

AD-A188 251

INTERNATIONAL CONFERENCE ON INDUSTRIAL AND APPLIED  
MATHEMATICS(U) OFFICE OF NAVAL RESEARCH LONDON  
(ENGLAND) E F BROWN 13 OCT 87 ONRL-7-838-C

1/1

UNCLASSIFIED

F/G 28/4

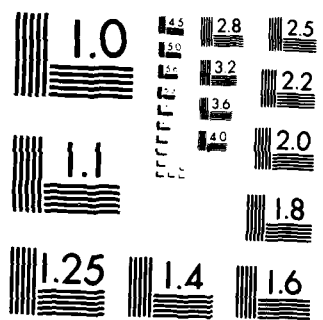
NL

END

DATE

FORM

28-



MICROCOPY RESOLUTION TEST CHART  
NATIONAL BUREAU OF STANDARDS 1963-A

DTIC FILE COPY

①



# ONRL Report 7-030-C

AD-A188 251

International Conference on Industrial  
and Applied Mathematics

Eugene F. Brown

13 October 1987

DTIC  
ELECTE  
NOV 24 1987  
S D  
CS  
H

Approved for public release; distribution unlimited

U.S. Office of Naval Research, London

87 11 10 056



CONTENTS

	<u>Page</u>
1 INTRODUCTION .....	1
2 VORTEX DOMINATED FLOWS .....	1
3 LARGE-SCALE COMPUTING OF AEROSPACE PROBLEMS .....	4
4 CONVERGENCE ACCELERATION METHODS .....	5
5 CALCULATION METHODS FOR TURBULENT FLOWS .....	6
6 LATTICE GAS METHODS .....	7
7 CONCLUSIONS .....	9
8 REFERENCES .....	9

## INTERNATIONAL CONFERENCE ON INDUSTRIAL AND APPLIED MATHEMATICS

### 1 INTRODUCTION

Four international societies of applied mathematics--die Gesellschaft für Angewandte Mathematik and Mechanik (GAMM)--of the German-speaking countries, the Institute of Mathematics and its Applications (IMA) of the UK, the Society for Industrial and Applied Mathematics (SIAM) of the US, and Société de Mathématiques Appliquées et Industrielles (SMAI) France, joined forces in Paris from June 29 through July 3 to conduct the First International Conference on Industrial and Applied Mathematics.

The popularity of the meeting exceeded all expectation, with more than 1500 attendees, more than 1000 papers (which were given in 17 parallel sessions), and 130 poster sessions. The topics ranged from vorticity, chaos, dynamical systems, robotics, and control to optimization, parallel processing, multigrid methods, algorithm development, stochastic modeling, lattices, and networking. In addition to the parallel sessions, there were plenary sessions which featured a total of 16 invited speakers.

The country with the greatest number of attendees (as might be expected) was France, with 310, followed by the US (250), West Germany (160), and UK (110). The remainder of the attendees came from 45 other countries in North, Central, and South America, Africa, Western and Eastern Europe, the USSR, Asia, and Australia.

I will be reporting on papers given in the area of fluid mechanics.

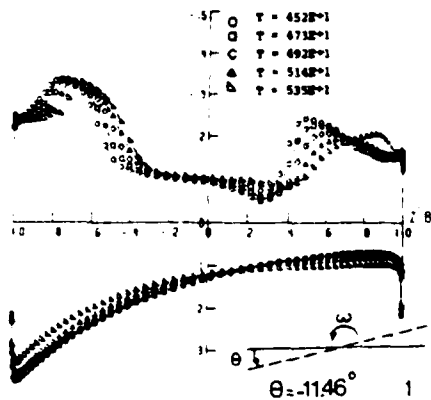
### 2 VORTEX DOMINATED FLOWS

In many high Reynolds high number problems, vortices play a dominant role in establishing the behavior of the flow. Examples are the role played by the leading edge vortex in producing a lift augmentation on delta-wing aircraft and the

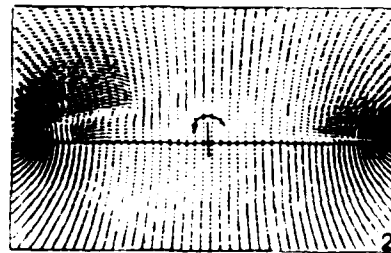
role of vortices in explaining and controlling the rate of mixing. Two contrasting approaches are possible in numerical vortex simulations. One is the so-called Vortex Dynamics approach in which the vortices are treated discretely in the context of a vorticity/stream function formulation of the incompressible Navier-Stokes equations and the vortex paths are traced by particle tracking methods. The other (less mathematically sophisticated) approach is to "capture" the vortices in the course of more or less conventional solutions of the incompressible or compressible Navier-Stokes or Euler equations in which sufficient local grid refinement has been introduced to resolve the vortex patterns.

