R&D No 6944-AN-03
DAJA45-92-M-0240

NATO Advanced Research Workshop

Vortex Flows and Related Numerical Methods
Ecoulements tourbillonnaires et méthodes numériques

Grenoble-St Pierre de Chartreuse
June 15-19, 1992

List of Participants /*Liste des participants*

Book of Abstracts / *Résumés*

Instructions for submission to the proceedings / *Format pour les actes*

Supported by / *Soutenu par*
DRET, CNRS, Ministère de l'Education Nationale, IMAG. Cray France, US Army, UJF, UH

92

NATO Advanced Research Workshop

Vortex Flows and Related Numerical Methods
Ecoulements tourbillonnaires et méthodes numériques

Grenoble-St Pierre de Chartreuse
June 15-19, 1992

List of Participants /*Liste des participants*

Book of Abstracts / *Résumés*

Instructions for submission to the proceedings / *Format pour les actes*

Supported by / *Soutenu par*
DRET, CNRS, Ministère de l'Education Nationale, IMAG, Cray France, US Army, UJF, UH

Carlos Albuquerque
CMAF
Univ. Lisboa
Av. Prof Gama Pinto, 2
1699 Lisboa Codex, PORTUGAL
Tel.: (351) 1 7950790
Fax: (351) 1 765622

Fons Alkemade
Laboratory for Aero and Hydrodynamics
Rotterdamseweg 145
2628 AL Delft, HOLLAND
Tel.: (015) 782961
Fax: (015) 782947
e-mail: fons@dutwg.tudelft.nl

Christopher Anderson
Department of Mathematics
University of California
Los Angeles, CA 90024, USA
Tel.: (310) 825 12 98
Fax: (310) 206 66 73
e-mail: anderson@math.ucla.edu

Pierre Baras
Laboratoire de Modélisation et de Calcul
Université Joseph Fourier, B.P. 53x
38041 Grenoble Cédex
Fax: 76 51 45 55
e-mail: baras@imag.fr

Claude Basdevant
LMD
Ecole Normale Supérieure
24, rue Lhomond
75231 Paris Cédex 05, FRANCE
Tel.: (1) 44 32 22 27
Fax: (1) 43 36 83 92
e-mail: basdevan@lmd.ens.fr

J. Thomas Beale
Department of Mathematics
Duke University
Durham, NC 27706, USA
Tel.: (919) 660 28 39
e-mail: beale@math.duke.edu

Christine Bernier
Département de Mathématiques
Université de Nancy 1
B.P. 239
54500 Vandoeuvre, FRANCE
Tel.: 83 91 21 52
e-mail: bernier@iecn.u-nancy.fr

Andrzej Bogustawski
LEMD-CNRS
25, Av. des Martyrs
B.P. 166x, 38042 Grenoble, FRANCE

Nathalie Bonneton
CERFACS
42, Av. Coriolis
31100 Toulouse, FRANCE
Tel.: 61 19 30 51
Fax: 31 19 30 00
e-mail: bonneton@orion.cerfacs.fr

Thomas F. Buttke
Institute for Advanced Study
Olden Lane
Princeton, NJ 08540, USA
e-mail: buttke@math.ias.edu

Emanuele Caglioti
Dipatimento Matematica
Universita di Roma 'La Sapienza'
00185 Roma, ITALIE
Fax: (6) 49 91 32 87

C. Carasso
Equipe d'Analyse Numérique
Faculté des Sciences
23, rue du Docteur Paul Michelon
42023 Saint Etienne, FRANCE
Fax: 77 25 60 71
e-mail: carasso@stroph.univ-st-etienne.fr

Mr. Cassot
INSERM
CHU Purpan
31059 Toulouse, FRANCE
Tel.: 61 49 89 26
Fax: 61 49 95 24

Thierry Charvet
LIMSI
B.P. 133
91403 Orsay, FRANCE
Tel.: (1) 69 85 81 30
Fax: (1) 69 85 80 88

Jean-Pierre Chollet
LEGI
B.P. 53x
38041 Grenoble Cédex, FRANCE
Tel.: 76 82 50 75
Fax: 76 82 50 85
e-mail: chollet@img.fr

Pierre Comte
LEGI
B.P. 53x
38041 Grenoble Cédex, FRANCE

Georges-Henri Cottet
Laboratoire de Modélisation et de Calcul
Université Joseph Fourier, B.P. 53x
38041 Grenoble Cédex
Tel.: 76 51 49 92
Fax: 76 51 45 55
e-mail: cottet@imag.fr

Olivier Daube
LIMSI
B.P. 133
91403 Orsay, FRANCE
Fax: (1) 69 85 80 88

David G. Dritschel
University of Cambridge
Silver Street
Cambridge CB3 9EW, G. B
Tel.: (223) 33 78 66
Fax: (223) 337918
e-mail: dgd11@phx.cam.ac.uk

Mr. Elsaesser
CNIM, Service Calcul Scientifique
Z.I. de Brégaillon, B.P. 208
83507 La Seyne, FRANCE
Tel.: 94 30 32 88
Fax: 94 30 31 00

Yves Gagnon
Centre Universitaire Saint Louis Maillet
Edmunton
New Brunswick, CANADA
Tel.: (506) 737 50 50
Fax: (506) 739 53 73

B.P. Gerasimov
The keldysh Inst. of Applied Sciences
4, Miusskaya Pl.
125047 Moscou, RUSSIA
Fax: (095) 972 07 37
e-mail: user@keldysh.msk.su

Thomas Y. Hou
Courant Institute
251 Mercer Street
New York, NY 10012, USA
Tel.: (212) 998 31 36
Fax: (212) 995 41 21
e-mail: hou@math14.nyu.edu

Serge Huberson
Laboratoire de Mécanique des Fluides
Université du Havre
2, rue Philippe le Bon
76000 Le Havre, FRANCE
Tel.: 35 53 69 37
Fax: 35 43 24 47

Fazle Hussain
Mechanical Engineering Department
University of Houston
4800 Calhoun Street
Houston, TX 77204-4792, USA
Tel.: (713) 743 45 45
Fax: (713) 743 45 03
e-mail: mecelw@jetson.uh.edu

M. Kermarec
IMFM, Souffleries de Luminy
163, Av. de Luminy
13009 Marseille, FRANCE
Tel.: 91 26 60 30
Fax: 91 41 16 91

Omar Knio
Dept of Mechanical Engineering
The Johns Hopkins University
Baltimore, MD 21218, USA
Tel.: (410) 516 77 36
Fax: (410) 516 7254
e-mail: knio@polaris.me.jhu.edu

Petros Koumotsakos
Graduate School of Aeronautics
CALTECH
Pasadena, CA 91125, USA
Tel.: (818) 356 44 62
Fax: (818) 449 26 77
e-mail: petros@frege.galcit.caltech.edu

Robert Krasny
Mathematics Department
University of Michigan
Ann Arbor, MI 48109, USA
Tel.: (313) 763 50 48
Fax: (313) 763 09 37
email: krasny@math.lsa.umich.edu

Henryk Kudela
Technical University of Wroclaw
Wybizeze S. Wyspianskieg 27
50-370 Wroclaw, POLAND
Tel.: (71) 20 25 54
Fax: (71) 22 36 64

E. Langlois
LIMSI
B.P. 133
91403 Orsay, FRANCE
Fax: (1) 69 85 80 88

John Lowengrub
Institute for Advanced Study
Olden Lane
Princeton, NJ 08540, USA

George Majda
Ohio State University
231 W 18th Avenue
Colombus, OH 43210-1174, USA
Tel.: (614) 397 46 24
Fax: (614) 292 14 79
e-mail: majda@shape.mps.ohio-state.edu

Carlo Marchioro
Dipartimento Matematica
Universita 'La Sapienza'
Piazzale Aldo Moro, 2
00185 Roma, ITALIE
Tel.: (6) 49 91 32 15
Fax: (6) 49 91 32 87

Sylvie Mas-Gallic
Centre de Mathématiques Appliquées
Ecole Polytechnique
91128 Palaiseau Cédex, FRANCE
e-mail: masgalli@cmapx.polytechnique.fr

Eckart Meiburg
Department of Aerospace Engineering
University of Southern California
Los Angeles, CA 90089-1191, USA
Tel.: (213) 740 53 76
Fax: (213) 740 77 74
e-mail: eckart@spock.usc.edu

M. Meneguzzi
CERFACS
42, Av. de Coriolis
31057 Toulouse, FRANCE
Tel.: 61 16 30 50
Fax: 61 16 30 30
e-mail: mngz@cerfacs.fr

Mathieu Mory
LEGI
B.P. 53x
38041 Grenoble Cédex, FRANCE
Tel.: 76 82 50 60
Fax: 76 82 50 01

Habib N. Najm
Texas Instruments
P.O. Box 743461
Dallas, TX 75374, USA
Tel.: (214) 995 52 61
Fax: (214) 995 19 16
e-mail: najm@spdc.ti.com

Monica Nitsche
Department of Mathematics
University of Michigan
Ann Arbor, MI 48104, USA
Tel.: (313) 769 11 42
Fax: (313) 763 09 37
e-mail: nitsche@math.lsa.umich.edu

Paolo Orlandi
Dipartimento di Meccanica e Aeronautica
Universita di Roma
via Eudossiana 16
00184 Roma, ITALIE
Tel: (6) 44 58 58 78
Fax: (6) 48 48 54
email: orlandi@orlandi.ing.uniroma.it

Philippe Pascal
rue de la Haie
57000 Metz, FRANCE

Valérie Perrier
LMD-Ecole Normale Supérieure
24, rue Lhomond
75231 Paris Cédex 05, FRANCE
Tel.: (1) 44 32 22 25
Fax: (1) 43 36 83 92
e-mail: perrier@lmd.ens.fr

Renzo Piva
Dipartimento di Meccanica e Aeronautica
Universita di Roma
via Eudossiana 18
00184 Roma, ITALIE
Tel.: (6) 44 58 52 25
Fax: (6) 48 48 54

Roy Reichenbach
US Army RSDG
223 Old Marylebone road
London NW15th, ENGLAND
Tel.: (071) 409 4423
Fax: (071) 724 1433

Giorgio Riccardi
Dipartimento di Meccanica e Aeronautica
Universita di Roma
via Eudossiana 18
00184 Roma, ITALIE
Fax: (6) 48 48 54

Raoul Robert
21 Av de la Plaine Fleurie
38240 Meylan, FRANCE

Frédéric Rouffi
LIMSI
B.P. 133
91403 Orsay Cédex, FRANCE
Tel.: (1) 69 85 81 28
Fax: (1) 69 85 80 88

Elie Rivoalen
Laboratoire de Mécanique des fluides
Centre havrais d'études et de recherches
B.P. 265
76055 Le Havre Cédex, FRANCE
Tel.: 35 53 69 50
Fax: 35 53 97 03

A. Zervos
Commission of the European Community
200 Rue de la Loi
B-1049, Bruxelles, BELGIQUE

Tentative Programme / *Programme provisoire*

Monday, June 15th

8:45-9:00 Opening address / *Ouverture* (G-H. Cottet)

9:00-9:50 M. Mory (LEGI, Grenoble): Vortex generation by suction in a rotating tank

10:00-10:50 C. Anderson (UCLA, Los Angeles): Computational investigation of vortex shedding

11:00-11:30 Coffee Break / *Pause café*

11:30-12:20 J.T. Beale (Duke University): Viscous splitting of the Navier-Stokes equations with boundaries

12:30 Lunch / *Déjeuner*

16:00-16:30 O. Daube (LIMSI, Orsay): Vorticity on the boundaries for the 2d Navier-Stokes equations in velocity-vorticity formulation

16:30-17:00 S. Mas-Gallic (Polytechnique, Palaiseau): A Particle in Cell Method for the compressible Euler equations

17:00-17:30 N. Bonneton (CERFACS, Toulouse): Simulation numérique des structures organisées d'un jet plan compressible

17:30-18:00 Coffee Break / *Pause café*

18:00-18:30 H. Najm (Texas Instr., Dallas): A hybrid vortex method with deterministic diffusion

18:30-19:00 M. Kermarec (IMFM, Marseille): A coupled potential-boundary layer calculation method for unsteady flows around airfoils

19:30 Dinner / *Diner*

Tuesday, June 16th

9:00-9:50 E. Meiburg (USC, Los Angeles): 3d vortex dynamics in free shear flows

10:00-10:50 P. Comte (LEGI, Grenoble): Vortex generation and evolution in numerical simulation of transitional shear flows

11:00-11:30 Coffee Break / *Pause café*

11:30-12:20 M. Meneguzzi (CERFACS, Toulouse): Dynamics of vortex tubes in 3d turbulence

12:30 Lunch / *Déjeuner*

16:00-16:50 R. Krasny (Ann Arbor): Vortex sheet computations

17:00-17:30 M. Nitsche (Ann Arbor): Axisymmetric vortex sheet roll-up

17:30-18:00 Coffee Break / *Pause café*

18:00-18:30 G. Majda (Ohio State Univ.): Particle approximation of the Vlasov-Poisson equations

18:30-19:00 E. Caglioti (Univ. La Sapienza, Roma): Statistical Physics of point vortices

19:30 Dinner / *Diner*

Wednesday, June 17th

9:00-9:50 C. Marchioro (Univ. La Sapienza, Roma): Point vortices and localization in Euler flows

10:00-10:50 R. Robert (CNRS, Lyon): Un modèle d'écoulement turbulent bidimensionel

11:00-11:30 Coffee Break / *Pause café*

11:30-12:20 C. Basdevant (ENS, Paris): Analysis and numerical simulation of turbulent flows using wavelets

12:30 Lunch / *Déjeuner*

Free Afternoon

19:30 Dinner / *Diner*

Thursday, June 18th

9:00-9:50 R. Piva (Univ. La Sapienza, Roma): A slightly diffusive contour dynamics

10:00-10:50 D. Dritschel (Cambridge): Simulation of 2d turbulence at near-infinite Reynolds number

