International Journal of Offshore and Polar Engineering
Vol. 11, No. 4, December 2001 (ISSN 1053-5381)

Numerical Simulation of 3-D Stall Mechanism on Wells Turbine for Wave-Power Generating System

M. Suzuki and C. Arakawa
Department of Mechanical Engineering, The University of Tokyo, Tokyo, Japan

ABSTRACT

The Wells turbine has a cascade whose stagger angle is 90º, that is, the blades are perpendicular to the axial velocity. Good performance is required from 0° to 90° of the angle of attack because the turbine is operated in the oscillating airflow produced with wave energy. Further, very interesting and complex flows are experimentally observed by the oil-film method in the large angle of attack where the performance is strongly influenced, especially the self-starting aspect. This paper tries to analyze the mechanism of these 3-dimensional flows around the turbine with numerical analysis, focusing on the off-design condition.

INTRODUCTION

The Wells turbine is an axial flow turbine mainly used for wave energy devices of the oscillating water-column type, because the turbine has a simple structure and can rotate in the same direction even if the air changes the direction of flow. For this reason it is called a self-rectifying turbine. In order to start the rotation, the turbine has to be operated in the oscillating flow from the air chamber in which the rotor's attack angle changes from 90° to 0°. Ideally, of course, the angle of attack is limited to an appropriate angle, such as less than 15° for the design condition. From the viewpoint of the self-starting aspect and the advanced design for extending the operation zone, analysis of the complex flows in the larger attack angles is required, including the mechanism of stall on the rotating blade.

The theoretical investigations of the Wells turbine were carried out as a first stage of these researches by using the general formulation with the momentum theory (Grant et al., 1981; Suzuki and Arakawa, 2000). The 2-D potential flow theory, the streamline curvature through-flow method and the actuator disk theory (Gato and Falcao, 1990) followed the momentum theory. The 3-D Navier-Stokes equations were numerically calculated to predict the characteristics of a Wells turbine with computational fluid dynamics (CFD) codes, including the commercial ones (Watterson and Raghnunathan, 1997a, b). They attempted to predict the performance and the flow around the Wells turbine in the design conditions, but they did not cover the application of the stall condition and a detailed prediction of performance.

Before researching numerical simulation, the experiments of the oil-film method in the water-circulating tunnel were carried out in order to understand the Wells turbine's complex flow. The flow visualization explained the influence of the attack angle, the solidity, and the difference between fan-shaped and rectangular wings (Suzuki et al., 1984a, b; 1985). The complex and interesting flow patterns on the rotating blade were observed during the operation of the large attack angle especially. These experimental results allowed the authors to analyze them with numerical simulation.

The objective of this paper is to analyze the mechanism of these 3-D flows around the turbine with numerical analysis. These complex flows of a Wells turbine are numerically solved using the incompressible Navier-Stokes equations and the domain decomposition of the grid system around its rotor with the tip clearance. This research will reveal the creative mechanism of the vortex in the Wells turbine's stall condition.

EXPERIMENTAL APPARATUS

Performance Test

Fig. 1 is a schematic view of the experimental apparatus. The inside diameter of the casing $D_c = 2R_c$ is 304.4 mm, and the hub diameter $D_h = 2R_h$ is 215 mm. Fig. 2 describes the rotor profile, which has 8 blades, a chord length of 73.5 mm, a wing section NACA0021, a solidity of 0.7 at the root mean square of the radius, $R_{RMS} = \left[(R_c^2 + R_h^2)/2\right]^{\frac{1}{2}}$, and a tip clearance of 1.0 mm. The wind tunnel is designed as a suction type. The blower charges the airflow to the turbine with the orifice and the flow control valve installed between turbine and blower. The measured data are the pressure drop across the turbine, the turbine torque, the rotational speed and the flow rate.

The Reynolds number is described with the representative velocity and length, which are assigned for the relative velocity at the tip, $(U^2 + V_a^2)^{\frac{1}{2}}$, and the blade's chord length, respectively. The Reynolds number is fixed at about $1.7\text{x}10^5$ when the angle of attack is less than 14.5°, and approximately $0.42 \times 10^5(1 + 1/\tan^2\alpha)^{\frac{1}{2}}$ when it is larger than 14.5°. The notations of U, V_a and α are the blade speed of the tip, the axial velocity and the angle of attack, respectively.

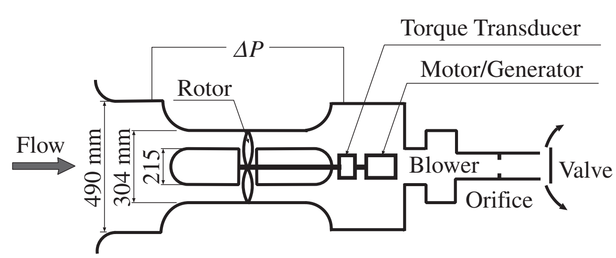

Fig. 1 Schematic explanation of experimental apparatus for performance test

Received November 6, 2000; revised manuscript received by the editors August 20, 2001. The original version (prior to the final revised manuscript) was presented at the Tenth International Offshore and Polar Engineering Conference (ISOPE-2000), Seattle, USA, May 28–June 2, 2000.
KEY WORDS: Wells turbine, wave power, 3-D, stall mechanism, numerical simulation.