In an invited paper, A. Majda of Princeton University (New Jersey) reviewed the applications of the vortex dynamics approach and presented some very recent results involving large-scale computations using this method. Specifically, he reported on calculations which he had done to model the evolution of the primary vortex on an elliptically loaded delta wing (with and without the effects of the fuselage) and to model the roll up of the Kirchoff vortex in a planar vortex sheet subjected to a periodic perturbation. Embodied in these calculations are what he termed "real breakthroughs" which involved (1) the use of streamlines of the inviscid theory to obtain nearly optimal convergence properties and (2) the use of multiple expansions to reduce the operator count in the vortex summation operation to order  $N$ , where  $N$  is the number of vortices used in the simulation.

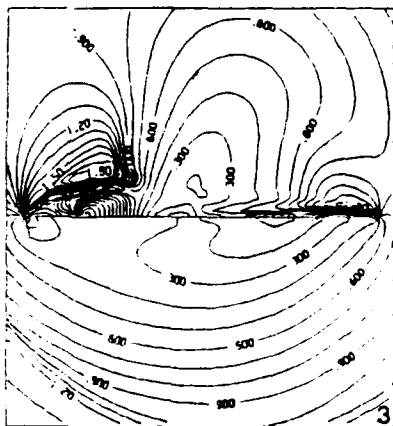
Incredibly complicated vortex patterns are produced when vortices of different sign are present such as in a delta wing calculation. This is illustrated in Figure 1, which shows the cross-flow velocity contours of a delta wing at an angle of attack of  $10^\circ$ . The calculation was done for a free stream Mach number of 2 with the additional complication that the wing was in rolling oscillation. The results were obtained by O.A. Kandil (Old Dominion University, Norfolk, Virginia) not with a vortex dynamics method but by means of a time-accurate solution of the



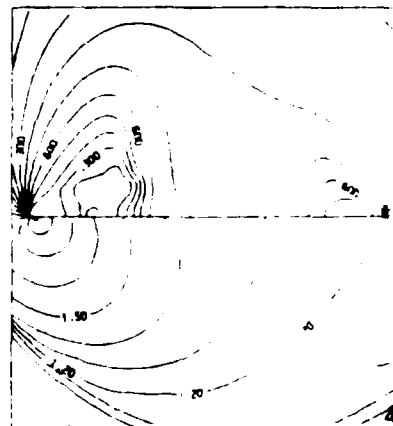
Surface Pressure



Cross-Flow Velocity



Cross-Flow Mach Contours



Static Pressure Contours

Figure 1. 70° Delta Wing at 10° Incidence in Rolling Oscillation at Mach 2.

Euler equations. These calculations are particularly interesting since it is the first time that such calculations have been attempted for a maneuvering wing. The calculations feature a three-dimensional, fully-vectorized computer program which solves the Euler equations for the relative motion in a moving frame of reference using central-difference, finite-volume discretization, and a four-stage Runge-Kutta integration scheme. Several possibilities exist for the choice of variables and (in the case of subsonic flow) the manner in which the condition of constant entropy is introduced.

Kandil found that only the algebraic entropy condition gave the proper (un-separated) leading edge solution for a rounded edge delta wing while for sharp-edged delta wings it didn't make any difference whether the constant entropy condition was introduced algebraically or a constant value was prescribed by means of a partial differential equation of entropy transport. In solving a similar problem (but for a nonmaneuvering case) C.-H. Bruneau of the Université de Paris-Sud (France) found that in order to get a stable solution of the steady Euler equations for both subsonic and supersonic

flows, he needed to solve an explicit energy transport equation. If, instead, a constant value of the stagnation entropy was enforced by means of an algebraic relationship, convergence could only be obtained for subsonic free-stream flow.

