CHAPTER I

INTRODUCTION

1.1 Aerodynamics of Rotors in Forward Flight

The prediction of rotor blade aerodynamic loads, especially in forward flight, requires accurate and efficient modeling of several distinct phenomena. This Problem is highly complicated because the flow field is inherently unsteady and three dimensional. On the advancing side, the combination of rotational speed and forward flight velocity may cause the flow in the tip region to be transonic with associated formation of supersonic pockets and shock waves. Shock induced separation and shock motion along the chord of the blade may occur. Because the inboard section of the retreating blade operates in a reversed flow condition that grows in size with increasing forward flight velocity, the outboard stations are required to operate at increasingly high angles of attack to generate sufficient lift, which may cause dynamic stall.

At low advance ratios, strong tip vortices in the rotor wake dominate the flow field and produce an unsteady and non-uniform induced velocity field at the rotor disk. The loads on a given blade are often impulsively modified by the passage of vortices shed by the other blades, a phenomenon known as blade-vortex interaction (BVI). The accuracy with which the rotor wake structure may be modeled greatly affects the performance, blade loads, vibration and acoustic characteristics of the rotor. Furthermore, rapid one-per-rev and two-per-rev variations due to the flow asymmetry between the advancing and the retreating sides may cause structural bending and torsional deformations, and must be included. Finally, the influence of the fuselage, which is often neglected, can produce significant modifications in the flow field around the blades. Thus, the problem of modeling rotors in forward flight is multi-disciplinary. The rotor loads are due to the nonlinear interaction between the rotor aerodynamics, trim, aeroelasticity and blade dynamics. As stated by Landgrebe [1], the accurate prediction of airloads and the many aerodynamic interaction effects that may influence the performance and dynamic loading of the vehicle has been the challenge in rotor aerodynamics. The variations in the aerodynamic interactions with forward speed add more complexity to advanced rotorcraft design.

In order to capture the above physical phenomena, the influence of the inflow, blade dynamics, elastic response, and the trim of rotor must all be properly included in any Computational Fluid Dynamics (CFD) analysis. The existing CFD tools developed to simulate this wide range of physical phenomena are not yet able to solve all the problems mentioned above. Typically, the rotor CFD methods either ignore some of the influences, or compute these influences with an analysis external to the CFD calculation, such as a comprehensive analysis based on lifting-line theory. The external computation is coupled to the CFD analysis in an open loop iteration process. In the case of hingeless and bearingless rotors, aeroelastic effects may significantly modify the blade loads and consequently the strength of the tip vortices. In such cases, open loop coupling may converge very slowly. Methods that tightly couple the aeroelastic analysis to the aerodynamic analysis are needed. At present, such a computation is still based on the combination of the CFD analysis with an appropriate comprehensive analysis, but entire calculation using a tightly coupled methodology should be available in the near future.

1.2 Comprehensive Analysis Methods

The simplest method of analyzing rotor wings uses simple momentum theory and blade element theory. This method can offer a good understanding of the rotor aerodynamic performance. In forward flight analyses, Theodorsen's unsteady two dimensional airfoil theory is often applied in rotor analysis. Loewy [2] extended the classical Theodorsen technique to rotors via a lift deficiency function that accounts for the influence of the vortex sheets underneath the rotor disk. The disadvantage of simple momentum theory based methods is that they can not give details of the aerodynamic field. The complicated three dimensional, skewed wake geometry and periodically varying free stream velocity in forward flight make the analysis of rotors quite different from fixed wings. Much effort has been put into developing three dimensional vortex theories that numerically compute the induced velocity field associated with lifting rotors [3, 4].

