Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 49(1) (2017) 3-4
DOI: 10.22060/mej.2016.852

Experimental Investigation of Flow Around a 3D Square Cylinder Using Five-hole Probe and Neural Networks Method

R. Eftekhari[1*], R. Taghavi[1], S. M. Nima Shojaee[2]

[1] Mechanical Engineering Department, Iran University of Science & Technology, Tehran, Iran
[2] Mechanical and Aerospace Engineering Department Science and Research Branch of the Islamic Azad University (SRBIAU), Tehran, Iran

ABSTRACT: In the present research, axial compressor blade cascade was analyzed by using numerical and experimental methods. The model includes three rotor blades of an axial compressor having the same geometry and profile of NGTE 10C4/30C5 aligned parallel with a ratio of chord length to the blade pitch (rigidity) 0f 0.8-1.2. The experiments were done in a subsonic wind tunnel. Before the tests, some reforms in the output of the wind tunnel were designed to install a new test section, suitable for cascade testing. The necessary considerations are made to let a hot-wire probe to go through the wind tunnel's wall move on the surface of the blades. The suction surfaces and pressure of the test blade have been equipped to the suitable tapping of the pressure to be connected through the connector hoses to the pressure transducers. Different situations between the blades were performed and at any stage of testing, the suction surface velocity and pressure profiles were measured in different longitudinal situations and distribution of surface pressure. Parallel with the experiments, the numerical analysis for different modes has been performed and the results were compared to experimental data. Combining the results of CFD and experimental measurements gives us a powerful tool for simulation, design and improvement.

Review History:

Received: 31 May 2014
Revised: 10 January 2016
Accepted: 15 November 2016
Available Online: 15 November 2016

Keywords:

Cascade
Axial compressor
Solidity
Hotwire
Wind tunnel

1- Introduction

In compressor design process, knowing the aerodynamic performance of the set of blades is of great importance. Designers must have a general idea of the effects of various parameters on the blades' fluid dynamics behavior. These parameters include fluid properties, especially in upstream, as well as the geometric properties. Upstream flow properties mainly include the Mach number, incidence angle, and Reynolds number, while geometric properties mainly include cross-sectional shape of the blade, pitch and installation angle. Hayashiba developed a model to measure the effects of Mach number on the stagnation pressure loss and entropy generation [1]. Whitehead proposed a method for calculating the energy and momentum coefficients in cascade compressors. Measurements were conducted for two different rigidities and were also compared with numerical solutions [2].

In this study, the effect of solidity, or the peripheral distance of axial compressor blades, was investigated as in all turbomachines, blade rows are forced to work at an equal incidence angle. However, the design point in most machines may change. Moreover, machines' off-design performance is also important because they may need to operate in off-design conditions as well.

2- Setup and Experimental Tests

The wind tunnel used in this study has a square cross-section with dimensions of 45×45 cm and a length of 90 cm. The blades' profile is NGTE 10C4/30C50, and blades were made of plexiglass. The nozzle has a dimension of 14.5×60×45 cm, and the lowest air outflow velocity without causing fluctuations at the exit is 34 m/s. In addition, the maximum air outflow velocity is 47 m/s. The ratio of the test length to test height in a range of 1 to 3, while the optimum recommended ratio is about 1.5. To measure the static pressure, 12 holes were created on the suction surface, and 11 holes were created on the pressure surface and were connected to plastic tubes using needles (as shown in Fig. 1).

Corresponding author, E-mail: aero2010.iust@gmail.com

Figure 1. Pressure tapping's configuration

3- Numerical Analysis

The realizable k-ε turbulence model was used. The model uses a new formulation for the turbulent viscosity. The reason for using this model is to better model the boundary layer and provide the results in places with strong adverse pressure gradient, where separation and recirculation occur [3]. The equations used in this study include Navier-Stokes equations with the assumption of viscous, incompressible and unsteady flow, and the solution method was second order implicit continuous method. Mixed grid was used for mesh generation. As can be seen in Fig. 2, in areas near the walls where the behavior of the boundary layer is important, structured grid was used, which is suitable for near-wall regions, while triangular grid was used in other regions. The inlet condition was realized using a velocity of 34.64 m/s, and the outlet boundary condition was realized by defining outlet static pressure. In this study, the static pressure at the outlet was assumed as the atmospheric pressure. It should be noted that boundary conditions used for pressure are relative to the reference pressure. Finally, after flow analysis and convergence of solutions, to ensure accurate and complete modeling of the boundary layer, the results obtained for Y^+, which is the dimensionless distance of the first grade on the surface, should have the correct value (less than 1). Otherwise, the computational domain should be re-meshed to generate a better grid near the walls [4].

