure 1, with $\alpha = 0$.

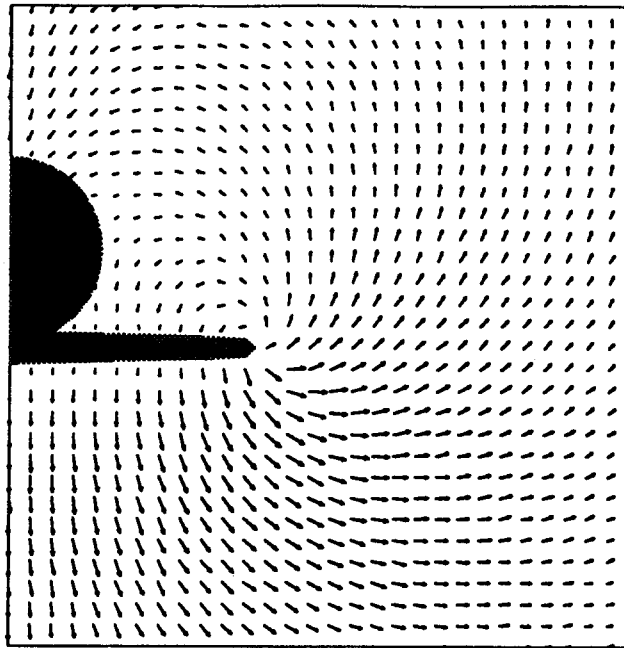


Fig. 3. Velocity field as seen in the cross-section plane located at 60% of the wing root chord. Body geometry and conditions are the same as those for figure 1, except $M = 0.8$.

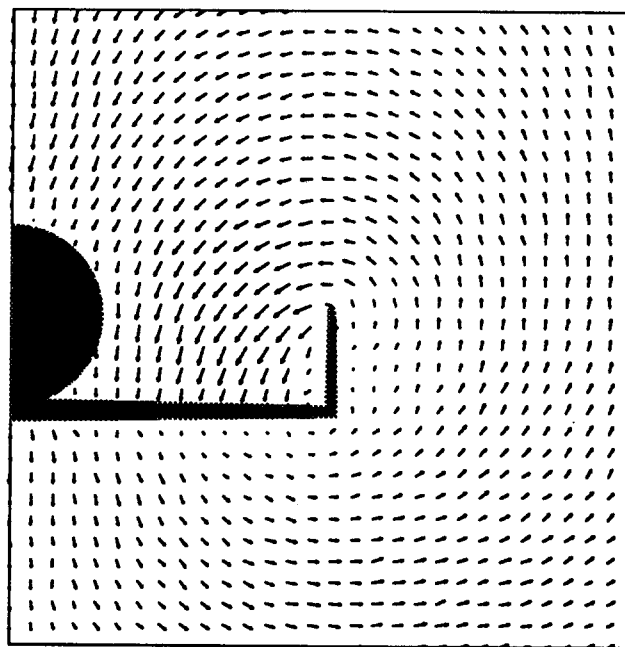


Fig. 4. Velocity field as seen in the cross-section plane located at 90% of the wing root chord. Body geometry and conditions are the same as those for figure 1, except $M = 0.8$.

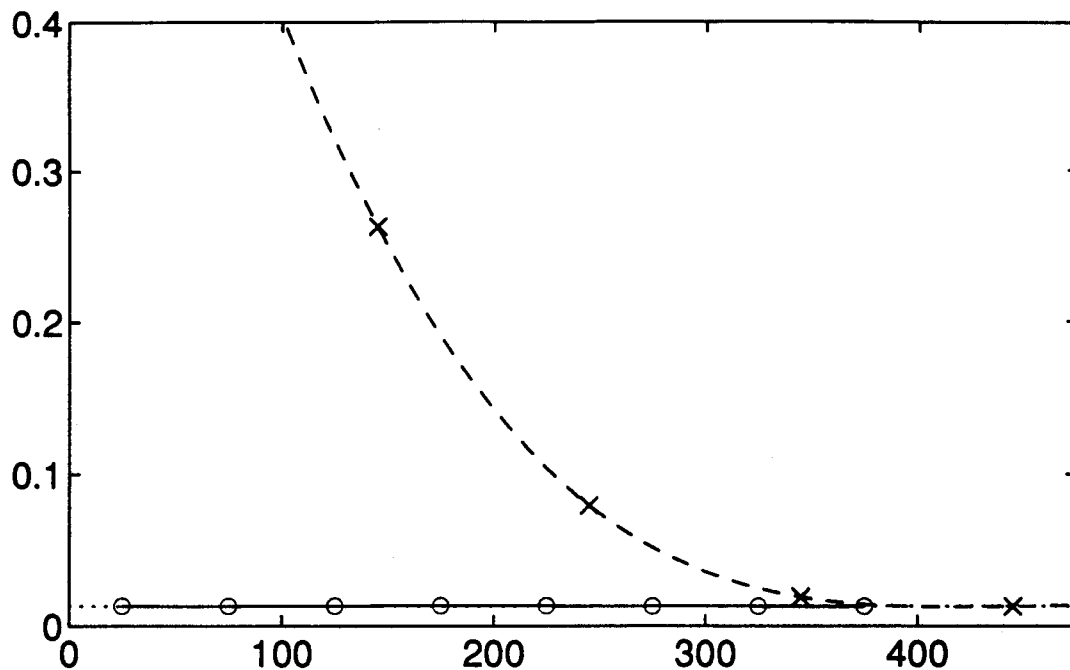


Fig. 5. Effectiveness of restart for a rarefied Prandtl-Meyer flow having a Knudsen layer. Time development of a conventional start (dashed curve) versus a restart (solid curve) is shown for the rms temperature difference with respect to a reference solution.