By far the most extensive vortex wing computations were reported by A. Rizzi of The Aeronautical Research Institute of Sweden. He carried out both Euler and Navier-Stokes calculations using his well-known finite-volume, cell-centered scheme with one-step, three-stage Runge-Kutta integration. He characterized his Euler calculations as being disappointing. He expected from the outset, of course, that the Euler calculations could not capture the secondary separation resulting from the cross-flow shock. However, it was expected that the overall character of the solution as evidenced by, say, the static pressure distribution on the wing would be more or less correct. This turned out not to be the case. The static pressure rise due to the primary vortex seemed to be too strong and located too far outboard on the wing, a situation which did not improve even when more than 1 million points were used in the calculation. Additional details of Rizzi's calculations can be found in (AIAA 41-1:22 [1987]).

Rizzi has just obtained the first results from his new Navier-Stokes code, which uses the same cell-centered scheme as the inviscid calculations. The derivatives appearing in the viscous terms were evaluated by means of a single-grid two-step application of the gradient theorem (Peyret and Taylor, 1983). A Mach 0.85 case at  $10^\circ$  incidence was run as a laminar calculation at a Reynolds number of  $2.4 \times 10^6$  employing 400,000 cells, 100,000 of which were in the boundary layer. Several qualitative differences from the earlier inviscid calculations were immediately apparent:

- o The origin of the primary vortex was at the wing apex rather than along the leading edge.
- o The Navier-Stokes equations picked up the secondary separation and perhaps even a tertiary one.

- o A cascade of vortices seemed to arise in the vicinity of the (rounded) leading edge of the wing.

The most striking difference, however, was in the greatly improved accuracy of the predicted static pressure. The new origin of the primary vortex resulted in the pressure peaks being moved inboard the proper location, and the level of the pressure rise was significantly (50 percent) less than the inviscid calculation and in excellent agreement with the experiment. Rizzi was pleased with these improved results and also with the fact that the addition of the viscous terms increased the CPU time by only 50 percent. An important conclusion which can be drawn from this work is that viscous effects play a much more important role in establishing the proper qualitative and quantitative behavior of delta wing flows than was previously thought and, in fact, are absolutely essential (at least for rounded leading edge wings) in order to establish the proper surface flow topology and static pressure distribution.

Rizzi then turned his attention to the detailed structure of the primary vortex, as revealed by his high-resolution (more than 1 million point) Euler calculation. The fundamental problem which arises in attempting to extract the detailed structure of the vortex from an Euler calculation is that, as the core of the vortex is approached, it becomes impossible to control the amount of numerical error which is present (due to the velocity gradient, which approaches infinity at the core of the vortex). Thus, a truly inviscid simulation of the vortex structure is impossible. Near the outer boundary of the vortex, however, the Euler calculations are relatively free from numerical dissipation and a more or less clearly defined vortex sheet can be distinguished from the calculations. As the vortex sheet spirals inward, however, the identity of the sheet is lost and diffused core is produced, the calculation of which is dominated by the effects of numerical viscosity.

It is surprising that the loss in total pressure predicted by Rizzi's

calculations agrees so well with the experimentally determined value. In a calculation of a 75° swept wing at an incidence of 10° and a Mach number of 1.95, Rizzi found a total pressure loss of 70 percent compared with the experimental value of 75 percent. Moreover, the magnitude of the loss in total pressure appeared to be independent of the level of numerical dissipation, a result subsequently confirmed by Murman (*ESN* 41-1:22 [1987]).

### 3 LARGE-SCALE COMPUTING OF AEROSPACE PROBLEMS

In separate but related presentations, P. Perrier and J. Periaux, both of Avions Marcel Dassault-Bréguet Aviation (AMD-BA), reviewed recent progress in algorithm development and computer design which have allowed computational fluid dynamics (CFD) to assume the vital role which it now plays in the design of today's aerospace vehicles. In 20 years, CFD has grown from zero to nearly 50 percent of the development cost of new aerospace vehicles. We have now reached the point where  $10^{12}$  operations can be completed in a 10-hour day and this, he said, is the threshold at which we can begin to obtain accurate simulations of the flow fields about physically realistic aerospace vehicles. Thanks to the strong interaction which has existed at AMD-BA with fundamental research organizations, such as Office National d'Etudes et de Recherches Aérospatiales (ONERA), Institut National de Recherche en Informatique et en Automatique (INRIA), and various European universities, fast and accurate solution of large sets of nonlinear partial differential equations is now possible. Although the details of the technique vary depending upon the application (aircraft, turbomachinery, missiles, or reentry vehicles), it is AMD-BA's position that the only method which is capable of the drastic mesh adaptations required to economically simulate relativistic aerospace vehicle flow fields is a non-structured finite element algorithm using domain decomposition and multigrid acceleration.

