Modification of the Euler equations for "vorticity confinement": Application to the computation of interacting vortex rings

John Steinhoff and David Underhill
The University of Tennessee Space Institute, Tullahoma, Tennessee 37388

(Received 29 June 1993; accepted 28 March 1994)

A new "vorticity confinement" method is described which involves adding a term to the momentum conservation equations of fluid dynamics. This term depends only on local variables and is zero outside vortical regions. The partial differential equations with this extra term admit solutions that consist of Lagrangian-like confined vortical regions, or *covons*, in the shape of two-dimensional (2-D) vortex "blobs" and three-dimensional (3-D) vortex filaments, which convect in a constant external velocity field with a fixed internal structure, without spreading, even if the equations contain diffusive terms. Solutions of the discretized equations on a fixed Eulerian grid show the same behavior, in spite of numerical diffusion. Effectively, the new term, together with diffusive terms, constitute a new type of regularization of the inviscid equations which appears to be very useful in the numerical solution of flow problems involving thin vortical regions. The discretized Euler equations with the extra term can be solved on fairly coarse, Eulerian computational grids with simple low-order (first- or second-) accurate numerical methods, but will still yield concentrated vortices which convect without spreading due to numerical diffusion. Since only a fixed grid is used with local variables, the vorticity confinement method is quite general and can automatically accommodate changes in vortex topology, such as merging. Applications are presented for incompressible flow in 3D, where pairs of thin vortex rings interact and, in some cases, merge.

I. INTRODUCTION

In many high Reynolds number flows, thin regions of concentrated vorticity often exist which convect through the flow field. Aerodynamic examples include many cases such as helicopter and aircraft flows where concentrated vortices are shed by lifting surfaces. Also, for basic turbulent flows, interacting thin, concentrated vortices may play a major role in the dynamics.[1]

Unfortunately, these flows are difficult to compute with conventional methods. These methods can be grouped into Eulerian and Lagrangian schemes. Eulerian use a fixed computational grid and discretized Euler or Navier–Stokes equations. Advantages of Eulerian methods are that they are very general and can automatically treat arbitrary vortical configurations. Disadvantages include the requirement of a high-order accurate discretization, together with a relatively large number of grid points within each vortical region. This is required mainly to prevent excessive spreading of vortices due to numerical diffusion errors.

Although standard Eulerian methods may be necessary for some flows where the details of the internal structure of the vortical regions must be resolved, there are features of flows with thin vortices that may allow the use of much more efficient computational techniques: If these vortical regions are much thinner than the main scales of the flow, the details of the internal structure may not be important. The important vortical features then may involve only the strengths and positions of the centerlines of these regions.

Lagrangian methods can easily take advantage of this simplification: A relatively small number of Lagrangian elements can be used to model the vortices (as vortex filaments). Then, effectively, the vortex internal structure will be modeled by the assumed structure of the elements. It is then expected that the solution will not depend significantly on this assumed structure as long as it is thin compared to the other length scales of the problem. Unfortunately, the treatment of general thin interacting vortical configurations, including changes in topology such as reconnection, is difficult with Lagrangian methods, and this technique becomes complicated unless the vortices remain separated and do not approach each other or any surfaces. Further, methods involving Lagrangian treatment of vorticity generally require *a priori* specification of points where vorticity separates from surfaces and generally cannot treat compressibility with shocks (see, however, Refs. 2–4).

Our objective is to develop a "vorticity confinement" method to treat these flows that is completely Eulerian and has the generality of conventional Eulerian methods, yet takes advantage of the relative lack of influence of the details of the internal dynamics of thin, separated internal structures, as Lagrangian vortex filament methods do. Such a method will treat vortices in a manner similar to that of shocks in Eulerian shock capturing methods, where the internal structure of an isolated shock is not important and not solved for in detail: only the strength and position of the centroid. The shock can then be resolved with a minimum number of grid points.