11:00-11:30 Coffee Break / *Pause café*

11:30-12:20 T. Buttke (Courant Institute, New York): Magnetization methods

12:30 Lunch / *Déjeuner*

16:00-16:50 F. Hussain (Univ. Houston): New aspects of vortex dynamics by numerical simulation: core dynamics, helical wave dynamics, and vortex fine-scale turbulence interaction

17:00-17:30 F. Alkemade (Delft Univ. of Technology): The vorton method: theory and application

17:30-18:00	Coffee Break / *Pause café*
18:00-18:50 vortex	P. Orlandi (Univ. La Sapienza, Roma): 3d simulation of free vortex rings and rings interacting with a wall
19:00-19:30	P. Pascal (ONERA, Chatillon): Modélisation d'un tourbillon en écoulement turbulent incompressible
19:30	Dinner / *Diner*

Friday June 19th

9:00-9:50	O. Knio (Johns Hopkins Univ., Baltimore): Stability analysis of differentially-heated asymmetric vorticity layers
10:00-10:50	J.P. Chollet (LEGI, Grenoble): Turbulent eddy structures, combustion and chemical reactions
11:00-11:30	Coffee Break / *Pause café*
11:30-12:20	B.P. Gerasimov (Keldysh Inst., Moscou): Computer simulation of fluid flows with heat and mass transfer in engineering and science
12:30	Lunch / *Déjeuner*
14:00-14:50	P. Koumotsakos (Caltech, Pasadena): Large scale simulations using vortex methods
15:00-15:30	H. Kudela (Technical Univ., Wroclaw): Vortex blob simulation of two dimensional flow in channel with complex geometries
15:30-16:00	E. Rivoalen (Univ. Le Havre): Estimation d'écoulements tridimensionnels avec une méthode particulaire à poids constants
16:00-16:30	Coffee Break / *Pause café*
16:30-17:20	T. Hou (Courant Inst., New York): Vortex method approximations for 2 phase flows with surface tension
17:30-18:00 finies	P. Cassot (INSERM, Toulouse): Etude numérique d'écoulements instationnaires autour de structures cylindriques par une méthode de superposition différences -particules.
18:00	Closing address / *Clôture*

Fons Alkemade (1), E. van Groesen (2), F.T.M. Nieuwstadt (1)

(1) Laboratory for Aero- and Hydrodynamics, Delft University of Technology, The Netherlands; (2) Faculty of Applied Mathematics, University of Twente, The Netherlands

THE VORTON METHOD: THEORY AND APPLICATIONS

The vorton method is a vortex method in which 3-D vortex flows are discretized by means of singular vortex structures (delta-functions), like in the 2-D point-vortex method [1]. However, a continuous "dipole" part has to be added in order to make the vorticity field divergencefree.

Departing from Helmholtz's vortex deformation equation, the dynamics of the vorton parameters (location and intensity of vorticity) is described by a set of nonlinear ordinary differential equations for vorton displacement and vorton deformation. A review of vorton theory is given in [2] and [3].

Several aspects of vorton theory have been or will be investigated: 1) **motion-invariants**: the vorton dynamics should show conservation of variables such as total kinetic energy and total helicity; 2) **convergence**: the vorton method solutions should converge towards the exact smooth solutions;

Some elementary vorton simulations have already been performed. If our newly derived vorton equations (submitted to Physics of Fluids) appear to give reliable and internally consistent simulations (e.g. of circular and elliptical vortex rings), it could be applied to the study of coherent structures in a turbulent boundary layer, the organized vortical structures in turbulence. For that purpose, investigation into the "chaotic" dynamics of vortons may be helpful. A convenient visualization method will be developed for this purpose, using AVS. This also gives us the possibility to study important scalars like helicity density.

Meanwhile, it is tried to derive vorton equations by making use of the fact that the 3-D Euler equations have a so-called Poisson-structure.

Along with the singular, or "hard", vortons, we also consider the theory of smoothed, or "soft", vortons.

An experiment is set up in order to study the behaviour of circular and elliptical vortex rings. Experimental results will be compared to vorton simulations.

[1] M.I. Aksman, E.A. Novikov, "Reconnections of vortex filaments". *Fluid Dynamics Research* 3(1988).

[2] F. Alkemade, "The world of vortons; origin, theory, applications, prospects". Laboratory for Aero- and Hydrodynamics, Delft University of Technology, 1992.

[3] F. Alkemade, E. van Groesen, "A new set of dynamical equations for 3-D vortex particles (vortons)". (submitted to *Physics of Fluids*).

Computational investigation of vortex shedding

Christopher Anderson

Department of Mathematics
University of California
Los Angeles, CA 90024
USA

In this talk a high order finite difference method for the solution of the incompressible Navier-Stokes equations in vorticity stream function form will be presented. We will then discuss how the numerical results obtained with this method have been used to investigate the inaccuracies associated with the use of the time-dependent Prandtl boundary layer equations to model vortex shedding - a procedure implicit in vortex sheet / vortex blob methods. We will also show how alternate computational strategies can reduce these inaccuracies.

Viscous Splitting of the Navier-Stokes Equations with Boundaries

J. Thomas Beale
Mathematics Dept.
Duke University
Durham, NC 27706 USA

June 12, 1992

We consider the approximation of the Navier-Stokes equations for incompressible flow by fractional time steps, without spatial discretization. Assume there is a smooth solution in a bounded domain in two or three space dimensions, with velocity zero on the boundary. We approximate by alternating partial steps: In the first, we solve the inviscid Euler equations with normal velocity zero on the boundary. In the second, we solve the linear Stokes equations with zero velocity on the boundary, even though the initial state for this step has tangential velocity at the boundary created during the Euler step. In joint work with Claude Greengard, of the IBM T. J. Watson Research Center, we have shown that this approximation is first-order accurate, that is, for small time step, the error has a bound proportional to the time step. The validity of such a fractional step method or splitting is presumed in some computational methods for viscous flow, including the transport-diffusion method and vortex methods. In proving the result, it seems necessary to work with low norms because of the inconsistency during the Stokes step between the initial state and the zero boundary condition.

Résumé pour le congrès "Ecoulements Tourbillonnaires et Méthodes Numériques", ayant lieu du 15 au 19 juin à Grenoble

Simulation numérique des structures organisées d'un jet plan compressible.

N. Bonneton, A. Kourta et H. HaMinh
C.E.R.F.A.C.S. , Centre Européen de Recherche et
de Formation Avancée en Calcul Scientifique
42, av. Gustave Coriolis
F-31057 Toulouse, France

Résumé

L'objectif de cette étude est l'analyse des mécanismes d'instabilité dans un jet plan compressible. La résolution de ce type d'écoulements compressibles, instationnaires, par simulation numérique directe nécessite un schéma numérique capable de prendre en compte des variations brutales des variables sans trop diffuser ni produire d'oscillations numériques importantes au voisinage de ces fronts.

Notre choix s'est porté sur la méthode de Transport par Flux Corrigés développée par BORIS et BOOK (1). C'est une méthode explicite, de différence finie, qui assure la positivité et la monotonie de certaines grandeurs physiques. Sa précision à l'ordre 4 en phase la rend tout particulièrement performante dans les régions de fortes discontinuités et de chocs. Elle effectue en tout point une pondération entre une solution du premier ordre (monotone mais diffusive) et celle du deuxième ordre (plus précise mais dispersive), afin de garantir la positivité et la monotonicité de la masse volumique. Son principe est d'ajuster localement le coefficient de diffusion en fonction du profil de vitesse et de la distribution de la variable convectée pour conserver à la fois la stabilité, la monotonicité et la précision du schéma. C'est une méthode essentiellement non linéaire.

Pour tenir compte de l'influence amont-aval en subsonique, deux types de conditions aux limites sont envisagés: le premier travaillant avec les relations dites "caractéristiques" sur les dérivées des variables, le second travaillant directement sur les variables caractéristiques ou Invariants de Riemann.

Le calcul du jet a été fait à un nombre de Reynolds basé sur le diamètre de 10000. L'instabilité des structures tourbillonnaires organisées a été bien captée comme l'illustre les isomachs figure 1 et les isovaleurs de la composante transversale de la vitesse figure 2. L'analyse de l'écoulement a permis de distinguer trois régions: La première, proche de la sortie où la vitesse sur l'axe reste sensiblement constante et égale à sa valeur à l'émission, dite "Zone à potentiel"; la seconde, loin à l'aval où les propriétés du jet obéissent à des lois de similarité relativement simples et bien connues; et la troisième "Zone d'interaction" qui sert de transition entre les deux. L'analyse spectrale réalisée sur le signal temporel de vitesse et de pression a permis de retrouver la fréquence caractéristique d'émission des structures organisées (cf figure 3).

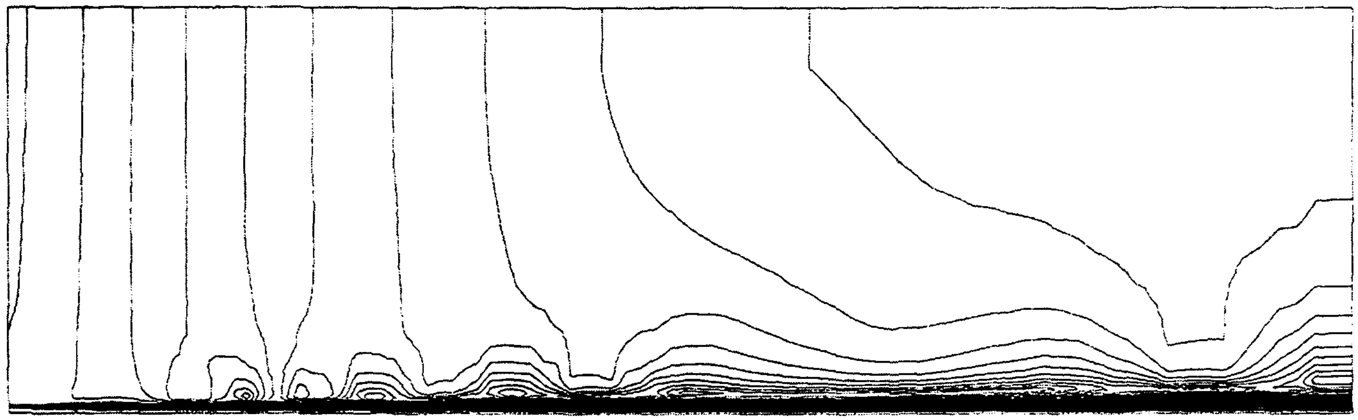


Fig.1: Isomachs

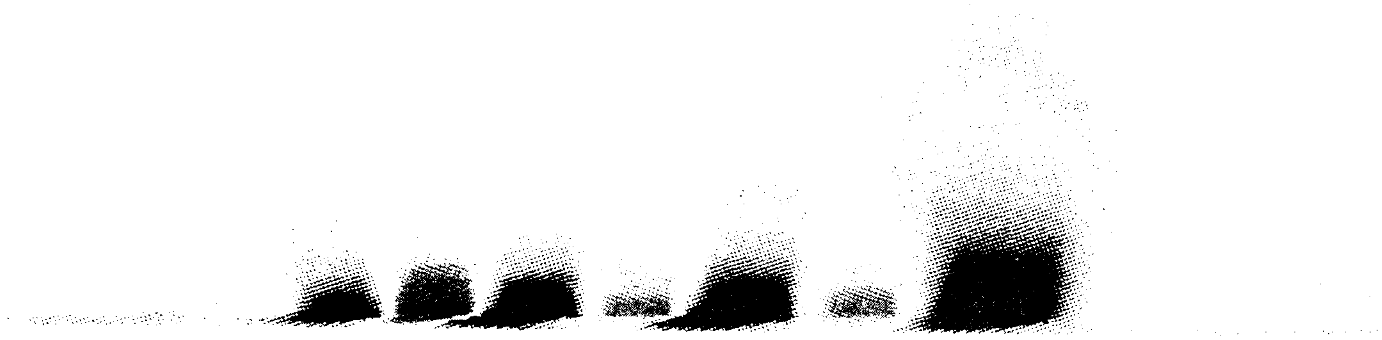


Fig.2: Isov

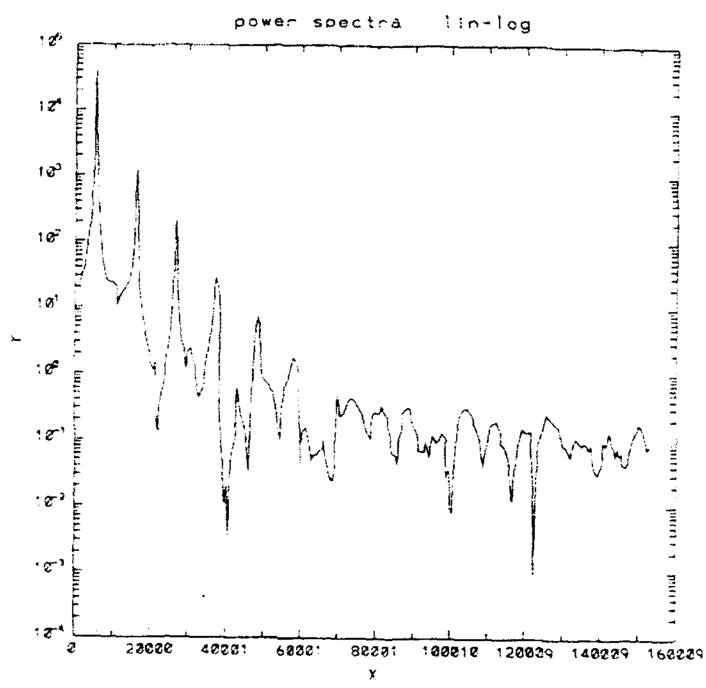


Fig.3: Spectre de la Pression, échelle lin-log

Analysis and numerical simulation of turbulent flows using wavelets

C. Basdevant

LMD-ENS
Rue Lhomond
75005 Paris

The wavelet transform decomposes a signal into contributions localized both in physical space and spectral space. For turbulent flows which exhibit a large spatial variability, it can, much better than the classical Fourier transform, separate active regions such as vortices from the background flow.This allows detailed space scale analysis; with the wavelet transform we are able to define locally in space and energy spectrum and locally in space and scale a Reynolds number. This will be illustrated by several examples. On the other hand, the wavelet transform can be used for information compression; indeed most of the non vanishing coefficients of the decomposition of a field are concentrated in active regions, when intermittency is strong, omly a few coefficients are needed to represent the field. This property is the basis of adaptive numerical methods using wavelets for numerical simulation of turbulent flows, we will review some of these methods.

