International Journal of Natural and Engineering Sciences (6)1 : 1-6 , 2012
ISSN: 1307-1149, E-ISSN: 2146-0086, www.nobel.gen.tr

Empirical and Numerical Study of Solidity Effects on Pressure Coefficient and Velocity Distribution of Axial Compressor Blades

Reza EFTEKHARI[1] Soheil FOULADI[2] Reza BARZEGAR[3]

[1] Faculty of Mechanical & Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, IRAN

[2] Faculty of Mechanical Engineering, Iran University of Science & Technology, Tehran, IRAN

[3] Faculty of Mechanical Engineering, Khaje Nasir Toosi University of Technology, Tehran, IRAN

***Corresponding Author**

e-mail: aero2010.iust@gmail.com

Received : September 01, 2011
Accepted : October 09, 2011

Abstract

In the present research, the axial compressor blade's cascade was analyzed in terms of numerical and experimental conditions. The model included three rotor blades of an axial compressor with the same geometry and profile of NGTE 10C4/30C5 and parallel to the ratio of solidity values, e.g. 1.2 and 0.8, have been studied. The experiments were done in a subsonic wind tunnel and, before it, the reforms in the output of a wind tunnel to install a new test section, suitable for cascade testing, were designed and then built. The necessary measures also for the possibility of a hot-wire probe crossing from the wind tunnel wall and its moving on the surface of the blades has been adopted. The suction and pressure surfaces of the test blade have been equipped to the suitable tapping of the pressure to be connected through the connector tubes to the pressure transducers. The suction and pressure surface velocity profiles in different longitudinal situations and distribution of surface pressure were measured. Parallel with the experiments, the numerical analysis of current also for different modes has been performed and the results compared. In a total sum can be said: by increasing the solidity, namely reducing the blades distance, the pressure reduces and the velocity will increase. According to the continuity equation can be said with increasing the solidity the input and output area ratio decreased and considering to the reverse relation of the area ratio with velocity, the velocity increases.

Keywords: Cascade, Axial Compressor, Hot Wire, wind tunnel, solidity

INTRODUCTION

In the design process of compressors, the knowledge of the aerodynamic performance of blades cascade is very important. Designer should have an overall insight to the effects of various parameters on the fluid dynamic behavior of the blades. These parameters include the fluid properties especially in the upstream flow and geometric properties. Upstream flow properties mainly include Mach number, Incidence angle, Reynolds number and geometric properties include, shape of blade section, pitch size and solidity.

In this study between these parameters has been addressed the effect of solidity or the environmental distance of axial compressor blades, because, the solidity is an important parameter in designing the cascades. Whatever the amount of solidity is smaller, therefore, the turbo-machine is consequently smaller and cheaper. On the other hand, the chord was controlled by the mechanical limitations and cannot be less than a certain amount (mainly, the centrifugal stresses which were applied to each blade) , also increasing the pitch of blades causes to exacerbate the secondary flow and separate the flow in the blades and decrease the efficiency therefore ,the solidity amount cannot be less than a certain amount.

It should be mentioned, one major problem of axial compressor design is the fluid stall phenomenon that mainly due to the lack of applying the aerodynamics principles in the design of blades occurs that is due to lack of stability of the air passing through the blades, because the air must move from the pre-predicted design angles and if the order of these angles to be disturbed, the vortex flows and stall in the stages of compressor occur and the air is unable to move forward [1]. If this phenomenon to be spread in the next steps of blades, the compressor will be in a complete stall status and not only can't deliver the high- pressure air, but will produce the vibrations combined with noises and many damages in the whole engine, thus studying the angle change effect in determining the velocity profile on the surfaces of cascade blades have the great importance. Hayashiba (1998) following their work, presented a model to measure the effects of Mach number on the stagnation pressure loss and entropy production [2]. White Head (1962) proposed a method for calculating force and momentum coefficients in the cascade compressor. He has performed his measurements for two different solidity and compared with the numerical solution [3].

