# Experimental and Numerical Investigations of Flow Incidence Effects on Surface Pressure Distributions and Velocity Profile of Axial Compressor Blades

Reza Eftekhari
Department of Mechanical and Aerospace Engineering , Science and Research Branch,
Islamic Azad University, Tehran , Iran
Email: aero2010.iust@gmail.com ,09395059318

**Abstract:** Cascade data are essential in design process of any turbo machine blades. These data consist of variations of losses, pressure rise, surface pressure distributions and outlet flow angle in terms of Reynolds number, inlet Mach number and flow incidence. Compressor designers use these set of data not only for selection of profile geometry to suit their predicted on design performance, but also to meet off design conditions. Blade incidence may change due to many factors. These factors mainly consist of rotational speed of the compressor axis and inlet flow conditions. Design point in majority of turbo machines does not necessarily correspond to zero incidences. As a result, cascade data at various incidences would be vital during design performance.

In the present research work, intensive investigations were performed on axial compressor blades from experimental and numerical point of views. In this paper, results are confined to variations of surface pressure distributions versus flow incidence.

In parallel to the wind tunnel tests flow characteristics were studied using computational fluid dynamics (CFD) technique. Reynolds averaged Navier-Stokes equations were solved using realizable K-ε turbulence modelling.

[Reza Eftekhari. **Experimental and Numerical Investigations of Flow Incidence Effects on Surface Pressure Distributions and Velocity Profile of Axial Compressor Blades.** Life Sci J 2013;10(4s):221-228]. (ISSN: 1097-8135). http://www.lifesciencesite.com. 33

**Keywords:** Compressor cascade flows, Surface pressure distribution, Incidence Effects, Turbulence modelling.

## 1. Introduction

The performance of NGTE 1065c40 compressor blade sections in cascade has been investigated systematically in a low-speed cascade tunnel. Cascade data are essential in design process of any turbo machine blades. These data consist of variations of losses, pressure rise, surface pressure distributions and outlet flow angle in terms of Reynolds number, inlet Mach number and flow incidence [1, 2]. Blade incidence may change due to many factors. These factors mainly consist of rotational speed of the compressor axis and inlet flow conditions. Design point in majority of turbo machines does not necessarily correspond to zero incidences. As a result, cascade data at various incidences would be vital during design performance [3, 4].

The design of an axial-flow compressor of high performance involves three-dimensional high-speed flow of compressible viscous gases through successive rows of closely spaced blades. No adequate theoretical solution for this complete problem has yet appeared nor considering the complexity of the problem does it seem likely that complete relationships will be established for some time. Various aspects of the problem have been treated theoretically, and the results

of those studies are quite useful in design calculations.
Some of the information required can be obtained only by experiment in single-stage and multistage compressors. Much of the information, however, can be obtained more easily by isolating the effects of each parameter for detailed measurement. The effects of inlet angle, blade shape, angle of attack and solidity on the turning angle and drag produced can be studied by tests of compressor blades in two-dimensional cascade tunnels. Cascade tests can provide many basic data concerning the performance of compressors under widely varying conditions of operation with relative ease, rapidity, and low cost. A number of successful high-speed axial-flow compressors have been designed using low-speed cascade data directly. A more refined procedure, however, would use cascade data, not as the final answer, but as a broad base from which to work out the three-dimensional relations. The main function of a compressor cascade is to raise the static pressure of the fluid across it. This should be achieved with minimum stagnation pressure loss or drag. Thus the compressor cascade behaves like an adiabatic diffuser. The static pressure rise for isentropic flow is given by this equation:

$$\Delta p_S = p_{2s} - p_1 = \frac{1}{2}\rho\left(c_1^2 - c_{2s}^2\right) \qquad (1)$$

$$\Delta p_S = \frac{1}{2}\rho c_1^2\left(1 - \frac{\cos^2\alpha_1}{\cos^2\alpha_2}\right) \qquad (2)$$

$$\Delta p_S = \frac{1}{2}\rho c_1^2\left(1 - \frac{c_{2s}^2}{c_1^2}\right) \qquad (3)$$