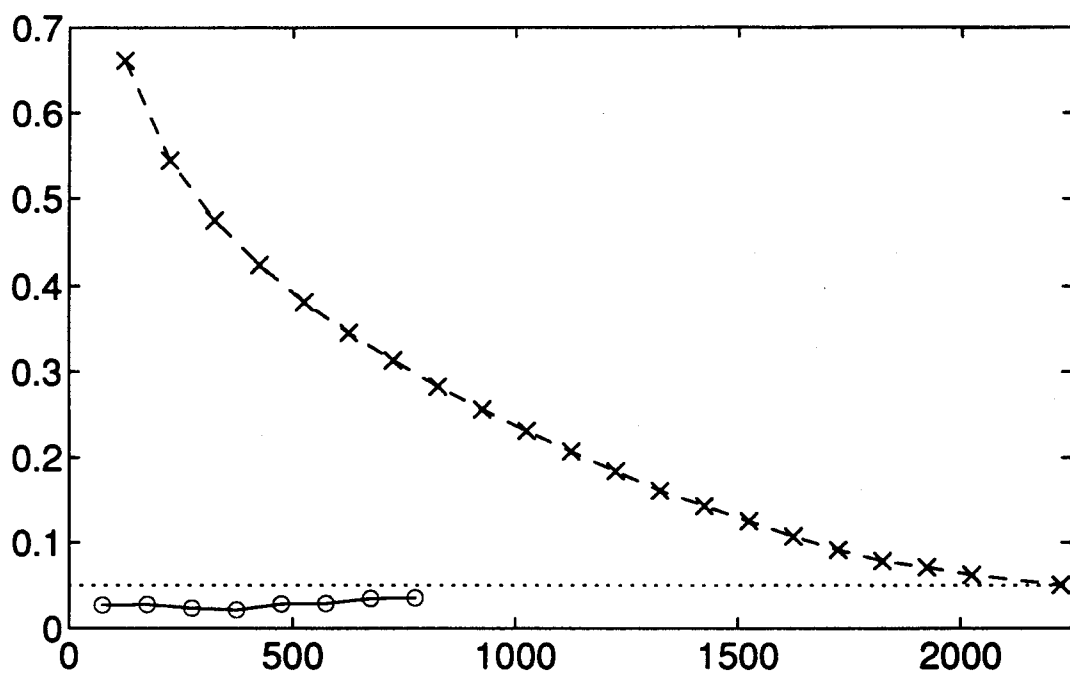


Fig. 6. Effectiveness of restart for a rarefied supersonic flow past a circular cylinder. Time development of a conventional start (dashed curve) versus a restart (solid curve) is shown for the rms temperature difference with respect to a reference solution.

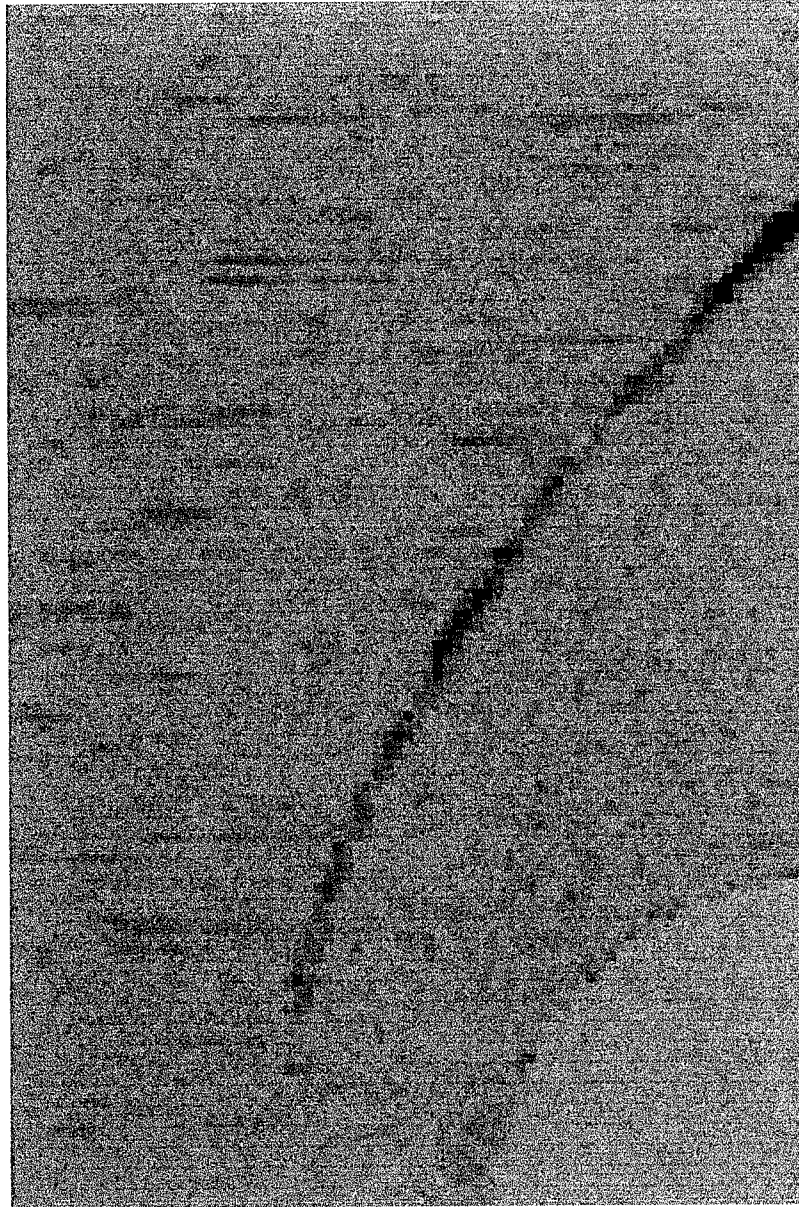
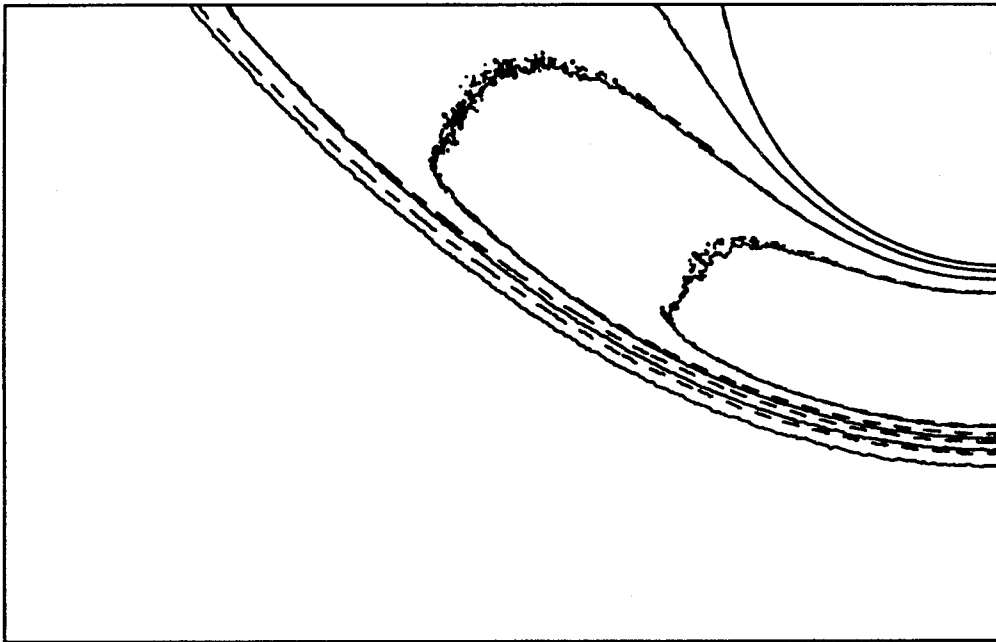
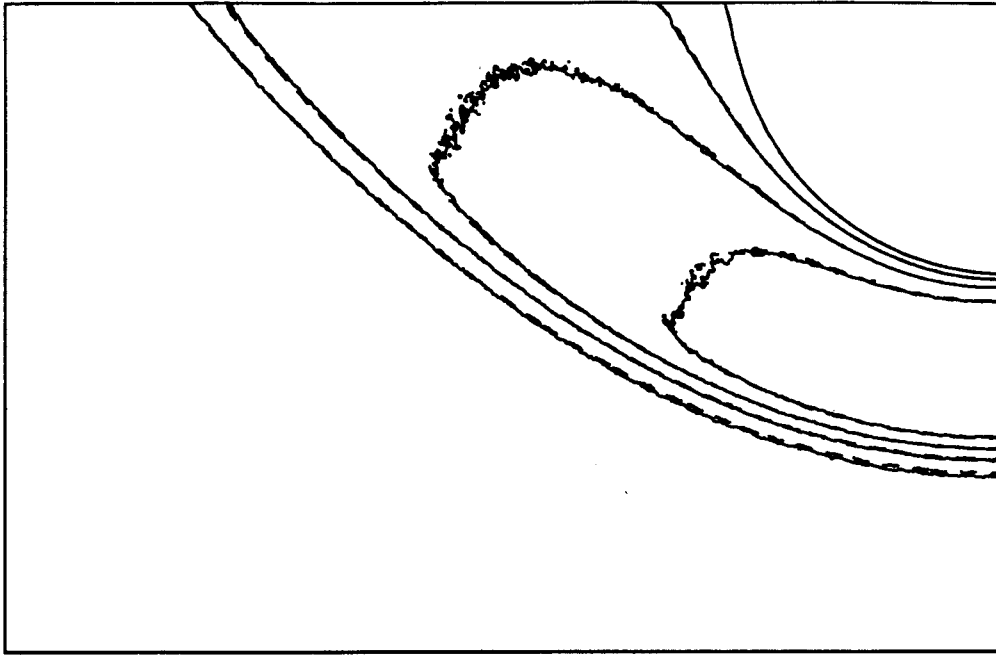