Periaux described several impressive examples of aerospace applications illustrating the current status of CFD technology beginning with the calculation of potential flow over a complete transport aircraft done more than 10 years ago, two- and three-dimensional Navier-Stokes simulations of transfer inlet flow calculations of the flow over a complete military and commercial aircraft, and calculations of three-dimensional reentry bodies. Despite the convincing nature of the calculation of the local static pressure, for example, upon closer examination the prediction of other quantities is not as satisfactory. For example, he showed the calculation of the flow in the vicinity of the wing root of a subsonic transport which exhibited an unwanted, and physically unrealistic, entropy increase. This can be traced to inadequate grid resolution. Similar difficulties occur in attempts to model the curvature of engine inlets and exhaust nozzles. Periaux concluded that although there is good reason to be excited about the design of possibilities of today's CFD methods, much research remains to be done on development of accurate algorithms to make optimum use of the current generation of supercomputers.

In the US, finite volume codes are much more common than finite element codes. S. Chakravarthy of Rockwell International (US) presented the results of his Euler and Navier-Stokes calculations using a class of numerical methods based on total variation diminishing (TVD) formulation. In the past 12 months he has imbedded this method into a unified code which has the capability of doing multizone, multigrid, and either explicit or implicit, time or space marching calculations. The results of his three-dimensional Euler calculations for supersonic flow over a complete shuttle/external tank/solid rocket booster assembly produced static pressures which agreed very well with flight test data. Calculations were also shown for a generic hypersonic transport configuration which involved the solution of the two-dimensional Navier-Stokes equations. The calculations were done for a Mach number of 25 and featured a combination of time

marching at the nose, patched to space marching for the remainder of the flow field. Having now obtained a comprehensive code for aerodynamic simulation, Chakravarthy's next step is to combine it with propulsion, structure, acoustics, and radar cross section codes in order to develop a truly comprehensive CFD design package.

J. Chattot of MATRA (France) described work undertaken to meet a much more modest objective--the development of an aerodynamic design code for high angle of attack missile operation. The basis of Chattot's code is an unsteady finite-volume structural mesh, Euler calculation structured meshes employing Lax-Wendroff-Ni integration procedure. Use of an explicit integration procedure, Chattot said, was motivated by his desire to facilitate the use of multi-zone solution techniques. As in Chakravarthy's code, multigrid was used to enhance convergence rate. The results showed that normal force and pitching moments were well predicted for models of considerable geometric complexity up to an angle of attack of  $15^\circ$ . To improve the predictions, a collaborative effort is being undertaken with ONERA to couple the code with an unsteady boundary layer calculation. Further details concerning this work can be found in (NSN 41-4:215 [1987]).

ONERA has also developed a series of Euler codes for the simulation of complex three-dimensional flows past aerodynamic configurations based on the Lax-Wendroff procedure. This work was described by P. Morice of ONERA-Chatillon. Each of the codes is well suited to a specific field of application, namely, low speed, transonic, or hypersonic flows. Application of the new three-dimensional Euler code to the calculation of a supersonic elliptical jet using jet surface "fitting" was described. In these calculations the surface of the jet was moved in accordance with appropriate characteristic relationships in order to match the prescribed constant-pressure boundary condition. A surprising amount of three-dimensionality was observed in the calculations for the relatively modest eccentricity of the nozzle cross section.

For supersonic and hypersonic applications ONERA has developed a second order of accurate, explicit, upwind (Van Leer-type) flux-split, nodal scheme. At the present time, real-gas equilibrium calculations are included, and work is underway to add the effects of chemical reaction. Two applications were shown, one to a rectangular supersonic jet exhausting into a supersonic cross stream and another involving a missile at a  $45^\circ$  negative angle of attack. For the jet calculation, a  $35 \times 40 \times 45$  grid was used, which revealed a considerable amount of fluid mechanics detail in the free-stream/jet interaction region. In particular, the expansion of the free stream around the back of the jet was well resolved, as was the cross-stream flow, as evidenced by cross-flow velocity contours plotted at several downstream occasions. The grid refinement procedure appeared to be far less sophisticated than that contained in AMD-BA's finite element codes. Morice reported that work is presently underway to add such a capability to the program as well as to develop improved turbulence modeling and transition criteria for the eventual viscous extension of these calculations. The objective, like that described by Chakravarthy, is to develop reliable aerodynamic simulation codes for industrial design use.