The basic approach involves modifying the continuum inviscid Euler equations of fluid dynamics to accommodate a new type of regularization. (Although the present application involves incompressible flow, the same ideas apply to compressible flow with some additional considerations conserving entropy. Preliminary results have already been obtained for these flows at subsonic speeds.[5]) The regularization in-

1070-6631/94/6(8)/2738/7/$6.00

volves adding two terms to the continuum momentum conservation equations. The first is a viscous term (this may only represent the inherent numerical diffusion of the solution method), and the second is a new term which depends only on local vorticity. The concept is very simple: The original Euler equations are very sensitive to numerical errors in their solution: Small amounts of diffusion originating from discretization errors cause thin vortices to undergo artificial spreading which continues to increase with time. However, the modified continuum equations, with the additional term, admit solutions which consist of thin vortices which convect in the local flow field with steady structure *without* spreading, even though there is a diffusive term. We call these steady structures, which are *co*nfined *vo*rtical regio*ns*, "*covons*." Since the continuum equations admit these *covon* solutions, numerical solutions of the modified equations, which may have numerical diffusion, also consist of the same type of thin vortical structures. Thus, while the unmodified equations require a very accurate, almost nondiffusive numerical method to solve for thin vortical regions, together with a large number of grid points, the modified equations do not: simple, efficient, low-order diffusive numerical methods can be used on relatively coarse grids.

Although numerical solutions, with the confinement term, of convecting vortices behave like finite-thickness Lagrangian vortex tubes when they are isolated over long times (*covons*), they appear to behave like solutions of the unmodified Euler equations when they interact closely with each other. (This may be because close interactions, such as those involving merger, occur on a convective time scale that is much shorter than the diffusive time scale at high Reynolds number[6] and during this short time, the additional vorticity confinement term remains balanced by the diffusion and does not have much effect. However, more study of this dynamics is required). For these reasons, the new technique allows the computation of flows with thin interacting vortices on fairly coarse computational grids (2–4 cells across a vortex tube) even when only a low-order discretization method (e.g., first or second order) is used.

It should be emphasized that we are not solving for the detailed dynamical evolution of a viscous fluid. Instead, we are taking a "zeroth-order" approach which is analogous to only solving the inviscid equation, with the idea of adding (weak) physical viscous terms later as an extension. These terms would have a special form consistent with our method and would be incorporated in the confinement term.

Our technique of adding a term to the equations of motion is in the same spirit of Carnevale,[7] who modifies the equations to extract energy with the goal of finding steady solutions.[8] Also, Harten[9] has modified the basic dynamic equations so that numerical diffusion in the computation of contact discontinuities (including vortex sheets) can be reduced.[10] Our modification, however, is only nonzero within vortical regions and has a natural multidimensional, rotationally invariant formulation. Further, it is formulated as an additive velocity correction, which apparently can easily be implemented in existing computer codes by a separate "CONFINE" subroutine called each time step during the numerical solution procedure.

II. VORTICITY CONFINEMENT METHOD

Some of the details of the basic method are presented in Refs. 11–14. Diffusion is an integral part of the basic method, and we include it in the continuum equations. (It represents the diffusive part of the numerical error when the equations are discretized.) Thus we really have a set of modified Navier–Stokes equations that represent the Euler equations with two regularization terms:

$$\nabla\cdot\mathbf{q}=0,$$
$$\partial_t\mathbf{q}=-(\mathbf{q}\cdot\nabla)\mathbf{q}-\nabla(p/\rho)+\mu\nabla^2\mathbf{q}+\epsilon\mathbf{k}, \tag{1}$$

where $\mathbf{q}$ is the velocity, p pressure, ρ density, and μ the diffusion coefficient. For the additional term, ϵ is a numerical coefficient which controls the size of the convecting vortical regions. The confinement term can take many forms. A particularly simple form is

$$\mathbf{k}=-\hat{\mathbf{n}}\times\boldsymbol{\omega},$$
$$\hat{\mathbf{n}}=\frac{\nabla\eta}{|\nabla\eta|}, \tag{2}$$

where

$$\boldsymbol{\omega}=\nabla\times\mathbf{q}$$

is the vorticity and η is a scalar field that has a local minimum on the centroid of the vortical region:

$$\eta=-|\boldsymbol{\omega}|. \tag{3}$$

Discretized numerical methods that we have developed to implement this correction are described in Refs. 11–14.