Fig. 7. Location of the largest errors in the early stages of a restart simulation for the case of supersonic flow past a circular cylinder. The quantity displayed is the *rms* density error between the restart and reference solutions.



(a)



(b)

Fig. 8: Effect of a low number of particles per cell in a simulation, where the test studies (dashed curves) made use of 1 particle per cell in the free-stream flow with (a) no macrocells and (b) macrocells, while the reference run (solid curves) made use of 16 particles per cell and no macrocells. The variable shown is temperature and the flow is about a circular cylinder.

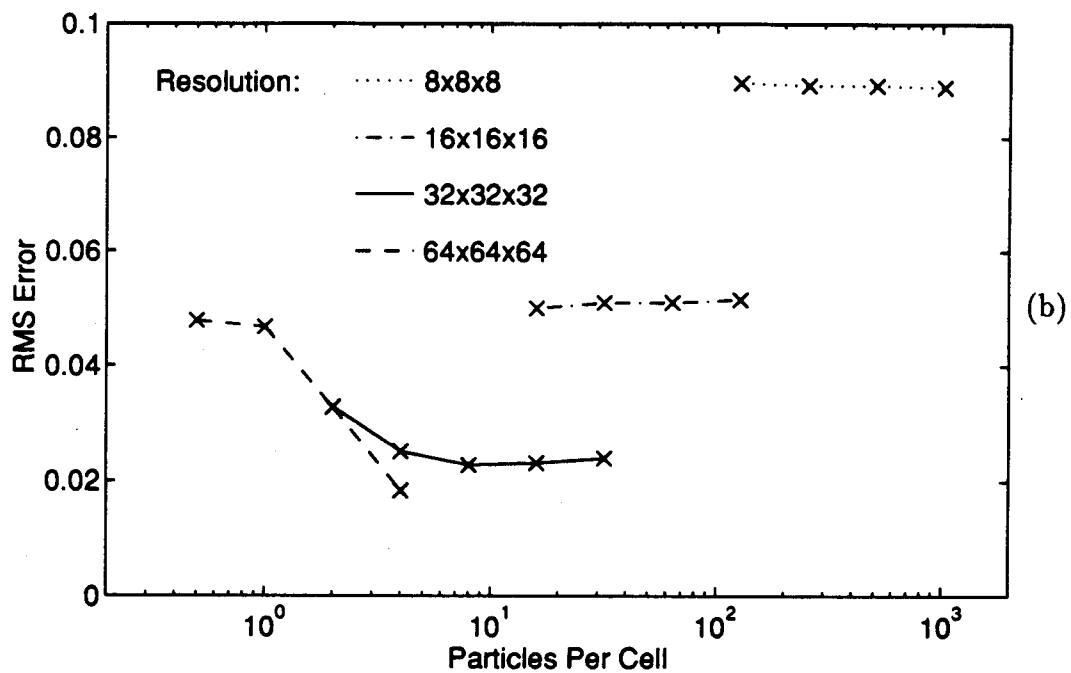
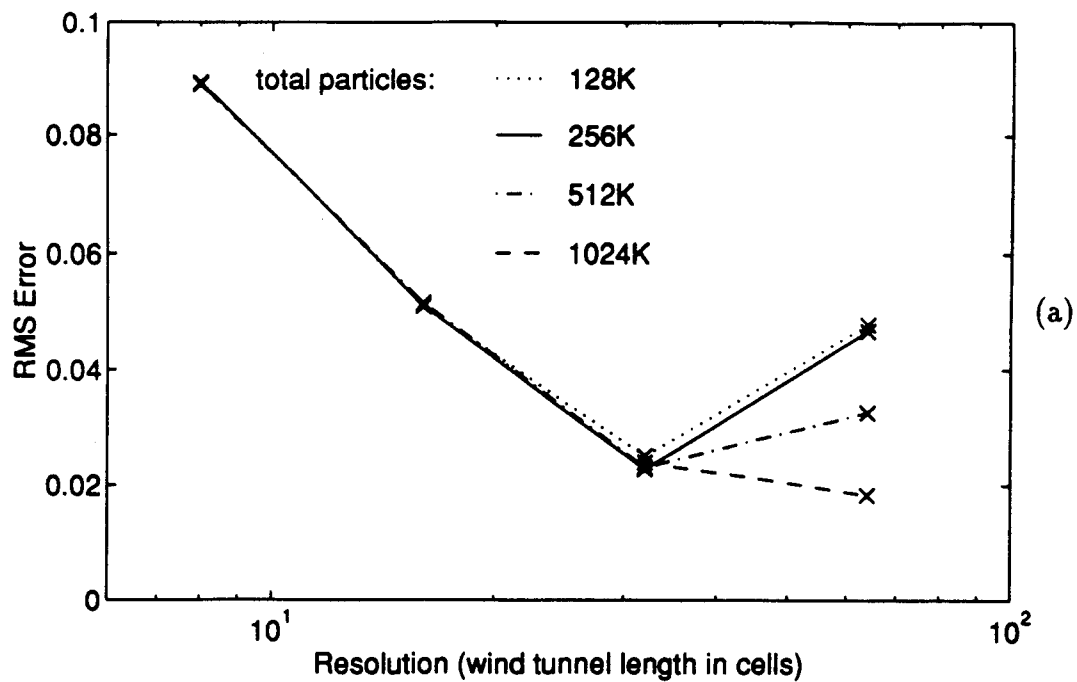


