AN EFFICIENT VORTICITY CONFINEMENT BASED LIFTING SURFACE METHOD FOR ROTOR WAKE COMPUTATIONS

John Steinhoff[1] and Yonghu Wenren[2]

[1] University of Tennessee Space Institute
Tullahoma, TN, USA
e-mail: jsteinho@utsi.edu

[2] Flow Analysis, Inc.
Tullahoma, TN, USA
e-mail: tiger@flowanalysis.com

Key words: Rotorcraft, Wake, Vorticity Confinement, Weak Solutions, Euler Equations

Abstract: There is a very important aspect of rotorcraft aerodynamics that cannot be computed, within engineering time constraints, with the engineering methods now in common use. This is the rotor wake.

Unlike for conventional transport aircraft, there are a very large number of flight regimes to be treated: rotorcrafts do not fly close to a specified design speed and a myriad of effects can be important. Thus, during a rotorcraft design phase, a very large number of computations must be done, so that each computation must be done in a short time. The computation of the wake is a major stumbling block to achieving this goal.

There are two main approaches currently used in aerodynamic computations: conventional CFD to approximate the discretized Euler equations, and panel/vortex lattice methods. Both are not able to solve the general wake problem without user input or within feasible computing time. However, there are basic ideas implicit in both approaches that can lead to a fully satisfactory, efficient treatment of all important effects of the wake. We first describe the features of each method that make it inadequate for the general problem, and the basic idea in each that can be used to make a new approach that will be effective. This new approach is known as "Vorticity Confinement".

1 INTRODUCTION

There is a very important aspect of rotorcraft aerodynamics that cannot be computed, within engineering time constraints, with the engineering methods now in common use. This is the rotor wake. This includes first the main rotor, but also the tail rotor. Further, multi rotor configurations are to be treated, as well as wake interactions with following blades, including blade-vortex interaction – (BVI), and interactions with the fuselage, ground and other nearby rotorcraft or personnel. Further, unlike for conventional transport aircraft, there are a very large number of flight regimes to be treated: rotorcrafts do not fly close to a specified design speed and a myriad of effects can be important. Thus, during a rotorcraft design phase, a very large number of computations must be done, so that each computation must be done in a short time.

Specific examples where the unsteady wake is important include:
-- Blade loading in hover (including body)
-- BVI
-- Interaction of the tail rotor with the main vortex following quick turns on deck
-- Interaction of another landing helicopter with a wake
-- Ability of ground personnel to walk in the strongly fluctuating wake near the ground caused by shed individual vortices.
-- Sand/snow pickup due to the individual shed vortices (brownout, whiteout)
-- Vibratory airloads caused by individual blade vortices impinging on the fuselage
-- Effects of vortices on operations such as crop dusting
-- And many more

There are two main approaches currently used in aerodynamic computations: conventional CFD to approximate the discretized Euler equations, and panel/vortex lattice methods. Both are not able to solve the general wake problem without user input or within feasible computing time. However, there are basic ideas implicit in both that can lead to a fully satisfactory, efficient treatment of all important effects of the wake. We first describe the features of each method that make it inadequate for the general problem, and the basic idea in each that can be used to make a new approach that will be effective. This new approach is known as “Vorticity Confinement” (VC).

The main objective is to demonstrate a computational method that is able to compute detailed wake effects in most of the problems listed above. This especially includes fluctuations due to individual blade tip vortices and shed vortex sheets. Further requirements are that no input involving the wake geometry be required. Thus, separate codes for separate problems with different configurations will not be required. Further, the computation is to run on a PC (~ 2GHZ) in less than a day for most of the problems.

In this paper the basic VC method will be explained and a number of results reviewed, including a new, VC based lifting surface method for the blades.

2 BASIC METHOD

In this section, first, the basic algorithm will be described. We feel a somewhat detailed description is necessary because it is quite different than currently used schemes. We first quickly review the two most widely used methods, and argue that each is inadequate. However, each employs an implicit assumption that, properly used, leads to an approach – “Vorticity Confinement” (VC) that has all the properties that we require.

2.1 Review of Panel/Vortex Lattice Methods (P/VLM’s)

These methods were first used in the early 1930's to treat the vortex sheet shed by a wing [5-6]. More extensive use, including representation of solid surfaces, followed from use of modern computers in the 1960's. A basic ingredient, the representation of the vortex as a vanishingly thin velocity discontinuity, was already developed much earlier, when it was realized that the details of the large velocity gradient in a boundary layer (BL) could be often ignored and that the internal structure of a thin, attached BL did not significantly affect the outer, inviscid flow. Thus inviscid methods could be effective for many problems.

The same basic assumption is, of course, used in P/VLM’s representation of shed vortex sheets as is used in the above "bound" vortex sheets – that the internal structures can be ignored, as long as they are thin.

We believe that these assumptions are very important for high Reynolds number flows, and have little to do with the main drawbacks of P/VLM's. These drawbacks do not come from the need to solve the internal structure of the vortical layers. Rather, we believe that P/VLM’s are not as useful as they could be for general problems because of the need to specify the topology of each panel or vortex sheet by a collection of marker coordinates, which lie on the sheet. Thus, typically, for realistic problems which shed thin vortex sheets, the number of sheets and topology of each must be specified beforehand (even though the locations and vorticity strengths of the markers are computed).

The main point in describing P/VLM's, which have been used very successfully in many applications, is to emphasise that they have demonstrated that there is no need to resolve the internal structure of the sheets. Rather, certain conservation laws must only be obeyed as integrals through the sheets (for example, that a sheet or filament convects with the flow can be shown from integrating the momentum conservation law over a thin "test function" containing the sheet). Thus, the solutions obtained with P/VLM's are a type of "weak" solution of the Euler equations.

2.2 Review of Eulerian Methods

For the reasons associated with marker allocation mentioned above, we feel we must use a fixed-grid Eulerian method, where vortex sheets are "captured" regardless of their location, in a similar way to shock capturing, described below. However, we still want to keep the P/VLM feature that they are treated as weak solutions, with no requirement that their structure be resolved. Then, as explained, there will be no need to specify the topologies, number of sheets etc., but a very efficient computation will be possible.

There is an important analogy between our treatment of vortex sheets (both bound and convecting) and shock capturing that will be explained in this section:

Initially, in the 1960's, when computers were first being used to treat 2-D transonic flow, "shock fitting" proved to be very effective. There, as with P/VLMs, a set of markers was used to define each shock surface. However, as more complex cases were attempted, especially in 3-D, the specification of the topologies, the number of shocks, etc, proved complex and "shock capturing" methods became more useful for general problems. In these methods, shocks were automatically captured wherever they formed as weak solutions, without trying to resolve the internal structure, which would have been prohibitive. The method described below, "Vorticity Confinement" (VC), represents the same type of advancement, but for vortex sheets. This new method, like shock capturing, will also be shown to overcome the problem of using a fine grid to reduce numerical dissipation inherent in trying to resolve the internal structure of thin vortices.

