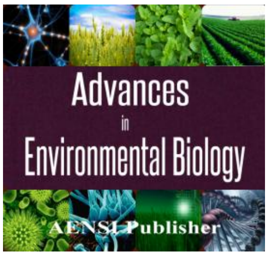
Advances in Environmental Biology, 8(11) Special 2014, Pages: 871-878

AENSI Journals

Advances in Environmental Biology

ISSN-1995-0756 EISSN-1998-1066

Journal home page: http://www.aensiweb.com/AEB/



Experimental Investigation of Flow Turbulence Effects on Outlet Velocity of Compressor Cascade Blades

[1]Abdolamir Bak Khoshnevis and [2]Seyed Mojtaba Hashemi

[1]Associate Prof. Mechanics Dept. Faculty of Engineering Hakim Sabzevari University, Sabzevar, Iran.
[2]Student of M.Sc. degree Program in Mechanics Faculty of Engineering Hakim Sabzevari University, Sabzevar, Iran.

ARTICLE INFO

Article history:
Received 15 June 2014
Received in revised form
8 July 2014
Accepted 4 September 2014
Available online 20 September 2014

Keywords:
Flow Turbulence Intensity Hot-Wire Anemometer, Turbulence Promoters, Compressor linear Cascade

ABSTRACT

In this paper (because of importance of fluid output velocity from each compressor layers) studies the effects of inflow turbulence intensity on the airfoil wake of compressor blades. First, the convex surfaces effect of blades on the velocity of behind the blade (in the state that inflow velocity is without turbulence) was examined .Then the experiment repeated by putting netted sheet on the upper of the flow and the effect of inflow turbulence intensity (in two states of inflow turbulence ٪3 and ٪5) was examined on the behind the blade velocity and was compared with the state of inflow without turbulence .It was observed that with increasing inflow turbulence intensity the velocity between the blades decreases and by following that, momentum of the flow decreases to overcome the pressure gradient between the layers of compressor, which this matter itself causes to create stall phenomenon.

To Cite This Article: Abdolamir Bak Khoshnevis and Seyed Mojtaba Hashemi, Experimental Investigation of Flow Turbulence Effects on Outlet Velocity of Compressor Cascade Blades. ***Adv. Environ. Biol., 8(11),*** *871-878, 2014*

INTRODUCTION

There are several factors involved in the stall or Surge phenomena in compressors, one of them is the velocity fall and consequently, reduction of flow momentum in overcoming the existing pressure gradients between compressor layers. Measurements between the two blades of turbo machines by hot wire Anemometer show that the real profile of the velocity between the two blades differs from the profile obtained by one-dimension theory. That is, velocity in concave surface of blade is lower than velocity in the convex surface of blade. In low flow rate, the boundary layer on concave surface is affected by the gradient of more intensive reverse pressure and is separated from concave surface of blades. In this way, the two high velocity and dead regions are created in the flow between the two blades and it is for this reason that this phenomenon is called jet and weak. In compressors with radial blades, the separation point of boundary layer is close to the entry edge of blades; while in backward blades, separation takes place in a point farther than the entrance edge and consequently, the dead region takes only a small surface; thus, although this phenomena is not very important in pumps, in compressors, the curved surfaces on velocity play an effective role in the compressor function.

2- History of the research:

Many researches have already been conducted on the effects of inlet flow turbulences on the objects sequences and studying the flows around airfoils (in particularly, around compressor blades) have become a focal point.

Akbari *et al* studies the effect of electromagnetic field on controlling flow separation on airfoils. By studying the numerical values of Lorentz Force emerging from electromagnetic field, they concluded that imposing Lorentz force delayed or deleted the separation flow on airfoils. They also observed that after controlling the airfoils NACA 0015, the lift coefficient increased while drag coefficient did not change, this in turn leads to increase in stall angle. [2].

