

EXPERIMENTAL AND NUMERICAL STUDY OF SOLIDITY AND INCIDENCE EFFECTS ON AERODYNAMIC PERFORMANCE OF AXIAL COMPRESSOR BLADE

*Reza Eftekhari¹, Reza Taghavi², Ghodrat Ghassabi³ Payam Hooshmand⁴
Farhud Shirinzadeh⁵

¹ PhD student, Faculty of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran.

² Associate Professor, Department of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran.

³ PhD student of mechanical Engineering, Department of Mechanical Engineering, Ferdowsi University of Mashhad, Iran

⁴ Young Researchers Club, Tabriz Branch, Islamic Azad University, Tabriz, Iran

⁵ PhD student, Faculty of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran.

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 chord length to the blade pitch or the 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 designs and actions also for the possibility of a hot-wire probe crossing from the *wind tunnel wall* and its moving on the surface of the blades, have been adopted and implemented. 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 experiments in different situations between the blades were performed and at any stage of testing, 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. Combining the results of numerical fluid dynamics and experimental measurements gives us a powerful tool for simulation, design and improvement. 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 according to the reverse relation of the area ratio with velocity, the velocity increases.

KEY WORDS: 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, setting angle. 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 mainly occurs by the lack of applying the aerodynamics principles in the design of blades that

*Corresponding Author: Reza Eftekhari, PhD student, Faculty of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran. Email : aero2010.iust@gmail.com , Tel: 00989395059318

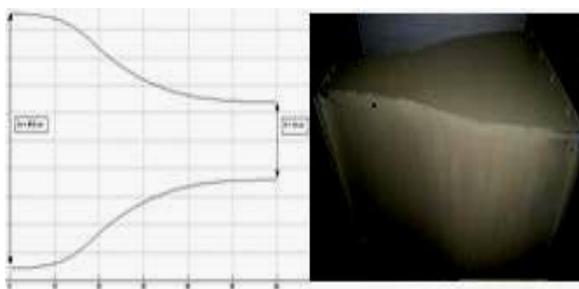
caused by the lack of stability in the air passing through the blades, because the air must move from the pre-predicted design angles and if the order of these angles to be disturbed, the vortex flows and stall in the stages of compressor occur and the air is unable to move forward [1]. If this phenomenon 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].

1- 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 *contraction nozzle* to achieve the higher velocities, has 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 container, 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 can disappear. 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. The Figure (1) shows the collection of cascade, which was made in the test section.



Figure(1):collection of cascade in test section



Figure(2): Nozzle

The nozzle used in this study as observed in the figure (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].

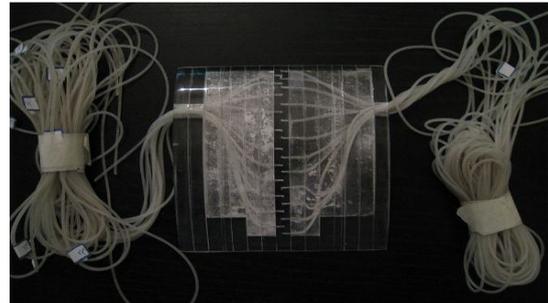
The boundary layer is created by passing the air flow over the test cross-sectional area on them, and this reduces the effective cross-section area and according to the continuity equation by decreasing the area of cross-section, air velocity and static pressure also increased. So to prevent from this 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 the figure (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.

1-1- 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 hot-wire probe used in this study schematic has been represented in figure (4).



Figure(3): Pressure tapping's configuration

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 ($\sigma.\alpha / \rho.c$) compare 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 $5 \mu m$ and 1.5 mm.



Figure(4): Hot-wire anemometer

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].