Magnetization methods

Thomas Buttke

Institute for Advanced Study
Princeton, NJ 08540
USA

There is a canonical structure for incompressible fluid flow which is canonical in any number of space dimensions. We describe lagrangian numerical methods based on the canonical hamiltonian equations and present numerical results for the two and three dimensional cases. In order to describe this canonical hamiltonian structure a new variable is introduced whch we call the velocity magnetization. The velocity magnetization has units of velocity, but is related to the vorticity of the fluid. A derivation of the continuous equation of motion for the velocity magnetization in three dimensions is presented. The continuous equation of motion is discretized using a regularization kernel in such a way that the hamiltonian structure is preserved in the discrete system and, more importantly, the discrete dipoles obtained do not have an infinite self-velocity. The discrete system is developed into a lagrangian numerical method which exactly preserves the kinetic energy, impulse and angular momentum of the incompressible fluid flow. We compare the method with standard vortex methods.

We also present results of a finite difference method based on the magnetization formulation of incompressible fluid flows. The magnetization formulation has advantages over a formulation in primitive variables in that the pressure does not appear explicitly in the equation of motion for the magnetization variable. We extend the continuous equations for the case of variable density flows and discuss the extension of the methods to the variable density case.

TURBULENT EDDY STRUCTURES, COMBUSTION AND CHEMICAL REACTIONS

Jean-Pierre CHOLLET

Laboratoire des Ecoulements
Géophysiques et Industriels (I.M.G.)*
B.P. 53 X, 38041 GRENOBLE Cedex, France

Turbulence strongly affects transport and dispersion of contaminants advected by velocity fields. For a fluid made of species which undergo chemical reactions, concentration fields evolve with time and space under competing effects of flow turbulence, chemical reaction and diffusion fluxes.

The mixing layer has been selected as a prototype of shear flows which develop vortex structures from unstability mechanisms. Such vortices are observed to strongly affect mixing and then chemical activity. First, binary reactions are considered in numerical simulations of three dimensional incompressible mixing layers using pseudo spectral methods with periodic boundary conditions (i.e. temporal approximation). Reaction rate is observed to be the quantity of interest to study chemical activity and to trace some properties of turbulent fields, especially in the strained regions. Chemical activity is observed to develop along vortex structures, with increasing spatial intermittency as the reaction gets faster (higher Damkhöler number). Depending on the way vorticity fields are triggered initially, resulting species concentrations evolve differentely with more or less three dimensionality. Results from a preliminary lagrangian particle tracing agree with this general trend. Chemical reactions emphasize also the contribution from diffusivity and viscosity (Schmidt numbers) and then enlights a discussion about direct versus large eddy simulations and the need for subgrid scale models.

Combustion differs from other reacting flows because of density variations induced by strong heat release. Chemistry is complex and temperature dependent. A numerical method has been derived specifically in order to compute evolutions in time and space, even with shocks and steep temperature or concentration gradients. Wide ranges of flow velocities have been considered up to supersonic conditions. Flow structure gets rather intricate with both evolving vortex structures and non stationary shock patterns. Simulations have been developed in two problems of interest for aircraft engine design, the underexpanded jet and the mixing layer confined between rigid walls. Results from simulations agree with those classicaly derived from instability theories. Moreover simulation provides pictures of time and space distribution of vorticity along the flow. Multiple step reactions have been also considered in the simulations, assuming either species partial equilibrium or explicit evolution of species mass fractions, depending on the ratios of charatestic chemical times to turbulence time scale.

* C.N.R.S., Université J.Fourier et Institut National Polytechnique de Grenoble
tél.: (33)76 82 50 75, E-mail: chollet@img.fr

VORTEX GENERATION AND EVOLUTION IN NUMERICAL SIMULATION OF TRANSITIONAL SHEAR FLOWS

Pierre COMTE

Institut de Mécanique de Grenoble
Laboratoire des Ecoulements Géophysiques et Industriels
B.P. 53X, 38041 Grenoble-Cedex
Tel : (33) 76 82 51 21, Fax : (33) 76 82 50 01, E-Mail: Comte@img.fr

The experiments of mixing layers performed at CalTech by Roshko and his group [1] showed the presence of large-scale organized vortices even extremely far downstream of their origin. It turned out that these vortices resulted from the growth of hydrodynamic instabilities yielding roll-up of fundamental vortices and successive pairings of the latter. These trends were retrieved numerically in particular by Normand *et al.* [2], who developed two-dimensional direct numerical simulations of a spatially-growing mixing layer forced upstream by a small random perturbation. However all natural plane shear layers are known to develop three-dimensionality, both in the large and small scales, characterized in particular by thin streamwise hairpin vortices streched between the large-scale rollers [3] and developed turbulence with $k^{-5/3}$ Kolmogorov spectra. We will present three-dimensional simulations of both incompressible and compressible mixing layers reproducing these features. They also point out a great sensitivity of the flow structure to initial conditions: in some cases, the flow pattern appears to be controlled by unstable oblique modes which are not the most amplified ones predicted by the linear theory of stability [4].

Nevertheless this does not necessarily contradicts a possible universality of transitional mechanisms leading to developed turbulence: we will also show results of simulations of jets and boundary layers (both in incompressible and compressible situations) in which analogous instability mechanisms are at work, yielding similar vortical structures (at least to the lowest order of approximation); in particular, the staggered or aligned Λ-vortices found in boundary layers close to the wall also result from the growth of oblique modes [5].

We will also present comparisons of results obtained by means of different numerical methods, in particular the vortex method used by Ashurst and Meiburg [6] which were found to reproduce satisfactorily the coherent structures of incompressible mixing layers, free jets and wakes during the early stages of transition. Nevertheless, when turbulence develops in the small scales, dissipative mechanisms have to be taken into account: the molecular diffusion, which will act at scales smaller than the mesh if the Reynolds number is high enough. In this case, it has to be modelled by means of appropriate techniques (subgrid-scale parameterization), which will be briefly discussed.

References:

[1] G.L. Brown and A. Roshko, *J. Fluid Mech.*, **64**, 775 (1974).

[2] Normand, P. Comte and M. Lesieur, *Rech. Aérosp.*, **6**, 45 (1988).

[3] L.P. Bernal and A. Roshko, *J. Fluid Mech.*, **170**, 499 (1986).

[4] P. Comte, M. Lesieur and E. Lamballais, "Large and small-scale stirring and mixing of vorticity and a passive scalar in a 3D temporal mixing layer", to appear in *Phys. Fluid A*.

[5] T. Herbert, *Ann. Rev. Fluid Mech.*, **20**, 487 (1988).

[6] W.T. Ashurst and E. Meiburg, *J. Fluid Mech.*, **189**, 87 (1988).

VORTICITY ON THE BOUNDARIES FOR THE 2D NAVIER STOKES EQUATIONS IN VELOCITY VORTICITY FORMULATION

O. Daube
LIMSI-CNRS
BP 133 ; 91403 ORSAY CEDEX ; FRANCE

Some results concerning the equivalence between the Navier–Stokes equations in primitive variables on one hand and in velocity voticity formulation on the oher hand, are recalled.

To compute the velocities, we made the choice to use Poisson equations. Thus, we present the conditions that the vorticity ω must satisfy on solid boundaries in order to enforce the definition of ω as the curl of the velocity, and therefore the mass conservation.

In the unsteady computations that are presented,(driven cavity, flow around a circular cylinder), these conditions are satisfied by means of an influence matrix technique that ensures at *each time step*, the definition of ω and the vanishing of the divergence of the velocity field.

Simulation of 2D Turbulence at near-infinite Reynolds number

David G. Dritschel

The "moment accelerated contour surgery" (MACS) algorithm has made possible an altogether new approach to 2D turbulence. It has enabled for the first time the simulation of 2D vortex flows at Reynolds numbers enormously higher than was previously possible.

MACS is an extended "contour dynamics" method based on the Lagrangian description of fluid mechanics. Material conservation of vorticity, the fundamental dynamical property of inviscid fluid motion, is satisfied automatically. MACS builds on contour dynamics by combining it with other techniques in a new way that results in a ten- to hundred-fold gain in computational efficiency.

The results of three large ensembles of calculations differing only in spatial resolution demonstrate rapid convergence to essentially inviscid dynamical behavior. The statistical vortex properties obtained by and large differ substantially from all previously-reported results for two-dimensional turbulence. In particular, many more small-scale vortices are observed, with the number density distribution not following a power law but rising progressively more steeply with decreasing vortex size. Also, and as a consequence, the time decay of basic flow properties (e.g., total enstrophy) is significantly slower, and, moreover, does not fit the recently-proposed universal scaling theory for two-dimensional turbulence.

The ability of MACS to model a greatly extended range of spatial scales, as compared to conventional models, and to preserve vorticity gradients from artificial numerical erosion, is what distinguishes its performance and potential from that of conventional Eulerian methods.

Computer simulation of fluid flow with heat and mass transfer in engineering and science

B.P. Gerasimov

The Keldysh Institute of Applied Mathematics
USSR Academy of Sciences, Moscow
Russia

Industrial applications require simple and unified computer simulation of different rather complex problems performed by users from common research and development staff without special training in numerical mathematics. Two multipurpose Applied Code Packages of Computational Fluid Dynamics named AEOL and NEPTUNE involve multi-dimensional models (the Euler equations for compressible gas and the Navier-Stokes equations for incompressible fluid) with extensive physics and chemistry. They may be easily appended and modified by a user in a straightforward manner and may be incorporated in Computer Aided Design systems or used for student training. The main distinction, which made them particularly suitable in engineering, is inplementation of the conservative homogeneous schemes on rectangular grids in the physical domain, allowing flow simulation in regions of complicated shape, even varying in time. The simplicity of algorithms provides implementation on transputor and multiprocessor systems, vector and array processors, etc ..

The code performances on PC may be demonstrated.

New Aspects of Vortex Dynamics by Numerical Simulation: ....cal Wave Dynamics, and Vortex-Fine Scale Turbulenc

by Fazle Hussain

By investigating a straight axisymmetric tube having an initially nonuniform core size along its axis, with and without superimposed fine-grained turbulence, we bring new life to local helicity concepts. While the relative helicity and helicity densities are of little use, we find that the separation of the flow field into right and left handed components by the "complex helical wave decomposition" is highly useful. This decomposition seems to open up an entirely new discipline of truly 3D vortex dynamics (including core and wave packet dynamics), as well as to provide a rich new set of tools for analyzing 3D vortical fields with coherent structures. We find that the evolution of the laminar vortex can be explained both by traditional means (such as coupling between swirling and meridional flow) and in terms of helical structures; in the context of such structures the dynamics is much simpler, more revealing, and allows better qualitative predictions of the flow's evolution than does the well-known vortex dynamics. We show that vortex core dynamics, ignored by all texts and most researchers, can have a significant effect on a vortex's evolution and interaction with others or with fine-scale turbulence. Core dynamics is better understood in terms of nonlinearly interacting polarized wave packets whose evolution equation is extracted from the vorticity transport equation by use of projection operators. The core size oscillation is controlled by viscous effects (decaying slower at high Re), the oscillation frequency decreasing with increasing Re, *but* having a finite limit as Re$\to\infty$. Internal dynamics disappears first near the outer edge, gradually retracting to an increasingly smaller region around the axis.

For the turbulent vortex we find the striking phenomenon of organization of incoherent vorticity in the boundary layer surrounding the vortex into spiral type structures that produce intermittency, and the inviscid generation of increasingly larger scales - a prime example of non-trivial coupling between large and small scales in turbulent flows. Our finding is in contrast with the prior suggestion of formation of sheet-like intermittency structures, and we find the azimuthally polarized smaller-scale vortices to anti-cascade into larger structures by pairing. The organization is accompanied by a strong tendency for the right and left handed components of the vorticity field to separate spatially. Moreover, the small scales take energy form the coherent vortex and can, if the Reynolds number is sufficiently high, feedback on the coherent vortex in such a way as to excite non-axisymmetric bending waves. This type of feedback or backscatter as a result of coupling between large and small scales can not be molded by any type of eddy viscosity. Our study raises some doubts about the validity of the local isotropy concept to turbulent shear flows. We conclude with a cascade model for turbulence scale hierarchy.

Vortex Method Approxmations for Two-Phase Flows with Surface Tension

Thomas Y. Hou

Courant Institute of Mathematical Sciences
251 Mercer St., New York, NY, 10012, USA
Co-Authors: J. T. Beale and J. Lowengrub

A variety of physical phenomena involve propagating interfaces. It ranges from the familiar water wave problem to problems in flame propagation and dendritic solidification. It is well-known that the vortex sheet problem is ill-posed and singularities are formed in a finite time without physical regularizations. Here we consider the stabilizing effect of surface tension for interfacial flows, including water waves, Hele-Shaws, and two-phase flows. The stabilizing effect of surface tension was not well understood before, partially due to the difficulty in distiguishing the numerical instability from a phyiscal one. In our study, we use a spectrally accurate point vortex method to approximate the governing equations. We show that this method is stable as long as the physical solution is smooth. In the process of proving this result, we develop a powerful analytical technique to analyze stability of vortex methods by a subtle energy estimate without using analyticity. Our study also indicates that there is a strong compactibility constraint that must be satisfied by a numerical method in order to be stable. Violation of this constraint will lead to numerical instabilities. For example, if one uses an alternating trapezoidal rule to approximate the velocity integral and uses a cubic spline approximation to the curvature, the resulting scheme is numerically unstable.