2.3 Vorticity Confinement Approach

First, for small scale vortices, VC allows us to eliminate numerical dissipation so that it minimizes the effect on the numerical solution, down to scales of a small number (~2) of grid cells sizes (h). In this way, VC allows us to maximize h for a given resolution. This is important because every factor of 2 increase in h leads to a factor of 16 decrease in computing time. For general small vortical scales, it is not possible to avoid numerical error over long convection times, even if expensive high order conventional CFD schemes are used. However, if the small scales in the field consist of vortex sheets and filaments, as is often assumed, this is easily accomplished – with VC.

3 VORTICITY CONFINEMENT – BASIC CONCEPTS

As explained, the basic VC concept is related to that of similar methods also involving thin structures—shock and contact discontinuity capturing. Accordingly, before describing the VC method, analogous, relevant features of these methods will be briefly described, since they have been used extensively for some time and are very familiar to the CFD community. Then, basic concepts of the new method (VC) will be reviewed. These points are known to people familiar with VC and more conventional discontinuity capturing methods, but may be helpful to people to whom it is new. We will use shock capturing as an example.

Shock capturing methods have, of course, received an extremely large amount of attention in the CFD community and have proven to be extremely important. These methods typically use only a moderately sized inviscid computational grid in the shock region. This is possible because only the *essential* physics of the shock (as far as the flow problem being solved) is retained. By “essential physics” we mean those features that affect the flow external to the shock interior. These features include computed shock thickness, which does not have to be as small as the physical thickness but, like the physical thickness, must be small compared to the main length scales of the problem. They also include the requirement that conservation laws, integrated through the shock, are preserved. In this way, for many problems that do not depend on the details of the shock internal structure, accurate flow solutions have been obtained with specially developed numerical “shock capturing” algorithms. In these methods the detailed, accurate solution of partial differential equations (pde)’s (for example, Navier-Stokes equations) for the internal shock structure have been avoided. This has been important since it avoids the requirements of a very fine computational grid within the structure, and very time consuming viscous computations there. These ideas, which go back to Von Neumann and Richtmyer (1950), Lax (1957) and others, involve the concept of “weak solutions” of pde’s where, in the inviscid limit, discontinuous features can be treated. After discretization these ideas typically allow the effects of shocks to be approximated over as few as 1-3 grid cells.

The question naturally arises as to whether similar efficient “capturing” treatments of thin vortical features are also possible, which would result in similar benefits. One difference, however, is that with shocks, unlike vortical regions, characteristics slope inward toward the shock, which naturally tends to steepen during a computation. As a result, modelling shocks is simpler than modelling thin vortical structures and other contact-like discontinuities which naturally tend to spread due to numerical discretization errors and require stabilizing numerical diffusion. This results in the requirement that a "steepening" or "Confinement" term be added to prevent artificial spreading.

Vorticity Confinement has been specifically formulated to effectively treat the difficult-to-compute concentrated vortical regions with the same basic philosophy as shock capturing. Although developed independently, the method, in its one–dimensional form, has some relation to Harten's "artificial compression" scheme for one-dimensional compressible flow [7]. However, the VC formulation is much simpler. Also, an important feature for capturing thin vortical regions is that VC is intrinsically multidimensional and rotationally invariant (in the continuum limit) and is most efficient at low speed, an important feature for rotorcraft. A number of recent papers [2, 8-14], mentioned below, describe the use of VC for incompressible flow. Further extensions to compressible flow, including supersonic, have been recently developed [4, 15-18] but will not be described here, since we want to concentrate on low speed flow with as few extra complications as possible. As with shock capturing, it is understood that the details of the internal structure of thin vortical regions will not be accurately treated, unless special models are developed for them. As stated above, we assume here that these details are not important and that simple capturing alone is sufficient, i.e, they are treated as "weak" solutions.

As background, we first mention some previous examples for which VC has been shown to be effective. These include convecting vortex rings, which can be convected with no spreading, yet can merge with no requirement for special logic [2]. They also include thin shed wingtip vortices which can be computed over arbitrarily long distances [19-20] and exhibit Crow instability, including merging [19]. Computations of both of the above phenomena show close agreement with experiment. For trailing vortex convection over very long distances (many kilometers) the method can also serve as a zeroth order approach in this case, since turbulence eventually induces a very slow spreading. This effect can then be simply modeled within the VC framework, again without using very fine grids or high order methods. Examples also include a very simple and inexpensive RANS substitute for attached and separating boundary layers. This is described in Refs. (17, 21-23).

An important point that should be emphasized is that, at high Reynolds number, most vortical regions will be turbulent. Hence, *any* computational method must involve, explicitly or implicitly, a numerical model for the small-scale structure, since it is not feasible to directly solve the Navier-Stokes equations for this (time dependent) structure. The structure obtained with VC when a small-scale vortical region is captured can, thus, be thought of as just such a model, but one that is very efficient to compute. Further, this model is intrinsically discrete, defined over only a few grid cells, and is not meant to be an accurate solution of a model pde. The rationale for taking this approach is that it is difficult to resolve pde's for a thin vortical structure over long distances, even with higher order methods, if it is spread over only a small number of grid cells (2-4). This is due to the well-known fact that the accuracy, or *order*, of a method is only an *asymptotic* estimate of the behavior of the error, valid for large N, the number of grid cells across the vortical region. N=2-4 is not sufficient to apply such an estimate. Since VC is meant to *capture* the feature, and not accurately solve a model pde, it gets around this problem. Further, many features of the flow external to the core, or interior of a vortical region are not sensitive to the details of the internal structure. For example, in 2-D, vortices tend to evolve to an axially symmetric state [24]. Then, the only requirements for accurately determining the induced flow external to the core are that the total circulation is conserved, that the vortex centroid have the correct location, and that the core does not spread due to numerical effects.

3.1 Vorticity Confinement Methodology

Two formulations of VC have been developed which have similar properties: The first, "VC1", involves first derivatives of velocity [8, 25], while the second, "VC2" involves second derivatives [26]. Starting from an initial condition more spread than the final structure, VC1 acts essentially as an inward convection and relaxes to the final structure more quickly than VC2, which acts, initially as a negative, second order diffusion. Only the VC1 version will be discussed in here, since it appears to be the more robust of the two versions and gives good results. (VC2 has some additional conservation properties which may be important for very long time convection but which are not necessary here.) We will refer to VC1 as just VC.

