THE 5TH ASIAN COMPUTATIONAL FLUID DYNAMICS
BUSAN, KOREA, OCTOBER 27~ 30, 2003

NUMERICAL SIMULATION AND VISUALIZATION OF THE FLOW AROUND THE DARIUS WIND TURBINE

Tetuya KAWAMURA and Mi Young LEE
Graduate school of humanities and sciences, Ochanomizu Univerity 2-1-1 Ootsuka Bunkyo-ku Tokyo
112-8610 JAPAN

ABSTRACT

Complex flow field around the Darius turbine rotating stationally are simulated by solving the three dimensional incompressible Navier-Stokes equation numerically. The rotating coordinate system is employed so that the boundary conditions on the blades of the rotor become simple. In order to impose the boundary condition on the blades precisely, the boundary fitted coordinate system is employed. Fractional step method is used to solve the basic equations.

The complex flow fields due to the three dimensionality of the geometry of the turbine and the rotation of the turbine are obtained and they are visualized effectively by using the technique of the computer graphics.

Keywords: DARIUS turbine, Navier-Stokes Equations, Numerical Methods

1. INTRODUCTION

The wind force is widely recognized as the environmentally friendly energy and attracts public attention. Actually, the number of wind power plants for electricity increases recently. Although the propeller turbine is the most widely used, the Darius turbine is also suitable for generating electricity because of its high performance. Moreover, as its axis is vertical to the wind, it can be rotated by the wind of any direction. The objective of the present study is to simulate and visualize numerically the flow field around the wind turbines in order to show the effectiveness of the numerical method and to obtain fundamental data for their design.

2. NUMERICAL METHOD

Incompressible Navier-Stokes equations are used for this simulation, since the flow speed is not very high. The rotating coordinate system which rotates on the same speed of the turbine is used in order to simplify the boundary condition on the blade. Additionally, the boundary fitted coordinate system [1]is employed in order to express the shape of the blades precisely. Fractional step method[2] is used to solve the basic equations. Third order upwind scheme is chosen for the approximation of the non-linear terms since it can compute the flow field stably even at high Reynolds number without any turbulence models[3].

At first, two-dimensional grid system in the plane perpendicular to the rotational axis is generated. As is shown in fig.1, this region is divided into two regions, i.e. inner region and outer region. The inner side of the blade is located at AB while the outer side is located at CD in this figure. The grid systems in both regions are obtained by deforming the grids of the polar coordinate. The shape of the blade is determined by using Joukovski transformation.

After obtaining the grids in each cross section, they are piled in the axial direction to get whole grid system. Fig. 2 shows the part of the grid system mainly used in this study. The grid points is $120 \times 63 \times 21$ in circumferential, radial and axial direction respectively. The far boundary in each cross section is the circle of radius 10R where R is the distance between the axis and the blade of the turbine. The lower and upper boundary locates z=0 and z=2L where L is the height of the turbine.
Initially, the flow is assumed to be uniform in whole region. The flow is also assumed to be uniform at far boundaries except the outflow boundary where the velocity is determined by the extrapolation. The pressure gradient at far boundaries is assumed to be zero. No-slip condition is imposed on the blade.

The pressure on the blade is determined by substituting the velocity condition (no-slip) into the momentum equations. The shape of the Darius turbine is symmetric with respect to the center plane, i.e. the plane which is perpendicular to the axis of the turbine and passes the midpoint of the axis. Therefore symmetric boundary condition is imposed on the center plane.

3 RESULTS

Examples of the results obtained by the present numerical method are shown here. The speed of the flow at far boundary is assumed to be 5m/s and the kinematic viscosity of the fluid is $0.03\, m^2/\mathrm{sec}$. This value is two thousand times as large as molecular viscosity of the air. However this is reasonable since the effective viscosity (i.e. eddy viscosity) of the air is not so small due to the turbulence.

Figure 3 shows the velocity vectors in the center (symmetric) plane. The flow comes from left side of the figure. Figure 4 is the pressure contours on the blades and those in the same cross sections as fig.3. In this case, the flow is also comes from left.

4. REFERANCES

[1] Thompson, J.F., Warzi, Z.U.A. and Mastin, C.W. (1985): Numrical Grid Generation, Foundations and Applications, North-Holland
[2] Yanenko, N. N (1971) The Method of Fractional Step, Springer-Verlarg
[3] Kawamura,T and Kuwahara,K (1984) : AIAA paper 84-0340

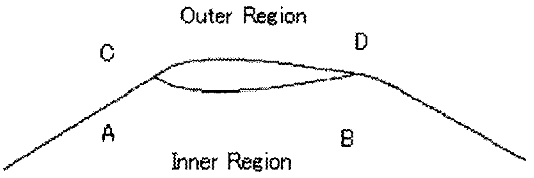

Figure1: Grid lines passing the blade of the Darius turbine

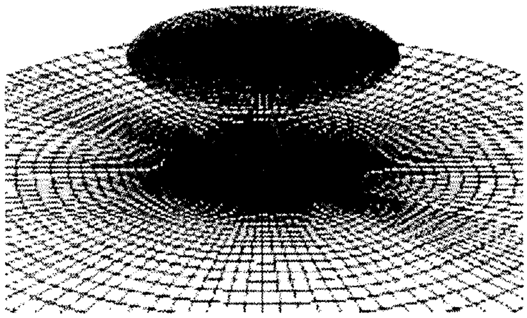

Figure2: Grid for the Darius rotor

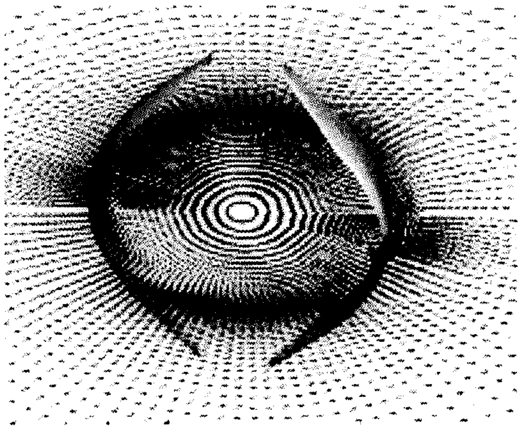

Figure3: Velocity vectors in one cross section normal to the axis of Darius

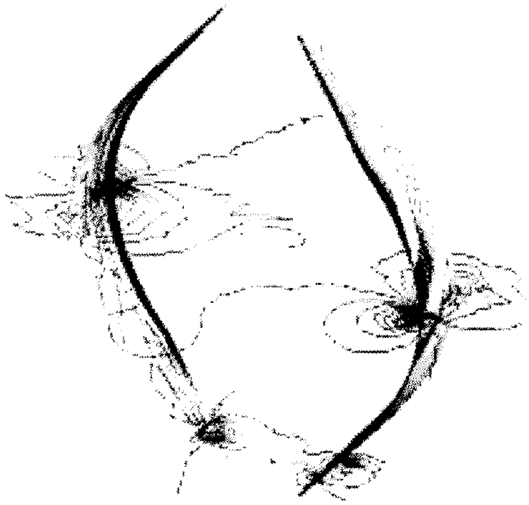

Figure4: Pressure contours on the blade and in two cross sections