Another aspect of our study is to investigate the question of whether or not surface tension can prevent singularity formation. This has been an unsettled question for a long time. Since our method is proved to be stable and spectrally accurate as long as the solution is smooth, this provides a very accurate method to compute the singularity if there is one. Our study shows that if surface tension is above some critical value, the interface problem has a smooth solution for all time. If surface tension is below the critical value, the curvature of the interface grows exponentially in time. This makes it extremely difficult to distinguish numerically the exponential growth from a finite time singularity.

One major difficulty in computations of interfacial flows with surface tension is the severe time step constraint; e.g. Δt must be less than h^3 for an explicit time discretization. In our study, we propose to use an implicit time discretization coupled with an efficient preconditioned iterative method. This combination provides a very efficient numerical algorithm – a factor of $O(N^2)$ faster than the usual explicit discretization. Here the choice of the preconditioner is crucial. This is constructed from the leading order solution operator in its normal and tangential coordinates, guided from our stability analysis. Extensive numerical experiments will be presented.

"A COUPLED POTENTIAL-BOUNDARY LAYER CALCULATION METHOD FOR UNSTEADY FLOWS AROUND AIRFOILS"

M. Kermarec, A.F. Decaix, P. Renon, D. Favier, C. Maresca
IM2-IMFM-UM34 CNRS, 163 Av. de Luminy, 13009 Marseille

A numerical method, based on coupling a potential model, and a boundary-layer calculation, has been developed to predict unsteady airloads of airfoils oscillating in two-dimensional flows. One original feature of the numerical approach consists in determining the geometry of the boundary at the junction between the two calculation domains. The coupling boundary location is specifically determined for flow zones including boundary-layer separation on the airfoil surface and downstream of the trailing-edge, in order to verify the following conditions : the coupling boundary represents a streamline in the frame linked to the oscillating airfoil; a constant pressure value is imposed on the separated boundary layer coupling line, equal to the value at the trailing edge; pressure values across the wake behind the trailing edge must be equal.

The numerical approach has first been validated in steady flow configurations around the model at rest. Comparisons between calculation and experiment have been performed on overall lift and drag coefficients. A new transition criterion has been deduced from boundary-layer velocity profiles measured on a flat plate model. Using this transition criterion within the numerical approach, a good agreement between calculations and experiments has been obtained on lift and drag coefficients of the airfoil including different airfoil geometry (NACA0012 and ONERA OA209) and Reynolds numbers varying from 10^5 to $3x10^6$.

For unsteady flow configurations, the first step of the numerical approach has been to introduce a modification of the classical mixing length turbulence model. This modified turbulence model has been based on physical arguments, and also deduced from velocity measurements performed within the boundary-layer of the flat plate model oscillating either in fore and aft motion or in pitching motion. The numerical method has been applied to the pitching airfoil case below dynamic stall, at low and moderated amplitudes of incidence oscillations. The calculated airloads coefficients are shown to well match the experimental hysteresis loops. Moreover, the comparisons between calculations and experiments also exhibited the strong dependence of the numerical results on the good prediction of the geometrical boundary location between the two calculation domains.

Finally, the present numerical coupling method has been checked for airfoils oscillating in pitch through dynamic stall. In this case, the boundary-layer calculation is not performed, and the unsteady motion of the turbulent separation point is deduced from local surface measurements on the upper side of the airfoil. Potential flow results obtained with such experimental data, and with the numerical determination procedure of the coupling boundary location, show the capability of the method to predict the experimental lift coefficient behavior through stall.

STABILITY ANALYSIS OF DIFFERENTIALLY-HEATED ASYMMETRIC VORTICITY LAYERS

Omar M. Knio
Department of Mechanical Engineering
The Johns Hopkins University
Baltimore, MD 21218

and

Ahmed F. Ghoniem
Department of Mechanical Engineering
Massachusetts Institute of Technology
Cambridge, MA 02139

ABSTRACT

The linear instability of a family of inviscid, two-dimensional, variable-density shear layers and wakes is investigated. Vorticity profiles corresponding to a monotonically increasing velocity profile are first examined. A larger family of initial vorticity distributions which model the merger of two unequal vorticity layers of opposite sign is then considered. The latter is obtained by superimposing on the former a wake component, characterized by a spread, δ, and a velocity deficit, W. The initial density distribution resembles a temperature spike and is described by a thickness, σ, and a temperature ratio, T_r. The stability properties of the layers are interpreted in terms of a four-dimensional parameter space (W,δ,T_r,σ). Finally, the non-linear evolution of the flowfield is illustrated using the transport element method.

Flowfield stability exhibits strong sensitivity to the details of the density distribution. In the absence of the wake component, the stability properties of the heated layer are divided into three categories according to the thickness of the density profile, σ, and the vorticity thickness, δ_w. For $\sigma >> \delta_w$, instability of the Kelvin-Helmholtz mode in a uniform-density flow is recovered. When $\sigma \sim \delta_w$, the shear layer mode is inhibited; while this trend persists for $\sigma < \delta_w$, the layer becomes characterized by the appearance of additional short-wavelength unstable modes which become dominant as σ decreases and T_r increases. Addition of a wake component is shown to alter this behavior, and to oppose the stabilizing effects of heat release. In this case, the shear layer mode always dominates the wake mode, and the presence of heated sublayer has a weak effect on the instability of the layer when δ is large, but may influence the phase speed of unstable waves whenever the zones of high vorticity and high density gradient coincide.

Large Scale Simulations Using Vortex Methods.

by

P. Koumoutsakos * and A. Leonard **

Graduate Aeronautical Laboratories

California Institute of Technology, Pasadena, California 91125

Abstract.

Simulations are presented for bounded and unbounded viscous incompressible flows. Our numerical method is based on vortex methods. The classical inviscid scheme is enhanced to account for viscous effects via the method of particle strength exchange. The method is extended to account for the enforcement of the no-slip boundary condition as well by appropriately modifying the strength of the particles. Computations are possible for extended times by periodically remeshing the vorticity field.

The particles are advanced using the Biot-Savart law for the evaluation of the velocity. Computations are made using $\mathcal{O}(10^5)$ vortex particles by efficiently implementing the method of multipole expansions for vector computer architectures.

The method is used to simulate the merging of Gaussian vortices in an unbounded fluid as well as unsteady flows behind circular cylinders for a wide range of Reynolds numbers. Direct comparisons are made with spectral methods for the unbounded flows and with a variety of numerical methods and experimental results for the bounded flows.

---

* **Graduate Student**

** **Professor of Aeronautics**

14 April 1992

Vortex Sheet Computations

Robert Krasny
University of Michigan

I will discuss recent work related to the computation of vortex sheet motion. The following problems will be addressed:

1. Periodic vortex sheet roll-up, convergence of the vortex blob method past the critical time, structure of the rolled-up spiral.

2. Wake patterns computed by the vortex blob method and comparison with experimental flow visualization by Couder and Basdevant (JFM, vol. 173, 1986).

3. Vortex sheet separation at a sharp edge, comparison with Pullin's computation of self-similar vortex sheet roll up past a semi-infinite flat plate (JFM, vol. 88, 1978), jet formation and instability, vortex pair formation due impinging flat plates.

VORTEX BLOB SIMULATION OF TWO-DIMENSIONAL FLOW IN CHANNEL WITH COMPLEX GEOMETRIES.

Henryk Kudela

Technical University of Wrocław

50-370 Wrocław, Wybrzeże Wyspiańskiego 27, Poland

Mostly the vortex blob method, originally proposed by A.J. Chorin, was used to numerical solution of the Navier-Stokes equations at high (as well as at low) Reynolds number in area with relatively simply geometry (a backward-facing step, a cylinder, a cavity). To fulfill the condition of cancellation of the normal velocity to the wall one has to add to the velocity generated by the blobs the auxiliary potential flow. For such simply area as mentioned above, that auxiliary potential flow can easily calculated. In order to be able to apply the vortex blob method to any geometry (in two dimension) we used the fast elliptic solvers together with the capacitance matrix technique. The solution of that boundary value problem for this auxiliary potential flow does not depend on any gradients of the flow. Potential velocity of each blobs were calculated by area-weighting scheme. The tangential boundary condition is realized through the generation of the vortex blob at the wall. The generations was carried out at points spaced at a distance h along the wall. We used $h=0.1$, and $\sigma=h^{0.95}$ where σ is cut-off radius. Ordinary differential equations for displacements of the blobs were solved by the first order Euler methods with time step $\Delta t=0.05$. Numerical grid for elliptic solvers was 100*20. The ratio of the length to height of the channel was as one to ten. We did not use the vortex sheet method for flow near the boundary. To test and verify of the program we studied the flow over backward-facing step for different Reynolds numbers(see Ghoniem,Cagnon (1987), and Sethian, Ghoniem (1988), J.Comp.Phys.) and we obtained good agreement with those work. We carried out also calculation for flow over rectangular step and for flat plate normal to the flow in channel. The results (velocity profiles, streamlines, instantaneous portrait of the vortex blobs) were promising.

Particle Approximation of the Vlasov-Poisson Equation

George Majda

Department of Mathematics
Ohio State University
Colombus, Ohio 43210-1174
USA

In this work we consider the spatially periodic 1-D single component Vlasov-Poisson equation from plasma physics. The initial condition is the analogue of a vortex sheet which we call an electron sheet. It consists of a delta function supported over a curve in x-v space. A number of exact solutions to this problem are known. Some of them exist for all time, while others have a finite time after which the solution of the Vlasov-Poisson equation develops singularities.

We approximate the Vlasov-Poisson equation with electron sheet initial data by a particle method. We demonstrate that the solution of the particle method converges to the exact solution over its time interval of existence. We discuss qualitative properties of the solution after the formation of a singularity.

Point Vortices and Localization in Euler Flows.

Carlo Marchioro
Dipartimento di Matematica, Università "La Sapienza",
Piazzale A.Moro 2, 00185 ROMA ,Italy

It is very old and natural the attempt to study an incompressible non-viscous fluid when the vorticity in concentrated in a subset of physical space of low dimensionality . For planar situation the first study of a fluid when the vorticity is sharply concentrated in very small regions (schematized as points) has been done in the last century, introducing the point vortex system . A formal connection between this model and the Euler equation is discussed in many textbooks. However only recently a rigorous connection have been proved. A short review of this topics is the argument of the present talk. In particular it is possible to prove a sort of "localization" of the motion in the sense that the evolved vorticity is concentrated in N regions of diameter d which vanishes as ε (the initial diameter) vanishes for any fixed time. For smooth problems this is a direct consequence of the continuity with respect to the initia datum. But our problem is singular because the initial datum is sharply concentrated and the time evolution (via the Euler Equation) has a logarithmic divergent kernel. Actually, when ε is very small, the field in each blob becomes very large and it is difficult to exclude that thin filaments of vorticity are pushed away. We can prove that this fact does not happen. As main consequence we can prove a rigorous connection between the Euler equation and the point vortex theory.

Some results in this direction has been obtained in the last ten years but only recently has been completely solved [1].

More exactly, consider an initial datum of the form:

$$\omega_\varepsilon(\underline{x},0) = \sum_{i=1}^{N} \omega_{\varepsilon;i}(\underline{x},0) \tag{1}$$

where $\omega_{\varepsilon;i}(\underline{x},0)$ is a function with a definite sign supported in a region $\Lambda_{\varepsilon;i} \subset \mathbb{R}^2$ such that

$$\Lambda_{\varepsilon;i} \equiv \operatorname{supp} \omega_{\varepsilon;i} \subset \Sigma(\underline{z}_i \mid \varepsilon) \quad ; \quad \Sigma(\underline{z}_i \mid \varepsilon) \cap \Sigma(\underline{z}_j \mid \varepsilon) \qquad \text{if } i \neq j \tag{2}$$

for ε small enough and where $\Sigma(\underline{z},r)$ is a circle of center $\underline{z}$ and radius r .

Moreover

$$\int d\underline{x}\, \omega_{\varepsilon;i}(\underline{x},0) = a_i \in \mathbb{R} \qquad (\text{ vortex intensity }) \tag{3}$$

and

$$|\, \omega_{\varepsilon;i}(\underline{x},0) \,| \leq \text{const } \varepsilon^{-\eta} \qquad \eta < \frac{8}{3} \tag{4}$$

We can prove the following result:

Theorem

Denote by $\omega_\varepsilon(\underline{x},t)$ the time evolution of $\omega_\varepsilon(\underline{x},0)$ via the Euler Equation. Then

i) for all $d > 0$ there exists $\varepsilon_0(d,T)$ such that, if $\varepsilon < \varepsilon_0(d,T)$, then

$$\operatorname{supp} \omega_{\varepsilon,i}(\underline{x},t) \subset \Sigma(\underline{z}_i(t) \mid d) \qquad \text{for any } t \in [0,T] \tag{5}$$

where $\underline{z}_i(t)$ is the solution of the ordinary equation system (called point vortex system)

$$\frac{d}{dt} \underline{z}_i(t) = - \nabla_i^{\perp} \frac{1}{2\pi} \sum_{j=1;j\neq i}^{N} \ln \mid \underline{z}_i(t) - \underline{z}_j(t) \mid$$
$$\underline{z}_i(0) = \underline{z}_i \tag{6}$$

ii) for any continuous bounded function $f(\underline{x})$

$$\lim_{\varepsilon \to 0} \int d\underline{x}\ \omega_\varepsilon(\underline{x},t)\ f(\underline{x}) = \sum_{i=1}^{N} a_i\ f(\underline{z}_i(t)) \tag{7}$$

We shortly comment the statement of the theorem. Position i) states that the blobs of vorticity remain localized until time T for any d and T provide we choose ε small enough. Position ii) states that

$$\omega_\varepsilon(\underline{x},t) \xrightarrow[\varepsilon \to 0]{\text{weak}} \sum_{i=1}^{N} a_i\ \delta(\underline{z}_i(t)) \tag{8}$$

where $\delta(\cdot)$ denotes the Dirac measure. This last statement gives a rigorous justification of the point vortex model.