In the implementation of VC, for thin vortical regions in incompressible flow, "Confinement" terms are simply added to the conventional, discretized momentum equation. The governing equations with VC are then discretizations of the continuity and momentum equations, with added terms:

$$\nabla \cdot \vec{q} = 0 \qquad (1)$$

$$\partial_t \vec{q} = -\vec{\nabla} \cdot (\vec{q}\vec{q}) + [\mu \nabla^2 \vec{q} - \varepsilon \vec{s}] \qquad (2)$$

where $\vec{q}$ is the velocity vector, p is the pressure, ρ is the density, and μ is a diffusion coefficient that includes numerical effects due, for example, to discretization of the first right hand side (convection) term. (We assume that the Reynolds number is large and that physical diffusion is much smaller than the added terms). For the last term, $\varepsilon\vec{s}$, ε is a numerical coefficient that, together with μ, controls the size and time scales of the convecting vortical regions or vortical boundary layers. For this reason, we refer to the two terms in the brackets as the "Confinement terms". The vector $\vec{s}$ is defined below.

Equation (2) involves constant μ and ε, which is sufficient for many problems. If these are not constant, such as, for example, when the grid spacing is not constant or models are used for them, then these quantities can be taken inside any differential operators in the corresponding terms.

The *pair* of confinement terms, which represent spreading, or positive diffusion and "contraction", or negative diffusion, together create the confined structures. Stable solutions result when the two terms are approximately balanced. In this way, corrections are made each time step to compensate for any perturbations to the vortical structure caused by convection in a non-constant external velocity, discretization error in the convection operator, or the pressure correction. The parameters μ and ε then essentially determine the thickness of the resulting vortical structure and the relaxation rate to that state.

It should be emphasized that stable, equilibrium structures result for a wide range of values of these parameters: i.e. – no "tuning" is necessary for the problems we will treat. In general, for boundary layers and isolated, convecting vortex sheets and filaments, computed flow fields *external* to the vortical regions are not sensitive to the internal structures, and hence to the parameters ε and μ, over a wide range of values, and they can serve as a good approximation to the physical vortex. For example, a general thin, concentrated vortex will physically tend to evolve to an axisymmetric configuration [24]. Further, even a rapidly rotating nonsymmetric configuration (computed or physical) will be approximately axisymmetric when averaged over a short time [27]. Then, it is well known that the flow outside an axisymmetric two-dimensional vortex core is independent of the vortical distribution, and hence, for the computed case, will not depend on ε and μ as long as the core is thin (and the filament curvature is large, so that the flow is approximately two-dimensional in a plane normal to the filament). Therefore, the issues involved in setting these parameters will be similar to those involved in setting numerical parameters in other standard computational fluid dynamics schemes, such as artificial dissipation in many conventional shock-capturing schemes, which, as explained, are closely analogous. Further, for turbulent blunt body wake flows, preliminary studies—described in Ref. (28)—suggest that ε can be used to parameterize finite Reynolds number effects, since it controls the intensity of the smallest resolved vortical scales (this is the subject of current research [29].

An important feature of the Vorticity Confinement method is that, for low speed (incompressible) flow, the Confinement terms are non-zero only in the vortical regions, since both the diffusion term and the "contraction" term vanish outside those regions. Thus, even if there is a second order isotropic numerical diffusion associated with the convection operator, and the diffusion operators are only second order, outside the vortical regions the resulting accuracy of these terms can be third or fourth order, since this diffusion is just the negative curl of the vorticity.

A final point concerns the total change induced by the VC correction in mass, vorticity and momentum, integrated over a cross section of a convecting vortex. A pressure – projection method [30] is used to solve equations (1) and (2), so that mass is automatically conserved. Vorticity is explicitly conserved because of the vanishing of the correction outside the vortical regions. Finally, (in the VC2 formulation), momentum is also exactly conserved [26] because the VC terms added to the momentum equations have a spatial derivative operator in front. This is not exactly satisfied in the VC1 formulation, but errors due to the lack of momentum conservation have been shown numerically to be small for the flows of interest here.

An important point, however, is that exact momentum conservation, in some cases, may not be as important as other features (such as, in our case, ensuring that a convecting vortex remain thin) and should not be regarded as an absolute requirement (see, for example, the basic CFD textbook—Ref. (31, pg. 60)).

Many basic numerical methods could be used for space and time discretization. We use a simple first order Euler integration in time and second order in space with, as stated, a pressure – projection method to enforce mass conservation. In conventional CFD schemes higher order methods often must be used, usually to reduce numerical diffusion and hence attempt to reduce spreading of thin vortical regions. Vorticity Confinement eliminates this problem and avoids the boundary condition complexity and computational cost of the higher order methods.

Another numerical issue involves the regularity of the grid. It is important to realize that, since a convecting vortex or separated boundary layer is captured directly on the grid, over a few grid cells, large grid aspect ratios or rapidly varying cell sizes need not be used. If these are avoided, VC will result in a dynamics that is close to rotationally invariant. Some modifications can be made, however, to accommodate non-uniform grids if the aspect ratio is not too large [32-34].

As explained, the two different formulations, VC1 and VC2, have somewhat different dynamics, since they differ in the order of the derivative in the contraction term. The one used for this proposal (VC1) has been described in a number of publications and only a few details will be presented here.

3.2 VC1 Formulation

This formulation involves an expression for the "contraction term", $\vec{s}$, that does not explicitly conserve momentum:

$$\vec{s} = \hat{n} \times \vec{\omega} \,. \tag{3}$$

For convecting vortices,

$$\hat{n} = \vec{\nabla}\eta \Big/ \left|\vec{\nabla}\eta\right| \tag{4}$$

where

$$\eta = \left|\vec{\omega}\right| .$$

For boundary layers, $\hat{n}$ is a unit vector parallel to the local normal. This term essentially convects vorticity within a thin vortical region either along its own gradient or along the local normal, from the edge, or region of lower magnitude, toward the center, or region of larger magnitude. As the structure contracts and the gradient increases, the "expansion" term, which is a linear diffusion, increases until a balance is reached. (This is a well-known property of convection-diffusion phenomena.) As explained, due to the rapid rotation of convecting concentrated vortices, any non-conservative momentum errors are almost completely cancelled and the method has proved to be sufficiently accurate for many problems.