Experimental Setup

Commonly, the blower wind tunnels were used for cascades and provide the consistent properties and conditions in the test section. In this study, the used wind tunnel which was in the type of subsonic and blower and with the output velocity of 12 (m/s) and the dimensions of 45 (cm) * 45 (cm) that according to the research objectives, this velocity is less than the optimum value, therefore, the nozzle to achieve the higher have been designed and developed. Studied blade profile was NGTE 10c4/30c50 that was made of Plexiglas, because this material has relatively less weight and a feature that can be drilled easily, also the blade surfaces which were made of this material do not need to be re-polished. These blades were installed in the test section, in such a form that their distance from each other compared to their cord, and the stagger angle respectively to be equal with one and 36°.The height and length of cascade are in the size that the available air can easily pass through it and the disturbing effects of tunnel walls will be disappeared. The change in the rate of solidity by maintaining the middle airfoil constant and changing the distance of upper and lower airfoil and maintaining them by the external clamps constant for testing is done. Fig.1 shows the collection of cascade, which was made in the test section.

The nozzle used in this study as observed in Fig.2 was with the dimensions of 45 (cm) * 60 (cm) * 14.5(cm) and has brought the minimum and maximum output speed of the air without causing fluctuations in the exhaust respectively to 34m/s and 47 m/s and for the tested nozzle to obtain the profile was used the low convergence ratio of 3.

The main body of test section is made of a transparent material, such as shatterproof glass, Plexiglas or talc and usually the lower surface of section was made of the harder materials such as iron and for gaining access to this area, the valves also are placed in lateral walls of the section. The test section length is as its height in the range of 1-3, that the recommended best ratio is about 1.5 [4].

The boundary layer is created by passing the air flow over the test cross-sectional area on them, and this reduces the effective cross-section area and according to the continuity equation by decreasing the area of cross-section, air velocity and static pressure also increased. So to prevent from this loss of pressure slightly makes the test section divergent [5]. The divergence angle is usually about half to one degree.

The wind tunnel used in this study, is with square dimensions of 45.6(cm) * 45.6(cm) and the length equal to 90 (cm). The test section length, usually between one to two time of the largest side of the test section is selected to decrease the turbulence of the air velocity and increase the flow uniformity and create the parallel flow lines to enter the test section, before the test section, an area of constant cross section and with the short length was inserted until the contraction nozzle outlet flow lines to be uniform.

To measure the surface static pressure of airfoil, 12 holes on the suction surface and 11 holes on the pressure surface were built so that can be seen in Fig.3 by some needles connected to the tubes as made of plastic. The tubes inside the airfoil are connected to the electronic pressure transducer which some of them are hollow shaped and the wall outside the test section is salient. The measurement accuracy of transducer is 0.1 mm of water.