Lasse and Niles studied the effects of inlet turbulences by simulating large eddy on airfoil NACA0015 wake. They studied the effects of inlet turbulences on parameters such as drag coefficient and lift coefficient. Their results showed that the inlet turbulences bring separation in flow close to stall state. [3]

Foroozesh *et al* studied the effects of inlet turbulences on the flow parameters in the back of an airfoil NACA0012 in zero attack angles in 38700 Reynolds number. They studied parameters such as average velocity,

Corresponding Author: Abdolamir Bak Khoshnevis, associate Prof. Mechanics Dept. Faculty of Engineering Hakim Sabzevari University, Sabzevar, Iran.

turbulence intensity, frequency response and Strouhal numbers in three different inlet turbulence intensities. Their tests showed that increase in turbulence intensity lowered Strouhal number. In addition, by increase in turbulence intensity, the intervals of eddy formation decreased [4].

Khoshnevis *et al* studied the velocity in the back of one layer of compressor blade cascades, model KH-4356 (in Reynolds 45500). They measured the average velocity, turbulence intensity, Strouhal number…parameters. The result of their studies showed that the best contact angle for this model of blade is between 20 to 30 degree angles [5].

Swalwell *et al* studied the impact of small turbulences on the performance of one cross section of airfoil NACA 2441 which is used for hydraulic turbine blade and their studies showed that even small turbulences could change the resultant force and significant changes took place in stall specifications as the inlet turbulences increased. [6].

Swalwell *et al* installed a Barometer on an airfoil NACA0021 in Reynolds number 3.5×10^5 in different attack angles and studied their drag and lift coefficients The airfoil was exposed to current with low turbulence (0.6%, free current) and turbulent flows (created through placing a network in different distances upstream model) in 4% and 7% intensity. Their studies showed that with respect to lift curves, by increase in turbulences in 35 degree impact angle, the lift coefficient shows decreasing process. [7]

Tutar *et al* studied the effects of inlet flow turbulence on circular cylinder sequence numerically, by simulating large eddies. They studied the effects of inlet turbulences on parameters such as drag coefficient, lift coefficient and the separation angle. Their results showed that by increase in inlet turbulences from 0.6% to 6%, the drag coefficient decreases 18.6%.

3- Research motivation:

With respect to the turbulence nature of flow that enters compressors, jet engines undergo excessive increase of inlet flow turbulences, especially in takeoff, landing and maneuvering that lead to stall; thus, studying the impacts of inlet flow turbulences on outlet velocity of compressors blade cascade layers has been considered by engineers for long time. For this purpose, different tests are performed on jet engines prior to installing them in aircrafts to study the accuracy of engine compressor performance in turbulence inlet flow conditions. For the reasons mentioned above, this research studies the effects of inlet turbulences on the outlet velocity from one layer of compressor cascades.

4-Validity:

To study the accuracy of wind tunnel performance, first, a cubic model (b/h=1) was tested in Reynolds number 8600 and draw the diagram of the time average of flow dominance velocity category ($\bar{U}$). As it is observed, the results are in agreement with the results of Saha *et al* and Shadaram *et al*. [9,10].

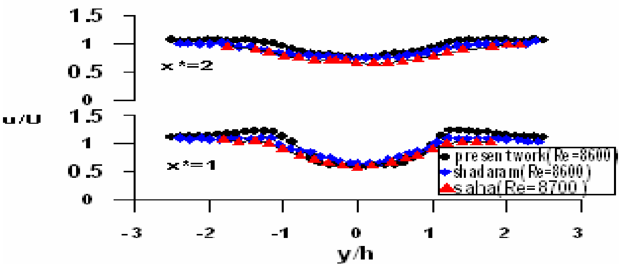


Fig. 1:

5-Test conditions and tools:

The wind tunnel which is used in this test is manufactured by Farasanjesh Saba and is open circuit blower. To perform the test, a chamber in 150cm length, 40cm width and 15cm height was made by using plexy glass and the wind tunnel was adjustable in range of 0-30 m/s. In addition, it should be mentioned that maximum nominal turbulences of free current for this device is 0.1 percent.