#### 4 CONVERGENCE ACCELERATION METHODS

Perhaps the most popular convergence acceleration method these days is the multigrid method. To set the stage for subsequent sessions on multigrid theory and application, a plenary session was organized in which an invited paper reviewing recent progress in multigrid theory was presented by W. Hackbusch of the Christian Albrechts University (Kiel, West Germany).

The problem with using a single grid, Hackbusch said, is that low frequency errors in the numerical solution are difficult to remove. The essence of the multigrid concept is that an error of wavelength  $\lambda$  is most easily eliminated on a mesh of size  $h$ , where  $\lambda$  is approximately equal to  $h$ . In view of this, a given

flow problem is solved on a sequence of grids in which the low-frequency error, when viewed from the perspective of a grid of increased coarseness, appears to be of higher frequency and therefore, easier to remove. By subsequent repetitions of a process in which a solution is sought on coarse grids and the results transferred back to serve as the first approximation for the solution on the next finest grid for the next sequence of calculations, dramatically enhanced convergence rates can be achieved. Although this concept has been around for some time, it has been implemented in a modern computational context by A. Brandt and placed on a firm theoretical basis only within the past decade. The past several years have seen the application of multigrid methods to the solution of elliptic, parabolic, and hyperbolic problems, singular perturbation problems, and integral equations of the second kind. Of particular interest to Hackbusch is the development of robust methods where convergence enhancement effects are achieved independent of mesh spacing and the geometry of the computational region. Particular difficulties occur in the application of multigrid methods to two-dimensional regions where high levels of anisotropy occur in both coordinate directions. For this he proposed the use of a new multigrid method and a novel prolongation method employing four contiguous grids.

A large number of the presentations focused on various aspects of multigrid refinements, such as the relative effectiveness of various solution techniques (called "smoothers" in multigrid terminology) and various methods for transferring coarse grid solution into the next finer mesh ("prolongation"). The most interesting engineering application of the multigrid method was presented by P. H. Gaskell of the University of Leeds (UK). He described a multigrid calculation for the flow in a lid-driven and a thermally driven cavity using a finite volume, staggered grid approach. In addition to showing the effectiveness of multigrid (a time savings of more than a factor of 100 was realized) the calculations showed the superiority of the quad-

ratic upwind interpolation (QUICK) scheme in contrast with the hybrid algorithm; it also showed the advantage of using symmetrical coupled Gauss-Seidel (SCGS) iterations in comparison with the uncoupled iteration technique in which an independent "pressure-correction" calculation is made. Gaskell found that multigrid, in combination with SCGS and the QUICK discretization scheme, allowed him to attain the nearly linear dependence of computational speed on the number of grid points that one would expect from a classical multigrid method. Unfortunately, in regions of high shear (for example, near the moving wall in the lid-driven cavity case) the unbounded characteristic of the QUICK scheme causes difficulty. This was seen in calculations of a turbulent lid-driven cavity case where unphysical oscillations produced negative values of the turbulence kinetic energy and would have caused the solution to diverge if no corrective action been taken. To overcome this difficulty, Gaskell has developed what he calls his Curvature Compensated Convective Transport (CCCT) discretization. It fulfills what he has identified as the minimum and necessary requirement for boundedness which he has called the Convection Boundedness Criterion (CBC). Unlike hybrid discretization, however, CCCT assures a bounded solution without the overwhelming numerical dissipation which is associated with the hybrid method. Further information about this work can be found in Gaskell and Lau (1988). Currently, work is underway to replace the point-by-point SCGS solver with a line-by-line technique and the use of quadratic interpolation for the prolongation and restriction operators which would further improve the performance of the technique.

## 5 CALCULATION METHODS FOR TURBULENT FLOWS

Two classical methods for calculating turbulent flows without the use of a turbulence model are direct and large eddy simulation. Of the two methods, direct simulation is by far the most computationally intensive since it has as

its objective the numerical resolution (insofar as possible) of all relevant turbulent scales. N. Satofuka of the Kyoto Institute of Technology (Japan) described his direct simulation calculations which employed the vorticity/stream function formulation of the Navier-Stokes equations. For the discretization of the spatial derivatives in the vorticity transport equation Satofuka used the method of lines. For the solution of the vorticity equations he used a 4th-order-accurate Runge-Kutta scheme, and for the solution of the stream function equation he used a spectral method. Satofuka was particularly interested in the decay of two-dimensional homogenous isotropic turbulence. In comparing the results of his totally-explicit calculations with the more familiar pseudo-spectral calculations, he found that the vorticity distribution maps were identical and that the explicit calculations required from only 1/4 to 1/5 of the time required for the pseudo-spectral calculations. His calculations, which were carried out at a Reynolds number of 25,000 and 60,000 were made with meshes of  $128 \times 128$ ,  $512 \times 512$  and  $1024 \times 1024$  points.