1-2- static calibration

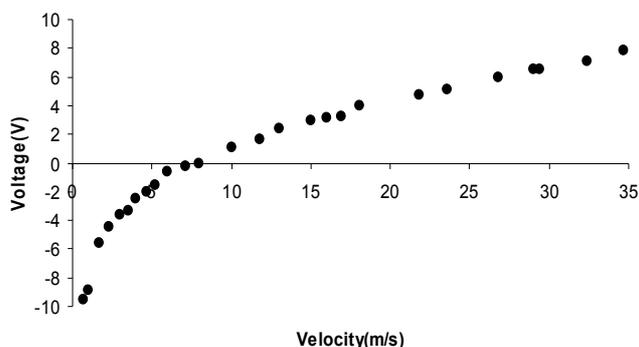
The software of the hot-wire anemometer can show the output of a hot-wire anemometer as oscillogram, so the frequency response of the hot-wire anemometer can be observed and regulated compared to a square wave through a computer system. Since the anemometer has a very high accuracy and the smallest change 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].



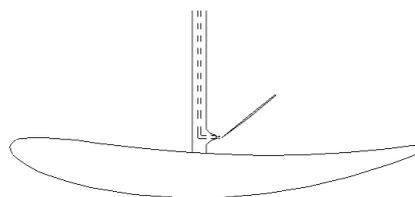
Figure(5): The wind tunnel used for calibration

In this study, the 27 points between the velocity of 1 m/s to 35 m/s were calibrated and in the figure (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 the Figure (6) the calibration curve of hot-wire anemometer schematic have been represented.



Figure(6): Calibration curve



Figure(7): the mechanism of determining the exact position of the hot-wire sensor compared to the surface

1-3- 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 the figure (7) is 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 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.

1-4- Probe traversing mechanism

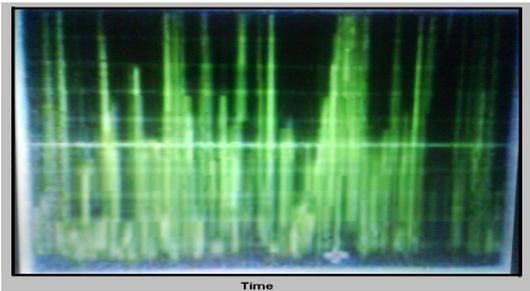
When measuring the velocity profile near the wall requires high accuracy in determining the location of the hot-wire sensor compared to the airfoil surface. For the probe can be placed in an appropriate location and velocity distribution or flow rate changes can be measured compared to the location, 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 have been represented in Figure (8).



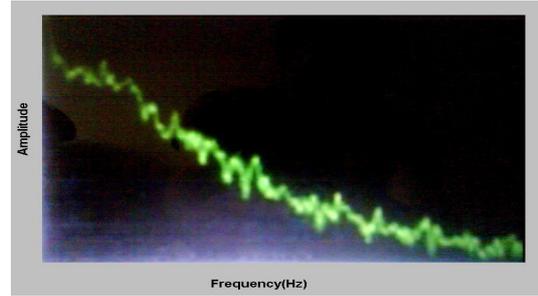
Figure(8):Probe traversing mechanism

1-5- 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 the figure (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, it's usually *better* to investigate it in the frequency domain according the figure (10) and then specify the amount of fluid flow velocity turbulence energy in a certain frequency certain and determine the quality of fluid flow. As specified in the figures (9) and (10) the flow had completely turbulence and the *dominant frequency* is about one 1 KHz .



Figure(9): The fluid flow turbulence in the time domain



Figure(10): The fluid flow turbulence in the frequency domain

2- 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 $k-\epsilon$ 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:

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

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

where $\frac{U^* k}{\epsilon}$ is now variable. (A_0 , A_s , and U^* are functions of velocity gradients).