Helicopter industries use comprehensive helicopter analyses that contain phenomenological models for the blade aerodynamics and the wake, coupled to structural dynamics models. The basis for the aerodynamic modeling in these analyses is blade element theory. A semi-empirical unsteady aerodynamic model developed by Leishman and Beddoes [5] is usually included for modeling unsteady aerodynamics, separated flow and dynamic stall. Classical inflow models such as Glauert theory, and various wake models that model the tip vortex as a rigid, prescribed or free wake are also included. In the 1970s, dissatisfaction with the first generation computer programs for predicting helicopter performance and dynamic behavior motivated the development of the CAMRAD (Comprehensive Analytical Model of Rotorcraft Aerodynamics and Dynamics) [6, 7], and later the development of 2GCHAS (Second Generation Comprehensive Helicopter Analysis System) [8]. The rotorcraft industries, in some instances, have also developed their own comprehensive analyses codes (e.g. Sikorsky's Rotorcraft System Dynamic Analysis (RDYNE) [9]). The aerodynamic models in the comprehensive analyses are not general enough to model the nonlinear and unsteady effects. The integration and expansion of these codes to incorporate the wake and airflow predictions from CFD codes is still an active area [10, 11].

1.3 Computational Fluid Dynamics (CFD) Models

With the advent of high speed computers, researchers and industry analysts began to use CFD techniques. A variety of techniques, ranging from potential flow to Navier-Stokes methods, have been developed and applied to rotors in forward flight.

1.3.1 Potential Methods

In order to compute the impulsive loads on the blades, it is important to accurately predict the position and strength of the shed vortices and feed this wake information back into an accurate unsteady flow solver. In potential flow methods, the effect of the wake is typically introduced as an inflow correction from an external wake model. Trim is also achieved externally using a comprehensive code and fed back into the aerodynamic analysis. Caradonna and Isom [12] first developed the transonic small disturbance solver for rotor flows. The potential flow code called FDR by Chattot [13] was coupled to a comprehensive code. The effect of the wake and trim were input to the solver as a table of angle of attack corrections. Sankar and Prichard [14] and Strawn [15-16] have developed unsteady full potential flow based rotor solvers, named RFS2 and FPR, respectively. In these analyses, the flow results were coupled to a comprehensive code called CAMRAD in an open-loop fashion. A review of the potential methods and comparison of some existed potential flow codes were given by Caradonna and Tung [17]. In contrast to the Navier-Stokes based methods, the potential flow analyses are limited to irrotational flows. This limits their use to problems where the shock waves and/or separation flow effects are known to be weak or nonexistent. The low computer CPU and memory requirements of the potential flow solver still make them popular in the helicopter industry.

The full potential flow methods are being extended to compute rotor flows and wake effects in a unified manner. Steinhoff and Ramachandran [18-21] applied a vorticity embedding technique to model and convect the wake as a series of markers in a

Lagrangian fashion. A hover version of their method called HELIX-I, and a forward flight version of their work called HELIX-II, have been developed.

The classical methods for capturing the wake influence use either a prescribed wake model or a free wake model. The prescribed wake models assume a wake shape which remains fixed throughout the calculations. Although rotor CFD simulations with a prescribed wake model do include the tip vortex effects, they neglect wake-to-wake interactions. Free wake models allow for wake distribution due to self induced velocities. Scully [22] developed a free wake model which formed the basis for Johnson's CAMRAD analyses. Saddler's model [23] considered the distortion of both the tip vortex and the inboard vortex sheet. The free wake analyses developed by Johnson [24] use one or two concentrated trailing vortices from each blade. A similar wake model [25] was also implemented in a comprehensive analysis UMARC (University of Maryland Advanced Rotorcraft Code). The tip vortices are modeled as straight line segments, and the undistorted inboard wake may be modeled by either vortex filaments or vortex sheets. Bagai and Leishman [26] developed a free wake model using a pseudo-implicit predictor-corrector relaxation method for the solution of the governing differential equations of the rotor free-wake problem. A general full span wake model approach called the Constant Vorticity Contour (CVC) approach for computing rotor airloads in forward flight was developed by Bliss and Quackenbush[27-28]. In their work, the wake structure is modeled as curved vortex elements. This approach has been included in an analysis code called rotorCRAFT [29]. Egolf [30] and Berry[31] modeled the entire wake using a vortex lattice method. Torok and Berezin [10] evaluated three free wake models (Egolf's model, Johnson dual-peak model, CVC model) for a UH-60A model rotor in low speed forward flight. Typical wake geometries are shown in Figure 1.1.