In three dimension the situation is less satisfactory. The naive generalization of point vortices would be filaments of vorticity. However a simple calculation show that they move (in general) with an infinite speed and so a Theorem like the previous one is hopeless. Some point systems, called vortons, has been studied, but they do not conserve some important quantities and so are less fundamental than the point vortex in two dimension. However the can be important for numerical purpose.

For a review of the topics discussed in this talk, see [2].

1. Marchioro C., Pulvirenti M.,: Vortices and localization in Euler flows, Commun. Math. Phys. (sumbitted to)

2. Marchioro C., Pulvirenti M., Mathematical Theory of Incompressible Non-Viscous Fluids, Berlin, Heidelberg,New York Springer Verlag (in press) (1992)

A PARTICLE IN CELL METHOD FOR THE COMPRESSIBLE EULER EQUATIONS

S. MAS-GALLIC (*), M. LOUAKED (**)

SUMMARY

The flow of an inviscid compressible fluid is governed by the compressible Euler equations,

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}.\nabla)\mathbf{u} + \frac{1}{\rho}\nabla p = 0,$$

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \mathbf{u}) = 0,$$

$$p = k\rho^{\gamma},$$

where $\mathbf{u}$, ρ and p are respectively the velocity, the density and the pressure of the fluid and where γ is the specific heat ratio. We consider the two dimensional case and introduce the Helmholtz decomposition of the velocity field $\mathbf{u}$

$$\mathbf{u} = \nabla\phi + \mathbf{v}$$

where $\mathbf{v}$ is the rotationnal part of the velocity. Introducing the vorticity $\omega = \nabla \times \mathbf{u} = \nabla \times \mathbf{v}$, the system of Euler equations is then written under the following form

$$\frac{\partial \rho}{\partial t} + \nabla(\rho(\nabla\phi + \mathbf{v})) = 0,$$

$$\frac{\partial \omega}{\partial t} + \nabla(\omega(\nabla\phi + \mathbf{v})) = 0,$$

$$-\Delta \mathbf{v} = \nabla \times \omega$$

$$\frac{\partial \phi}{\partial t} + \frac{1}{2}|\mathbf{v} + \nabla\phi|^2 + k\frac{\gamma}{\gamma - 1}\rho^{\gamma-1} = q,$$

$$\Delta q = \omega^2 + \nabla\omega \times (\nabla\phi + \mathbf{v}).$$

Under this form we notice that both the density ρ and the vorticity ω are convected by the flow with the velocity $\mathbf{u} = \nabla\phi + \mathbf{v}$. Then we consider the equation which gives the evolution of the potential ϕ. Introducing the hamiltonian

$$H(\psi) = \frac{1}{2}|\psi|^2,$$

this equation is rewritten

$$\frac{\partial \phi}{\partial t} + H(\nabla\phi) + \mathbf{v}.\nabla\phi = q - \frac{1}{2}|\mathbf{v}|^2 - k\frac{\gamma}{\gamma - 1}\rho^{\gamma-1},$$

and we recognize a Hamilton-Jacobi equation with a convection term.

The method of resolution of the Euler compressible system is based on these two remarks. The convection equations are solved by a particle method whereas the Hamilton-Jacobi equation is solved by the finite difference scheme proposed by S. Osher and J. Sethian and the Laplace equation by a classical finite difference scheme. A grid-particle operatorand a particle-grid operator are then defined, as in any particle in cell method. The grid-particle operator associates a particle approximation to the quantities defined on the grid, the particle velocities are computed with this procedure, whereas the particle-grid operator defines a grid approximation for the quantities associated to the particles. In order to ensure the conservativity of the scheme, the particle-grid operator is defined following the ideas of J.U. Brackbill and H.M. Ruppel.

(*) Centre de Mathématiques Appliquées, Ecole Polytechnique, 91128 Palaiseau Cedex, France
(**) Laboratoire d'Analyse Numérique, T55-65, 5è étage, Université Pierre et Marie Curie, 4 Place Jussieu, 75252 Paris Cedex 05, France

Three dimensional vorticity dynamics in nominally axisymmetric jets

E. Meiburg and James E. Martin

Department of Aerospace Engineering
University of Southern California
Los Angeles, CA 90089-1191
USA

We study the inviscid mechanisms governing the three dimensional evolution of round jets by means of vortex dynamics simulations. A typical simulation starts from an unperturbed axisymmetric layer of vortex filaments. For non-swirling jets, the filaments initially have the form of rings, while helical vortex filaments introduce swirl into the jet. The axisymmetric jet shear layer can then be perturbed in different ways by modulating the strength of the filaments (corresponding to acoustic forcing in the axial direction) or by displacing the filament centerlines (thus simulating the effect of corrugated or indented nozzle shapes or acoustic forcing in the azimuthal direction). The spatially periodic calculations provide a detailed picture of the processes leading to the concentration, reorientation, and stretching of the vorticity for different combinations of perturbations. In the purely axisymmetric case, a wavy perturbation in the streamwise direction leads to the formation of vortex rings connected with braid regions, which become depleted of vorticity. An additional azimuthal wave leads to a fully three-dimensional evolution, which results in concentrated braid vortices as well as a potential vortex ring instability. For non-swirling jets subject to axisymmetric and azimuthal waves, we observe the formation of counterrotating streamwise braid vortex pairs, while the presence of helical waves or swirl in the jet leads to the formation of braid vortices all of the same sign.

VORTEX GENERATION BY SUCTION IN A ROTATING TANK

by Mathieu MORY
LEGI/IMG
(Laboratoire de l'INPG et de l'UJF associé au CNRS)
BP 53 X
38041 GRENOBLE Cédex

Abstract

Vortex generation in a rotating tank by a convergence of vorticity caused by suction through a tube centered on the rotation axis is reconsidered in view of an analysis of Ekman layers dynamics. New experimental studies, based on flow visualisations and velocity profile measurements by Laser Doppler Anemometry, are presented, which show that the whole secondary flow is passing through the Ekman layers. Ekman layers therefore govern the structure of the flow. A model, inspired by Stern (1975), establishes that the circulation of the vortex is equal to the flowrate divided by the Ekman layer depth. When the Ekman-number, build on the radius of the suction tube, is smaller than one, the radius of the vortex core is approximately fixed by the radius of the vortex tube. When the diameter of the orifice tube is small, a sudden increase of the vortex diameter occurs, in agreement Maxworthy's observations (1972). The latter related this phenomena to vortex breakdown. The present experimental investigation determines the range of parameters for which this phenomenon occurs.

A Hybrid Vortex Method with Deterministic Diffusion

Habib N. Najm
Texas Instruments, SPDC, Dallas, TX

In the general class of particle methods, the vortex method is most convenient for modeling high Reynolds (Re) number vortex flows. This is particularly due to the following: (1) the Lagrangian solution of the Navier-Stokes equations eliminates the need to discretize the non-linear inertia terms, leading to good numerical stability at high Re, and (2) the restriction of computational elements to the regions of the flow exhibiting shear and finite vorticity leads to significant numerical efficiency.

The present work maintains the above advantages of the vortex method while proposing an accurate scheme for modeling the diffusion component of the flow equations. The method is based on a fractional step solution of the two-dimensional vorticity transport equation. The advection step is implemented using lagrangian propagation of vortex elements, while the diffusion step uses a hybrid lagrangian-eulerian implementation. In this diffusion scheme, both the vorticity field, $\omega(x, y)$, and its Laplacian, $\nabla^2\omega(x, y)$, are evaluated from the summation of the individual analytical elemental fields at any point in the computational domain. In particular, both quantities are computed analytically at the cell centers of a suitable grid overlaid on the computational domain. A second order time integration of the diffusion equation is then utilized to arrive at the new values of the vorticity at the cell centers. These are used to generate a new set of vortex elements at each time step, in the same fashion utilized in the initial discretization of the vorticity field at time $t = 0$. This process of continuous regeneration of vortex elements at each time step, based on the diffusion of the overall vorticity field, allows for creation of new elements where necessary to maintain a time accurate representation of the flow. Further, the absence of grid based discretization of $\nabla^2\omega$ maintains the desireable grid-free solution of the Navier-Stokes equation at each time step.

The scheme is presented and demonstrated on an incompressible flow problem that involves the viscous decay of a vortex tube in an unbounded two-dimensional domain. The analytical solution of this problem is known, and is used to compute the error associated with the numerical solution, to study its convergence characteristics, and to compare the present results against those available from the random vortex method[1], and the deterministic vortex method due to Cottet and Gallic[2,3]. Results demonstrate the improvement in accuracy provided by this hybrid scheme over a wide range of Reynolds number.

References

[1] A. Chorin, *J. Fluid Mech.*, 57, 785 (1973).
[2] G-H. Cottet, in *Mathematical Aspects of Vortex Dynamics*, ed. R.E. Caflisch, p. 128, SIAM (1989).
[3] J.P. Choquin and B. Lucquin-Desreux, Int. J. Num. Meth. in Fluids, 8, 1439 (1988).

Abstract submitted to the International Workshop on Vortex Flows and Related Numerical Methods, to be held in Grenoble, June 15-19, 1992 :

AXISYMMETRIC VORTEX SHEET ROLL-UP

Monika Nitsche
University of Michigan

The evolution of 3-d axisymmetric vortex sheets is studied numerically for the special case of flow without swirl. The axisymmetric sheets roll up at their boundaries to form a circular vortex ring. The computations use a vortex blob method applied to axisymmetric vortex sheet motion.

Two examples are studied. The first one is the evolution of an initially flat circular vortex sheet, which has been produced by setting a circular plate impulsively into motion in an ideal fluid and then removing the plate. The vortex sheet rolls up into a ring. We study the effect of the numerical parameters upon the ring's propagation velocity, radius, and vorticity distribution. The computations are compared to analytical results by Taylor (J. Appl. Phys, vol. 24, 104) and Saffman (Stud. in Appl. Math., Vol XLIX, No 4).

The second example is the formation and roll-up of a vortex sheet at the edge of a circular tube. Well-documented experiments have been performed by Didden (ZAMP, vol 30, 101), in which a piston drives fluid out of a tube, and dye particles are used to visualize the vortex sheet formed. We compare the numerical results to the experiments, and find good agreement in the shape of the spiral and several parameters describing it, as well as in the behaviour in time of the center of the spiral.

3-D SIMULATION OF FREE VORTEX RINGS AND VORTEX RINGS INTERACTING WITH A WALL

P. Orlandi and R. Verzicco

Dipartimento di Meccanica ed Aeronautica, Universita' di Roma

The vortex ring is a basic coherent structure and the detailed comprehension of its 3-D evolution is a basic feature to understand, when it evolves in an unbounded domain, the entrainment of external fluid into round turbulent jets and, when it interacts with no-slip boundaries, the bursting event in turbulent boundary layers. The aim of the paper is to perform finite difference numerical simulations second order accurate in time and in space to have a full description of the complex vorticity field.

An axisymmetric calculation has been, initially, performed in both cases with the initial azimuthal vorticity $\omega = \frac{\Gamma}{4\pi\sigma^2} e^{-r^2/\sigma^2}$. Because the Gaussian distribution is not a solution of Euler equation, in the case of a free ring, the study has been focused on analysing which $\omega/r = f(\psi)$ relationship, (ψ is the Stokes streamfunction in a coordinate system translating with the ring) is obtained by a direct numerical simulation performed at a $Re_\Gamma = \Gamma/\nu = 5500$. As a first check of the precision of the numerical simulation we have shown that the calculated dimensionless translation velocity of the ring with $\sigma = 0.413$, $V_t = 2.23$, agrees with the analytical expression $V_t = 2.49$. Moreover vorticity contour plots shows that immediately after the ring starts its translatory motion, a wake is generated; this wake was experimentally observed by Maxworthy; from the simulation we conjectured that the wake is formed by the readjustment of the initial vorticity distribution to that of equilibrium, for which scatter plots of $\omega/r = f(\psi)$ show a good collapse of the data. We obtained a function f which smoothly goes from 0, in the region external to the core, to a linear function in the region within the core of the ring.

In the case of the ring interacting with a non-slip wall we performed the simulation at different Reynolds numbers and we compared the trajectories of the centres of primary, secondary and tertiary vortices with in the experiment of Walker *et al.* Even if at all Reynolds number, the secondary ring, generated at the wall, causes the rebound of the primary ring from the wall, only at high Reynolds number a pairing between secondary and tertiary rings forms a new ring with enough circulation to migrate far from the wall region.

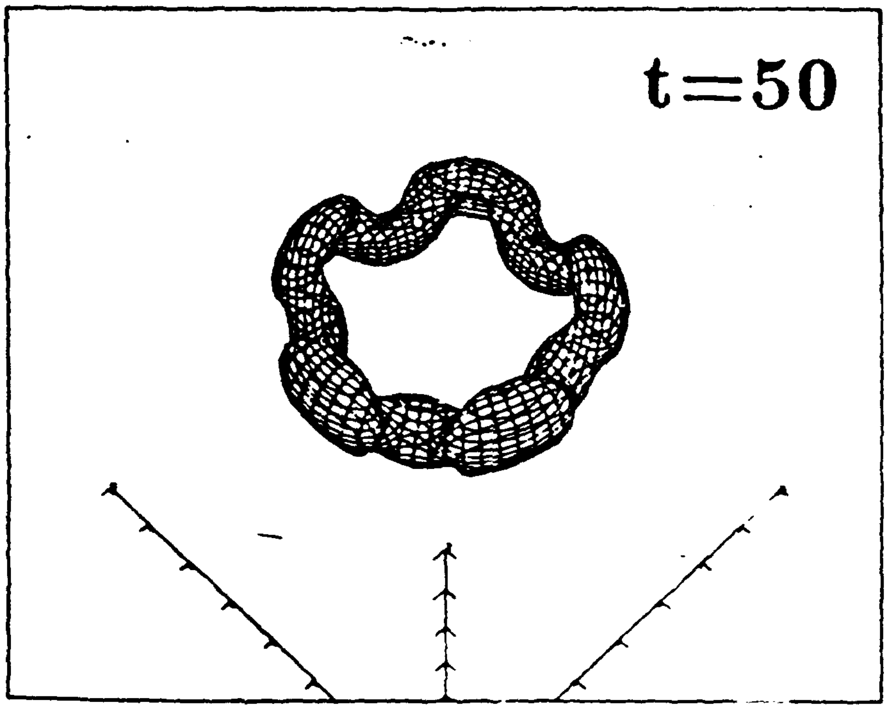
The three-dimensional simulation of the Navier-Stokes equations for the free ring, with periodic conditions in the direction of translation, has been performed with several initial perturbations in the azimuthal direction. In all cases the translation velocity does not change from that in the axisymmetric case. The linear stability analysis predicts that, for this translation velocity, the most unstable mode is that for $n = 5$. The numerical simulation shows that, giving a $n = 5$ sine waves or a random disturbance with the same amplitude $\epsilon/r = 0.02$ the perturbation grows in time maintaining in the first case the $n = 5$ wave (Fig.1a) and in the second case choosing a $n = 6$ wave (Fig.1b). A further case has been performed with $n = 3$ and we observed that the the $n = 3$ mode is dissipated and the $n = 6$ mode is that which grows.