The second approach, where the small scales are modeled and only the large-scale turbulence is calculated, is called large eddy simulation. An example of such a calculation was presented by J.P. Chollet of the Institut de Mécanique de Grenoble (France). The first case which Chollet described was a two-dimensional incompressible plane mixing layer. To visualize the flow, he used both isovorticity maps and color contours illustrating the convection of a passive scalar. To simulate the effects of three-dimensional disturbances, which were, of course, absent in this two-dimensional calculation, he introduced random, small-scale perturbations in the subgrid scale model. These were responsible for initiating the Kirchoff-Taylor instability in the mixing layer from which there followed the appearance of discrete vortices and subsequent vortex pairing. Development of an appropriate subgrid scale model is by no means a closed issue and Chollet presented two methods for doing this, one which made use of a 5/3 power

decay of turbulence kinetic energy with wave number, and the other (and far more computationally intensive) used the EDQNM closure model. In his review of the renormalization group (RNG) theory, V. Yakhot, Princeton University (New Jersey) indicated his support of the large eddy simulation philosophy. He said that he believed that no one could *ever* afford to compute a direct simulation of an aerodynamic flow at full-scale Reynolds number. RNG, he believes, offers an ideal subgrid scale model for large eddy simulations. RNG allows the evaluation of transport coefficients and transport equations for the large-scale (small wave number) modes by dynamic scaling and invariance together with iterated perturbation methods. He reported on the use of RNG in conjunction with a large eddy simulation in a calculation involving  $32^3$  points at (wall unit) Reynolds numbers of 460 to 1000. (A comparable direct simulation would have required almost 10 million points!) The RNG method performs extremely well in the near-wall region where it is particularly useful in unsteady problems where the log law cannot be expected to hold. In such situations the RNG theory provides excellent prediction of such quantities as the phase lag and wall shear stress. Excellent predictions of the convective heat transfer at the wall can also be obtained.

Yet another approach involves modeling turbulent velocity fields with stochastic partial differential equations and stochastic boundary conditions. This approach was described by J. Hunt of the University of Cambridge (UK). In particular, Hunt talked about the so-called Largevin models in which the form of the particle acceleration spectrum is provided by random walk or random Fourier modes. This, Hunt said, provided a "cheap" turbulence model, and he described conclusions which could be drawn from such calculations.

## 6 LATTICE GAS METHODS

Unlike the classical approach in which the fluid is treated as a continuum, the lattice gas method represents the

fluid as a collection of discrete fluid particles moving on a discrete physical grid, with discrete velocities in accordance with specific interaction rules and conservation laws. In other words, instead of writing the Navier-Stokes equations for a continuous medium and then solving them by a discretization process, one obtains discretized equations of motion directly by discretizing the medium and its associated velocity and physical space first.

The history of the lattice gas method was reviewed by J.-P. Rivet of the Observatoire de Nice (France). In the first implementation of the lattice gas method (Hardy et al., 1973) particles were constrained to travel on a rectangular grid. This was a natural choice given the architecture of the specialized computers (cellular automata) on which these calculations were carried out. However, the resulting stress tensor when viewed from a macroscopic perspective failed to exhibit the isotropy which must be observed if correspondence with the Navier-Stokes equations is to be obtained. In 1986, however, Frisch (Frisch et al., 1986) discovered that the desired isotropy properties could be recovered if a triangular (hexagonal) instead of a rectangular grid were used (see Figure 2).

Recent theoretical developments have eliminated other restrictions of the early lattice gas formulations. Early models were unable to accommodate thermal

effects (such as the adiabatic compression experienced in shock waves) due to the existence of a single particle translation velocity on the matrix. The development of more flexible collision rules has avoided this problem. Another advance has been the calculation of three-dimensional flows. Up until the past few months, this seemed to be impossible since no three-dimensional lattice was known to exhibit the required isotropy. An ingenious solution was found for this problem by d'Humières (d'Humières et al., 1986) in which a three-dimensional solution was found as the projection of an isotropy-preserving four-dimensional one.