New transport equation for dissipation rate, ϵ :

$$\rho \frac{D\epsilon}{Dt} = \underbrace{\frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right]}_{\text{Convection}} + \underbrace{\rho c_1 S \epsilon}_{\text{Generation}} - \underbrace{\rho c_2 \frac{\epsilon^2}{k + \sqrt{v} \epsilon}}_{\text{Destruction}} + \underbrace{c_{1\epsilon} \frac{\epsilon}{k} c_{3\epsilon} G_b}_{\text{Buoyancy}}$$

Convection Diffusion Generation Destruction Buoyancy

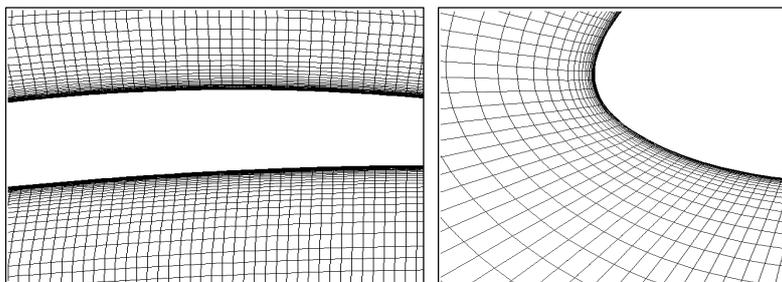
Typically values for the model constants $C1\epsilon$ and $C2\epsilon$ of 1.44 and 1.92 are used [10]. In this study, the numerical analysis using a house code based on the finite volume method has been done.

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 of the compound network mesh ,in the regions near the wall that the boundary layer behavior is important, of organized networks (suitable for areas near the wall) and in other resolved regions of the Triangle network has been used. 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

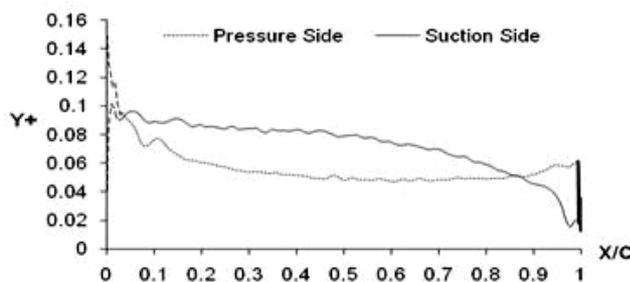
in one to one form, also in the areas where there is the severe flow gradients, the distribution of network is denser and grid lines distortion is low[11].

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 figures (11).



Figure(11): Grid domain near the blade surfaces schematic

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 figure (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.



Figure(12): the diagram of Y^+

3- 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:

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

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 CONCLUSIONS

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 the figures (13) and (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.

By investigating the diagrams (15) to (18), it is observed that reducing the solidity causes to increase the boundary layer on the pressure surface. The boundary layer thickness is low on this surface at the attack edge; However, this amount becomes significant by approaching to the trailing edge. On the suction surface according to the surface curvature near the edge of attack, the results of experiment with the numerical solution have the error over 10%.

