# Computational Studies of Horizontal Axis Wind Turbines

Quarterly Progress Report

Covering the period

December 1, 2001 – February 28, 2002

Contract No.  XCX-7-16466-02

Submitted to the

National Renewable Energy Laboratory
Attn: Dr. Scott Schreck
1617 Cole Boulevard
Golden, CO 80401-3393

Prepared by

Lakshmi N. Sankar and Sarun Benjanirat
School of Aerospace Engineering
Georgia Institute of Technology
Atlanta, GA 30332-0150

March 6, 2002

# I. INTRODUCTION

A computational research program is underway at Georgia Tech in the area of horizontal-axis wind turbine aerodynamics. The research focuses on understanding the flow mechanisms that affect the performance of wind turbines in non-axial and non-uniform inflow, and the development of modern, efficient computational techniques that complement existing combined blade element-momentum theory.

The computational effort is based on the extension of a 3-D hybrid Navier-Stokes/potential flow solver that has been developed at Georgia Tech for helicopter rotor and propeller applications to horizontal axis wind turbines. In this approach three-dimensional unsteady compressible Navier-Stokes equations are solved in a small region, on a body-fitted grid surrounding the rotor blade. Away from the blades, the potential flow equation is solved. The vorticity shed by the blades as a result of dynamic stall, and the spanwise and azimuthal variations of circulation are captured using vortex filaments. These filaments are freely convected by the local flow. Since the costly Navier-Stokes calculations are done only in regions close to the wind turbine blades, and because much of the vorticity is tracked using Lagrangean techniques, this method is an order of magnitude more efficient than full blown Navier-Stokes methods.

The specific objectives of this research project during the period May 1, 2001 – April 30, 2002 are:

1) Improve the viscous separated flow modeling accuracy by replacing or augmenting the zero-equation and one-equation turbulence models in the analysis with higher order two-equation models,

2) Improve the load predictions in the blade root region,

3) Validate the analysis through a simulation of the NREL Phase VI rotor under yaw conditions.

## SUMMARY OF ACTIVITIES DURING THE RESEARCH PERIOD

During this reporting period, the following work was completed:

1. Work on the development of advanced turbulence models (e.g. k-ε) continued. No numerical results are available for this model at this time.
2. Work began on the implementation of a root vortex in the hybrid analysis, to assess its effects on the blade loads, blade performance, and the flow details in the root region. Calculations have been done for the NREL Phase VI rotor for a two wind speeds of 10 m/s, and 7 m/sec at zero yaw. The inboard vortex, and the tip vortex geometries were assumed to be non-contracting helical structures. The axial placement between successive helical rings was determined from an estimate of the inflow velocity field from combined blade-element-momentum theory. The hybrid methodology does have the option for using a free wake model that allows the tip vortex and the inboard vortex to move and deform according to local flow conditions and velocities. However, at high wind speeds, the rigid wake model has been found to be more than adequate.

   Table 1 shows the computed and measured rotor torque at these two wind conditions. In general, the calculations under-predict the torque by 10% to 15%. Other simulations, including our own predictions at the NREL Blind Run Workshop show considerable variations in the computed torque. As seen in figure 2, the present simulations are an improvement over our earlier predictions at the workshop (shown in pink), largely due to a much finer grid that is being used in the most recent set of results.

From Table I, it is seen that the inclusion of the root vortex has very little influence on the total torque. The radial distribution of lift and drag forces, and the root bending moment, were also affected negligibly if the root vortex effects are included.

The root vortex does have a substantial effect on the angle of attack that the inboard sections see. The angles of attack tend to decrease at the root when the root vortex is included, leading to a somewhat smaller separation zones, as seen in figures 3a-3b. An attempt was made to determine if the addition of root vortex eliminates or modifies the centrifugal pumping of the low velocity separated flow in the root region. It appears that a significant amount of centrifugal pumping is still present (as evidenced by the outward deflection of fluid particle streaklines (shown in red in figure 3-b).