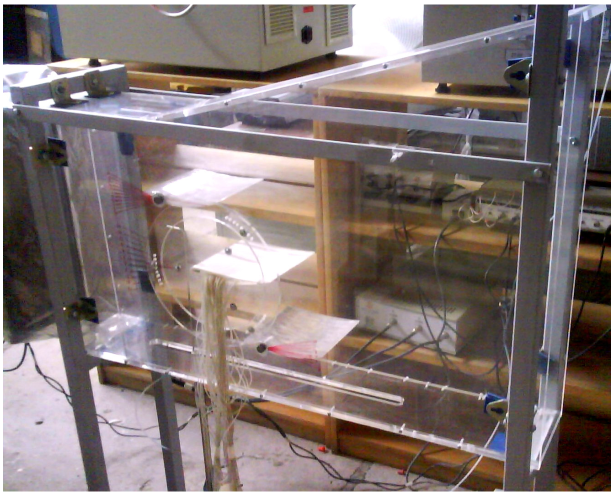

Fig.1. Collection of cascade in test section

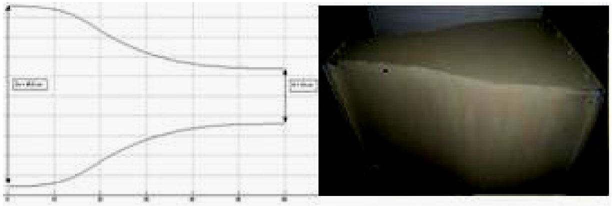

Fig.2. Nozzle

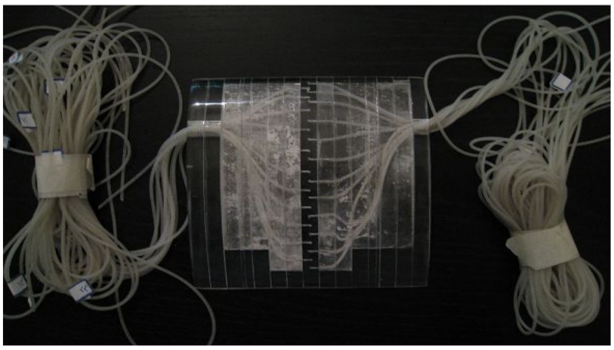

Fig.3. Pressure tapping's configuration

Hot-Wire Anemometer

By the hot-wire anemometer, the instantaneous fluid velocity with high frequency, signal ratio compared to the low noise and in a wide range and using the measured instantaneous velocity, average velocity, of fluid flow turbulence, Reynolds stress can be measured. The hot-wire anemometer accuracy in good condition is about 0.1% - 0.2%. The probe Hot-wire used in this study schematic have been represented in Fig.4.

In this study, should be used SN, which measures the average velocity and the fluid flow turbulence in one dimension and then the boundary layer probe which measures the characteristics needed by boundary layer should be used. This study also used of Tungsten sensor which its time constant ($\sigma.\alpha / \rho.c$) compared to the changes in the fluid flow was small and consequently, the frequency response is quick. This sensor was bare and respectively has the diameter and length about 5 μm and 1.5 mm.

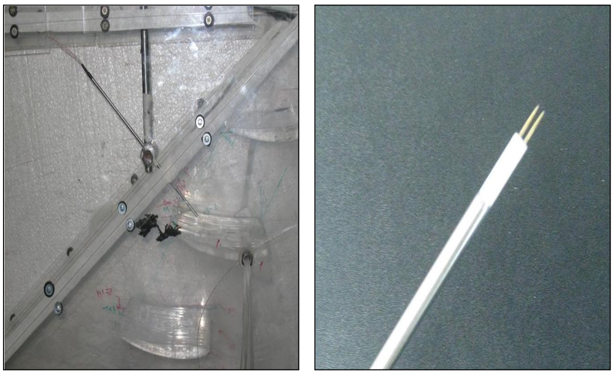

Fig.4. Hot-wire anemometer

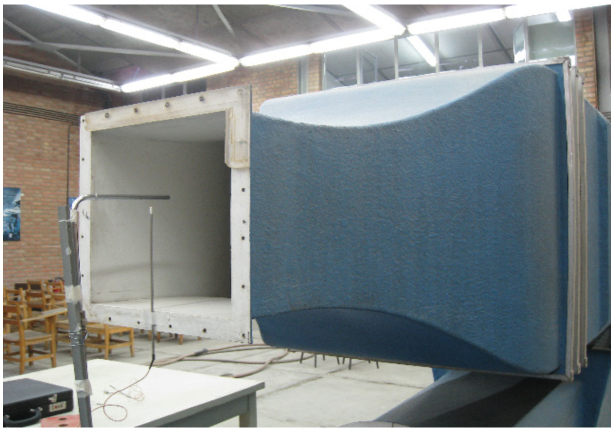

Fig.5. The wind tunnel used for calibration

In this study, hot-wire anemometer is a kind of constant temperature anemometer, the constant temperature electronic circuit (CT) includes Wheatstone bridge, differential amplifier, frequency regulator and flow amplifier. Under constant temperature conditions, thermal inertia of the sensor element while changing fluid flow conditions will automatically be adjusted. This feature will be along with major advantages in the measurement conditions. The performance status by putting a differential amplifier factor in the circuit of hot wire to obtain a rapid change in the heater flow of sensor is calculated to compensate the sudden changes in fluid flow rate. According to this that the hot-wire anemometer can measure the instantaneous flow velocity with the frequency higher than 30 kHz, according to Naikost law to receive the correct data, the minimal sampling rate should be twice of the hot-wire anemometer frequency and the frequency is determined and marked in the signal conditioner circuit by low pass filter. According to the type of fluid flow and Reynolds number, the frequency of fluid flow velocity turbulence was specified and according to it, the sampling rate is determined[6].