The pressure rise through the cascade can be expressed as a dimension less quantity known as the pressure or pressure recovery coefficient [5, 6]. Thus the ideal pressure recovery coefficient for the cascade is given by:

$$C_{ps} = \frac{\Delta p_S}{\frac{1}{2}\rho c_1^2} = 1 - \frac{\cos^2\alpha_1}{\cos^2\alpha_2} \qquad (4)$$

**2. Experimental setup**

At the beginning of each experiment, it is essential to attain reasonably uniform conditions at entry to the cascade. This is obtained by means of adjusting both top and bottom suction rates and also the included angle of the bottom diffuser. Given low free-stream turbulence intensity, sufficiently low values of Reynolds number increase the laminar separation over most of the blade suction surface. This separation may occur at low incidence which results high losses [6]. Horlock, Shaw, Pallard and Lewkowicz have shown that there is a minimum value of Reynolds number to have no effect on the performance this is a conclusion which was reached also by Rhoden [7].

**2.1 The Wind Tunnel**

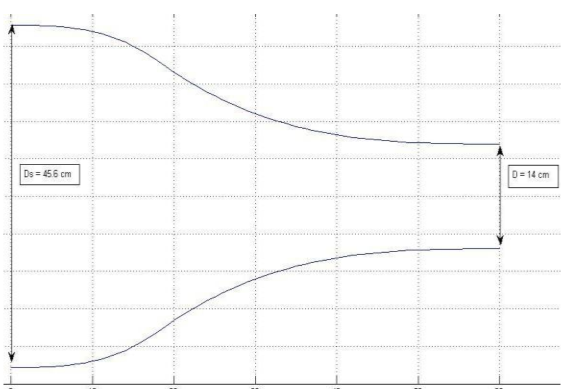
In this paper, the wind tunnel which is used for testing our cascades is blowing type which creates uniform flow in Test Section. The used wind tunnel is subsonic (TE44) with velocity of 12m/s at 45 cm X

45 cm test section. This test section speed and also the shape of test section are not sufficient for the designed experimental scheduling, so a contraction nozzle and suitable test section designed and added to the end of the test section.

**2.2 The Convergent Nozzle**

This part of wind tunnel plays a vital and essential role in quality of airflow in test section and it should meet two basic needs. First, it should provide the needed uniformity in outlet since the stream immediately enters into test section after convergent nozzle, thus a uniform flow without variation needs for outlet; and the second is the dispersion of flow which should not be occur on the walls.

The shape of this nozzle should be selected in such a way that it could create a continuous increase in velocity from a fixed section before nozzle toward test section in order to prevent from severe destruction of boundary layer and its detachment. It is always appropriate that for prevention of detachment and in order to increase an opportunity for flow propagation, to use a longer nozzle length, But production cost and boundary layer thickness restrict this parameter. The best solution is reducing curvature at both ends of nozzle, particularly in the nozzle outlet as well as using attenuation section with a length about half of inlet diameter to improve flow in test section. Reynolds number and test section dimensions determine convergence proportion in wind tunnel nozzles. The greater convergence proportion is the better method, but it also increase production cost and some issues like increase in vibration, acoustic pollution and flow separation at both ends of nozzle. Similarly, decrease in proportion of inlet to the outlet Area may cause reduction in flow homogeneity at outlet part and increase the risk of separation at both ends of nozzle. Contraction ratio of 3 is selected for the tested nozzle. The nozzle which used in this study has outlet dimensions of 14.5 cm × 45cm×60cm with the minimum air velocity of 34m/s and without creating any fluctuation on outlet. Designed Nozzles profile and actual nozzle have been represented in Figures 1 and 2.




profile

**Figure2**: Actual nozzleFigure 1: **N**ozzle

**2.3 The Test section**

The test section which used in this study is made of Plexiglas which is highly resistant and provides a better vision ability from inside part of the test section. Because of importance of this part of wind tunnel, it is tried to keep this part of wind tunnel away from destructive factors, thus some shock absorbers which prevent from conveyance blow and/ or vibration to test section, are also used in production of test section. In this project, with respect to requirement and aerodynamic considerations, the test section is made rectangular with length-to- width ratio of 3.