4 IMMERSED BOUNDARY MODEL

VC can also be especially effective as a simple treatment of "immersed" surfaces embedded in a regular, coarse, non-confirming grid. To enforce no-slip boundary conditions on immersed surfaces, first, the surface is represented implicitly by a smooth "level set" function, "*F*", defined at each grid point. This is just the (signed) distance from each grid point to the nearest point on the surface of an object – positive outside, negative inside. Then, at each time step during the solution, velocities in the interior are simply set to zero. In a computation using VC, this results in a thin vortical region along the surface, which is smooth in the tangential direction, with no "staircase" effects.

The important point is that no special logic is required in the "cut" cells, unlike many conventional schemes: only the same VC equations are applied, as in the rest of the grid, but with a different form for $\hat{n}$, which is computed as the local surface normal. Also, unlike many conventional immersed surface schemes, which are inviscid (i.e. cannot use no-slip conditions) because of cell size constraints, there is effectively a no-slip boundary condition. This results in a boundary layer with well-defined total vorticity and which, because of VC, remains thin, even after separation.

The method is especially effective for complex configurations with separation from sharp corners. Also, even with constant coefficients, it can treat separation from smooth surfaces, as shown in Ref. (28).

Results (tangential velocity contours) are presented in Ref. (35), for a computation on a uniform Cartesian with 128x128 cells, for flow over an immersed, oblique flat plate in zero pressure gradient. For this flow, the velocity is simply set to zero at nodes below the surface. The resulting large, diffusive numerical errors can be seen, for the case with no VC. It can be seen that these are eliminated, to plottable accuracy, for the case with VC. All grid artefacts such as "staircase" effects and large diffusive boundary layers are eliminated because VC now has a smoothing effect in the tangential direction, but compresses the vorticity in the normal direction, maintaining a thin, smooth boundary layer. A number of projects involving blunt bodies immersed in regular coarse grids have been done using VC. These are reviewed in Sec. 6. They make it clear that VC is effective for treating rotor bodies and their effect on the rotor wake.

5 ROTOR BLADE TREATMENT

Besides being able to treat the wake and body, it is important to treat the rotor blades efficiently. We have implemented two approaches, which will serve as options in the final code. The first involves a Chimera treatment, where a blade fitted grid is used near the blades, which is tightly coupled to an outer Cartesian grid through interpolation of velocities at each time step [43]. This will not be described here. The second, which is more economical, is a lifting surface treatment where the blades move through a uniform Cartesian grid. The latter has been loosely coupled to a blade fitted grid computation to determine the circulation at each span station of the lifting surface representation. As many users have their own circulation computational methods, such as table lookups, these could also serve as economical options.

5.1 Lifting Surface Method

The VC based lifting surface method uses a localized momentum source rotating at the blade position at each time step. This source, together with the basic flow solver on the grid, give the same effect (for incompressible flow) as a Biot-Savart based panel method – except for numerical diffusion. Two difficulties in conventional solvers with this approach are that vorticity created by the momentum source diffuses away both from the local momentum source, as well as the shed vorticity in the wake. This requires a VC correction. To demonstrate this, a computation was first done for an isolated wing to test this concept. A vorticity isosurface is shown in Fig. (1 a) with no VC and Fig. (1 b) with VC (same isosurface level). A level of 20% of that value is shown in Fig. (2) to show the shed vortex sheet (with VC). In the center plane spanwise vorticity contours are shown (along with the grid) for a computation without VC and with VC (Fig. (3)). It can be seen that the momentum source is effective at creating the vorticity distribution (and, hence velocity field) and that VC is important in confining it.

6 COMPUTATION OF FUSELAGE FLOW

The first objective for this computation was to quantify the model used for treating body surfaces in Cartesian grids: in particular, the ability to accurately predict surface pressure distributions. First, predicted flow was compared to the exact solution for a Cauchy Riemann flow about a two-dimensional circular cylinder. This is reported in Ref. (43). Then, the computation of convecting viscous flow over an ellipsoid was computed and compared with experiment. This is reported in Refs. (44, 45). After this, computations of flow over a test case helicopter fuselage were done and predicted surface pressures at various streamwise stations compared to wind tunnel data [43]. Then, the computation of flow over a complete Apache helicopter, including rotor, fuselage and wake was computed. Flow around a circular cylinder was also computed using both VC1 and VC2. Finally, the effects of vorticity shed from the pylon on a Comanche helicopter were computed and compared (by Ted Meadowcraft Ref. (46)) to measured values.

7 ROTOR WAKE COMPUTATION

This case was presented at the 2005 ERF [47]. It is being shown here for comparison to the coarser grid computations, discussed below and because it exhibits a recently discovered phenomenon – rotor wake “twinning”.

We fix the circulation and examine the ability of Vorticity Confinement to capture the tip vortices which spiral downward. A lightly loaded 2 bladed rotor was simulated with a uniform Cartesian grid of 128x128x128 cells. The computing time was about 2-3 hours per revolution on a PC (Pentium 4, 2.0 GHz, 1GB RAM). The vortices remained compact, spread over only ~4 grid cells (at the 25% contour vorticity level). The 25% iso-surface is shown in Figure (6 a) from a perspective and in Figure (6 b) from a side view. The contour (down to 25% of maximum magnitude) with the computational grid is shown in Figure (4). The contraction and downward motion of the vorticity are about as expected. A set of contour plots, which shows the development of the rotor wake, are shown in Fig. (5).

A very important feature of this computation is the instability, which results in two consecutive spirals eventually wrapping around each other. This is seen in wind tunnel experiments.

Another point concerns numerical issues: As the number of the blades is increased, or the downwash is decreased, (for example, by decreasing the loading), the distance between the spiral turns decreases. There is, of course, a limit where there will not be a sufficient number of grid cells between the turns, and there will be an interaction between spirals that is a numerical artefact:

For simple single tip vortices, the current theory and method are in close agreement and results are excellent -- no other Eulerian method known can achieve stable convection over arbitrary long times with a vortex core of only ~ 3 cells in diameter. When there are multiple vortices in close proximity, the low level "tails" away from the cores interact over a long time and certain effects emerge. One effect is the transition observed from a spiral to concentric ring geometry for an isolated rotor wake in hover when the inter-vortex distance is only a few grid cells. The rings still form a good approximation. However, the rings can "leap frog", as predicated both theoretically [1] and numerically [2]. Since the angle of the actual spiral is small and it behaves almost like a set of rings, one would expect a similar phenomenon in the spiral. As explained, this has recently been observed experimentally [3], and was given the name "twinning". Thus there seems to be a connection between the well-known leap frogging and twinning.