In a comprehensive code, the free-wake is usually implemented as a module with prescribed blade motion, and a prescribed bound circulation obtained from blade element theory. When the wake model is coupled with the CFD code, the tip vortex strength of the wake elements can be calculated directly from the flow solver at each time step. Once the inflow from the wake is computed, it is used within the CFD codes in a number of ways: the angle-of-attack approach; the lifting surface or transpiration velocity approach; the branch cut approach; the split potential or perturbation approach. The full potential rotor flow solver, RFS2, with the velocity transpiration approach of Sankar and Malone [32] was used by Hassan et al [33] and Tadghighi et al [34] to study self-generated blade-vortex interactions. The potential flow code, FPR, also was used in various BVI studies by Torok et al [10], and Hassan et al [35]. Comparisons of results from different potential flow codes for the Operational Loads Survey (OLS) model rotor were reported by Yu et al [11]. The angle-of-attack approach and velocity transpiration approach are usually used because of their computational efficiency. They only require the vortex induced velocity at the blade surface grid points. In contrast, the split potential approach requires a large computational effort, because of need to calculate induced velocity due to wake elements at every grid node inside the computational domain.

1.3.2 Euler/Navier-Stokes Methods

Most, if not all, of the current methods for solving Euler or NS equations are capable of capturing the rotor wake as part of the solution. They do not require empirical parameters such as the tip vortex core size required in the wake model. Sankar et al [36, 37] developed the first Euler analysis for rotors in hover and forward flight. Other Euler analyses for rotors have been reported by Murman et al [38], Agarwal et al [39], and Chang [40]. The first Navier-Stokes solution of isolated rotor blades was reported by Wake and Sankar [41], and was followed by Srinivasan and McCroskey [42], and Agarwal and Desse [43]. Srinivasan et al [44, 45] have developed a widely used NS solver for isolated rotors in hover called TURNS (Transonic Unsteady Rotor Navier-Stokes). Work is in progress for improved modeling of the rotor wake in hover from first principles. Wake and Baeder [46] used a 3rd order Navier-Stokes approach to study the UH-60 rotor with about 10^6 grid points. Ahmad and Strawn [47] studied the same cases using an overset grid system with about 8×10^6 grid size and a 4th order method. Hariharan and Sankar [48] have developed a 7th order spatially accurate ENO (Essentially Nonoscillatory) methodology and assessed its advantages over a previous 5th order spatial ENO scheme [49]. In recent rotor forward flight analyses, a Navier-Stokes solver was used by Bangalore and Sankar [50] to study a UH-60A configuration and a leading edge slatted rotor in forward flight. Ahmad and Duque [51] have also studied forward flight flow fields using a version of the NASA Ames code OVERFLOW with an embedded overset grid technique. All these calculations use information regarding the blade dynamics from flight test or wind tunnel data, and no attempt has been made to trim the rotor.

In addition to the application to isolated rotor blades, Navier-Stokes methods have also been used to model the interaction of the rotor flowfield with the fuselage. Hariharan and Sankar [52] used a Chimera grid system for predicting the aerodynamic interaction between a cylindrical body and a two-bladed rotor. Duque and Dimanlig [53] use OVERFLOW to compute the airloads on the Comanche fuselage with an actuator disk model for the time-averaged rotor downwash. The Chimera grid and actuator disk model was also used by Chaffin et al [54] to compute a simple configuration for various values of advance ratio. Recent developments at ONERA for the prediction of helicopter rotor-fuselage aerodynamic interactions are documented in Ref. [55].