The main objective is to convect $\boldsymbol{\omega}$ back towards the centroid as it diffuses outward. This convection will increase the second derivatives of the vorticity near the center and, therefore, the diffusion term. Subsequently, a steady-state form (in a steady external velocity field) will result when the two effects become balanced. This will be a concentrated vortical region, or *covon*, that will convect with the local flow field without spreading, even with the numerical diffusion.

In the confinement term, $\hat{\mathbf{n}}$ is a unit vector pointing away from the centroid of the vortical region. The cross-product term (2) results in an added velocity along vorticity magnitude contour lines [three-dimensional (3-D) surfaces], which effects the vorticity convection without directly convecting fluid inward into the core and altering mass balance. The simple form (2) just specifies an added velocity along the contour lines that is proportional to the local vorticity magnitude. Other forms which, for example, go to zero at the vortex center can easily be defined.

For the numerical implementation reported here, we start with an efficient incompressible primitive variable Euler equation solution method. This involves, each time step, numerically convecting the velocity components and computing a pressure term to enforce mass conservation. It is similar to the "split-velocity" method of Ref. 15. The vorticity confinement method is then used to compute a correction each time step that controls the numerical diffusion in the convection. Other basic flow solvers can also be used. We have had good results with a standard compressible solver.[5] In addition to being very efficient, the use of an additive correction allows a smooth transition to an unperturbed, conventional primitive variable Navier–Stokes scheme where the grid is fine enough to accurately resolve the flow: The coefficient, ϵ, is made to depend on grid size so that the correction is simply turned off in those regions.

It can be seen (for steady external flow) that steady-state axisymmetric *covon* solutions exist even with diffusion present, for *any* (positive) value of ϵ, and that the size of these regions depends on ϵ. The basic point is that it may make more sense to discretize this set of regularized equations (1)–(3) which have thin, well-behaved vorticity distributions, even in the presence of numerical diffusion, than to discretize the unmodified, inviscid Euler equations which only admit vortical regions that continue to spread, if there is any numerical diffusion. Solutions to the regularized equations should be accurate for a range of *covon* sizes as long as they are thin—just like vortical solutions are accurate for a range of assumed filament sizes for a Lagrangian method as long as the filaments are thin. This accuracy over a range of *covon* sizes will result in an accurate solution for a range of values of the coupling parameter, ϵ.

The main features of the method can be seen by looking at an isolated axisymmetric vortex in a two-dimensional (2-D) uniform flow. We have, in a frame convecting with the vortex,

$$\partial_t \mathbf{q} = \mu \nabla^2 \mathbf{q} - \epsilon \hat{\mathbf{n}} \times \boldsymbol{\omega}.$$

If $\epsilon=0$, the solution is

$$T(r,t) = \frac{T_0}{r}\,(1 - e^{-r^2/2\mu t})$$

where we define

$$\mathbf{q} = \hat{\mathbf{e}}_\theta T(r,t) + \mathbf{q}_\infty,$$

where $\mathbf{q}_\infty$ is a uniform free stream velocity and $\hat{\mathbf{e}}_\theta$ is a unit vector in the azimuthal direction. This, of course, results in a continually spreading vortical region with radius $\sim \sqrt{2\mu t}$ and no nontrivial steady solution.

When $\epsilon > 0$, we can write an equation for the steady solution with $\partial_t \mathbf{q}=0$: The solution which is finite at $r \to 0$ is

$$T(r) = \frac{T_0}{r}\left[1 - \left(1 + \frac{r}{a}\right) e^{-r/a}\right],$$

where

$$a = \mu/\epsilon$$

is the (constant) length scale. This is the basic *covon* for axisymmetric 2-D flow.

This continuum solution should be a good approximation to the actual solution of the discretized equations with numerical diffusion and the vorticity confinement correction. This has been verified in Ref. 11. Other closed-form solutions for confined vortex sheets and some simple numerical tests of the confining method for vortex blobs are presented in Refs. 11–14. Also, other similar, local forms for the confinement term can easily be derived which result in velocity distributions that go to zero in the center of the vortex ($r \to 0$).