A rotor wake from a 4 bladed rotor is presented in Fig. (7), where the spiral separations are smaller. The flow condition and computational setup are the same as the above 2 bladed rotor case. The resulting set of disjoint, concentric rings, not quite the spiral of Figure 6, can be seen. However, this should be a weak effect because the actual spiral angle would be small. In fact, sets of concentric rings have been widely used in the past with the Biot-Savart law as a model for the spiral in vortex filament simulations because they can easily be computed.

7.1 Coarse Grid, Rapid Computations

We feel that it is very important to know the resolution limitations of a computational method. To this end we implemented the same VC Lifting Surface code, whose results were described above, on a 64x64x64 (256k nodes) grid. The VC method degrades gracefully with decreasing resolution and reasonable looking (and probably reasonable quantitative) results are still obtained. We are now down to ~12 min./rev., or 2 hrs for 10 revolutions. Results for a 2-bladed rotor (vorticity contours in a plane) are shown in Fig. (8), along with the grid. (As expected, the vortices transitioned into rings). A body was then immersed in the grid and a computation done on the same grid (in the same time). Vorticity isosurfaces are shown for this computation in Fig. (9). The effects of the body on the shed vortices can be seen.

8 CONCLUSION

A new method has been developed to treat rotor blades and wings. The method is based on Vorticity Confinement (VC) and Lifting Surface theory. Unlike earlier panel/vortex lattice methods, the VC based one does not use the Biot-Savart law. Instead, it is based on a primitive variable, Eulerian fixed grid approach. Like recently developed methods, it uses a momentum source, interpolated onto grid nodes near the blade. However, unlike other methods, it uses VC to confine the created vorticity to a small region near the blade surface. It is shown that without VC, the vorticity can create an effective blade cross section that is much larger than the actual blade, because of numerical diffusion.

An important feature of VC is that computations can be done for full rotorcraft in a medium-size grid (~1 M nodes). These can be done on a single PC in about 2-3 hrs/rev., so that a full solution (~10 rev.) can be done in about 20-30 hours. Where data is available, these agree with experimental. The accuracy of the computations is consistent with the state of the art for similar blunt body flows with turbulent wakes.

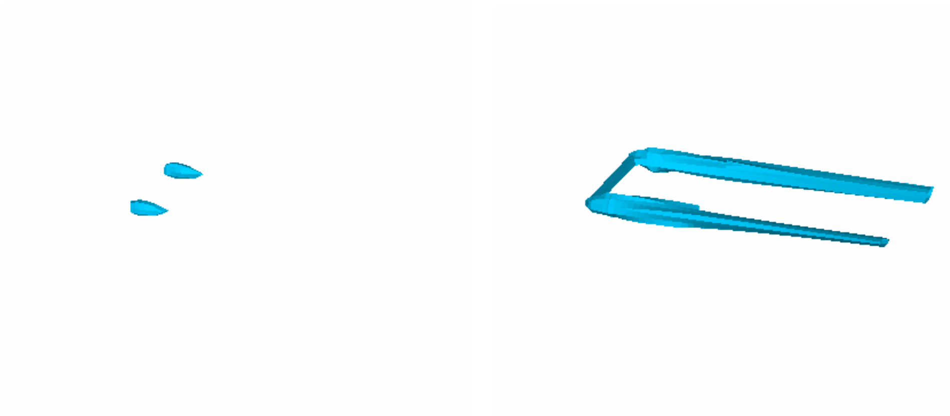
A second result is a possible explanation of the recently observed “twinning” phenomenon in isolated rotor wakes in hover. The fact (which is known) is that a spiral wake is closely approximated by a set of concentric vortex rings lead us to relate the twinning to a well-known behaviour of the rings – “leap frogging”. A spiral wake that exhibits twinning computationally was first demonstrated. This was shown to go to concentric rings when computed on a coarser grid which did not have as much resolution. The coarser grid results were shown to exhibit leap frogging.

Finally, the possibility of using a very coarse grid (64x64x64) for a full rotorcraft including blades and body, for a very fast VC based computation was explored. Although quantitative comparisons have not been made yet, the qualitative results appear to be promising.

ACKNOWLEDGEMENT

Development of the method and the work presented here were partially supported by the Army Research Office, the Army Aeroflightdynamics Directorate, NASA and Army SBIR grants.

9 FIGURES



a) No VC b) with VC

Figure 1. Vorticity Isosurface of Wing Computation

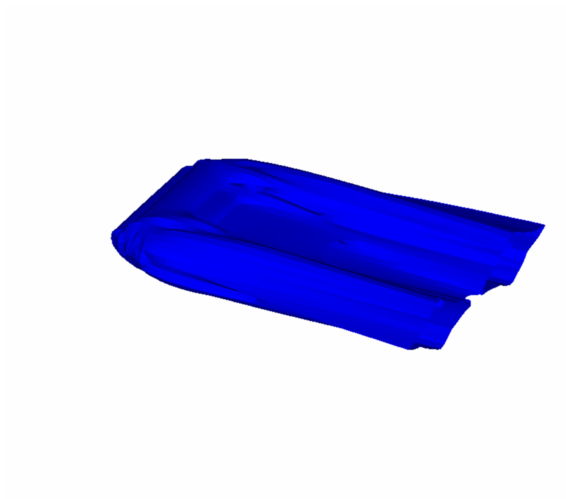
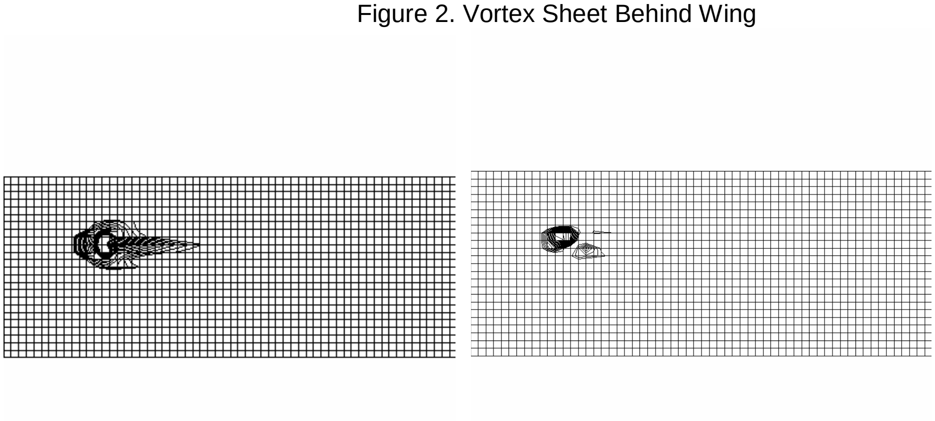