Fig. 9. Effect of spatial resolution (cell size) and the number of particles used in a simulation (particles per cell) on the quality of the simulated results, where improved spatial resolution for a fixed number of particles is shown in (a) and increase in the number of particles holding the spatial resolution fixed is shown in (b).

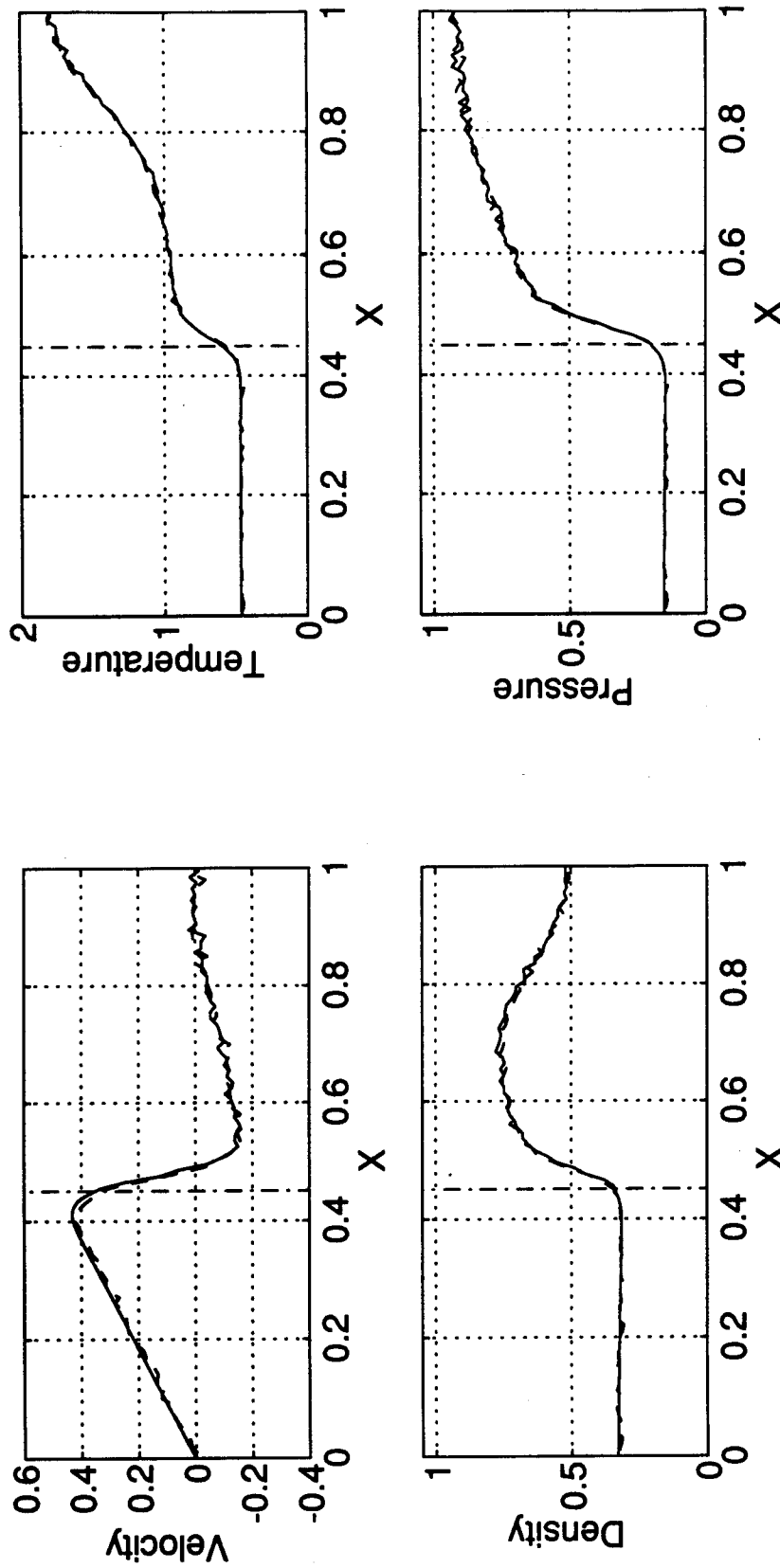


Fig. 10. Comparison of a hybrid solution (solid curve) with a full DSMC solution (dashed curve) at the time the reflected shock wave just reaches the interface position given by the vertical line.