An important feature of the vorticity confinement method is that the correction is limited to the vortical regions. Another important feature concerns the total change induced by the correction in mass (δI_ρ) and vorticity ($\delta \mathbf{I}_\omega$), integrated over the vortical regions: In general, 3-D flow, because of the vanishing of $\mathbf{k}$ outside the vortical regions, these terms are zero. Another important quantity that can be conserved with the method is momentum. For the class of $\boldsymbol{\omega}$ distributions that have two axes of symmetry (such as elliptical distributions) in each (2-D) cross section, this term will vanish due to symmetry. The confinement term is intended to be used where thin vortical regions are convected over relatively long distances and where the velocity (except for that due to the vortex) is slowly varying on the scale of the vortex diameter. In that case, we would expect the viscous terms (either due to the basic numerical convection process or added explicitly) to symmetrize the $\boldsymbol{\omega}$ distribution since any strong, concentrated vortex will be spinning rapidly. Numerical results for a number of cases in both 2D and 3D confirm that the total momentum added to the flow by the confinement term is small even though the term itself is not. This point should still be tested for a wider range of cases. A correction scheme has been formulated to drive the added momentum to zero if it becomes significant, but has not yet been required.

The basic solutions in 2D to the modified flow equations are axisymmetric blobs with vorticity that decreases exponentially with radius from the center. Since vorticity is conserved, long 3-D *covon* "tubes" will have the same structure in each 2-D cross section. A very important feature of the confinement method, of course, concerns the interaction of these vortices with each other. The vortex interaction feature in 2D can be studied by considering the interaction of vortex pairs. For example, in the high Reynolds number limit, corotating vortices that are far apart should stay apart for a relatively long time and ones that are close should quickly merge.[16] We desire that *covons* approximate inviscid flow, except when they finally merge, when there should be a viscous-like behavior. This feature is necessary for a realistic vortex computation method. Numerical results for two corotating vortices are presented in Ref. 14, where it is shown that they have the desired behavior.

The roll-up of a thin vortex sheet with elliptical circulation distribution is also a standard test case for vortex dynamics methods. Numerical results of this flow are also presented in Ref. 14. An important feature here is the lack of sensitivity of the final main vortex position to the confinement parameter, ϵ, and that the results reproduce well the salient features of some similar experimentally measured flows.

Further results, reported in Refs. 14 and 5, concern the interaction of confined vortices with solid surfaces in 2D and 3D, respectively. Also, a preliminary application of the confinement method in 3D in generalized coordinates is described for helicopter rotor flow in Refs. 11–13.

It should be mentioned that for 3-D curved vortex tubes a self-induced velocity component is present which depends logarithmically on the tube thickness. This component will depend on the computed thickness in our method. For cases where this component is slowly varying or small, results will be independent of the value of ϵ over a range, as explained above. When this component is important and is not slowly varying, the tube thickness must be accurately represented. Then, an evolution equation for ϵ can easily be formulated to give the correct thickness.

The present version of the method uses a parameter ϵ that is independent of vorticity magnitude, and only depends on grid cell size. If this version is used with a low-order, diffusive, basic solver (1st or 2nd), it will concentrate large, spread vortical regions. As such, it is intended for problems with no actual spread vortices but only thin concentrated ones. These comprise a large set of important problems by themselves. Cases with diffuse, spread vortical regions will require a functional dependence of the confinement parameter on vorticity, such as a cutoff for lower values or smaller gradients. Such cutoffs are often used to limit the effects of lower order artificial viscosity terms to the shock region in shock capturing schemes. Alternatively, the "underresolved" components of the vorticity could be filtered out and treated separately (with confinement) with the resolved, nonconcentrated components treated in a standard way, without confinement.

III. NUMERICAL SOLUTIONS

Incompressible flow problems in 3D were solved for interacting pairs of initially identical circular vortex rings, for two initial configurations with parallel axes: coaxial and coplanar.