To measure the flow parameters hot wire Anemometer with fixed temperature (manufactured by Farasanjesh Saba Company). The probe which was used in this test is one dimensional with a sensor of 1.25mm length and 5micrometre diameter. It should be mentioned that the tests were performed in 17 centigrade degree temperature.

To move probe in different points, a transmitter with 0.01mm displacement precision and three degree freedom was used. This mechanism moved parallel to the blades installation lines so the data could be registered by hot wire ammeter. The mentioned mechanism was installed on a separate frame from wind tunnel stands to prevent transmission of possible vibrations of body of the tunnel to probe and ensure higher precision data collection. The compressor cascade had three blades of KH-4356 type. It should be noted that the test was performed in Reynolds 45500.

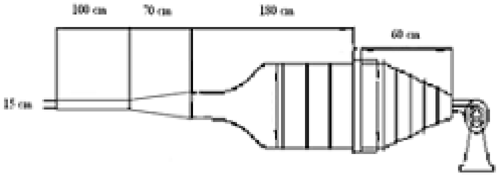


Fig. 2: Schematic figure of wind tunnel which is used in this research).

Fig. 3: figure of wind tunnel which is used in this research).

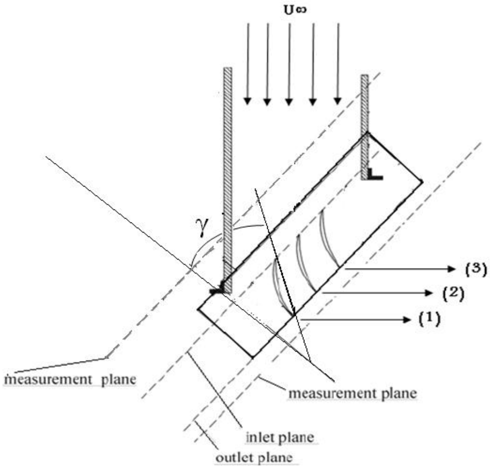


Fig. 4: Schematic figure of blade arrangement used in this research (from up)).

6-Turbuleance parameters for introducing the physics of flow:

Turbulence parameters are important in studying the physics of turbulent flow. For this reason, parameters such as turbulence intensity, fluctuation velocity, average velocity….have been introduced and their calculation in turbulent flows are explained:

The velocity of inlet flow: This is the velocity in entry point of test chamber and is obtained by using following formula:

$$U_{ref} = \sqrt{\frac{2}{\rho}(P_{total} - P_{static})} \qquad (1)$$

In turbulent flows, due to random fluctuations of velocity and pressure, it is very difficult to solve Navier Stocks equations even by using numerical analysis. For this reason, for an easier solution of Navier Stocks equations, velocity is expressed by using a series of average time velocity and one fluctuation part. For example, the momentum velocity of fluid in three directions could be expressed as follows:

$$\begin{aligned} u &= \bar{u} + \acute{u} \\ v &= \bar{v} + \acute{v} \\ w &= \bar{w} + \acute{w} \end{aligned} \qquad (2)$$

In these relations, the time average of parameters that are shown as $\bar{x}$ is in quantity and the parts that are shown as $\acute{x}$ mean the fluctuation parts of that quantity in the givne time. In this way, the averge time speed could be defined as follows:

(3) $$\bar{u} = \frac{1}{N}\sum_{1}^{N} u(n)$$

Or

(4) $$\bar{u} = u_{mean} = \frac{1}{T}\int_{0}^{T} u\,dt$$

Since velocity fluctuations are repeated in their frequency period, if the average of fluctuation part of velocity is extracted:

(5) $$\bar{\acute{u}} = \frac{1}{T}\int_0^T (u - \bar{u})\,dt = \acute{u} - \acute{u} = 0$$

According to the above formula it could be seen that $\overline{u'}$ =0; therefor,e to study mixed tubulances, a quantity is deifned as follwos as the averge squares of the fluctuation part:

(6) $$\overline{\acute{u}^2} = \frac{1}{N-1}\sum_{1}^{n} (u(n) - \bar{u})^2$$

Or

(7) $$\overline{\acute{u}^2} = \frac{1}{T}\int_0^T (u - \bar{u})$$

Since the quantity dimension is defined in $(m/s)^2$, to make the quantities co-dimensional with velocity, the square root is taken and it could be said that average square root of velocity fluctuations specifies the intensity of fluids flow turbulences, which is shown by u_{rms}.