By the direct simulation we proved that for thick free rings the fundamental and higher modes are excited but only the most unstable persists and grows in time. In all cases flow visualisation of isolevel surfaces of ω_θ and $\omega_a = \sqrt{(\omega_r^2 + \omega_z^2)}$ show the deformation of the ring structure. ω_z gives the greater contribution to ω_a and ω_z initially is located in very long and thin structures at the interior of the ring. These patches of opposite sign vorticity are generated near the centre of the ring core, are deformed by the wake and become elongated structures which by mutual induction migrate towards the centre where are dissipated.

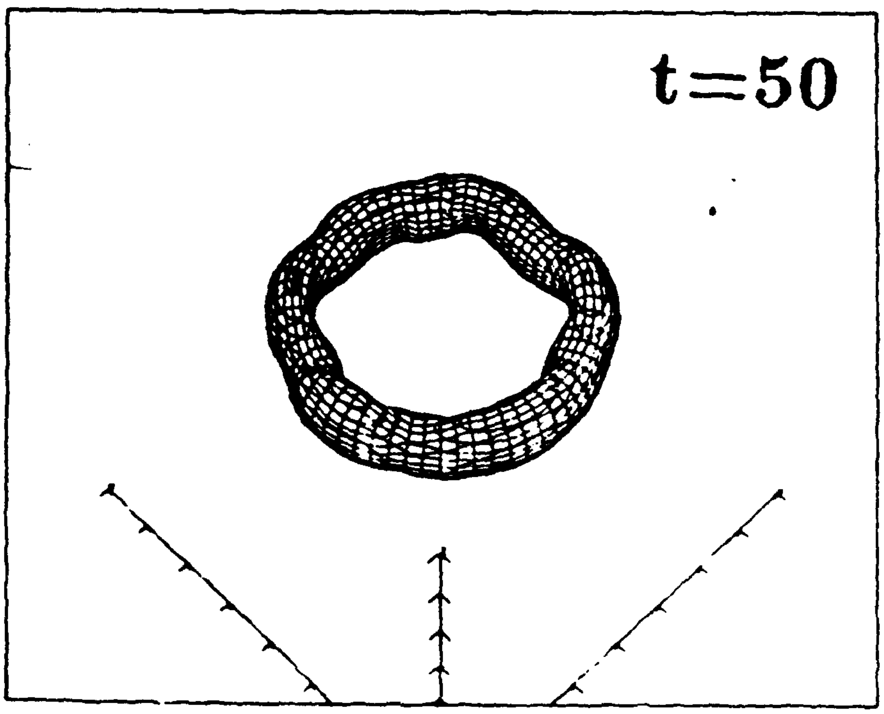
In the 3-D simulation of the vortex ring impacting a solid wall we observed that azimuthal instabilities are effecting mainly the secondary ring, which is advected upwards by the primary ring; in this first stage (Fig.2a) the initial $n = 5$ perturbation grows. In a second stage, while the secondary ring is advected and compressed at the interior of the primary ring (Fig.2b), the radius decreases and a well defined $n = 10$ mode is observed. When the secondary ring is very close to the wall a substantial amount of ω_z is generated and five rings of smaller dimensions are generated that interact and pair with the tertiary ring during its translation towards the centre (Fig.2c). The tertiary ring and the smaller new rings pair and produce a structure with enough circulation to move far from the wall (Fig.2d). This complex interaction of secondary vorticity patches induces on the primary ring a $n = 10$ deformation. For the case of vortex rings interacting with solid wall there was no any theoretical analysis but we found that our vorticity magnitude contour plots resemble very close the flow visualisation of Walker *et al.* At the workshop comparison between the ejection velocity in the axisymmetric and that in the 3-D case will be presented, together with the differences when a random initial perturbation is given. Furthermore will be presented vorticity, and r.m.s. velocity distributions which are very difficult to measure.

References

1. Walker, J.D.A., Smith, C.R. , Cerra, A.W., and Doligaski, J.L., "The impact of a vortex ring on a wall" J. Fluid Mech. 181, 99-140, (1987).



a)



b)

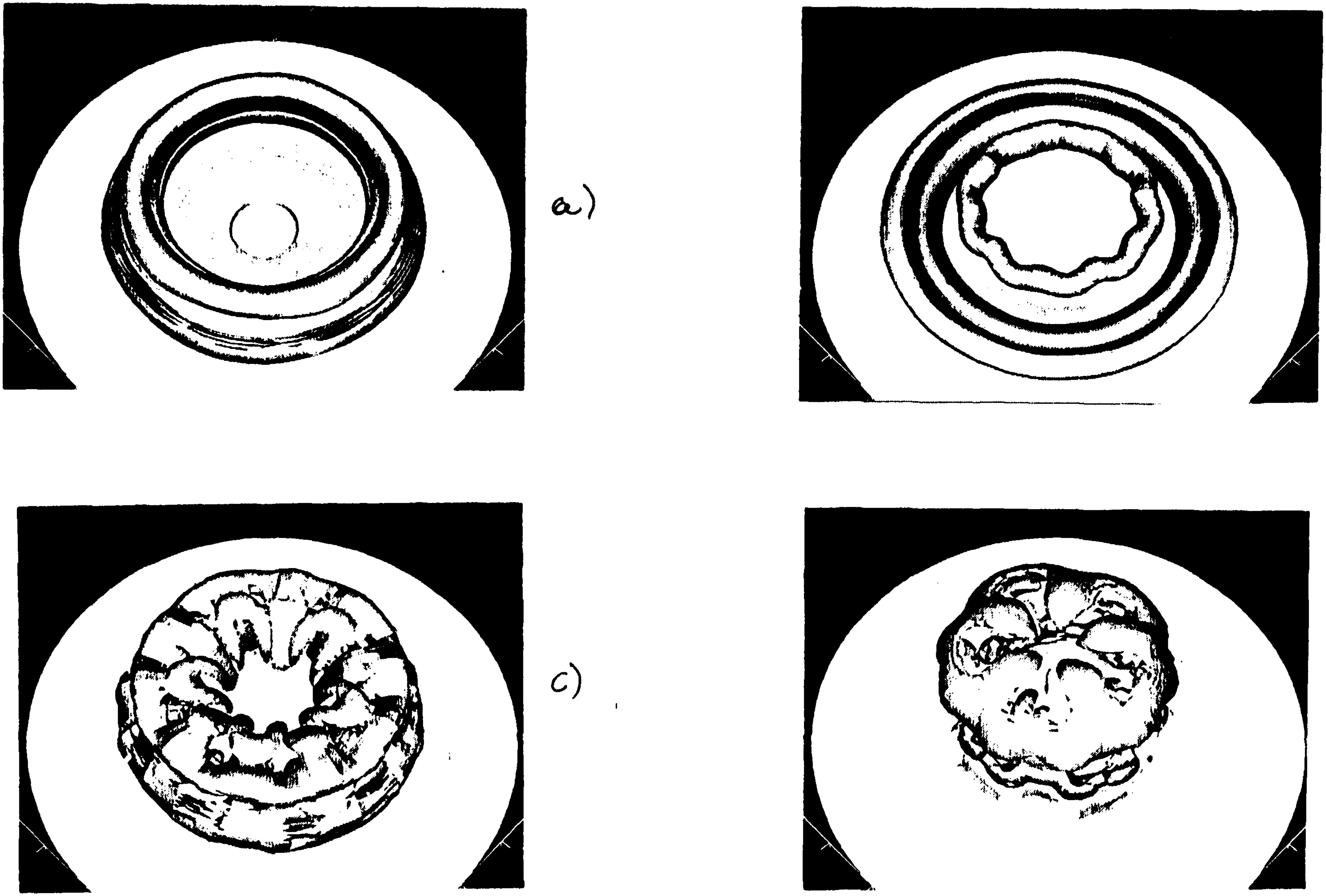


Fig. 2

THE STABILITY OF VORTEX DIPOLES

P. Cavazza+, G.J.F. van Heijst! and P. Orlandi *

+ Istituto di Fisica dell'Atmosfera del CNR Roma Italy
! Department of Technical Physics University of Eindhoven The Netherlands.
* Dipartimento di Meccanica e Aeronautica, Università di Roma Italy

Abstract

Vortex dipoles are common features of geophysical flows, and they play an important role in their dynamics. In the ocean, dipolar vortices may be generated in various ways (e.g. as a results of shedding from unstable boundary currents or due to localized wind forcing) and they provide an important mechanism in the transport of physical properties (heat, pollutants). In the atmosphere, dipolar flow structures in the form of blocking systems tend to have a stabilizing influence on the local weather. Within the context of stability of such flow structures it is of importance to know whether the structure, once perturbed, relaxes towards its initial (stable) state. It is easy to show that any functional relationship $\omega = f(\psi)$ between the vorticity ω and the streamfunction ψ satisfies $J(\omega, \psi) = 0$ and is thus a stationary solution of the inviscid Euler equations. In the present study we consider the Lamb dipole, which travels with a constant velocity while preserving its shape. For this dipolar vortex the relation between ω and ψ, in a reference frame travelling with the dipole, is a linear one. i.e. $\omega = k^2\psi$.

Analytical approaches based on linear approximations are not very helpful to study the stability of dipoles as has been recently shown by Nycander (1992) for modons, which are dipoles conserving their properties in a beta plane. Numerical approaches are more useful, in fact different types of perturbations can be applied to see in which case the dipole reaches a new steady configuration and which functional relationship $f(\psi)$ is yielded. We have performed numerical simulations with two different initial configurations out of equilibrium. In the first one the vorticity is confined within an ellipse with the longer axis aligned with the direction of translation and in the second one the longer axis is orthogonal to the translational direction. In both cases we observed that, at $Re = 10000$, a value very close to the inviscid conditions, the structure in a first stage sheds a tail and later on forms a steady dipole. The time necessary to reach the steady configuration depends on the disturbance. In the case of an ellipse elongated in the direction of translation the same initial $f(\psi)$ is obtained, (Fig.1a) while in the other case a relationship with two slopes is reached (Fig.1b). The reason of the reduced slope is due to the greater distance between the two vorticity peaks than those for the Lamb dipole. Couder and Basdevant (1986) found a similar flattening near the axis in their numerical simulations. The core of the dipole on the contrary is well restored.

A different perturbation has been given by letting two dipoles to collide at different angles, as done in the experiment by van Heijst and Flor (1989). In this case during the impact the

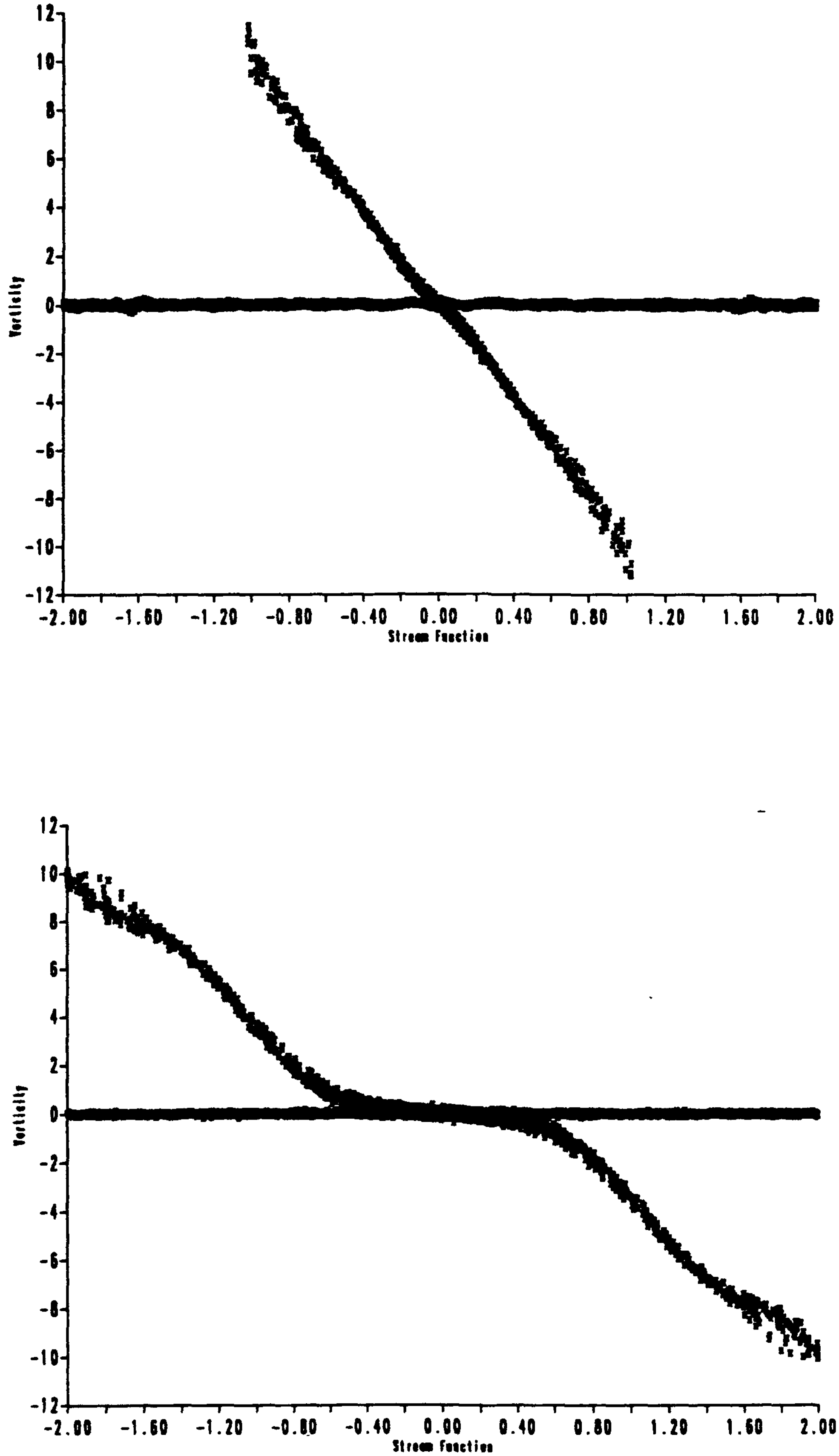
dipoles are deformed, interchange partner and finally travel again along a straight trajectory. The numerical simulation has been performed by assuming that the dipole is impacting with a free-slip wall. We considered the collision at three different angles 90, 60 and 30 degrees. The scatter plots are given in Fig.2 a-c. Fig.2a shows that for the collision at 90^0 the law $f(\psi) = k^2\psi$ is reached. On the other hand Figs.2b and 2c show that, when the dipole impacts the wall at an angle a functional relationship with a discontinuity on the slope is obtained and that the region responsible of the higher gradient is that close to the wall. To obtain these plots we analysed the field of the dipole which was closer to the wall and was moving faster.