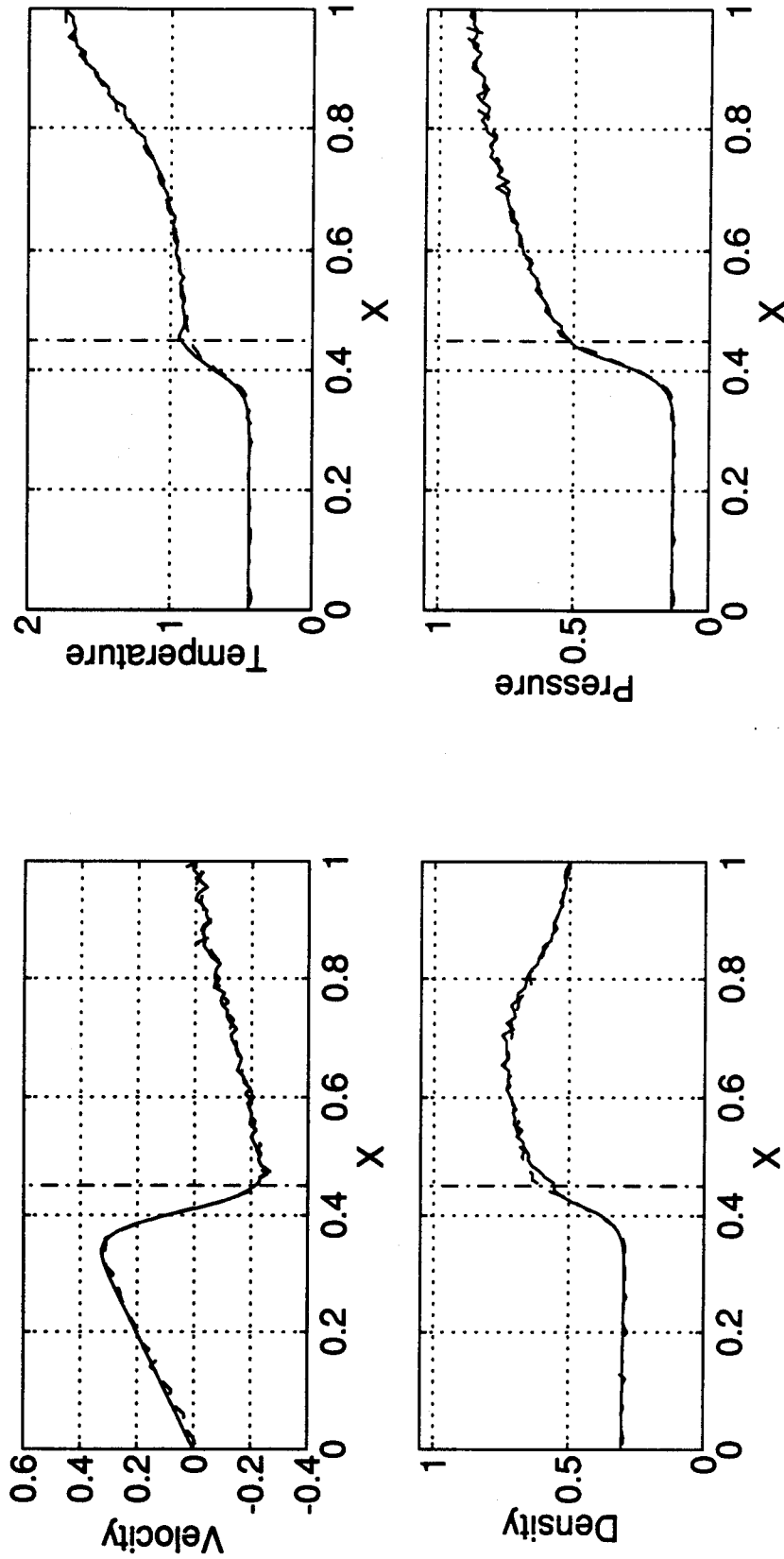


Fig. 11. Comparison of a hybrid solution (solid curve) with a full DSMC solution (dashed curve) at the time the reflected shock wave just passes the interface position given by the vertical line.