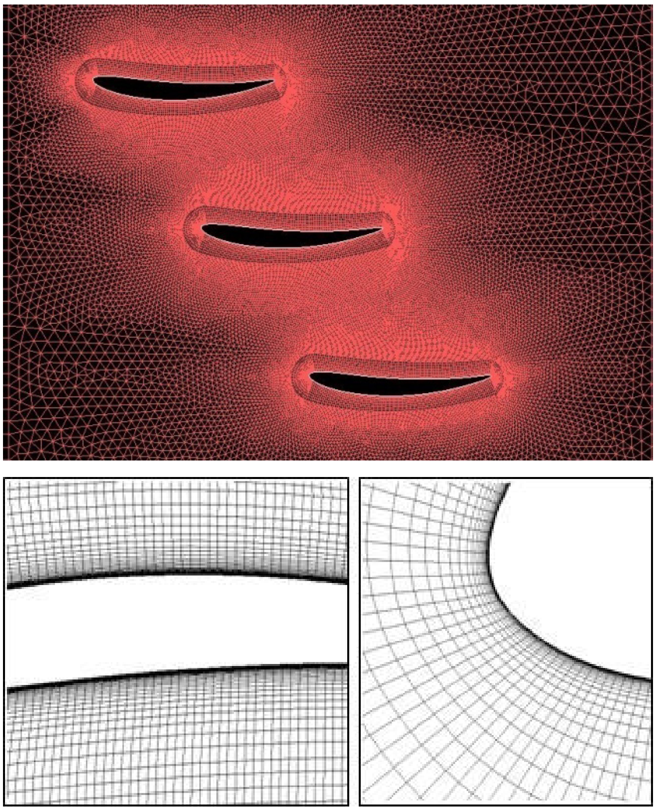

Figure 2. Grid domain near the blade surfaces

4- Results and Discussion

The effect of solidity on the pressure distribution at the pressure and suction surfaces were studied. Experimental measurements and numerical analysis were conducted on the middle blade in a 3-blade.

Comparisons show that maximum error between numerical and experimental results is 5%, indicating the accuracy of results. On pressure surface, increasing the solidity reduces the pressure coefficient. As we move toward the trailing edge, difference in pressure coefficients decreases, so that they have approximately equal values at the trailing edge. On the suction surface, increasing the solidity reduces the pressure coefficients. It is also important to note that the increased solidity displaces the stagnation point from the suction surface to the pressure surface. It is noteworthy that the conditions of tests done in 2D, considering the kinematic and dynamic ratios used in the reference, is similar to 3D, and only changes in parameters on profiles can be relied on. Velocity distribution near the surface was then compared from two perspectives of experimental and numerical studies. Due to the difficulties in working with hot wire sensor near the surface, it is not possible to compare the numerical solutions in this regions with those obtained from experimental measurements. Comparison of solutions obtained at a distance of more than 0.1 mm by hot wire anemometer and numerical solutions showed a good agreement.

Results show that reducing the solidity increases the thickness of the boundary layer on the pressure surface. The boundary layer thickness on this surface is low in the leading edge; however, the value increases significantly by approaching the trailing edge. On the suction surface, considering the curvature near the leading edge, the error between the experimental results and numerical solutions is more than 10 percent. However, by comparing the velocity distribution graphs on this surface on other sections, acceptable differences (less than 5%) were observed in the solutions. It is also observed that increasing the solidity reduces boundary layer thickness on the suction surface. Boundary layer thickness also increases by approaching the trailing edge. It should be noted that, at a constant distance from the surface of the blade, the velocities at every sections of the suction surface are greater than that of the pressure surface.

5- Conclusion

In this study, numerical and experimental investigations of solidity effects on aerodynamic performance of axial compressor cascade are performed. It is seen that by increasing the solidity, pressure decreases while velocity enhances. According to conservation of mass, the velocity must increase since the ratio of inlet area to outlet area is lessened.

References

[1] Hayashibara, S., "*Cascade flow Simulation and Measurement for the Study of Axial Compressor Loss Mechanism*", Wichita State University, 2003.

[2] Whitehead, D. S, "*Force and Moment Coefficients for Vibrating Airfoils in Cascade*", University of Cambridge, 1962.

[3] Athanasiadis, A.N., D.G. Koubogiannis and K.C. Giannakoglou "One- and two-equation turbulence models for the prediction of complex cascade flows using unstructured grids", 2001.

[4] Rodrick V.Chima, "*Analysis of Inviscid and Viscouse Flows in Cascades with and Explicate Multiple-Grid Algorithm*", Lewis Research Center Cleveland, Ohio, 1984.

Please cite this article using:

R. Eftekhari, R. Taghavi, S. M. Nima Shojaee, "Experimental and Numerical Investigation of Rigidity Effects on Aerodynamic Performance of Axial Compressor Cascade", *Amirkabir J. Mech. Eng.*, 49(1) (2017) 3-10.

DOI: 10.22060/mej.2016.852