Figure 2. Vortex Sheet Behind Wing



a) No VC b) with VC

Figure 3. Spanwise Vorticity Comparison

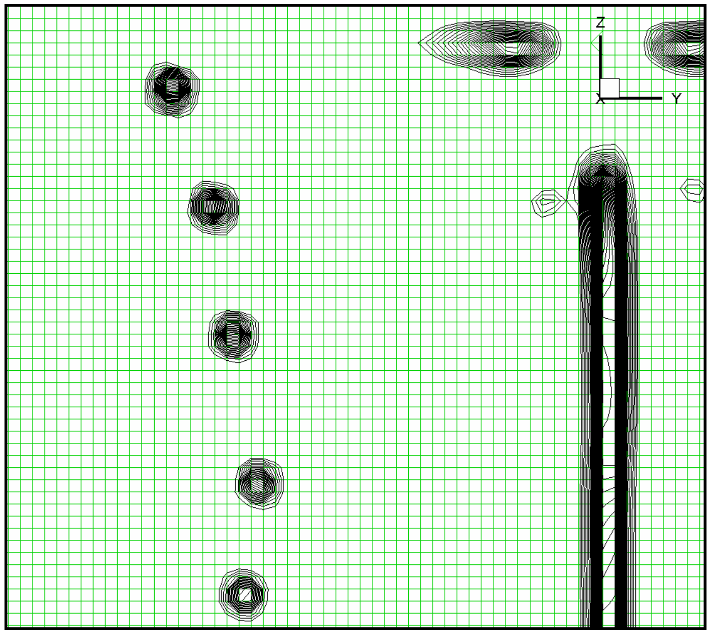


Figure 4. 2-Bladed Rotor Wake Vorticity Contours with Computational Grid

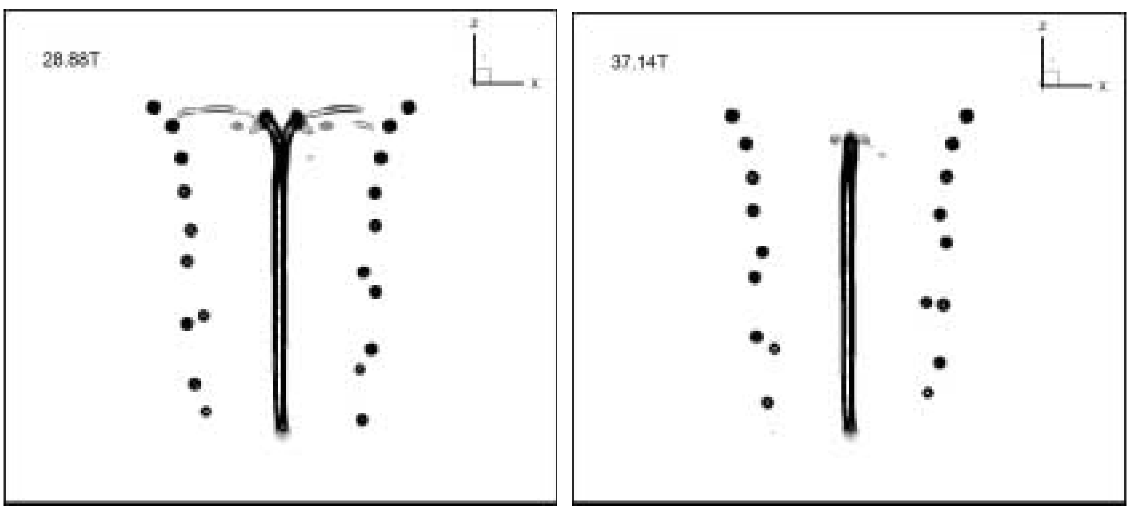
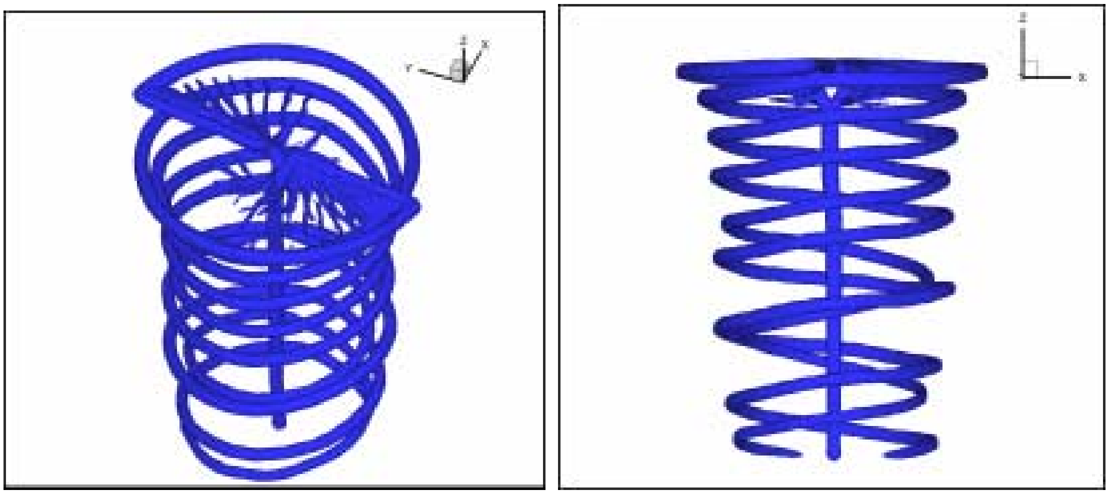
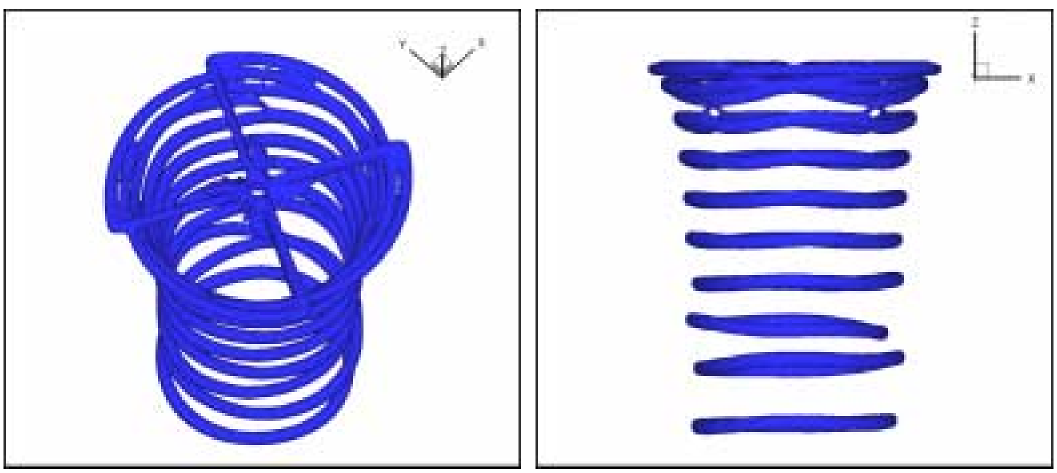