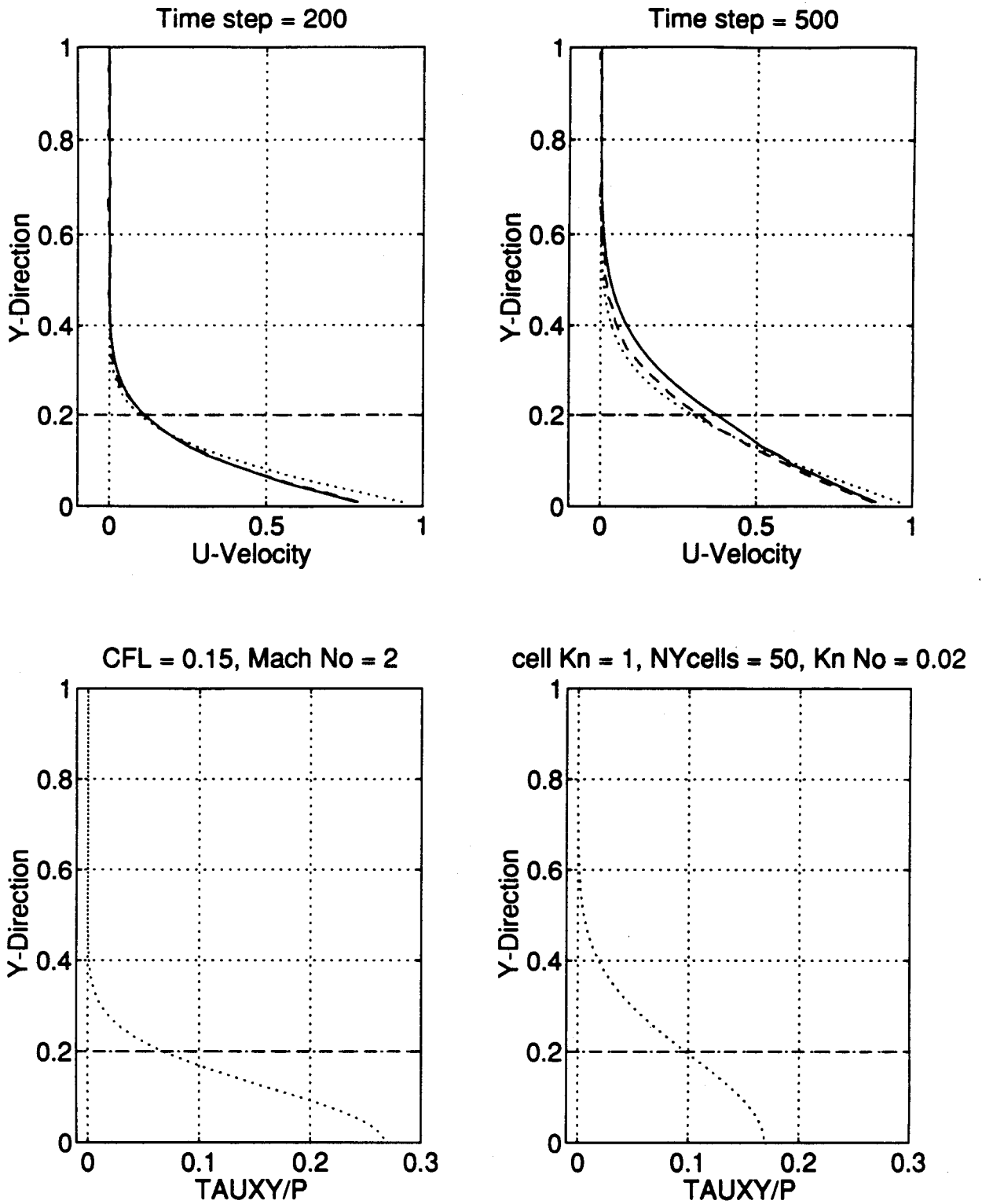


Fig. 12. Comparison of a hybrid solution (solid curve) with a full DSMC solution (dashed curve) for the problem of an impulsively started flat plate and for two time periods: an early time (200 time steps) when $\tau/p < 0.1$; and a later time (500 time steps) when $\tau/p > 0.1$.

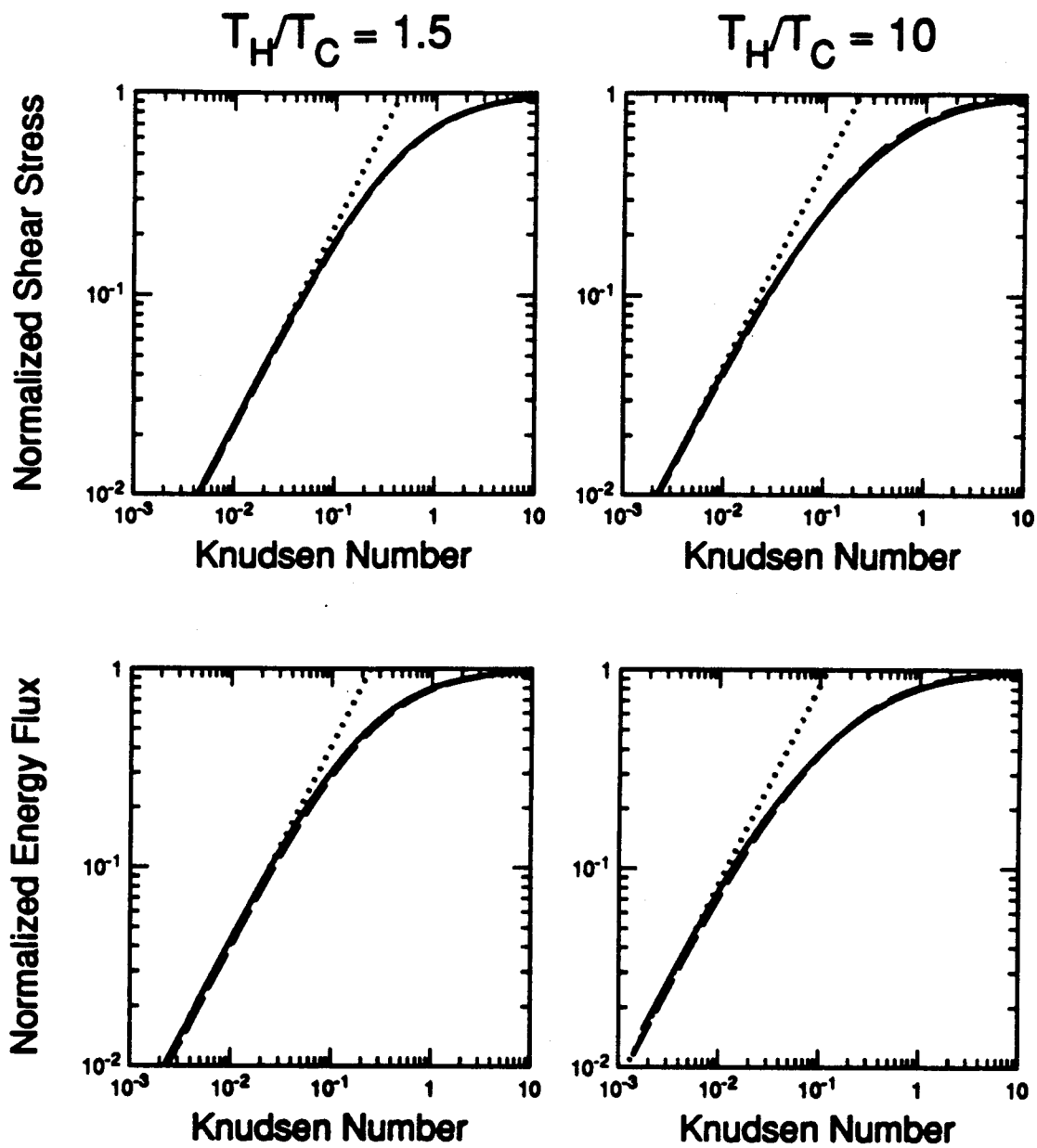


Fig. 13. Comparison of a commonly used approximation (dashed curve) with the full solution (solid curve) for Couette flow, $M = 0.8$ and the hard-sphere molecular model. The Navier-Stokes result (dotted line) is included for reference.