The prediction capability of the current generation of rotor Euler/NS analyses is not at a level acceptable to the helicopter industry because the rotor is not trimmed, and the blade dynamics and aeroelasticity are not adequately modeled. Furthermore, due to the presence of numerical diffusion which occurs as a result of spatial discretization of the governing equations, these wake-capturing approaches suffer from excessive numerical viscosity. The tip vortex can not be carried for long distances after it is shed. Finally, although Euler/Navier-Stokes simulations make fewer assumptions about the flowfield than potential flow method, they require significant computational memory and time, especially in forward flight.

Although CFD simulations based on the Navier-Stokes equations have been available for some time, it has not been possible to accurately capture the tip vortices over the convection distance required for BVI modeling. Capturing the tip vortex without numerical diffusion is an active area of research where researchers are trying a number of approaches. The idea behind these methods is to improve the spatial accuracy by increasing either the order of the spatial operator, or improve the grid resolution. Encouraging methods available to date include high order spatial accuracy schemes [48, 49, 56], grid refinement, overlapping structured grids with additional grids at the vortices [57-58], unstructured grids which are vortex-adapted [59], and hybrid structured and unstructured grid methods [60]. Unstructured grids are particularly suitable for dynamic unsteady remeshing and grid adaptation, because they permit easy addition and removal of grid points within the tip vortex region. An excellent survey article by McCrosky [61] summarizes the state of the art in wake modeling.

More recently, Caradonna [62] gave a rough estimate of the exceedingly large grids required for the global wake prediction problem and pointed out that the problem is far bigger than can be solved with existing computational models. New approaches which consider the method of preserving, instead of resolving, the vortex cores and strengths are needed. Two vortex core models have been proposed. The first approach, called the vortex embedding technique, replaces the wake with embedded vortex sheets [63-65] and requires a fairly coarse grid. Bridgeman and Caradonna [66] demonstrated the application of this approach to a BVI problem. The difficulty in this method is the explicit construction of the vortex structure through a Clebsch function. This method requires cumbersome and costly searching of the wake markers in computational cells. The second approach is the vorticity confinement model [67] which modifies the flow equations in the vortical regions by adding a body force term which pushes the vorticity towards its core. This method was applied to a two-dimensional BVI problem by Steinhoff et al [68].

Another weak point in the Navier-Stokes based methods is the prediction of dynamic stall. Although the comparison of CFD calculations with experimental data is acceptable for the light stall, the details of the dynamic stall phenomenon are still not fully understand, especially for moderate and deep dynamic stall. Development of appropriate turbulence and transition models is important for the prediction of stall on rotor blades. Ekaterinaris et al [69] used a Navier-Stokes flow solver with a variety of turbulence models for a 2D dynamic stall problem, and emphasized the importance of turbulence models. Narramore and Sankar [70] have applied a three dimensional Navier-Stokes solver to study the retreating blade stall for a rotor configuration.

1.4 Methods that Account for Aeroelastic Effects

Most of the rotorcraft codes developed to predict aeroelastic response use phenomenological models for the wake structure and geometry, the induced velocity calculation and the blade aerodynamics. A quasi-steady strip theory approximation is commonly used. The dynamic inflow wake modeling [71] is usually employed for capturing low frequency, unsteady rotor wake effects. These approaches are inadequate for calculating higher harmonic airloads and aeroelastic stability near stall. More detailed computational methods for the rotor wing aerodynamics are needed to calculate the airloads in regions involving complex flow phenomena such as flow separation, reverse flow, transonic flow and dynamic stall. It is important that the models used for the various parts of aeroelastic analysis have a consistent level of sophistication. Unfortunately, in rotor forward flight, many rotor aeroelastic analyses use a complex structural model but a simple quasi-steady, two dimensional aerodynamic theory. Unlike the fixed wing, the rotor blades encounter the oscillatory aerodynamic loads even in steady forward flight. An appropriate unsteady aerodynamic model capturing detailed flow physics is needed. This need has lead to the coupling of rotor dynamics analyses with highly detailed CFD methods.

Some efforts have already been made to tightly couple dynamics and elastic calculations with CFD codes in fixed wing application. One of the notable codes called ENSAERO was developed by Guruswamy [72]. A deforming mesh algorithm [73] that uses a spring analogy was developed to move the mesh so that it continuously conforms to the instantaneous shape of the aeroelastically deforming wing.