There are exact, steady *covon* solutions for axisymmetric 2-D flow and infinite straight tubes in 3D. However, *covon* solutions in the shape of single, isolated vortex rings, in 3D, while quickly relaxing to a typical shape (in about 10 time steps), slowly spread afterwards (over hundreds of time steps). This is apparently due to the self-interaction of different parts of the ring with the low-level vorticity surrounding other parts (recall that in 2D the steady-state vorticity decreases exponentially with distance for the *covon* solutions). This means that although there are stable 2-D and corresponding straight 3-D vortical configurations, other 3-D configurations where the exponential vortical "tails" overlap should slowly change due to shedding of low level vorticity. This shedding, however, is much slower than the loss due to numerical diffusion if vorticity confinement were not used and represents, we believe, a real physical effect for vortices with the exponential distribution that we have. It should vanish like the exponent of $1/a$ where a is the thickness, as a becomes small.

The determination of a proper value of ϵ for our problems is similar to the determination of proper values of a multiplying constant for artificial viscosity terms in shock capturing methods: There is a maximum value of the parameter for the particular grid and numerical method used, beyond which solutions are not stable or not smooth. Near but within this limiting value accurate solutions are obtained over a large range of values, so that "tuning" to find a particular value is never required. For our case, above a maximum value of ϵ; $\epsilon_{\max}$, vorticity distributions are not smooth. For $\epsilon_0 \leqslant \epsilon \leqslant \epsilon_{\max}$, the vortex cores are confined to about two grid cells, and accurate results are obtained. In our experience, $\epsilon_0 \sim \epsilon_{\max/4}$. As ϵ is reduced below ϵ_0 thicker vortices result that are still essentially stable. As ϵ approaches 0, of course, we recover the original numerical method with numerical diffusion, with vortices continuing to spread without limit.

In the study presented here, ϵ was kept fixed and no attempt was made to simulate vortex tubes with dynamically changing thickness due to stretching. As a result, the self-induced velocity will be approximately constant. The effects of variations in this component can easily be incorporated by computing values of ϵ (as a function of position) to give the changes in thickness due to stretching. This should not be large for the cases treated here.

For each of the two cases presented below, essentially the same results were obtained with two different values of ϵ differing by a factor of 2 (except that the vortex thicknesses varied).

In each of the cases studied here, the flow field of a single vortex ring which had converged to the (approximately) steady "*covon*" solution was used as a starting point. The flow field due to this single ring was then translated and superimposed with itself to form the initial conditions.

Periodic boundary conditions were used in a cubical domain with a Cartesian computational grid. In each case the CFL number, based on maximum velocity in the field, was 3–5. As a check, solutions were obtained for a time step differing by a factor of 2. In each case, the effect of changing the time step was found to be small. All runs were done on a SPARCstation 2 work station and required less than one hour CPU time for a complete solution. In each case, no explicit dissipation term was added and the dissipation inherent in the basic numerical solution method served, together with the added confinement term, to define the *covon* solution.

A. Coaxial rings

Coaxial vortex rings were simulated on a 48^3 grid. The single confinement coupling parameter, ϵ, which controls the thickness of the rings, was set to give vortex rings with thickness (defined by 30% maximum $|\boldsymbol{\omega}|$ contour) of 3 grid cell spacing for this case. In Fig. 1, a sequence of plots of the 3-D contour corresponding to this vorticity magnitude contour (the maximum value of vorticity magnitude only changed slightly during the run). It can be seen that the vortices "leap frog" through each other several times.

In the limit where there is no internal deformation of the vortices (other than changes in the radius) and the vortical fields of the rings to not overlap, this leap frogging would continue indefinitely.[17] However, the *covon* rings have a vorticity field that decreases exponentially with distance and there is some overlap and, hence, interaction. This is probably the main reason that the rings eventually coalesce (although dynamics of the internal structure may also be a reason). For larger rings with small thickness the effect should become exponentially small.