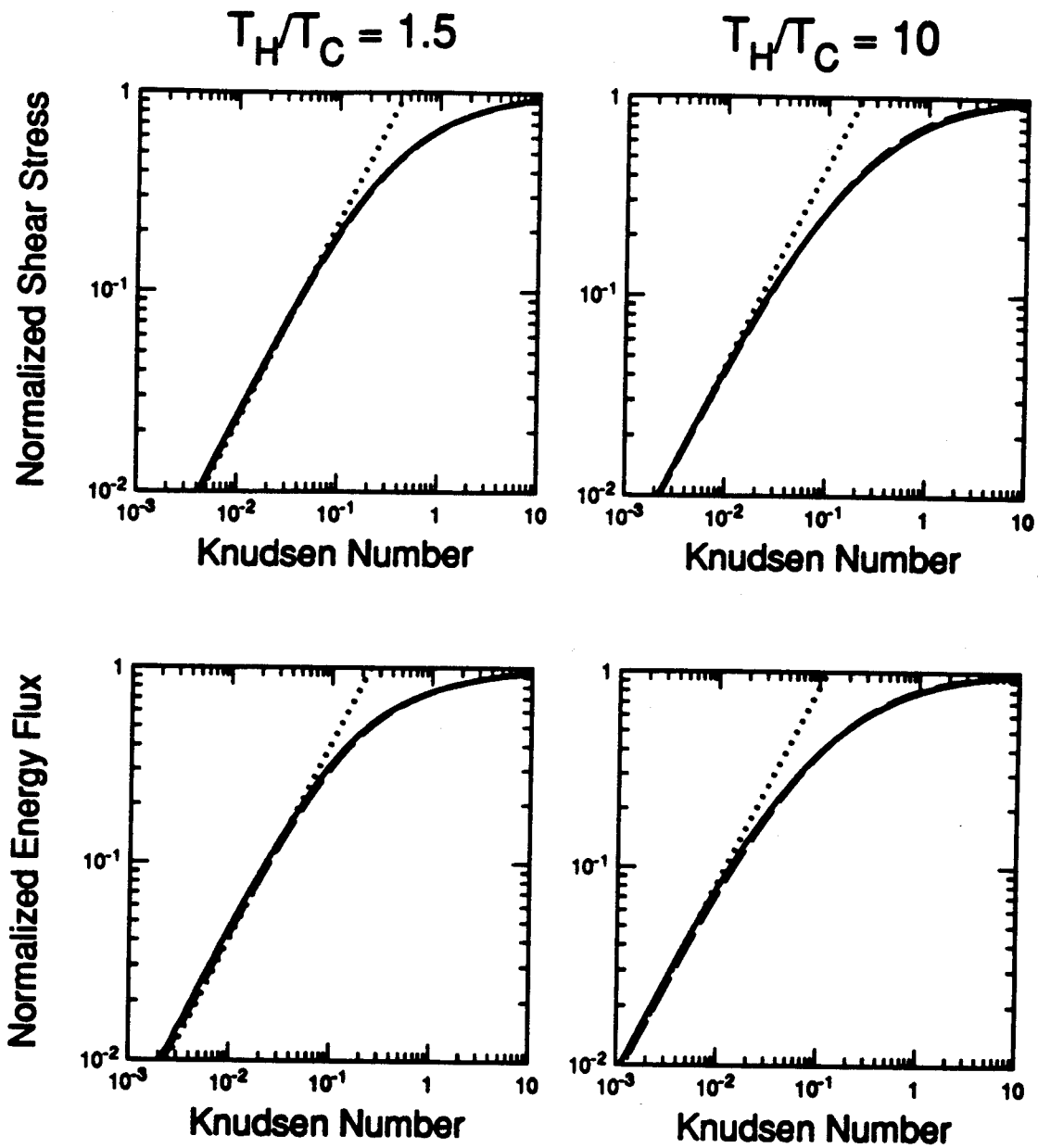


Fig. 14. Comparison of a commonly used approximation (dashed curve) with the full solution (solid curve) for Couette flow, $M = 2.0$ and the hard-sphere molecular model. The Navier-Stokes result (dotted line) is included for reference.

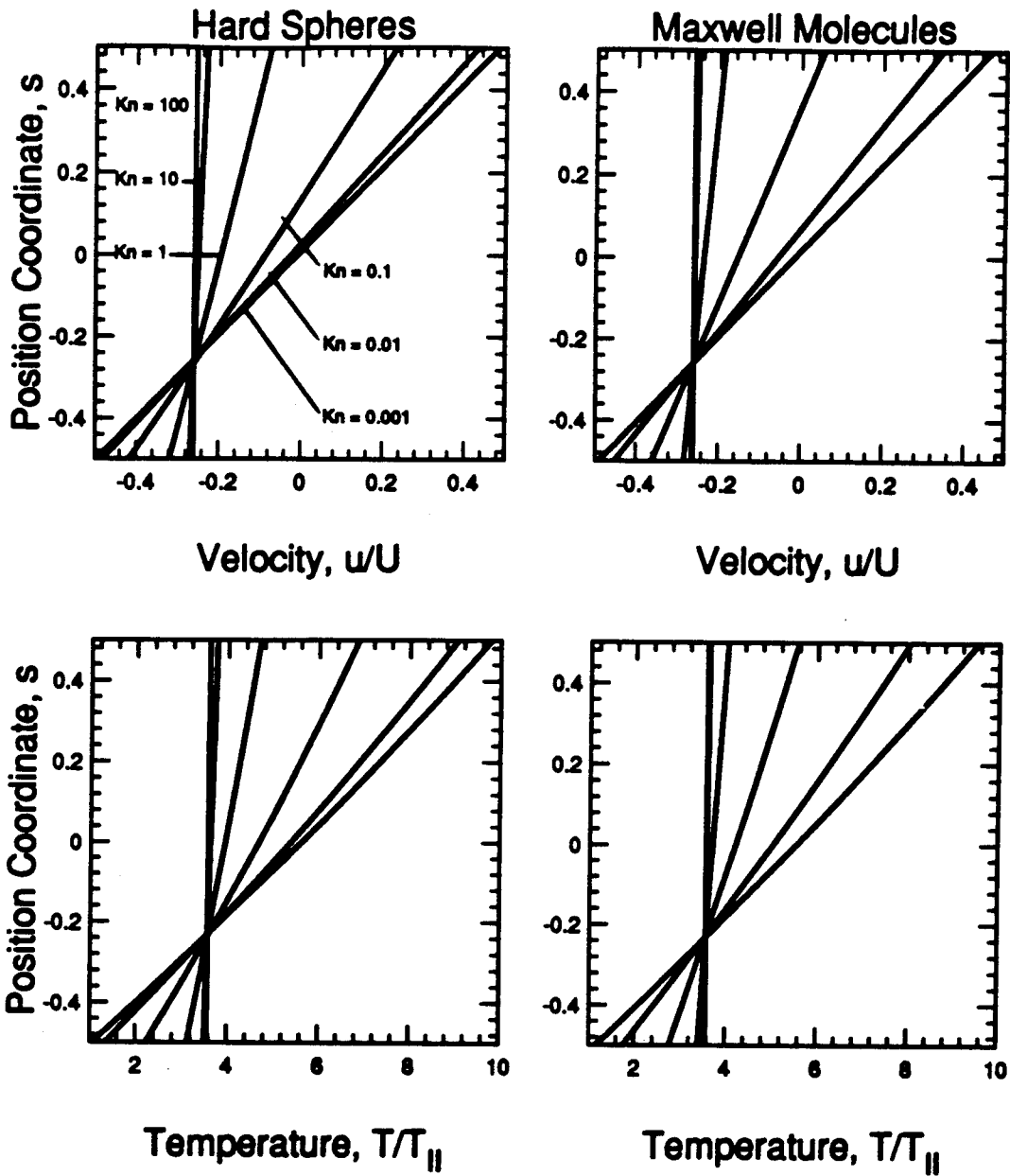


Fig. 15. Velocity and temperature profiles, as determined by the full solution for Couette flow, when using the spatial variable defined by $ds = dy/\mu$. The upper plate has a temperature 10 times that of the lower plate and its Mach number is 2.0, based on the lower plate temperature.