In summary, based on the limited number of studies we have done with and without root vortices, inclusion of the root vortex has only a negligible effect on the rotor performance (Power vs. Speed, torque vs. speed), blade loads, and root moments. Root vortex modeling does not significantly increase the flow simulation time, and should be included if the flow field in the root region is of interest.

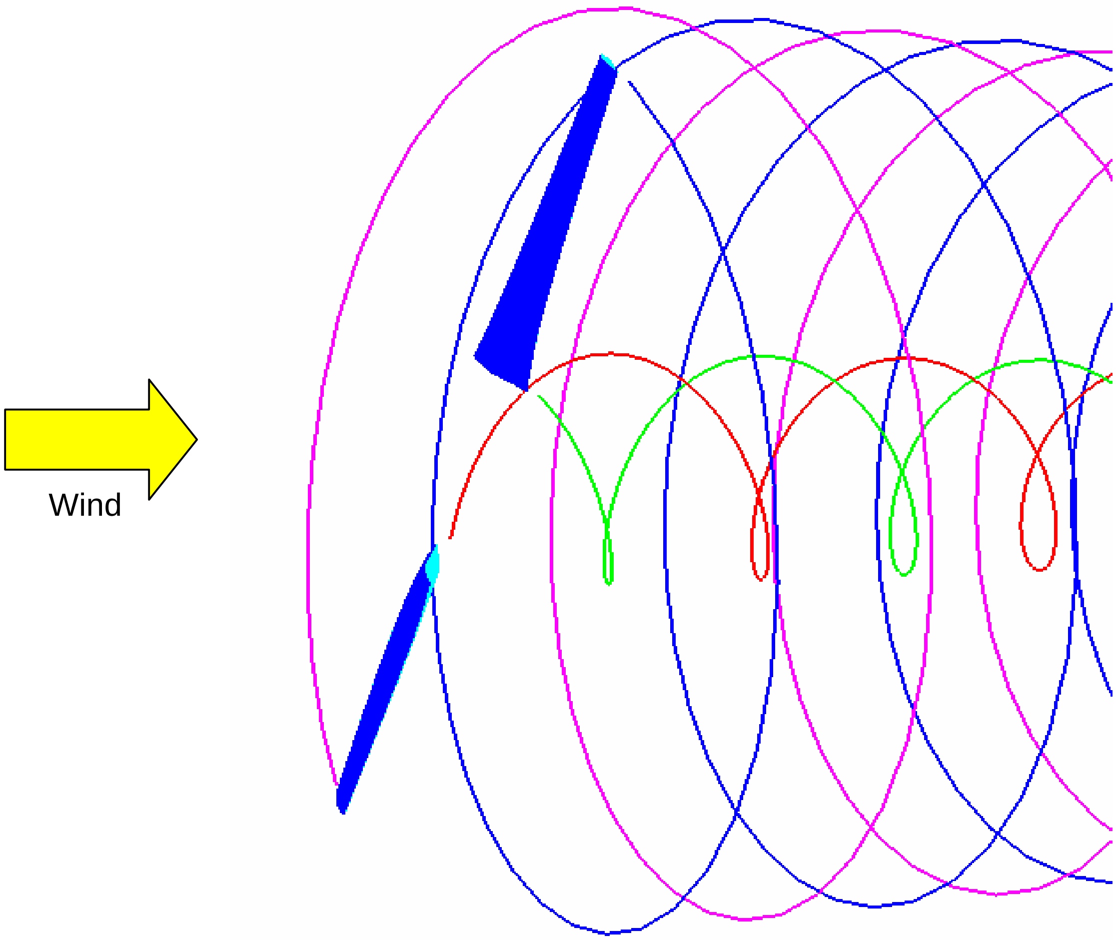


Figure 1. Geometry of Tip and Root Vortexes for the Phase VI Rotor at 7m/s Wind

| Wind Speed (m/s) | NREL Measurements (Newton-Meters) | Without root vortex (Newton-meters) | With Root vortex Newton-Meters |
|---|---|---|---|
| 7 | 801.2651 | 705.2228 | 706.0739 |
| 10 | 1341.1022 | 1239.120 | 1246.1038 |

Table 1. Low Speed Shaft Torque for the Phase VI Rotor Calculated with and without Root Vortex