In the test section, some holes on the lateral walls created to provide the opportunity to keep the upper and lower walls at the desired angles. This mechanism carried out to testing the blades in different angles. test section and cascade blades have been represented in Figures 3 .

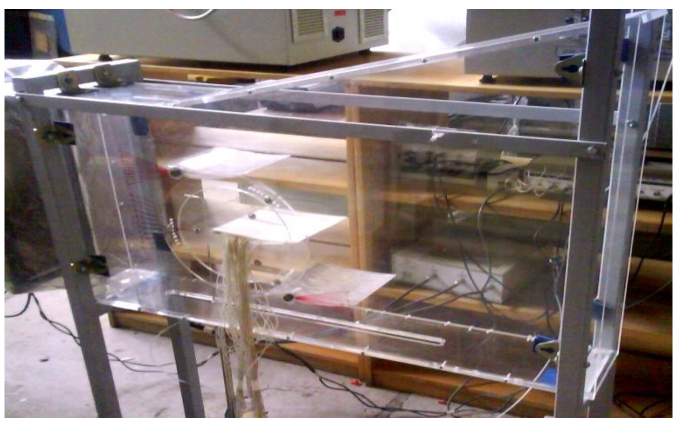


**Figure3**:Test section and Cascade blades

To measure static pressure on airfoil surface, 12 holes on suction surface and 11 holes on pressure surface were provided and as it observed in the figure 4, they are connected to some plastic pipes (tubes) by some needles. Those pipes are raised from airfoil inside (which some of them are hollow) and test section wall, and connected to an electronic converter. The accuracy rate of this converter is 0.1( H2Omm).

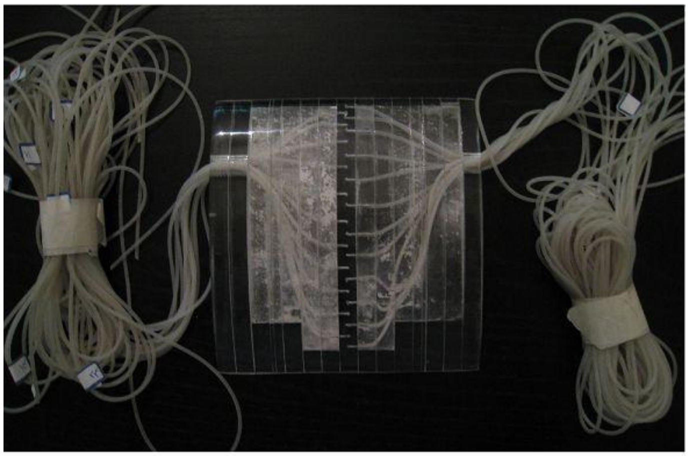


**Figure4**: Pressure tapping's configuration

3. **Numerical Solution Method**

Most turbo machinery flows encountered in practice are turbulent and three-dimensional in nature. In addition, turbulence is dissipative and diffusive. As a result, because of the complex nature of the turbulence, especially in compressors, simple turbulence models, which are based on simple shear layers, fail to capture all turbulence effects. Turbulence can not be modelled by any of the approaches, which utilize the zero, one and simple two-equation turbulence models. The realizable k-ε model has different equations from k-ε model for simulation of eddy viscosity and turbulent energy dissipation. This kind of turbulence modelling is relatively simple to implement; moreover, provides superior performance for flows involving boundary layer flows with large adverse pressure gradients, separation and circulation. Since this model accounts for the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a more rigorous manner than one-equation and other two-equation models, it has greater potential to give accurate predictions for complex flows. The realizable k-ε model was designed specifically for turbo machinery applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients [6,7].