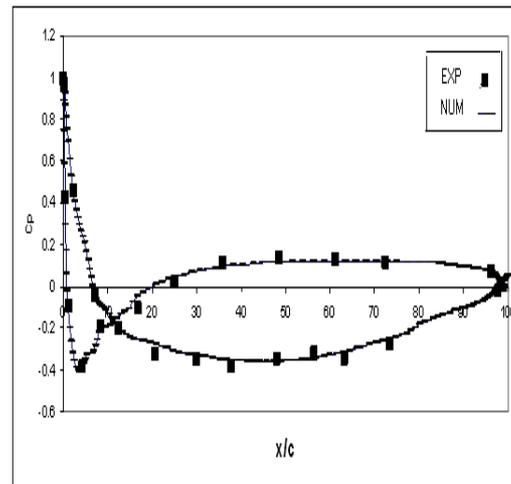


Figure 13: Variation of pressure coefficient at solidity=0.8

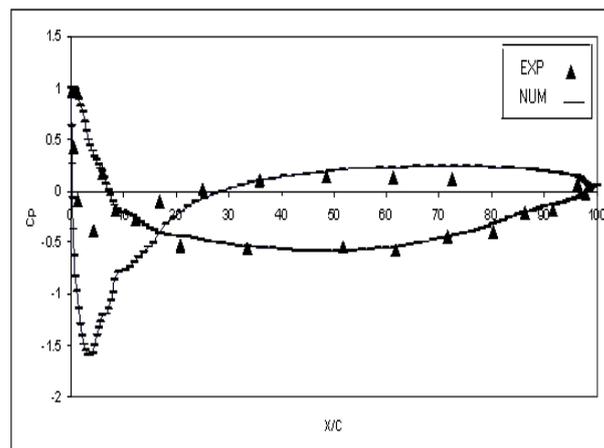
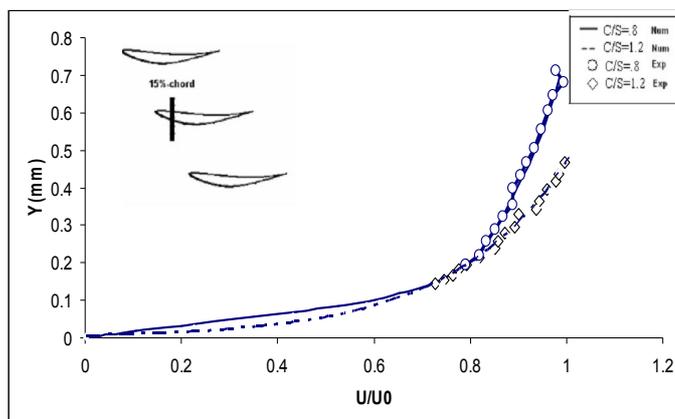


Figure 14: Variation of pressure coefficient at solidity=1.2

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 the diagrams (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.

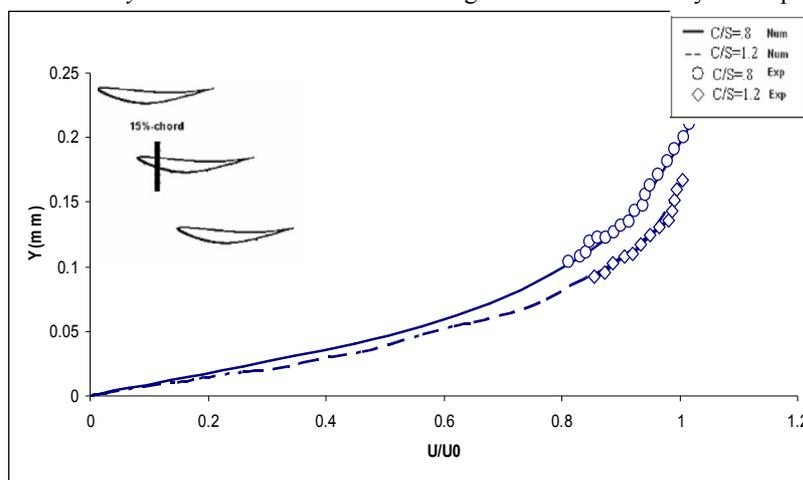
Due to the problems with hot-wire sensor near the surface, comparing the answers obtained from the numerical solution in this region by laboratory measurement is impossible and the results obtained from the incidence angel of flow in the following paragraphs will be addressed.

The Pressure distribution obtained from experimental data and numerical simulation is shown in figure 19 to be compared. This figure shows the pressure distribution in various incidence angles on both sides of blades. The comparison of results shows that the maximum error is about 5%. Such an error indicates the high accuracy of numerical solution and data-acquisition system.



Figure(15):the experimental and numerical comparison of solidity effect in the velocity distribution on the pressure surface at 15 % of chord.

An increase in the pressure coefficient is apparent with the increase in incidence angle on the pressure side of blade. With approaching to the trailing edge, differences between pressure and suction pressure coefficients become lower and lower. This is continued so that they have equal values at the trailing edge point. Conversely, on the suction side of blade, the pressure coefficient decreases when the incidence angle increases. Furthermore, the pressure coefficient on both sides decreases with an increase in the velocity. It should be noticed that the increase of incidence angle would shift the *stagnation point* from suction side to the pressure side. As the kinematics and dynamics ratios of the used reference are observed and fixed strictly, the test conditions in 2-dimensional case are similar to the 3-dimensional case, and it could be cited only on the changes of parameters on profiles. The evaluation will be continued by comparison of the *velocity distribution* in the near-wall regions both numerically and experimentally.