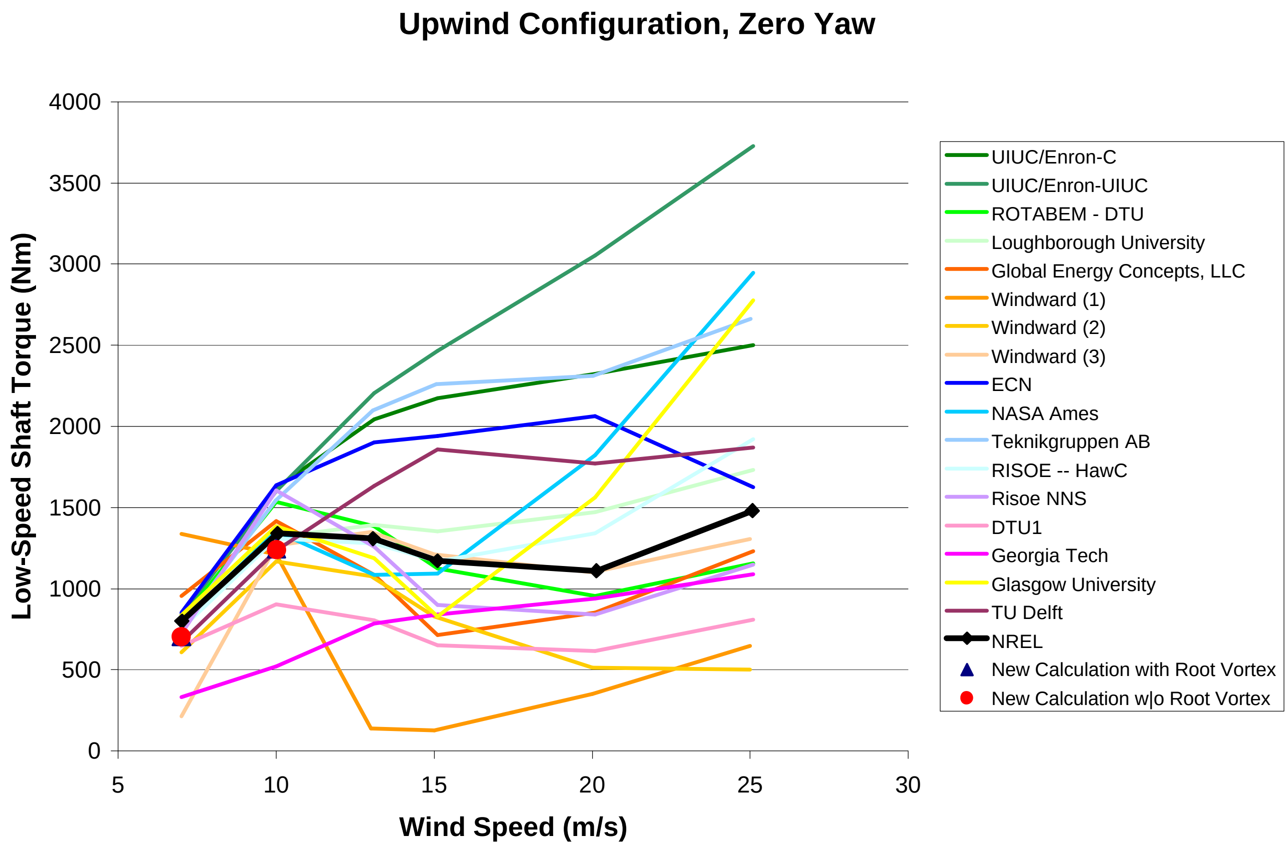


Figure 2. Predictions of Low Speed Shaft Torque for the NREL Phase VI Rotor

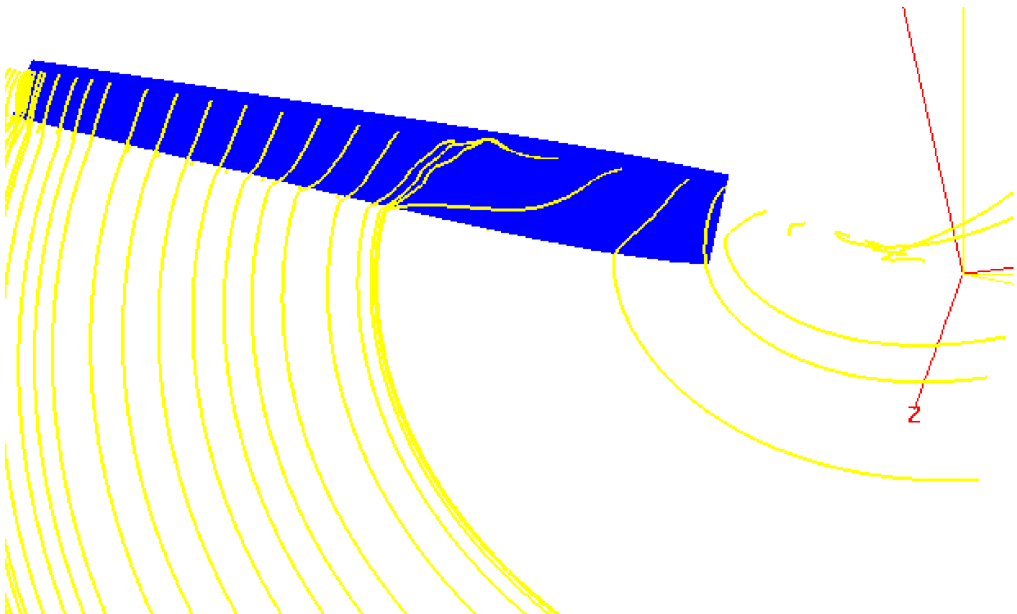

Figure 3 a) Calculated flow field without root vortex markers for the Phase VI Rotor at 10m/s
A larger separation zone at the root is predicted at the root.