Another recently solved problem was that the lattice gas equivalent of the Navier-Stokes equations failed to exhibit Galilean invariance. That is, the nature of the lattice gas equations was changed when they were transformed to a uniformly translating frame of reference. This can be traced to a nonunitary value of the parameter which is used to scale the values of time. By means of a new lattice gas model employing "rest" particles, d'Humières and Lallemand (1986) were able to increase this value to unity and thus restore the desired invariance. Of a more fundamental nature appears to be the limitation of the lattice gas method to low Mach number, low Reynolds number flows. The low Mach number limitation arises due to the necessity of neglecting terms which depend upon the Mach number if correspondence with the Navier-Stokes equations is to be achieved. The low Reynolds number limitation results from the necessity of limiting the number of particles participating in the collision processes in order to have a workable set of interaction rules. Modest increases in the Reynolds number are possible by the introduction of rest particles, however at the present time it is unclear how simulations at Mach numbers more than a few tenths and Reynolds number of more than a few thousand can be achieved.

Progress in the past 2 years has been spectacular with lattice gas simulations being made to investigate buoyancy effects, calculate seismic P-waves, model magnetohydrodynamic flows, solve reaction-diffusion problems, and model

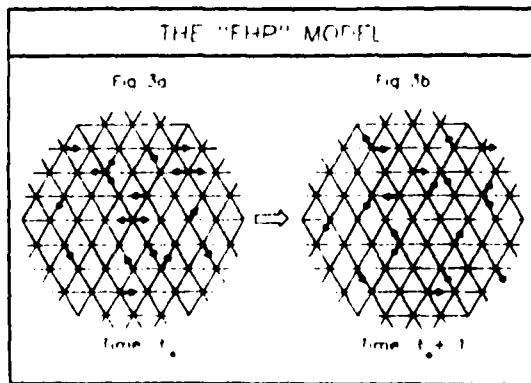


Figure 2. Interaction Rules of the FHP Model. (Frisch, et al., 1986)

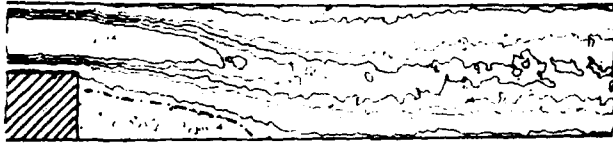


Figure 3. Lattice Gas Simulations of the Flow over a Backward Facing Step. (Reynolds Number = 150).

interface and combustion phenomena. P. Lallemand of the Laboratoire de Physique de l'Ecole Normale (Paris) described several recent applications including the calculation of developing Poiseuille flow at the entrance of a rectangular channel at a Reynolds number of 200 (see Figure 3), flow over a backward facing step at a Reynolds number of 150, flow around a flat plate at a Reynolds number of 90, the occurrence of Kelvin-Helmholtz instability in planar shear flow, and the appearance of Rayleigh-Taylor waves in a buoyancy-driven flow.

## 7 CONCLUSIONS

The contributions which applied mathematics have made to the field of computational fluid mechanics particularly in the fields of discrete vortex methods, multigrid methods, and lattice gas hydrodynamics were in clear evidence throughout the meeting. Computational methods have become a very popular technique among applied mathematicians, and numerous fruitful collaborations between the mathematical and fluid mechanics communities have resulted in the development of new algorithms and the development of enhanced convergence procedures for existing ones. Much yet remains to be done, particularly in developing innovative theories (such as the renormalization group theory) for efficient calculation of turbulent flows and methods for the calculation of turbulent flows with reaction. It will be interesting to see how much progress has been made on these topics when the second international conference on industrial and applied mathematics is held in 4 years.