(8) $$u_{rms} = \sqrt{\overline{\acute{u}^2}} = (\frac{1}{T}\int_0^T (u - \bar{u})^2\,dt)^{0.5}$$

The dimensionless value of by u_{rms} is shown in percentage and reveal the intensity of fluids flow turbulences.

(9) $$Tu = \frac{u_{rms}}{\bar{u}}$$

7- Information obtained from test:

The present research studies the effects of convex surfaces on the velocity of flow passing the surfaces in the two states of turbulent and non-turbulent states. The data is collected in x/d=0.25 (x, distance of probe position from airfoil escape edge and d is the length of blade chord) in three stages. In first stage, the blades are in non-turbulent flows, in second stage, they are in 3 percent turbulence flow and in third stage they are in 5 percent turbulent flow. The velocity is studied in back of cascade once in 35 degree angle, followed by 25 degree angles and the results are analyzed accordingly.

35 Degree Impact Angle:

As figure 5 shows, the velocity under the impact of convex surface of blades increases with acceleration and with high acceleration as it goes beyond the reference velocity (approximately 16 percent) and as it approaches the concave surfaces, the velocity decreases with slower slope.

Figure 6 shows that by increase in turbulence of inlet flow to 3 percent, the effect of convex surface on increasing the speed decreases. It is also observed that the velocity between the two blades decreases; nevertheless, it is still around 4 percent faster than reference velocity.

According to figure 7, by increase in turbulence in inlet flow to 5 percent, it could be seen that the effect of convex surface on inlet flow continues its decrease as close to convex surface, the velocity is almost equal to reference velocity.

As figure 8 (diagram of comparing the three stages mentioned above), the flow between the two blades in the inlet flow state will be less than one percent higher than reference velocity and by increase in the turbulence, the velocity between the two blades decreases; however, it is still higher than reference velocity and when turbulence reaches 5 percent, the velocity between the two cascade blades becomes equal to the reference velocity.

25 Degree impact angle:

As the impact angle decreases to 25 degree, the effect of convex surface to the state of 35 degree attack angle will be less and the increase in velocity caused by convex surface will be around ten percent more than reference velocity. (Figure 9)

According to figure 10, by increase in turbulence to 3 percent, the velocity between the blades decreases and reaches to almost the reference velocity. As figure 11 shows, when turbulences in inlet flow reaches 5 percent the velocity of the blades decreases (about 5 percent less than reference velocity).

By comparing the diagrams of inlet flow states and turbulences less than one percent to the inlet flow with 5 percent turbulence (with 25 degree impact angle), it is observed that the velocity between blades changes from ten percent more than reference velocity to five percent less than reference velocity. (Figure 12).

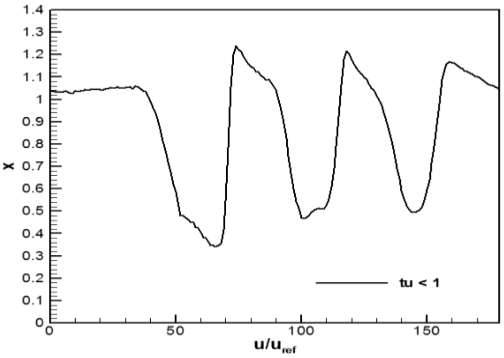


Fig. 5: $^{u}/u_{ref}$ diagram ($\frac{x}{d} = 0/5$, Tu < 1% , I =35).