Static Calibration

The software of the hot-wire anemometer can show the output of a hot-wire anemometer as oscillogram, so the frequency response of the hot-wire anemometer can be observed and regulated compared to a square wave through a computer system. Since the anemometer has a very high accuracy and the smallest change in flow conditions of probe placement status, pollution of measurement environment and so on can affect the amount of output voltage, so the calibration before each experiment seems to be necessary[7].

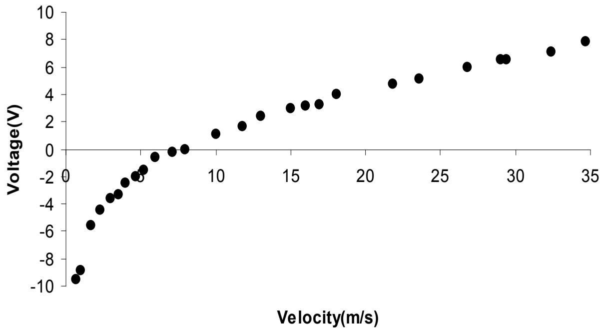

Fig.6. Calibration curve

In this study, the 27 points between the velocity of 1 m/s to 35 m/s were calibrated and in Fig.5 can be seen that the hot-wire anemometer has been calibrated in the other wind tunnel which was in the type of closed circuit and the velocity about 2m/s to 40m/s were accessible in it.

The calibration equation of hot-wire anemometer is a polynomial equation degree 4 that the accuracy and simplicity of using the equation is so that the approximation error of curve is declined and in the very short time, the flow velocity using the output voltage of hot-wire anemometer can be determined. In Fig.6 the calibration curve of hot-wire anemometer schematic has been represented.

Determining the Exact Position of Hot-wire Sensor Compared to the Airfoil Surface

One problem with hot-sensor is the measurement of near wall velocity. Determining the reference point for calculating the position of hot- sensor compared to the airfoil surface was one of the most important activities, which should be done. In the studies presented by researchers, They consider the reference as a point for the contact of probe bases with the surface, that this type of determining the reference point due to the physical contact of sensor with the wall and low mechanical strength enhances the probe possibility of burning[8].

In this study, the functional characteristics of vector data system and hot-wire sensor are used to detect the actual position and this method required no physical contact with the sensor surface, Therefore, the possibility of burning the sensor reaches a minimum. As in Fig.7 observed, the air with pressure from the hole which was built to measure the pressure on the airfoil surface is blown out and in the same mechanism has been shown in the Figure, closes to the hole and is placed on it, the tip of rod has a groove so that the air jet after the collision is diverted.

In fact, the air flow becomes as a plane jet that its height is about 0.2 mm, if the sensor is placed at the front of the jet, then

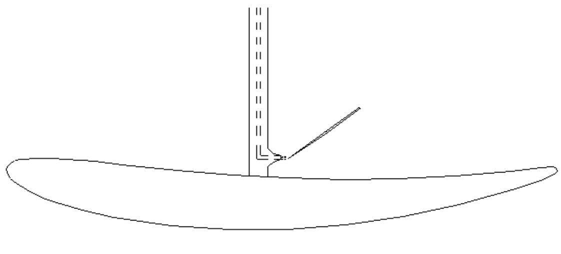

Fig.7. The mechanism of determining the exact position of the hot-wire sensor compared to the surface

the maximum velocity in the center of the jet will be measured. As regards, the exact distance of the sensor to the surface is determined; the sensor distance to the airfoil surface is to be determined.

Probe Traversing Mechanism

When measuring the velocity profile near the wall requires high accuracy in determining the location of the hot-wire sensor compared to the airfoil surface. For the probe can be placed in an appropriate location and velocity distribution or flow rate changes can be measured compared to the location, we need the mechanism of probe transfer in order to ensure of the probe displacement and prevent the hot-wire collision to the surface, the mechanical watch with the accuracy of 0.01 mm and parallel with the above system shows the exact amount of vertical displacement. The probe transfer mechanism schematic has been represented in Fig.8.