Figure 5. Development of Tip Vortices with Twinnig



a) Perspective View b) Side View

Figure 6. 2-Bladed Rotor Wake Vorticity Isosurface



a) Perspective View b) Side View

Figure 7. 4-Bladed Rotor Wake Vorticity Isosurface (underresolved)

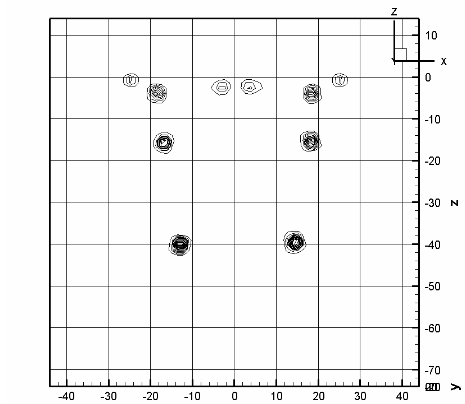
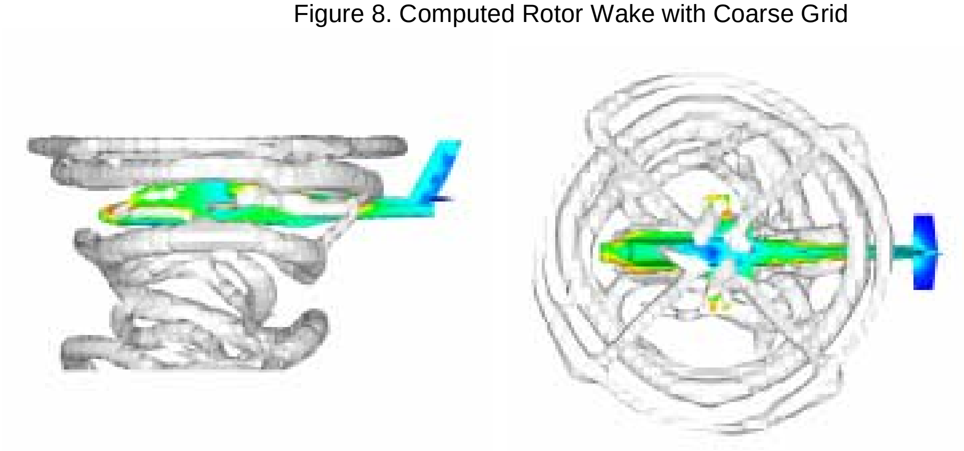


Figure 8. Computed Rotor Wake with Coarse Grid



a) Side View b) TopView

Figure 9. Rotor-Body Computation with Coarse Grid

10 REFERENCES

[1] Sommerfeld, A., “Mechanics of Deformable Bodies”, Lectures on Theoretical Physics, Vol. II, Academic Press, 1950.

[2] Steinhoff, J., and Underhill, D., “Modification of the Euler equations for vorticity confinement application to the computation of interacting vortex rings”, *Physics of Fluids*, 6, 1994. pp.2738-2743.

[3] Caradonna, F., Private communication

[4] Costes, M. and Kowani, G., “An Automatic Anti-diffusion Method for Vortical Flows Based on Vorticity Confinement”, *Aerospace Science and Technology*, Vol. 7, 2003.

[5] Rosenhead, L., “The formation of vortices from a surface of discontinuity”, *Proc. R. Soc. London Ser. A*, 134:170-192, 1931.

[6] Westwater, F. L., “Rolling up of surface of discontinuity behind an aerofoil of finite span”, *Aero. Res. Comm. Rep. Memo.*, 1692, 1935.

[7] Harten, A., “The Artifical Compression Method for Computation of Shocks and Contact Discontinuities III, Self-Adjusting Hybrid Schemes”, *Mathematics of Computation* 32, 142, 1978.

[8] Steinhoff, J., Senge, H. & Wenren, Y., “Computational Vortex Capturing”, UTSI preprint, 1990.

[9] Steinhoff, J., Mersch, T., Underhill, D., Wenren, Y. & Wang, C., “Computational Vor-