From these two different perturbations we conclude that the region close to the peak vorticity is very stable, while the region close to the centre of the dipole varies depending on the perturbation given. Dipoles with a flat region are those that can easily loose the coherence and split into two monopoles.

We were also interested to understand which is the global quantity representing, in the best way, when the steady state is reached. We have thus calculated, in the region with the dipolar structure, the excess energy, the enstrophy and the integral of $\omega log(\omega)$ that represents the entrophy of the flow field. We found that when the minimum for each quantities is reached a well defined $f(\psi)$ is observed. Time histories of these quantities will be presented at the conference.

References

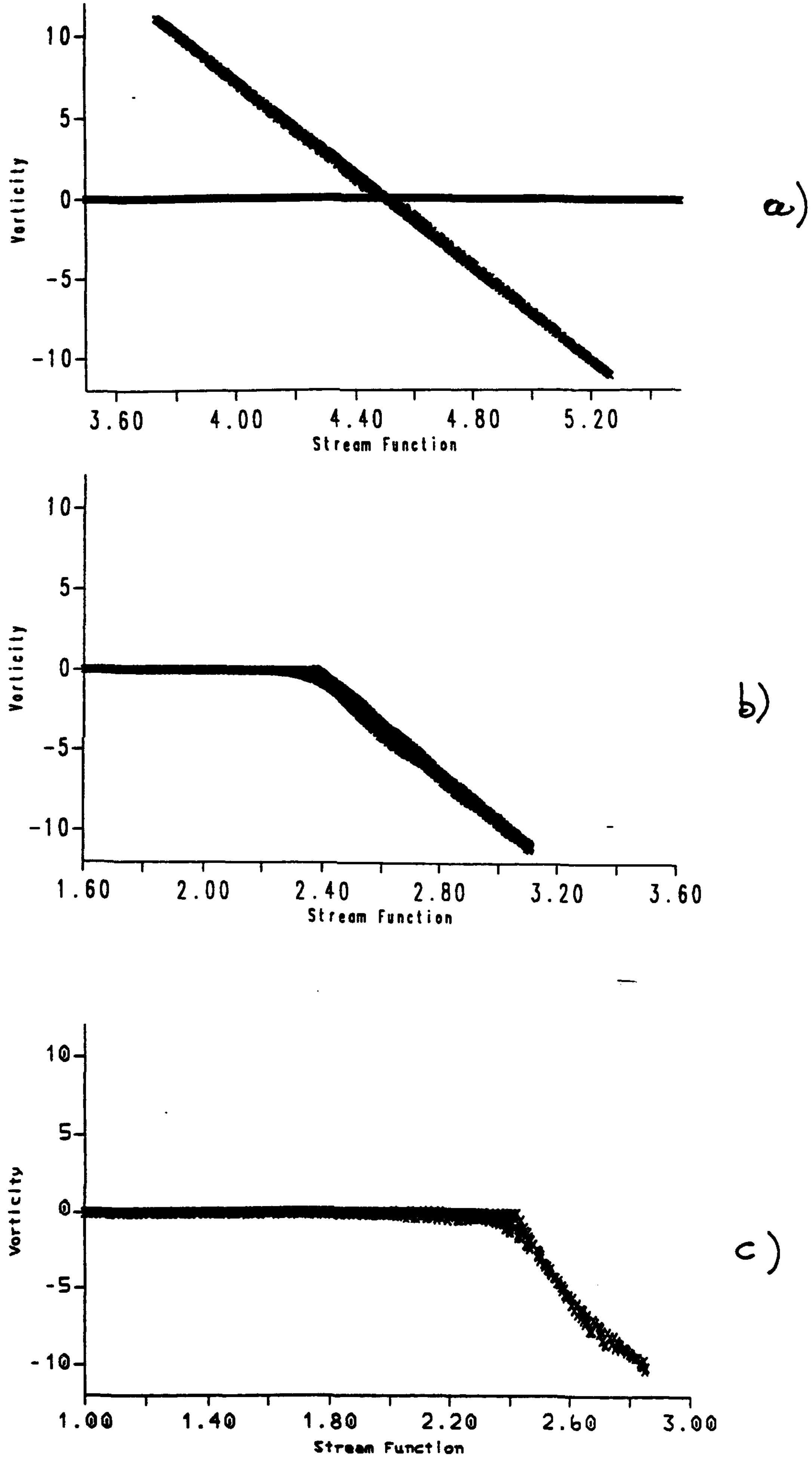
- Nycander ,J. 1992 Refutation of stability proofs for dipole vortices. Physics of Fluids, A(3),pp.467-476.
- Couder,Y. and Basdevant,C. 1986 Experimental and numerical studies of vortex couples in two-dimensional flows. J.Fluid Mech.,173,pp.225-251.
- van Heijst, G.J.F. and Flor, J.B. "Laboratory experiments on dipole structures in a stratified fluid" in Mesoscale/synoptic coherent structures in geophysical turbulence, Edited by J.C.J. Nihoul and B.M. Jouart (Elsevier Amsterdam, 1989), pp. 591-608.


Vorticity
Stream Function
12
10
8
6
4
2
0
-2
-4
-6
-8
-10
-12
-2.00
-1.60
-1.20
-0.80
-0.40
0.00
0.40
0.80
1.20
1.60
2.00

a)

b)

Fig. 1



10
5
0
-5
-10
Vorticity
3.60
4.00
4.40
4.80
5.20
Stream Function
a)
10
5
0
-5
-10
Vorticity
1.60
2.00
2.40
2.80
3.20
3.60
Stream Function
b)
10
5
0
-5
-10
Vorticity
1.00
1.40
1.80
2.20
2.60
3.00
Stream Function
c)

Fig. 2

MODELISATION D'UN TOURBILLON EN ECOULEMENT TURBULENT INCOMPRESSIBLE

par Philippe PASCAL

Résumé

Les écoulements résultant de l'évolution d'une nappe fluide, engendrée par un décollement, donnent naissance à des structures tourbillonnaires enroulées, que l'on nomme d'une manière plus courante: tourbillons.

En Aérodynamique, ces tourbillons peuvent avoir des répercussions importantes sur les conditions de vol à grande incidence des avions et des engins munis d'ailes à forte flèche. En effet, sous l'action des champs de pression associés à la voilure, ces tourbillons sont susceptibles de subir une désorganisation brutale, plus connue sous le nom d'éclatement tourbillonnaire. Ce phénomène entraîne une modification profonde de l'écoulement sur l'aile, dont la prévision reste très incertaine. L'objet de ce travail est de proposer une méthode de calcul capable de décrire ce type d'écoulement. Une étude théorique a donc été réalisée, dans le but de comparer les différentes méthodes de prédétermination de l'évolution d'un tourbillon, dans son processus d'éclatement. Les méthodes proposées sont basées sur l'hypothèse d'axisymétrie de l'écoulement, et sur un traitement statistique classique de la turbulence avec fermeture en un point. Trois approches différentes ont été menées:

- en utilisant les équations statistiques du mouvement écrites avec les approximations de la couche limite, et en introduisant un modèle de turbulence algébrique,
- en appliquant les mêmes équations, mais en utilisant l'A.S.M,
- en résolvant directement les équations statistiques, sans approximation et avec un modèle algébrique. Une confrontation des résultats du calcul avec certaines données expérimentales permet de vérifier l'adéquation entre le calcul et l'expérience, ainsi que les tendances sur la validité des modèles de turbulence.

Une étude expérimentale a été réalisée pour mieux appréhender les différents phénomènes constituant un tourbillon, et notament analyser et quantifier l'influence de la turbulence sur le développement d'un tourbillon stable non éclaté. Pour cela, on a utilisé un dispositif permettant de produire un tourbillon de bout d'aile. La structure interne du tourbillon présente, au centre, une survitesse qui tend à diminuer lorsque le tourbillon évolue dans sa direction axiale. Le champ de vitesse extérieur est bien représenté par un tourbillon potentiel. Toutefois, on a remarqué que les mouvements latéraux d'ensemble de l'écoulement doivent être pris en compte dans l'interprétation des résultats expérimentaux, car ils induisent des fluctuations à basse fréquence, qui se superposent à la turbulence proprement dite.

A slightly diffusive Contour Dynamics

G. Riccardi and R. Piva

Dipartimento di Meccanica e Aeronautica
Universita' di Roma "La Sapienza", Italy

The use of a piecewise constant approximation of the vorticity field gives a simple and fruitful approach to the Lagrangian simulation of the coherent structures motion in a two-dimensional *inviscid* context. The way is a straightforward generalization of the Zabusky idea to multi-levels distribution of vorticity based on the linearity of the vorticity-velocity rule. In the following we outline a natural extension of the Contour Dynamics to a *slightly diffusive* fluid by using a simple approximation of the vorticity fluxes between adjacent contours.

The velocity field induced by a piecewise constant vorticity

$$\omega_t = \sum_{i=1}^{n} [\omega]_i \chi_{D_i(t)}$$

is given in terms of the contours $\{\partial D_j\}_{j=1,\dots,n}$ and vorticity jumps $\{[\omega]_j\}_{j=1,\dots,n}$ across these by

$$u_t(x) = -\sum_{i=1}^{n} [\omega]_i \int_{\partial D_i(t)} \tau(s) G(x-y) ds$$

in which $G(x) = \frac{1}{2\pi} log|x|$. An approximate description of the vorticity dynamics is given by using a straight panel discretization of each contour ∂D_j with a suitable panel duplication procedure.

The fascinating aspect of this method is the absence of any form of numerical viscosity that enables to follow the small scales motion as clearly shown in the collision of Lamb dipoles of *Fig*.1. This suggests to modify the Contour Dynamics technique in order to study the small scales motion in presence of a slight diffusion due to *real* viscosity. A simple way is to consider the vorticity jumps as time dependent

$$\omega_t \simeq \sum_{i=1}^{n} [\omega]_i(t) \chi_{D_i(t)}$$

(we assume $D_0 \supset D_1 \supset \cdots \supset D_n$) and to integrate the Helmholtz equation in all the annular regions $D_{i-1} - D_i$, $i = 1, \dots, n$ and also in D_n. By using a suitable approximation of the vorticity fluxes exchanged by adjacent levels finally we obtain a first order differential system in terms of $\left\{\frac{d[\omega]_i}{dt}\right\}_{i=1,\dots,n}$.

The accuracy of the time integration as well as the effects on the numerical solution of the vorticity fluxes approximation are evaluated computing some "global quantity" of the flow that in an inviscid context play the role of first integrals of the motion. These computations can be carried out in a simple way as long as the limit Γ of the circulation around spheres B_R for $R \to \infty$ is *numerically* retained. The most useful relations involve the second moment of vorticity $I = \int_{R^2} |x|^2 \omega dx$

$$I(t) = I(0) + \frac{4}{Re} \Gamma t$$

and the excess energy $\mathcal{E}_f(t) = \frac{1}{2} \int_{R^2} \psi \omega dx$ for which we have

$$\mathcal{E}_f(t) = \mathcal{E}_f(0) - \frac{1}{Re} \int_0^t E d\tau$$

where E is the enstrophy of the flow.

As an example of calculation of the previous quantities we show in *Fig*.2 the axisymmetrization of an elliptical non uniform vortex in an inviscid and in a viscous flow. In this case the effects of the vorticity fluxes approximation appear clearly negligible.