Temporal Analysis

The study of instantaneous velocity of fluid flow can be done in the time and frequency domain. In the time domain, showing the instantaneous velocity according to Fig.9 is as oscillogram and using the oscillogram can be studied the instantaneous velocity changes of the fluid flow compared to the time. To determine the exact instantaneous velocity is the better to investigate it in the frequency domain according Fig.10 and then specify the amount of fluid flow velocity turbulence energy in a certain frequency certain and determine the quality

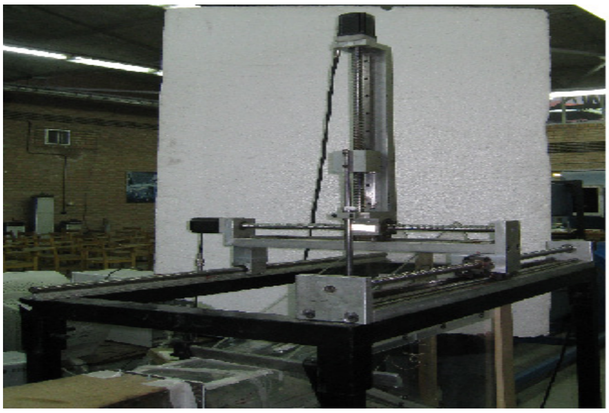

Fig.8. Probe traversing mechanism

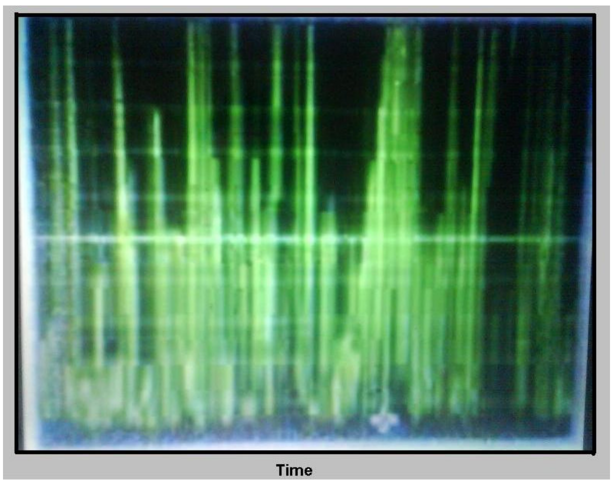

Fig.9. The fluid flow turbulence in the time domain

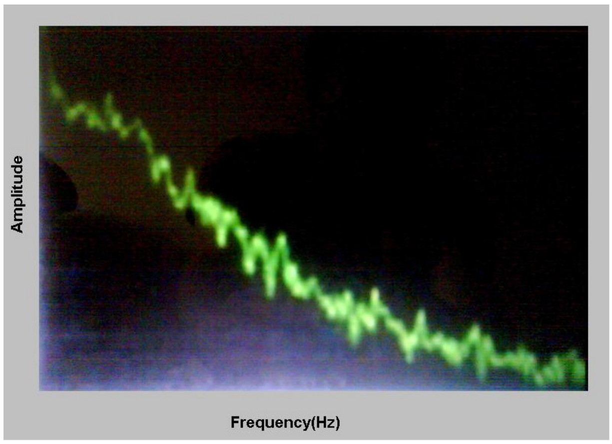

Fig.10. The fluid flow turbulence in the frequency domain

of fluid flow. As specified in Fig.9 and Fig.10, the flow was completely turbulence and the dominant frequency is about 1 KHz.

Numerical Analysis

The numerical, computational fluid dynamics methods are the suitable design methods for the turbo machines to be analyzed and the equations used for turbo machines are the same equation which was used for the other flows. But, there is a difference that the boundary conditions for turbo machines include the most complex cases in numerical fluid dynamics topics and the turbulence model used in this study is Realizable k-epsilon. The Realizable $K-\varepsilon$ model includes a new formulation for the turbulent viscosity. The reason for using this model is due to the better modeling of boundary layer and, representation of the better results occurs where the strong pressure gradients are reversed and in parts that the re-separation and re-rotation occurs[9].

Alternative formulation for turbulent viscosity:

$$\mu_t \equiv \rho C_\mu \frac{k^2}{\varepsilon}$$

where:

$$C_\mu = \frac{1}{A_o + A_s \frac{U^* k}{\varepsilon}}$$

is now variable. (A_0, A_s, and U* are functions of velocity gradients).