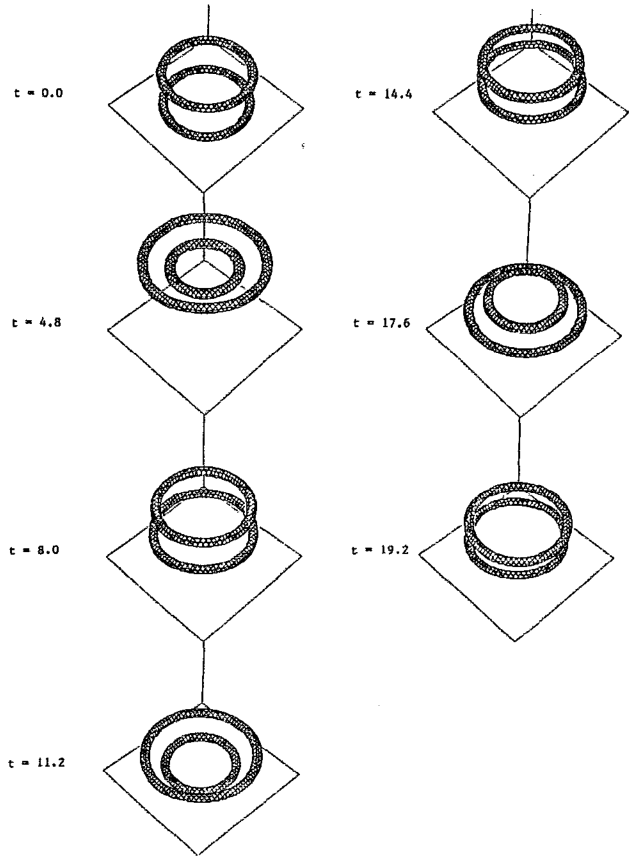

FIG. 1. Coaxial vortex rings.

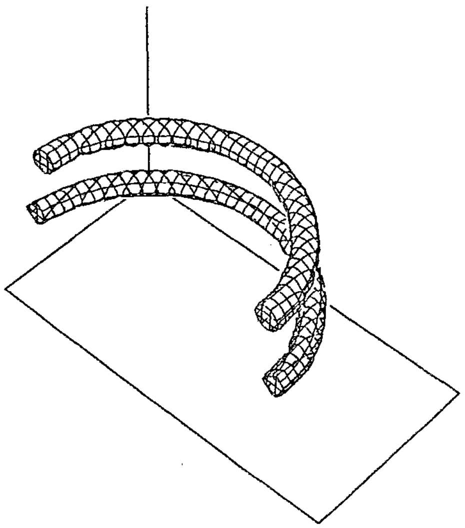

FIG. 2. Cutaway: high level contours.

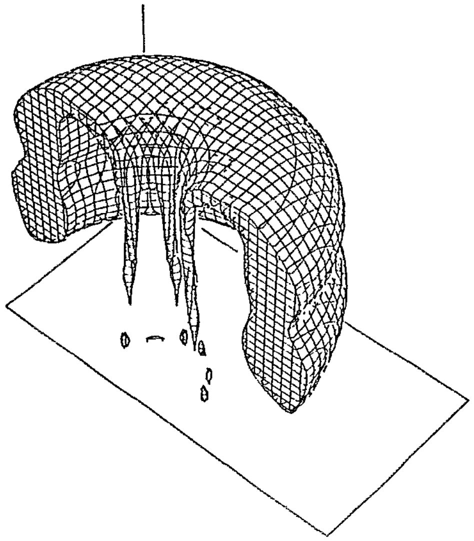

FIG. 3. Cutaway: low level contours.

To demonstrate the efficiency of the method, a cut-away view of the 30% contour for this case $t=14.4$ is displayed in Fig. 2. It can be seen that most of the vorticity remains confined to about 2 grid cells. A case with thicker rings and smaller initial separation was run and a lower level (5%) vorticity contour displayed in Fig. 3, with a cut-away view. The interaction for this case is clearly seen. This low level interaction, which should become very small when the ring is large compared to its thickness, was expected, as explained above. (Note that this level is much lower than usually presented in published computational results, such as Ref. 18.)

B. Coplanar rings

Two identical vortex rings were placed close to each other in the same plane as starting conditions. As in the above case, the computation was done on a 48^3 grid. The evolution of this flow can be seen in Fig. 4 where the contour corresponding to 30% of maximum value of vorticity magnitude is plotted for a sequence of time steps (as in the first case, this maximum value did not change significantly during the run). The results are very close to the computational results published in Ref. 18 as well as the experimental results published in Ref. 19. In the latter, velocity values were measured at a set of points forming a grid in space and then vorticity values computed from these velocities. Contours of constant vorticity magnitude were then plotted, as in our computation.