In the rotary wing arena, Kwon [74] coupled a panel method to the aeroelastic model of Hodges, with a prescribed tip-vortex wake model. The same test case was studied by Smith [75] with a fourth-order accurate Euler/Navier-Stokes solver, coupled to an elastic rotor blade beam structural model. More recently the study by Murty et al [76] used the airloads obtained from a parallel adaptive finite element Euler solver, coupled with a multi-body structural dynamics code called DYMORE. The computational structural dynamics (CSD) code determined the displacements along spanwise locations and provided a new rotor blade geometry at each time step. A deforming mesh algorithm can be used to account for the geometry changes and the resulting mesh was supplied to the CFD solver. Bauchau and Ahmad [77] coupled the

CFD code OVERFLOW with an overset grid to DYMORE for studying the aeroelastic effects on rotors in forward flight. Other studies include research by Chopra [78-79]. In their work, the aerodynamic analysis was done by a potential flow code, and coupled to a structural dynamic analysis of the rotor blade. The attempts to use a coupled analysis with comparable sophistication in both structural and aerodynamic models are hampered by enormous computational requirements. Strategies for efficient, first principles based aerodynamic modeling are still needed to obtain periodic and trimmed solutions using reasonable amounts of CPU time and memory.

1.5 Hybrid Methods

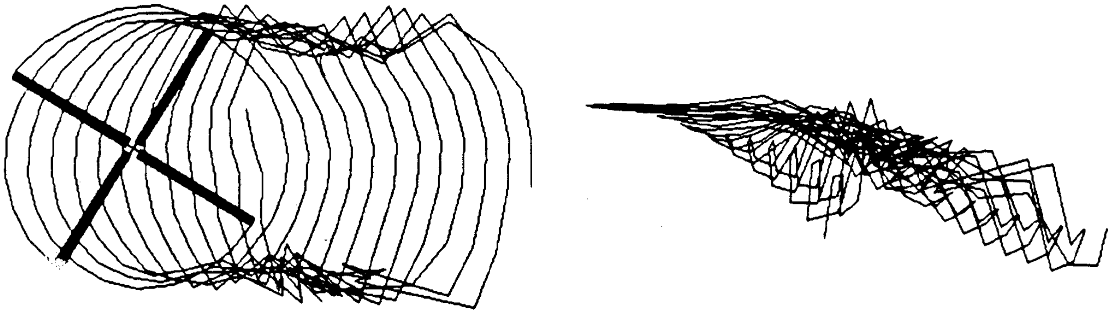
Although the Navier-Stokes based methods inherently contain all the necessary physics and details of the flowfield, large computational requirements still prevent the full Navier-Stokes codes from industry use. Considerable expertise and time are needed for Chimera grid and embedded grid system manipulation to control the numerical diffusion of the vortical wake. Hybrid methodologies employ the most appropriate numerical models in different flow regions to retain solution quality, with a large reduction in computer time. Berezin and Sankar [80] developed a hybrid rotor solver using Navier-Stokes equations near the blades and a potential flow model elsewhere. Only the near wake was captured. The viscous region near the blade is needed to capture flow separation, shock waves, and the tip vortex formation. In the full potential region, there is very little entropy change. However, potential flow models can not admit distributed vorticity fields, thus a wake model is necessary. This hybrid solver was coupled to a comprehensive code called RDYNE [9] to properly account for the far wake and trim effects through angle of attack corrections. Berkman and Sankar [81] improved the hybrid technique by modeling the entire wake from first principles, and obtained good results for rotors in hover. Moulton and Caradonna [82-84] coupled HELIX-I to the Navier-Stokes solver TURNS to form a hybrid solver for modeling rotors in hover. The vorticity embedding approach was used to preserve the vorticity in the outer potential zone. Bangalore et al [85] extended this methodology through the use of an overset grid method for the prediction of advancing rotor flows and avoided the grid skewness problems near the far-field boundaries. In most of these calculations (except in ref. 80) the rotor was not trimmed, and the elastic effects were not included.