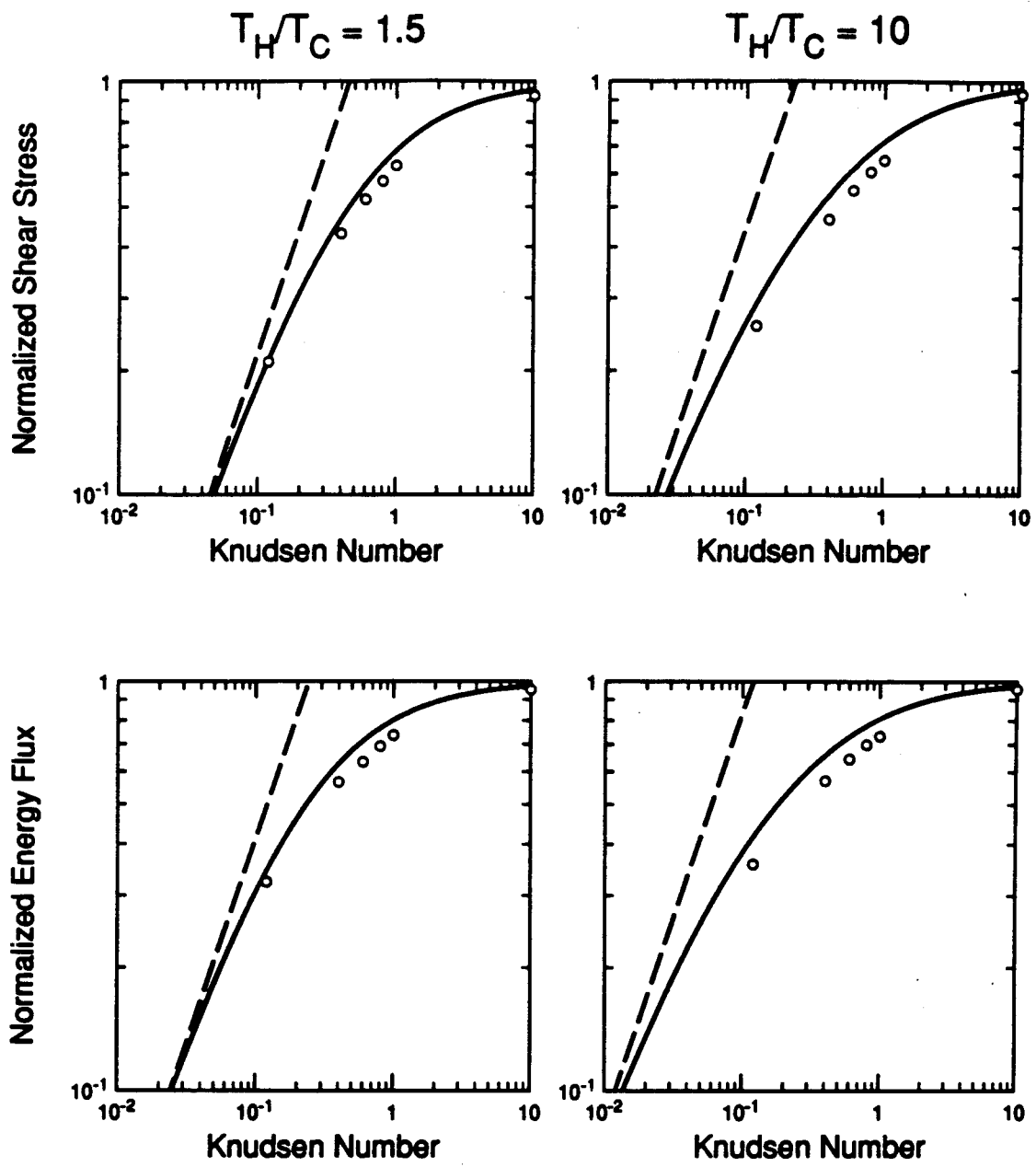
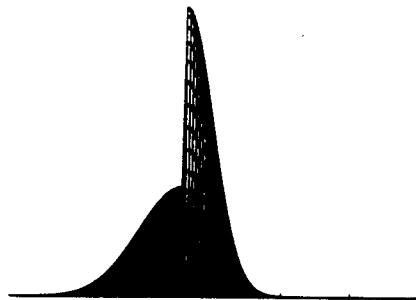
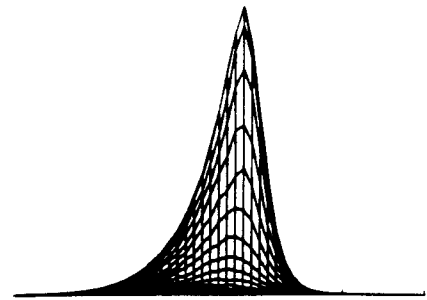


Fig. 16. Comparison of the full solution (solid curve) with results from a DSMC simulation (open circles) and the Navier-Stokes prediction (dashed line). The case is for Couette flow, $M = 2.0$, and the hard-sphere molecular model.

Side View

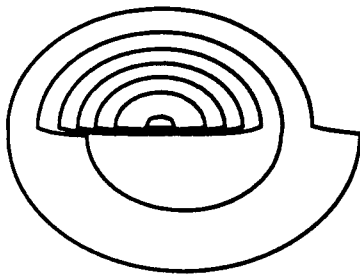


Lees-Liu

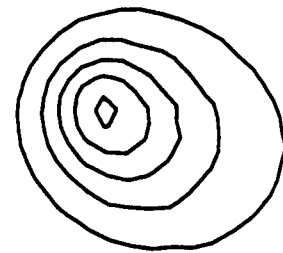


DSMC

Top View



Lees-Liu



DSMC

Fig. 17. Comparison of the velocity distribution function as determined by a DSMC simulation with that found from the full solution using the Lees-Liu two-sided Maxwellian. The data are for the central plane in Couette flow, with $M = 2.0$, $Kn = 0.1$ and a temperature ratio of 10.