New transport equation for dissipation rate, ε:

$$\rho\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho c_1 S\varepsilon - \rho c_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} + c_{1\varepsilon}\frac{\varepsilon}{k} c_{3\varepsilon} G_b$$

Convection Diffusion Generation Destruction Buoyancy

Typically values for the model constants C1ε and C2ε of 1.44 and 1.92 are used [10].

The used equations, Navier–Stokes equations, assuming the viscous flow are non- reversible and non-permanent and numerical methods to solve the equations are continuous implicit method of the order 2.

In this analysis is used a house code based on finite volume method and used of the structure mesh ,in the regions near the wall that the boundary layer behavior is important, of structure

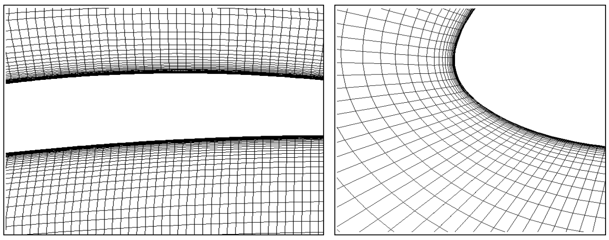

Fig.11. Grid domain near the blade surfaces schematic

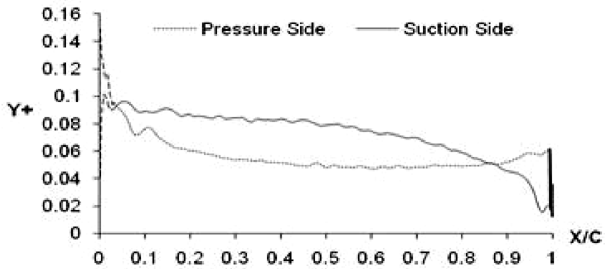

Fig.12 . The diagram of

mesh (suitable for areas near the wall). On the near wall regions of the structured and organized mesh with compacted mesh on the areas near the surface of blade was used .so that a mapping is in one to one form, also in the areas where there is the severe flow gradients, the distribution of mesh is denser and grid lines distortion is low[11].

In this study, input condition for the input velocity 34.64 m/s and the pressure outlet boundary condition is achieved by defining the static pressure. In this study, the static pressure on exit is assumed atmospheric pressure. It should be mentioned that the boundary conditions used for pressure compared to the reference pressure was specified. Grid system near the blade surfaces schematic have been represented in Fig.11.

Finally, after analyzing the flow and convergence of the solution to ensure of the accurate and complete modeling of the boundary layer, the value obtained was the number y^+ , that is a distance without the dimension after the first grade (grid) on the surface must have a good amount (less than one), otherwise should again another mesh be done for the solution range and a suitable mesh in the near wall region to be produced. In this study, value of Y^+ on the blade surfaces schematic have been represented in Fig.12. If the value of Y^+ is greater than the required value, must a new second mesh for the solution domain to be done and the suitable mesh in the near wall region to be produced.

Stagnation Pressure Loss

One of the most important factors in the optimal design of compressors is to minimize the loss resulted from stagnation pressure and the amount of the stagnation pressure loss by integration is as follows:

$$\Delta \overline{p}_0 = \frac{\int_0^s (p_{01} - p_{02}) \rho c_x dy}{\int_0^s d\dot{m}}$$

The pressure loss coefficient with the total stagnation pressure changes ratio in the direction of the flow line on the total dynamic pressure is defined as follows [12]:

$$C_{ps} = \frac{\Delta p_S}{\frac{1}{2} \rho c_1^2} = 1 - \frac{\cos^2 \alpha_1}{\cos^2 \alpha_2}$$

RESULTS AND CONCLUSION

In this research, the solidity effect on the pressure distribution over the blade suction and pressure surface has been discussed and the study was done by the experimental measurement and the numerical analysis on the middle blade of the cascade blades set with number 3. In Fig.13 and Fig.14, the achieved pressure distribution by the analysis of experimental and numerical results in the different solidities on the suction and pressure surfaces have been compared together. By comparing the answers can be seen that the maximum error in the answers obtained is 5 % which shows the high accuracy of numerical solution and the results obtained from the experiment.