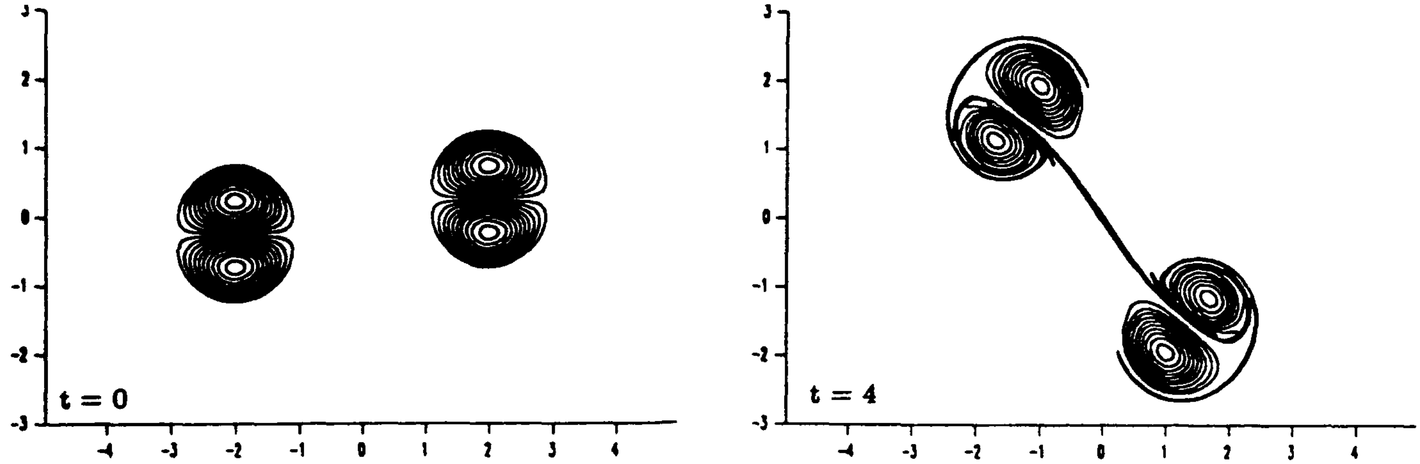


Fig. 1 : Collision between Lamb dipoles. Initially the axes of the dipoles are parallel at distance of half dipole radius.

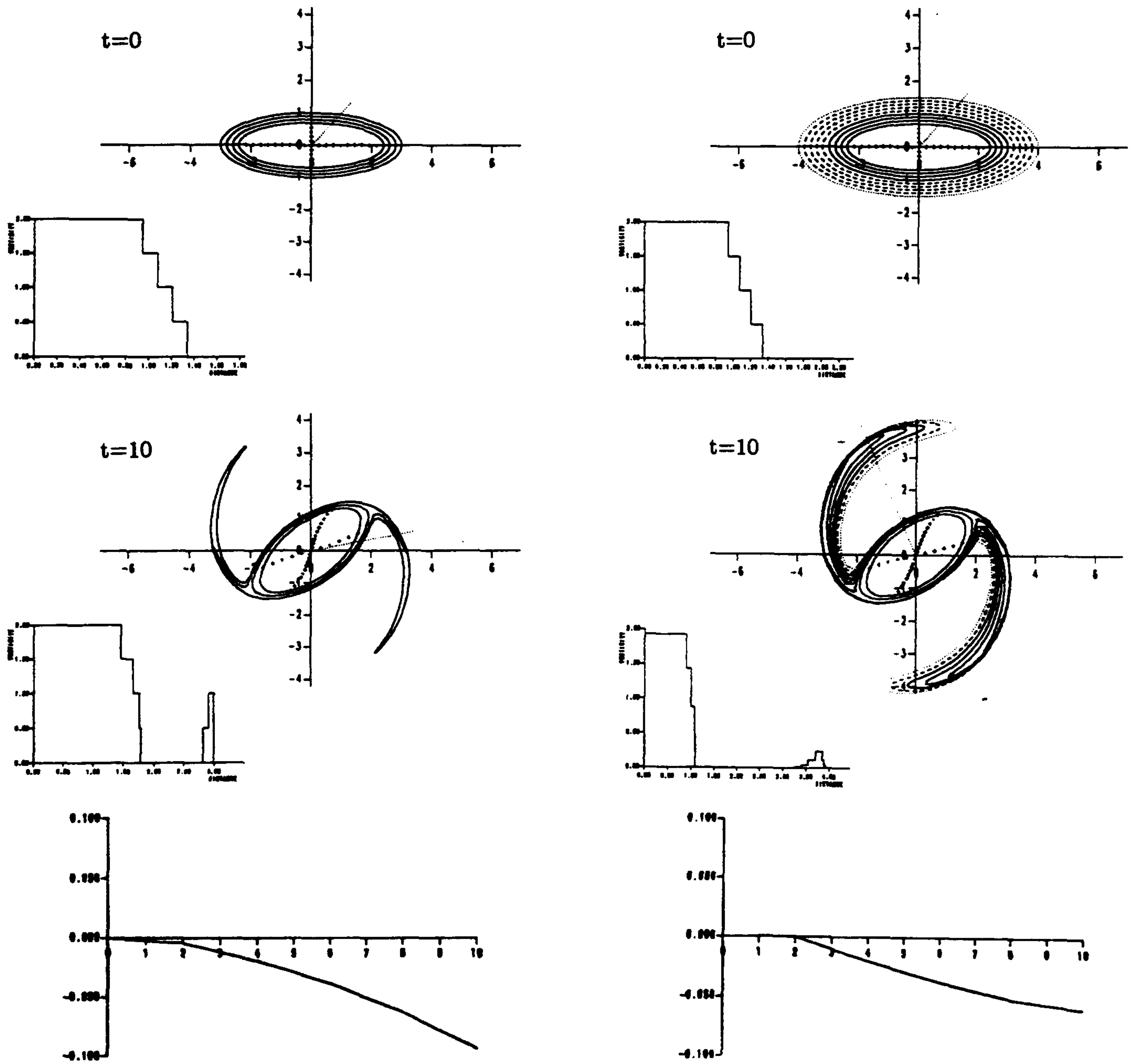


Fig. 2 : Axisymmetrization of an elliptical non uniform vortex in an inviscid flow (left) and in a viscous one (right) with $Re = 1000$. Configurations of the vorticity levels and behavior of the [illegible]

Estimation numérique d'écoulements tridimensionnels avec une méthode particulaire à poids constant

E. Rivoalen

1 Introduction

L'objectif de ce travail est la mise au point de méthodes numériques susceptibles dêtre utilisées pour calculer l'évolution d'un tourbillon émis d'un profil portant sur des distances importantes. En effet le calcul de transport d'un sillage tourbillonnaire pendant des temps relativement grands pose des problèmes à la fois de précision des algorithmes utilisés et de viscosité numérique. Les imprécisions peuvent être acceptables si l'on ne s'intéresse qu'aux caractéristiques hydraudynamiques du profil générateur de sillage , mais ne le sont plus si l'on regarde l'intéraction de ce sillage avec d'autres corps comme par exemple une hélice, une carène de bateau ou encore une surface libre. Quand on s'intéresse au transport du sillage sur des distances relativement grandes il apparait également nécessaire de prendre en compte les phenomènes de diffusion si l'on veut donner une description réaliste de l'écoulement. Nous présentons dans cette étude une modélisation de l'écoulement tourbillonnaire tridimensionnelle par la méthode des tourbillons ponctuels. La nécessité de conserver le tourbillon sur de très longues distances nous amène à travailler avec une méthode utilisant une description lagrangienne de l'écoulement. Nous utilisons ici le cadre de la méthode de discrétisation particulaire tridimensionnelle mise au point par Rehbach en 1977.

2 Transport du tourbillon à poids constant

L'objectif numérique est de construire une méthode d'intégration de l'équation de Helmholtz. Dans ce cas l'équation de transport du tourbillon comporte un terme supplémentaire, le terme de déformation. Ce dernier est décomposé en un champ tangent au tourbillon local qui représente l'élongation du tourbillon et un champ orthogonal qui représente une réorientation du tourbillon local donc une transformation à module constant. On remarque que le terme tangent peut sinterpréter comme une redistribution des particules le long des filets tourbillonnaires du fait des élongations et contraction que l'écoulement externe lui fait subir. Ce terme apparaissait déjà dans le terme covectif de l'équation de transport, il disparait donc de cette équation. Sous forme discrète l'équation de transport du vecteur tourbillon s'écrit :

$$\left\{ \begin{array}{lcl} \frac{d\vec{X}}{dt} & = & \vec{U}_{orth} \\ \frac{d\vec{\Omega}}{dt} & = & (\vec{\Omega} \cdot \vec{\nabla})\vec{U}_{orth} \end{array} \right.$$

Cette décomposition revient a ne considérer localement que le champ de vitesse orthogonal au tourbillon.

3 Modélisation de la diffusion

La prise en compte des phénomènes de diffusion est traitée comme un réamenagement de la répartition des points de discrétisation. Nous proposons une méthode mixte dans laquelle la composante tangente du vecteur tourbillon est diffusée par une méthode de dépacement déterministe (l'équation de diffusion pure est couplée à une équation de transport) et les deux autre composantescomme une rotation du vecteur tourbillon (transformation à module constant). Ce modèle permet de rendre compte de l'évolution de la vortité sous l'effet de la diffusion et en particulier de l'expension d'un sillage.

Un modèle d'écoulement turbulent bidimensionnel

Raoul Robert

Université Claude Bernard
69000 Villeurbanne Cédex

On utilise la théorie des états d'équilibre statistiques de l'équation d'Euler 2D pour construire un modèle de turbulence en évolution pour un fluide bidimensionnel incompressible faiblement visqueux.

THE USE OF THE "NATOKAP" STYLE FILE FOR DELIVERING MANUSCRIPTS IN LaTeX

The use of the NATOKAP style file for delivering manuscripts in LaTeX:

1. Use standard LaTeX. Anything allowed in LaTeX may be used.
Begin your article with:

```
\documentstyle{NATOKAP}
```

The following items are different:
a. 'Opening'-environment.
This environment helps to print the title, author, affiliation and date in the required format.

Use:

```
\begin {opening}
\title {title of your paper}          % This is a new command
\subtitle {if there is any}           % This is a new command
\author {one author's name}
\institute {address of your institute}  % This is a new command
\end {opening}
```

b. Do not use the command.
c. Acknowledgements.

The **acknowledgements** command can be used for making these. The text can be written by starting on the next line.

d. The commands:

```
\article
\book
\inproceedings
\bibitem{}          % The brackets must be empty!
```

have been implemented to ease the use of bibliographical data. They have been 'borrowed' from BibTEX, so BibTEX need not to be used. If the standard items required by the first three commands are not available (see your LaTeX manual), you are kindly requested to use **bibitem** and do the formatting yourself (e.g. with titles of articles).
N.B. If you would like to use BibTEX then you are invited to do so, but you need to indicate where the BibTEX files are.
If you encounter problems, please call: (78) 334 316
or send a message to email: SURF406@KUB.NL
Your document should be structured along the following lines:

```
\documentstyle{natokap}

% The 'OPENING'-environment
\begin{opening}
\title{}
\author{Author1}
\author{Author2}
\institute{Institute of Author1 and Author2}
\author{Author3}
\institute{Institute of Author3}
% etcetera
\end{opening}

\begin{document}

\begin{abstract}
Tekst of abstract.
\end{abstract}

\section{The Sectiontitle Should Not Be Typed with Capitals Only}

\begin{figure}
\vspace{??cm}  % You can indicate here the space your figure needs
\caption[]{caption text}
\end{figure}

\begin{table}
\caption[]{caption text}
% Here the data and structure of the table
\end{table}
% N.B.: You may use the "tabular" environment as well

\acknowledgements
If you would like to thank people or organizations

\begin{thebibliography}{}
% Space between second pair of brackets is empty
\end{thebibliography}

\end{document}
```

NO^TES FOR THE PREPARATION OF CAMERA-READY TYPESCRIPTS ON A LASERPRINTER

A. N. AUTHOR
Kluwer Academic Publishers
P.O. Box 17
3300 AA Dordrecht
The Netherlands

ABSTRACT. These instructions are intended to help and guide contributing authors and their typists when preparing articles as camera-ready output on a laserprinter.

1. Format Instructions

1.1. INSTRUCTIONS FOR TYPISTS

This document has been output on a QMS PS-800 laserprinter using the software *Flexicon* with justified text, automatic centering, italics, and boldface facilities. Margins were set to a width of 15.2 cm (36 picas), and each page was typed in Times 11 points letter for the main text with a 13 points base-to-base spacing of the lines. The small typeface was typed in Times 10 points letter with a 11 points base-to-base spacing of the lines. Each page was typed to a page depth of 21.6 cm (51 picas, 47 text lines). The headings used are:

2. First-Order Headings

These are typed in bold, upper and lowercase letters, with two lines of space above the heading, and one line below. The text after the heading begins at the left-hand margin (i.e. it is not indented as for new paragraphs).

2.1. SECOND-ORDER HEADINGS

These are typed in light small capitals, at the left-hand margin. There is one line of space above, and one line of space below the heading.

2.1.1. *Third-Order Headings*. Typeset in italics, with capital initial letters. Run the following text on in the same line, and leave one blank line above.

New paragraphs are to be indented 1 pica from the left-hand margin, with no extra space between the paragraphs.

Whatever your final decision on the layout of headings, etc., please ensure that you type each page to the correct dimensions, as found on this model. Set the word processor margins to the appropriate width, and type to the same overall depth on each page.

3. Paper

If you are using a word processor to produce your text on a laserprinter, you can use the normal A4 (210 × 297 mm) standard paper. The layout should be according to this model and the instructions mentioned in Section 1.1.

4. Corrections

It is assumed that, if you are using a word processor, you will re-output, where necessary, after editing and correction on-screen.

5. Special Problems

5.1. SPECIAL CHARACTERS AND SYMBOLS

It can happen that some equations or formulae in your work cannot be easily typed because they require the use of symbols which are not available on the apparatus you are using. Some common equipment, for example, will not handle the accents necessary for typing most European languages. A decision will have to be made, therefore, on how to treat these occurrences. Obviously, this is a matter of degree: if you are only citing names and/or titles of non-English-language works, for instance, it is possible and perfectly acceptable to insert accents by hand on the finished typescript, or to leave them out altogether.

For most mathematical and/or logical writings, there often exists an alternative formulation which avoids the use of 'difficult' characters. In cases where it is genuinely unavoidable, there is a wide range of unusual symbols and letters available on dry-transfer lettering. Handwritten equations are strongly discouraged.