Figure(16):the experimental and numerical comparison of solidity effect in the velocity distribution on the suction surface at 15 % of chord.

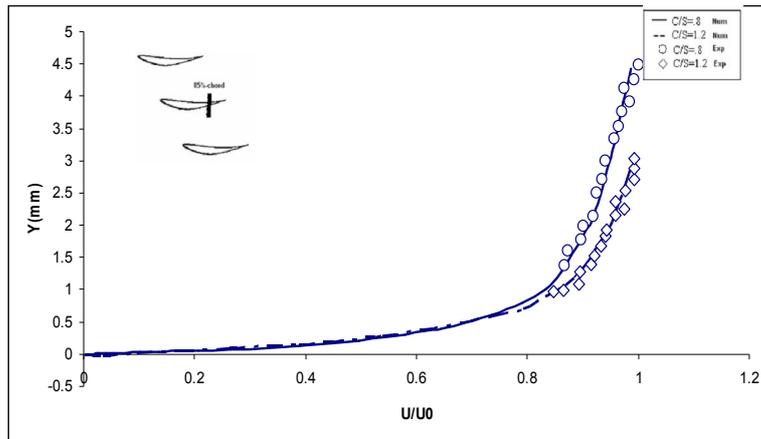
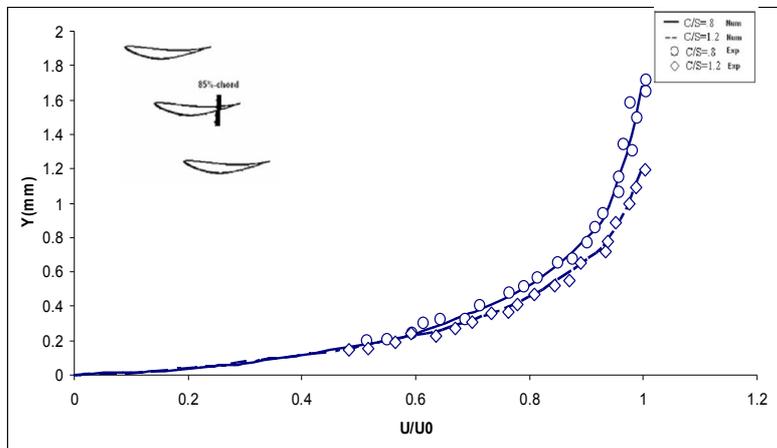


Figure (17):the experimental and numerical comparison of solidity effect in the velocity distribution on the pressure surface at 85 % of chord.



Figure(18): the experimental and numerical comparison of solidity effect in the velocity distribution on the suction surface at 85 % of chord.

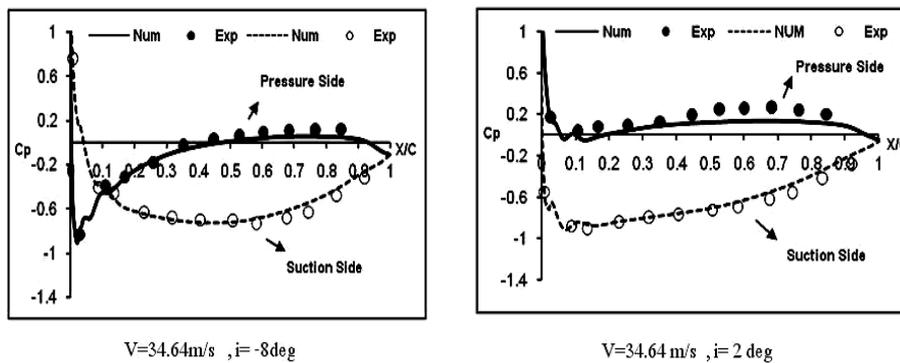


Figure 19 – the comparison between experimental and numerical results of pressure coefficient in various *incidence angles*

The comparison between solutions obtained from the *numerical simulation* and *experimental measurements* is not possible due to the problems existed with the hot-wire sensor near the surface. By

neglecting the values obtained from the surface to 0.1 mm (height), all other measurements are in accordance with our numerical results.