After the last time shown in Fig. 4, ($t=22.4$), there was no more significant bridging in our computation and the vortical distribution remained in basically the same shape. In other simulations we have seen that by varying the initial conditions, such as increasing the circulation, and moving the rings closer to each other, there was continued bridging and vortical shape changes. Since the experimental results of Ref. 19 were not presented beyond approximately $t=22.4$, we only displayed results to this point.

IV. CONCLUSION

A method has been presented for computing flows with thin, concentrated vortical regions, which should be important for many high Reynolds number flows. The basic idea involves modifying the continuum Euler equations by adding a simple, local term to the momentum equations. The modi-

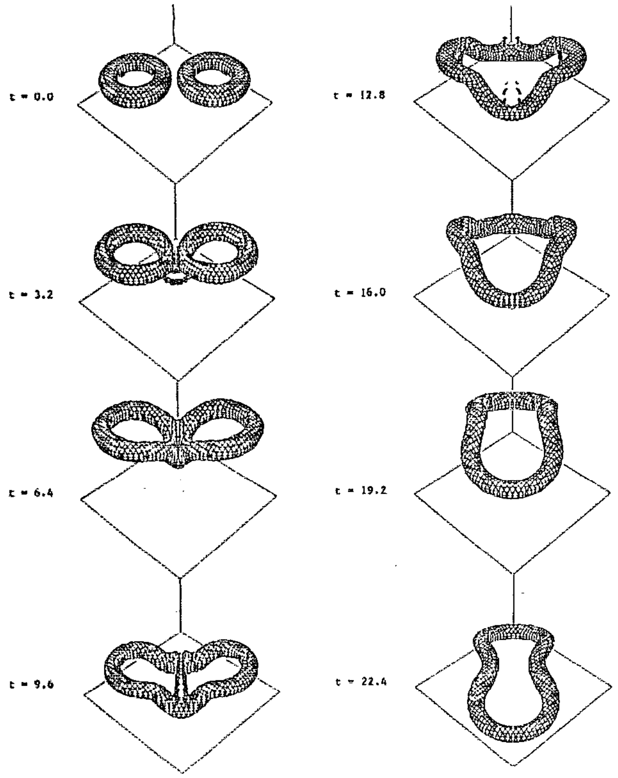

FIG. 4. Coplanar vortex rings.

fied equations, even with the addition of diffusion, admit solutions—*covons*—which are concentrated vortical regions which convect with fixed internal structure—without diffusing. It is argued that these modified, or "regularized" equations may be better suited for numerical solution than the unmodified ones—which result in vortices that spread if there is *any* numerical diffusion.

For the numerical solution, discretized mass and momentum conservation equations are solved on a fixed Eulerian grid, as in conventional Euler/Navier–Stokes methods, using a conventional code. However, a "vorticity confinement" correction is applied to the velocity each time step, by calling an added CONFINE subroutine. The effect of the vorticity confinement term is to confine concentrated vorticity to *covon* states or thin regions extending over a small number of grid cells, as they convect through the flow. The internal structure of these vortical regions attains a fixed, steady-state form without spreading, even though the basic, discretized momentum equations involve numerical diffusion.

Applications of the method to incompressible flows involving vortex ring interactions in 3D were presented. These show the effectiveness of the method even when coarse computational grids are used. In general, the vortices behave like inviscid solutions: However, at small scales, when the vortices finally merge, salient features of viscosity are automatically simulated. An important point is that the computed vortex positions are not sensitive to the confinement parameter over a large range of values. This is related to an independence of the solution on the exact thickness of thin vortices, which is exact for axisymmetric 2-D vortices and straight 3-D tubes, and approximate for general 2-D vortices and curved 3-D tubes. Further, the main features of conventional high-order calculations with higher resolution were replicated, as well as observed experimental results.

The method has already been used in a preliminary application to a realistic helicopter rotor flow in 3D. However, additional testing is required for more complex flows and for applications to compressible transonic flows. For example, the use of general nonisentropic, compressible flow solvers may require an additional entropy confinement term to avoid entropy diffusion away from concentrated vortical regions.