ticity Confinement: A Non-Diffusive Eulerian Method for Vortex Dominated Flows", UTSI preprint, 1992.
[10] Wang, C. Steinhoff, J. and Wenren, Y., "Numerical Vorticity Confinement for Vortex-solid Body Interaction Problems", *AIAA Journal*, Vol. 33, August 1995.
[11] Steinhoff, J., Puskas, E., Babu, S., Wenren, Y. & Underhill, D., "Computation of thin features over long distances using solitary waves", AIAA Proceedings, 13th Computational Fluid Dynamics Conference, 1997.
[12] Steinhoff, J., Wenren, Y. and Wang, L., "Efficient Computation of Separating High Reynolds Number Incompressible Flows Using Vorticity Confinement", AIAA-99-3316-CP, 1999.
[13] Wenren, Y., Fan, M., Wang, L. and Steinhoff, J., "Application of Vorticity Confinement to the Prediction of Flow over Complex Bodies", *AIAA Journal*, Vol. 41, May 2003.
[14] Steinhoff, J., Fan, M., Wang, L. and Dietz, W., "Convection of Concentrated Vortices and Passive Scalars as Solitary Waves", *SIAM Journal of Scientific Computing*, Vol. 19, December 2003.
[15] Hu, G., Grossman, B. and Steinhoff, J., "A Numerical Method for Vortex Confinement in Compressible Flow", *AIAA Journal*, Vol. 40, October 2002.
[16] Dadone, A., Hu, G. & Grossman, B., "Towards a Better Understanding of Vorticity Confinement Methods in Compressible Flow", AIAA-2001-2639, 2001.
[17] Dietz, W., "Application of Vorticity Confinement to Compressible Flow", AIAA-2004-0718, 2004.
[18] Robinson, M., "Application of Vorticity Confinement to Inviscid Missile Force and Moment Prediction", AIAA-2004-0717, AIAA Reno meeting, January 2004.
[19] Wenren, Y., Steinhoff, J. & Robins, R., "Computation of Aircraft Trailing Vortices", SBIR Final Report, 1995.
[20] Hass, S., "Computation of Trailing Vortex Over Long Distances Using Vorticity Confinement", Master thesis, Institut fur Luft- und Raumfahrt, RWTH Aachen, Germany, 2003.
[21] Wenren, Y., Fan, M., Dietz, W., Hu, G., Braun, C., Steinhoff, J. & Grossman, B., "Effeicient Eulerian Computation of Realistic Rotorcraft Flows Using Vorticity Confinement: A Survey of Recent Results", AIAA-2001-0996, 2001.
[22] Dietz, W., Fan, M., Steinhoff, J. & Wenren, Y., "Application of Vorticity Confinement to the Prediction of the Flow Over Complex Bodies", AIAA-2001-2642, 2001.
[23] Fan, M., Wenren, Y., Dietz, W., Xiao, M., and Steinhoff, J., "Computing Blunt Body Flows on Coarse Grids Using Vorticity Confinement", *Journal of Fluids Engineering*, 124, No. 4, pp.876-885, December 2002.
[24] Melander, M. V., McWilliams, J. C. & Zabusky, N.J., "Axisymmetrization and Vorticity-gradient Intensification of an Isolated Two-dimensional Vortex through Filamentation", *Journal of Fluid Mechanics* 178, 1987.
[25] Steinhoff, J., Mersch, T., and Wenren, Y., "Computational Vorticity Confinement: Two Dimensional Incompressible Flow", Proceedings of the sixteenth Southeastern Conference on Theoretical and Applied Mechanics, 1992. pp.III.II.73-III.II.82.
[26] Steinhoff, J., Dietz, W., Haas, S., Xiao, M., Lynn, N. & Fan, M., "Simulating Small Scale Features In Fluid Dynamics And Acoustics As Nonlinear Solitary Waves", AIAA-2003-0078, 2003.
[27] Misra, A. & Pullin, D., "A Vortex-based Stress Model for Large-eddy Simulation", *Physics of Fluids* 9, 8, 1997.
[28] Steinhoff, J., Lynn, N., Wenren, Y., Fan, M., Wang, L. and Dietz, W., "Turbulent Flow Simulations Using Vorticity Confinement", Chapter 12 in *Implicit Large Eddy Simulation:Computing Turbulent Flow Dynamics*, Edited by F. F. Grinstein, L. G. Margolin,

and W. J. Rider, Cambridge University Press, New York, NY, 2006
[29] Fan, M. & Steinhoff, J., "Computation of Blunt Body Wake Flow by Vorticity Confinement", AIAA-2004-0592, 2004.
[30] Kim, J. & Moin, P., "Application of a Fractional-Step Method to Incompressible Navier-Stokes Equations", *Journal of Computational Physics* 59, 2, 1985.
[31] Tannehill, J., Anderson, D. & Pletcher, R., "*Computational Fluid Mechanics and Heat Transfer*", Taylor & Francis, 1997.
[32] Steinhoff, J. & Raviprakash, G., "Navier-Stokes Computation of Blade-Vortex Interaction Using Vorticity Confinement", AIAA-95-0161, 1995.
[33] Morvant, R., Badcock, K., Barakos, G., and Richards, B., "Aerofoil-Vortex Interaction Simulation Using the Compressible Vorticity Confinement Method," 29th European Rotorcraft Forum, Friedrichshafen, Germany, September 2003.
[34] Morvant, R., "The Investigation of Blade-Vortex Interaction Noise Using Computational Fluid Dynamics," PhD Dissertation, University of Glasgow, United Kingdom, February 2004.
[35] Steinhoff, J., Lynn, N. and Wang, L., "Large Eddy Simulation Using Vorticity Confinement", Chapter 4 in *Implicit Large Eddy Simulation:Computing Turbulent Flow Dynamics*, Edited by F. F. Grinstein, L. G. Margolin, and W. J. Rider, Cambridge University Press, New York, NY, 2006
[36] Oshima Y. & Izutsu N., "Cross-linking of two vortex rings", Physics of Fluids 31, 1988.
[37] Ferguson, S., Rotorwash Analysis Handbook, Volume I, Development and Analysis, DOT/FAA/RD-93/21, June 1994.
[38] Kufeld, R., Balough, D., Cross, J., Studebaker, K., Jennison, C., and Bousman, W., "Flight Testing of the UH60A Airloads Aircraft," American Helicopter Society 50th Annual Forum, Washington, D.C., May 1994.
[39] Ramachandran, K., Owen, S., Caradonna, F., and Moffit, R., "Hover Performance Prediction Using CFD," AHS 50th Annual Forum, Washington, D.C., May 1994.
[40] Strawn, R., and Djomehri, M., "Computational Modeling of Hovering Rotor and Wake Aerodynamics," AHS 57th Annual Forum, Washington, D.C., May 2001.
[41] Caradonna, F.X., and Tung, C. "Experimental and analytical studies of a model helicopter in hover." Technical report, NASA TM-81232, 1981.
[42] Srinivasan, G.R., Baeder, J.D., Obayashi, S. and McCroskey, W.J. "Flowfield of Lifting Rotor in Hover: A Navier-Stock Simulation." *AIAA Journal*, 30(10): 2371-2378,Oct,1992.
[43] Wenren, Y. and Steinhoff, J., "Application of Vorticity Confinement to the Prediction of the Wake of Helicopter Rotors and Complex Bodies", AIAA-99-3200, 17th Applied Aerodynamics Conference, July 1999.
[44] Steinhoff, J., Wenren, Y., Wang, L., Fan, M., Xiao, M., and Braun, C., "Application of Vorticity Confinement to the Prediction of the Wake of Helicopter Rotors and Complex Bodies", *Computational Fluid Dynamics JOURNAL*, Vol. 9, April 2001. Japan.
[45] Braun, C., "Application of Vorticity Confinement to the Flow over a 6:1 Ellipsoid at High Angles of Attack", Master thesis, Institut fur Luft- und Raumfahrt, RWTH Aachen, Germany, 2000.
[46] Duque, E. and Meadowcraft, E., "Numerical Evaluation of Main Rotor Pylon Flowfields on the RAH-66 Comanche Helicopter," in Proceedings, the AHS 55 th Annual Forum and Technology Display, May 1999.
[47] Wenren, Y., Steinhoff, J. and Caradonna, F., "Application of Vorticity Confinement to Rotorcraft Flows", 31st European Rotorcraft Forum, 12-16 September, 2005, Florence, Italy.