As can be seen in the figure 20, reducing the *incidence angle* causes the boundary layer to be thickened on the *pressure surface* side. Boundary layer thickness is as comparatively low on the leading edge of this side but when approaching the trailing edge, this value becomes more significant.

There is a *measurement error* about 10% between the experimental and numerical results of suction side. This is because of the surface curvature around the trailing edge. However, the value of 10% error is limited only to this area. In other sections of the suction side, the error was reduced to its acceptable value (<5%) again. Finally, it can be realized that with an increase in the *incidence angle*, the thickness of boundary layer is increased too. Furthermore, there is an increase of the boundary-layer thickness when the flow proceeds to the trailing edge.

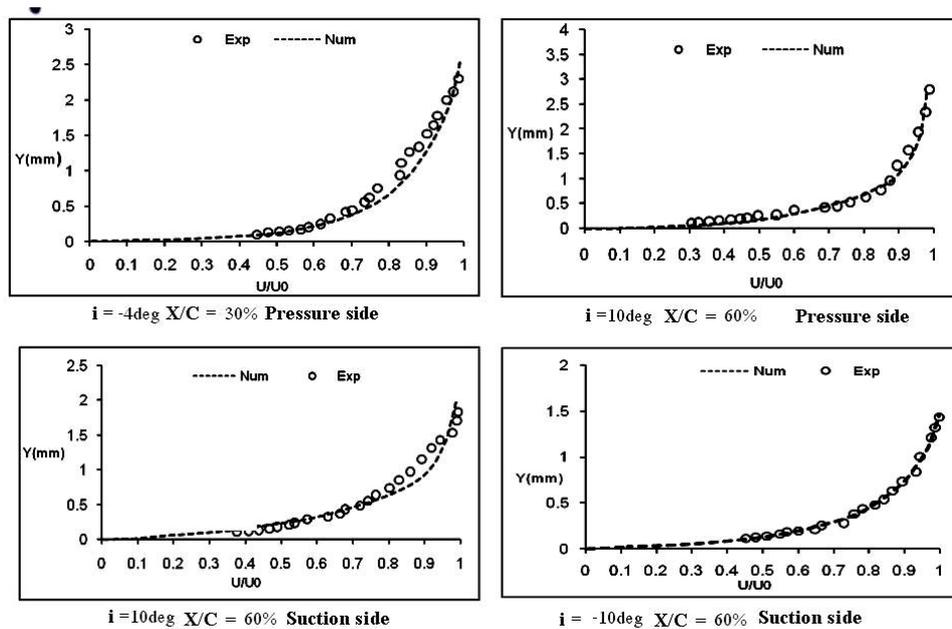


Figure 20 - Comparison between experimental and numerical results of velocity distribution in various velocities and incidence angles

By comparing the results obtained by comparison of the distance in more than 0.1 mm from measuring by the hot-wire anemometer, a good accordance with the numerical solution is observed.

Acknowledgements :

I am extremely grateful to Mr. Hamed Almasi (Translator , Email : Tarjomeh4u@gmail.com , Tel : 00989388681183), for his great support in translation and submission process of my article.

REFERENCES

- [1] Aungier Ronald H. “ Axial Flow Compressors ”, the American society of Mechanical Engineers, New York,2003
- [2] Shigeo Hayashibara, “Cascade flow Simulation and Measurement for the Study of Axial Compressor

Loss Mechanism”, Wichita State University, 2003.

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

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

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

[6] Collis, D.C. and Williams, M.J., “ Two –dimensional convection from heated wire at low Reynold numbers”, J.Fluid Mech .,6.357-384.1959.

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

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

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

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

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

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