Interesting extensions include the simple possibilities of having vorticity-dependent upper and lower cutoffs for the coupling constant, ϵ. These should, respectively, accommodate "waterbag" constant-vorticity models and smoothly varying background vorticity distributions. Further extensions could include extra terms to "encode" in an evolution equation for ϵ, as well as additional parameters, desired features of the internal dynamics of the simulated vortices, such as turbulence models or physical laminar viscous effects which result in a slow physical growth of vortex sizes.

ACKNOWLEDGMENTS

This work was partially supported by the U.S. Army, Contract No. NAS2-13079, and Flow Analysis, Inc., Contract No. FAI91-1. The computer code used for the study was written by the second author with assistance from Thomas Mersch. Also, the computer runs were done on a Sun SPARCstation 2 work station and required less than one hour of CPU time each.

[1] A. Pumir and R. M. Kerr, "Numerical simulation of interacting vortex tubes," Phys. Rev. Lett. **58**, 1636 (1987).

[2] J. Steinhoff, K. Ramachandran, and K. Suryanarayana, "The treatment of convected vortices in compressible potential flow," Proceedings of the AGARD Symposium on Aerodynamics of Vortical Type Flows in Three Dimensions, Rotterdam, Netherlands, 1983.

[3] J. Steinhoff and K. Ramachandran, "Free wake analysis of compressible rotor flows," AIAA J. **28**, 426 (1990).

[4] J. Steinhoff and K. Ramachandran, "A vortex embedding method for free wake analysis of helicopter rotor blades in hover," Vertica **13**, 133 (1989).

[5] C. Wang, J. Bridgeman, J. Steinhoff, and Y. Wenren, "The application of computational vorticity confinement to helicopter rotor and body flow," Proceedings of the 49th American Helicopter Society Meeting, St. Louis, Missouri, May 1993.

[6] W. T. Ashurst and D. I. Meiron, "Numerical study of vortex reconnection," Phys. Rev. Lett. **58**, 1632 (1987).

[7] G. K. Vallis, G. F. Carnevale, and W. R. Young, "Extremal energy properties and construction of stable solutions of the Euler equations," J. Fluid Mech. **27**, 133 (1989).

[8] This was pointed out by Dr. J. Z. Wu.

[9] A. Harten, "The artificial compression method for computation of shocks and contact discontinuities: III. Self-adjusting hybrid schemes," Math. Comput. **32**, 363 (1978).

[10] This was pointed out by Professor S. Osher.

[11] J. Steinhoff, H. Senge, and Y. Wenren, "Computational vortex capturing," UTSI preprint (1991).

[12] J. Steinhoff, Y. Wenren, T. Mersch, and H. Senge, "Computational vorticity capturing: Application to helicopter rotor flow," AIAA Paper No. AIAA 92-0056, Reno, Nevada, January 1992.

[13] J. Steinhoff, H. Senge, and Y. Wenren, "Computational vortex capturing—reduction of numerical diffusion," Proceedings of the First International Conference on Computational Fluid Dynamics, Davis, California, October 1991.

[14] J. Steinhoff, C. Wang, D. Underhill, T. Mersch, and Y. Wenren, "Computational vorticity confinement: A non-diffusive Eulerian method for vortex-dominated flows," submitted to Intl. J. Num. Methods Fluid (1992).

[15] J. Kim and P. Moin, "Application of a fractional-step method to incompressible Navier–Stokes equations," J. Comput. Phys. **59**, 308 (1985).

[16] E. A. Overman and N. J. Zabusky, "Evolution and merger of isolated vortex structures," Phys. Fluids **25**, 1297 (1982).

[17] A. Sommerfeld, *Mechanics of Deformable Bodies* (Academic Press, New York, 1950).

[18] S. Kida, M. Takaoka, and F. Hussain, "Collision of two vortex rings," J. Fluid Mech. **230**, 583 (1991).

[19] Y. Oshima and N. Izutsu, "Cross-linking of two vortex rings," Phys. Fluids **31**, 2401 (1988).