1.6 Scope of the Present Work

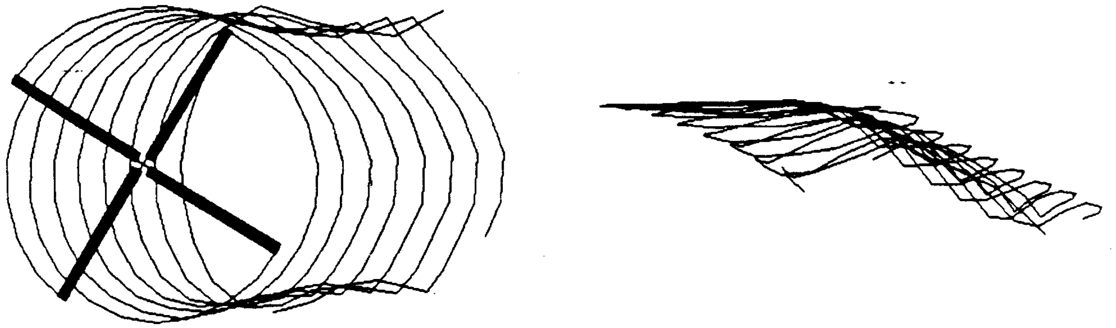
The goal of this research is to develop an efficient solution methodology that will accurately predict the aerodynamic loads on a rotor in forward flight. For reasons discussed earlier, the model should simulate the complex blade dynamic motion in forward flight, include a trim process, have a self-consistent wake model, and account for the aeroelastic deformations of the blade. This method is an extension of the hybrid methods discussed in Ref. [80] and Ref. [81]. This hybrid Navier-Stokes/full potential solver has been modified to model rotors in forward flight.

The hybrid method is an appropriate phenomenological methodology for capturing the appropriate flow phenomena in different flow regions, as illustrated in Figure 1.2. A Navier-Stokes analysis is used for modeling the viscous flow and near wake in a small region surrounding the single rotor blade. A finite volume form of Navier-Stokes equation is solved using Roe's approximate Riemann solver with third order or fifth order spatial accuracy. In the outer zone of the computational domain, a potential flow analysis is used to model the inviscid isentropic flow and carry the acoustic and pressure waves generated by the blade to the far field. A velocity decomposition approach is implemented that consists of the freestream velocity, disturbance potential velocity, and induced velocity caused by the far wake. The grid velocity terms in the viscous and inviscid zones are implemented in a general form to take into account the fact that the cell faces are moving with respect to an inertial observer, and are deforming as a result of aeroelastic effects in forward flight. Care must be taken to satisfy the Geometric Conservation Law (GCL) in order to avoid spurious production of mass, momentum and energy within the computational cells. A Langragean wake approach is implemented as part of the hybrid methodology to capture the effects of the tip vortex once it leaves the viscous zone, and convect it without diffusion in the inviscid zone and the far field. There is no need to link the analysis to an external rotor wake simulation code in order to include the far wake effects. This transonic, potential-based method with a Lagrangean wake model can convect the wake vorticity in the far field without numerical dissipation. This strategy also permits the use of a fairly sparse grid in the far wake.