As normally used for cascade flows, the H-O-H grid system near the blade surfaces is adopted and unstructured grids are used for the flow field which is far away blades. As shown in figure 5. using these grids system allows a fine mesh round the blade (i.e., in the blade passage) and fits the blade shape very well. Near the blade surface, the mesh is set even finer. All these mesh characteristics lead to improved resolution of flow information from the computation, thus are advantageous to better predict boundary layer development.

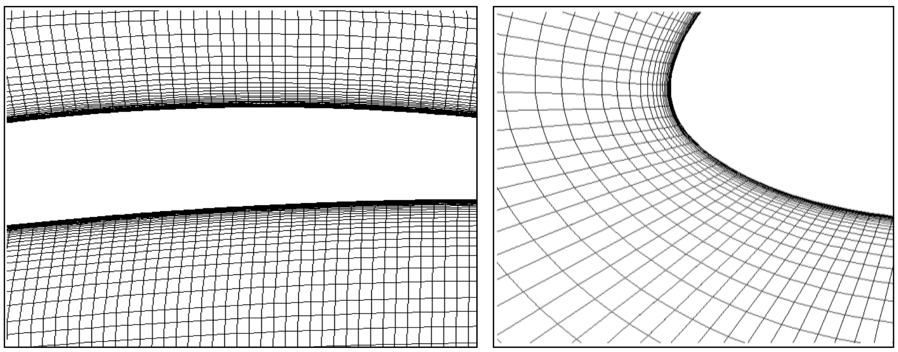


**Figure5**: Grid domain near the blade surfaces schematic

## RESULTS AND CONCLUSIONS

Pressure distribution obtained from experimental data, and numerical simulation is shown in figure 6 to compare. This figure shows pressure distribution in various incidence angles on both sides of blades. Comparing the results shows that the maximum error is about 5%. Such an error indicates high accuracy of numerical solution and the data-acquisition system.

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

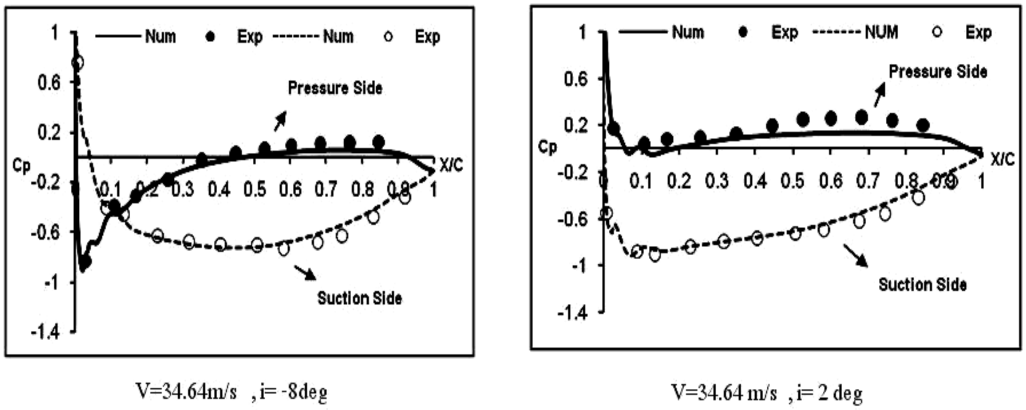



**Figure 6**: Comparison between experimental and numerical result of the pressure coefficient in various incidence angles

The comparison between solutions obtained from numerical simulation and experimental measurements is not possible due to problems with hot-wire sensor near the surface. Neglecting the values obtained from the surface to 0.1 mm height, all other measurements correspond to numerical results fairly.