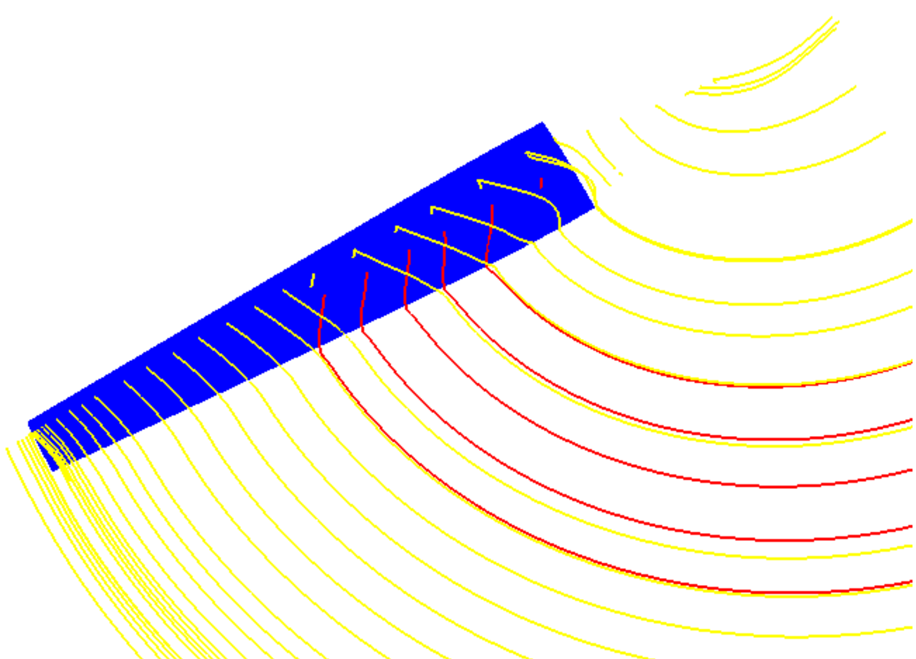

Figure 3b) Calculated Flow Field for the Phase VI Rotor at 10m/s with Root Vortex Markers.
A smaller separation zone is seen.
(Yellow trace lines are above the separation zone, red trace lines are in the separation region, centrifugal pumping effects are confined to separation zones)