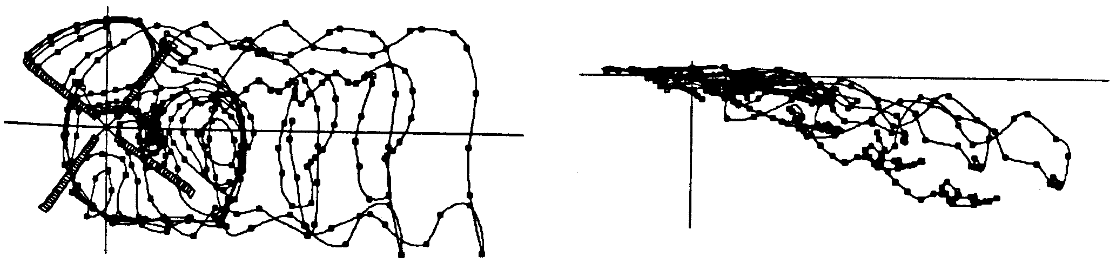
This thesis is organized as follows. In chapter II, the mathematical and numerical formulation behind the Navier-Stokes solver and the potential flow solver will be given. In chapter III, extensions to and implementation of the hybrid method to forward flight will be addressed. A dynamic motion module has been developed to account for the complex blade motion. Various wake models for calculating the induced velocity have also been developed. The strategy of trimming the rotor has been introduced into a potential flow code and tested. Chapter IV and V discuss code validation studies. This work finishes with a list of conclusions and recommendations for future study.



(a) Wake geometry using FREEWAKE model



(b) Wake geometry using Johnson Model



(c) Wake geometry using rotorCRAFT model

Figure 1.1: Wake Geometry for the UH-60A Rotor in Forward Flight at μ=0.1
(reproduced from reference [10])

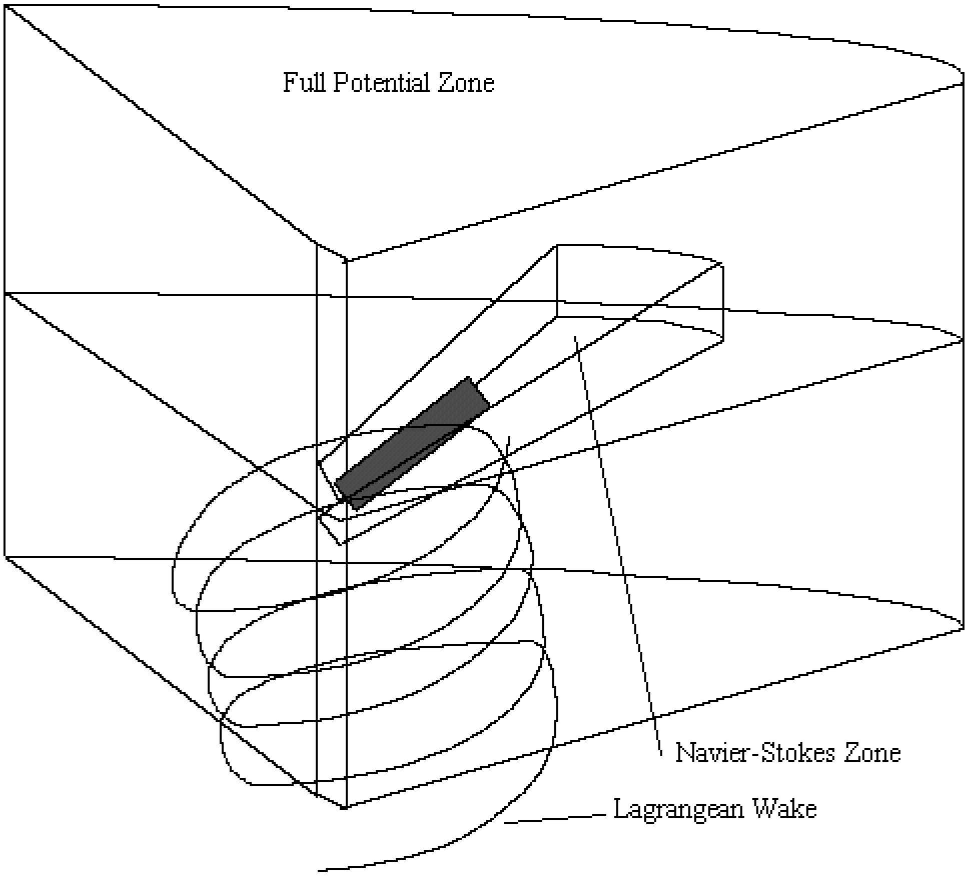


Figure 1.2: Flow Field Partitioning for the Present Hybrid Method