As it is seen in figure 7, reducing the incidence angle causes the boundary layer to be thickened on the pressure side. Boundary layer thickness is comparatively low on the leading edge of this side but when approaching trailing edge this value becomes more significant.

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

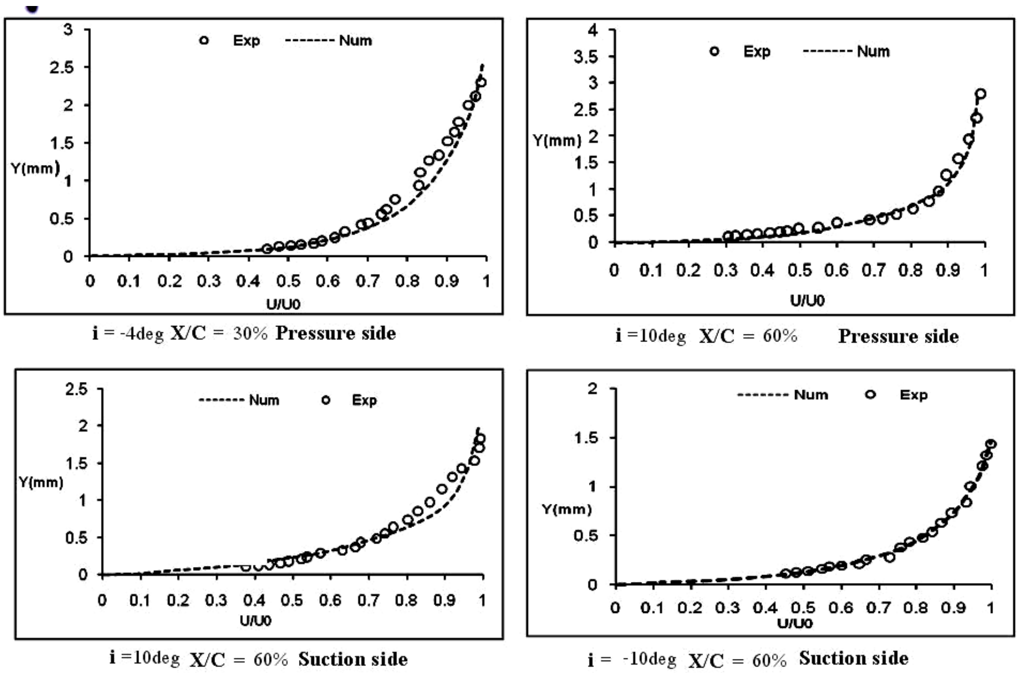



**Figure 7:** Comparison between experimental and numerical results of velocity distribution in various velocities and incidence angles.


**References**

[1] Erwin, John R., and Yacobi, Laura A.: Method of Estimating the Incompressible Flow pressure Distribution of Compressor Blade Sections at Design Angle of Attack. NACARML53F17, 1953.

[2] Aerodynamic Design of Axial-Flow Compressors", revised, NASA SP-36, 1965.

[3] Howell, A. R.: Design of Axial Compressors. Lectures on the Development of the British Gas Turbine Jet Unit Published in War Emergency Issue No. 12 of the Institution of Mechanical Engineers. A.S.M.E. Reprint, Jan. 1947, pp. 452-462.

[4] Lakshminarayana B, An assessment of computational fluid dynamic techniques in the analysis and design of turbomachinery, ASME Journal of Fluids Engineering, Vol.113, 1992. 5. h.

[5] Umit Kus and Jacques Chauvin, Performance Prediction of Turbomachinery Cascade as Affected by Upstream Turbulence, journal de physique III, October 1995.

[6] S M Yahya, Turbine Compressor and Fans, Second Edition,2003.

[7] Rae Jr., W.H., Pope, A., 1984, Low-Speed Wind Tunnel Testing, SE,John Wiley & Sons, ISBN 0-471-87402-7.