As in the results determined, on the pressure surface by increasing the solidity, the pressure coefficient will reduce and whatever we go forward the trailing edge, the difference of pressure coefficients is less so that in the trailing edge has roughly equal amounts. On the suction surface by increasing the solidity, the pressure coefficients reduce. It is important that, increasing the solidity will transfer the stagnation point from the suction surface to the pressure surface. It is noteworthy that the test conditions which were performed in two dimensional state, According to observing the kinematic and dynamic ratios with the used references, were similar to the three dimensional conditions and only can be cited to the changes of parameter on the profiles.

And more, the comparison of velocity distribution near the surface from both experimental and numerical point of view has been discussed.

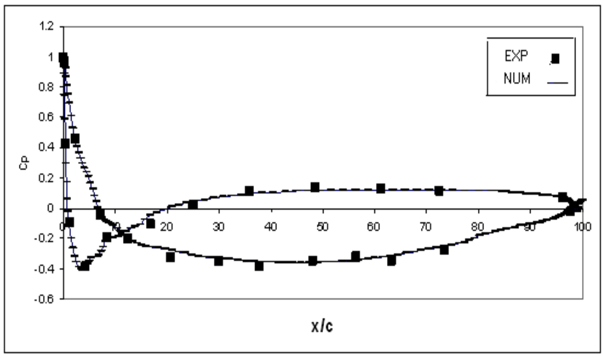

Fig.13. Variation of pressure coefficient at solidity=0.8

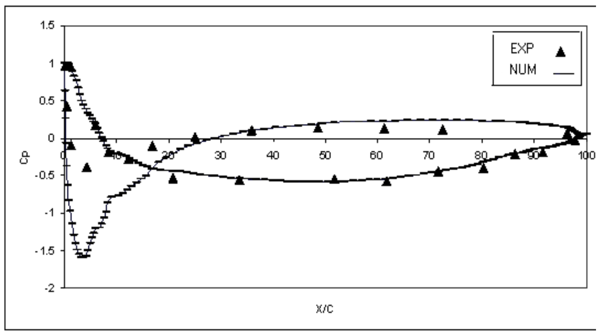

Fig.14. Variation of pressure coefficient at solidity=1.2

By investigating figures 15 to18, it is observed that reducing the solidity causes to increase the boundary layer on the pressure surface. However the boundary layer thickness is low on this surface at the leading edge, this amount becomes significant by approaching the trailing edge. On the suction surface according to the surface curvature near the leading edge, the results of experiment with the numerical solution have the error over 10%.

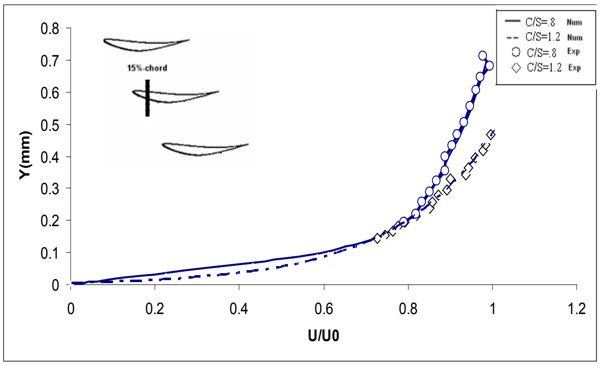

Fig.15. The experimental and numerical comparison of solidity effect in the velocity distribution on the pressure surface at 15 % of chord

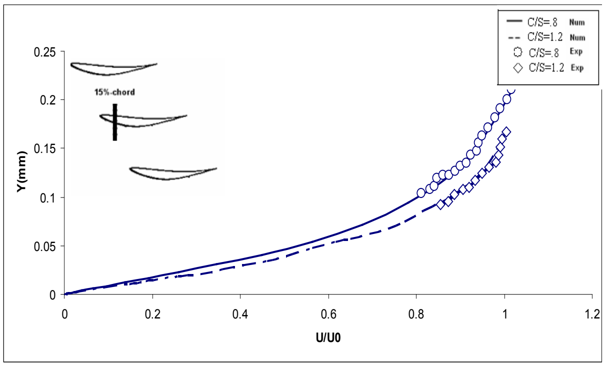

Fig.16. The experimental and numerical comparison of solidity effect in the velocity distribution on the suction surface at 15 % of chord.