## 8 REFERENCES

- Clouqueur, A., and D. d'Humières, "R.A.P.1, a Cellular Automaton Machine for Fluid Dynamics," *Proceedings of the Workshop Modern Approaches to Large Non-Linear Systems*, (Santa Fe, 26-29 October 1986; to appear in *Complex Systems*).
- d'Humières D., and P. Lallemand, "Numerical Simulations of Hydrodynamics with Lattice Gas Automata in Two Dimensions," *Proceedings of the Workshop Modern Approaches to Large Non-linear Systems*, (Santa Fe, 26-29 October 1986; to appear in *Complex Systems*).
- d'Humières, D., P. Lallemand, and G. Searby, "Numerical Experiments on Lattice Gases Mixtures and Galilean Invariance," *Proceedings of the Workshop Modern Approaches to Large Nonlinear Systems*, (Santa Fe, 26-29 October 1986; to appear in *Complex Systems*).
- d'Humières, D., P. Lallemand, and U. Frisch, *Europhysics Letters*, 2:291 (1986).
- Frisch, U., D. d'Humières, B. Hasslacher, P. Lallemand, Y. Pomeau, and J.F. Rivet, "Lattice Gas Hydrodynamics in Two and Three Dimensions," *Proceedings of the Workshop Modern Approaches to Large Nonlinear Systems*, (Santa Fe, 26-29 October 1986; to appear in *Complex Systems*).
- Frisch, U., B. Hasslacher, and Y. Pomeau, *Physics Review*, 56:1505 (1986).
- Gaskell, P.H., and A.K.C. Lau, "Curvature Compensated Convective Transport: SMART, A New Boundedness Preserving Transport Algorithm," accepted for publication in: *International Journal for Numerical Methods in Fluids*, (1988).
- Hardy, J., Y. Pomeau, and O. de Pazzis, *Journal Mathematical Physics*, 14:1746 (1973).
- Peyret, R., and T.D. Taylor, *Computational Methods for Fluid Flow*, (Springer-Verlag Inc., New York 1983).

REPORT DOCUMENTATION PAGE				
1a REPORT SECURITY CLASSIFICATION <b>UNCLASSIFIED</b>		1b RESTRICTIVE MARKINGS		
2a SECURITY CLASSIFICATION AUTHORITY		3 DISTRIBUTION AVAILABILITY OF REPORT <b>Approved for public release; distribution unlimited</b>		
2b DECLASSIFICATION/DOWNGRADING SCHEDULE		4 PERFORMING ORGANIZATION REPORT NUMBER(S) <b>7-030-C</b>		
5 NAME OF PERFORMING ORGANIZATION <b>Office of Naval Research Branch Office, London</b>		6a OFFICE SYMBOL <i>(If applicable)</i> <b>ONRBRO</b>	7a NAME OF MONITORING ORGANIZATION	
6b ADDRESS (City, State, and ZIP Code) <b>Box 39 FPO, NY 09510</b>		7b ADDRESS (City, State, and ZIP Code)		
8a NAME OF FUNDING SPONSORING ORGANIZATION		8b OFFICE SYMBOL <i>(If applicable)</i>	9 PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER	
8c ADDRESS (City, State, and ZIP Code)		10 SOURCE OF FUNDING NUMBERS		
		PROGRAM ELEMENT NO	PROJECT NO	TASK NO
				WORK UNIT ACCESSION NO
11 TITLE (Include Security Classification) <b>International Conference on Industrial and Applied Mathematics</b>				
12 PERSONAL AUTHOR(S)				
13a TYPE OF REPORT <b>Conference</b>	13b TIME COVERED FROM _____ TO _____	14 DATE OF REPORT (Year, Month, Day) <b>13 October 1987</b>	15 PAGE COUNT <b>10</b>	
16 SUPPLEMENTARY NOTATION				
17 COSATI CODES		18 SUBJECT TERMS (Continue on reverse if necessary and identify by block number)		
FIELD	GROUP	SUB GROUP		
<b>20</b>	<b>04</b>		<b>Multigrid</b>	
<b>01</b>	<b>01</b>		<b>Lattice Gas</b>	
			<b>Viscous Flows</b>	
			<b>Turbulence</b>	
			<b>Vortex Flows</b>	
19 ABSTRACT (Continue on reverse if necessary and identify by block number) <p>This conference, held in July 1987 in Paris, France, gave clear evidence of the contribution of applied mathematics to the field of computational fluid mechanics, particularly in the fields of discrete vortex methods, multigrid methods, and lattice gas hydrodynamics. Selected presentations are reviewed.</p>				
20 DISTRIBUTION AVAILABILITY OF ABSTRACT <input checked="" type="checkbox"/> UNCLASSIFIED UNLIMITED <input type="checkbox"/> SAME AS RPT <input type="checkbox"/> DTIC USERS		21 ABSTRACT SECURITY CLASSIFICATION <b>UNCLASSIFIED</b>		
22a NAME OF RESPONSIBLE INDIVIDUAL <b>C.J. Fox</b>		22b TELEPHONE (Include Area Code) <b>(44-1) 409-4340</b>	22c OFFICE SYMBOL <b>310</b>	



Distribution/Availability Codes  
Avail and/or Special  
**A-1**

**DAI  
FILM  
2**