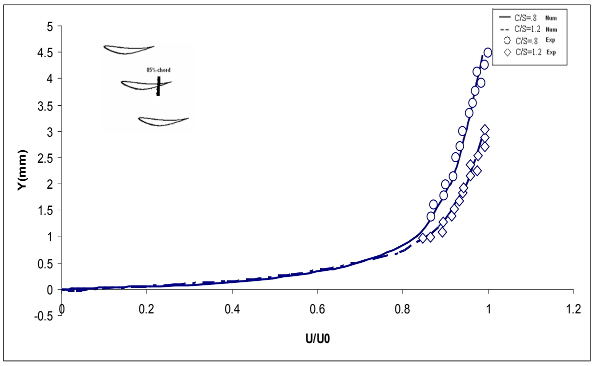

Fig.17. The experimental and numerical comparison of solidity effect in the velocity distribution on the pressure surface at 85 % of chord.

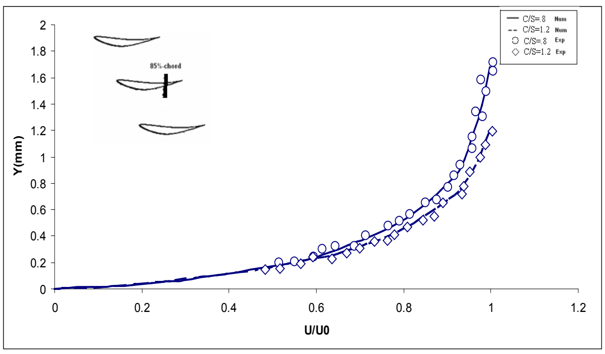

Fig.18. The experimental and numerical comparison of solidity effect in the velocity distribution on the suction surface at 85 % of chord.

However, by comparing the diagrams associated with the velocity distribution in this surface on the other sections, an acceptable difference (less than 5 %) obtained in responses can be observed. By investigating figures 15 to 18 can come to the conclusion that increasing the boundary layer thickness solidity on the suction surface decreases. Furthermore, by approaching to the trailing edge, the boundary layer thickness increases and at a constant distance from the blade surface should be mentioned, the velocity in all sections of the suction surface is higher than the pressure surface.

Due to the problems with hot-wire sensor near the surface, comparing the answers obtained from the numerical solution in this region by laboratory measurement is impossible. By comparing the answers obtained by comparing the distance in more than 0.1 mm from measuring by the hot-wire anemometer, a good accordance with the numerical solution is observed.

REFERENCE

[1] Aungier Ronald H.2003. Axial Flow Compressors, the American society of Mechanical Engineers, New York.

[2] Shigeo Hayashibara. 2003. Cascade flow Simulation and Measurement for the Study of Axial Compressor Loss Mechanism, Wichita State University.

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

[4] Pollard, D. and Gostelow, J.P.1967. “Some Experiments at Low Speed on Compressor Cascades”, Transactions of the ASME, journal of Engineering for Power, Vol. 89,series A,pp. 427-436.

[5] Bammert, K. and Milsh, R.1972. Boundary Layers on Rough Compressor Blades, ASME Paper No. 72- GT-48.

[6] Collis, D.C. and Williams, M.J.1959. Two –dimensional convection from heated wire at low Reynold numbers.,J. Fluid Mech .,6.357-384..

[7] Bruun H.H., 1996 .Hot-Wire Anemometery Principles and signal analysis ,Oxford university press, pp.19-127.

[8] Papadopoulos G.and Otugen M.V. 1993.A Simple automated hot-wire positioning technique for near-wall measurements .Experiments in Fluids, 15.459-461.

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

[10] Shabbier, a , Zhu, J and Celestina, M.1996.Assessment of Three Turbulence Models in a Compressor rotor , presented at the international Gas Turbine and Aeroengine congress & Exhibition Birmingham, Uk.

[11] R. F. Kunz, B. Lakshminarayan, 1992 .Computation of Supersonic and Low Subsonic Cascade Flows Using An Explicit Navier-Stokes Technique amd The k-E Turbulence Mode, Department of Aerospace Engineering Pennsylvania State University.

[12] S. M. Yaha , Turbines ,Compressors and Fans.2002.,Tata MaGraw-Hill Publishing company Limited.