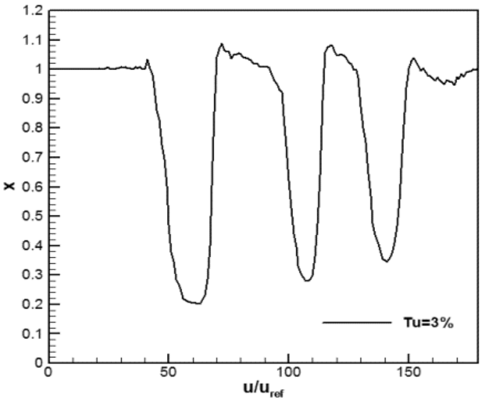


Fig. 6: $^{u}/u_{ref}$ diagram ($\frac{x}{d} = 0/5$, Tu = 3% , I =35).

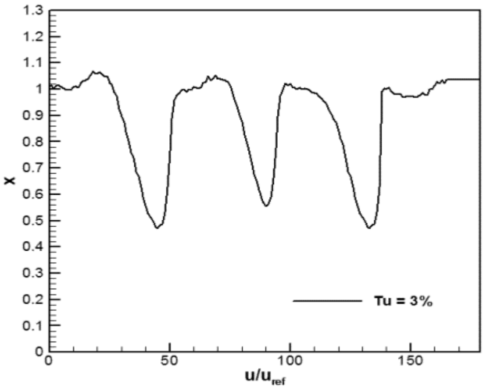


Fig. 7: $^{u}/u_{ref}$ diagram ($\frac{x}{d} = 0/5$, Tu = 5% , I =35).

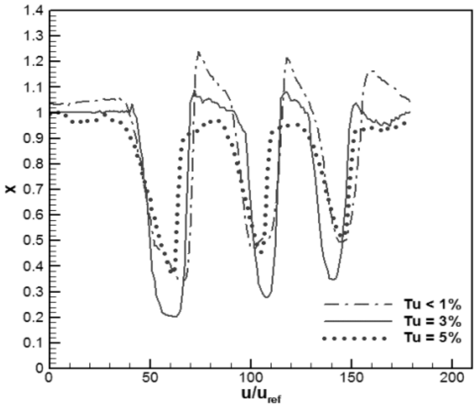


Fig. 8: $^{u}/_{u_{ref}}$ diagram ($\frac{x}{d} = 0/5$, Tu = 3% , Tu = 5% ,Tu < 1% , I =35).

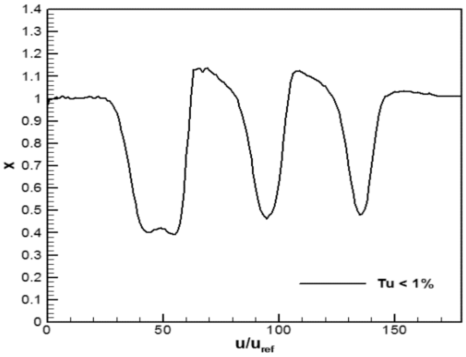


Fig. 9: $^{u}/_{u_{ref}}$ diagram ($\frac{x}{d} = 0/5$, Tu < 1% , I =25).

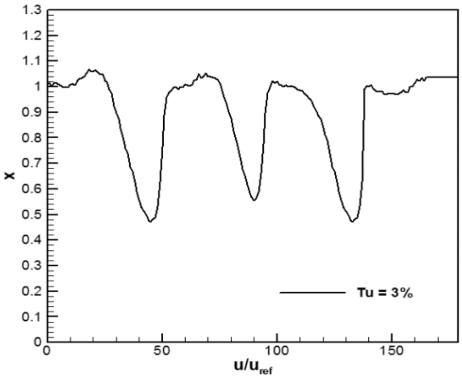


Fig. 10: $^{u}/_{u_{ref}}$ diagram ($\frac{x}{d} = 0/5$, Tu = 3% , I =25).

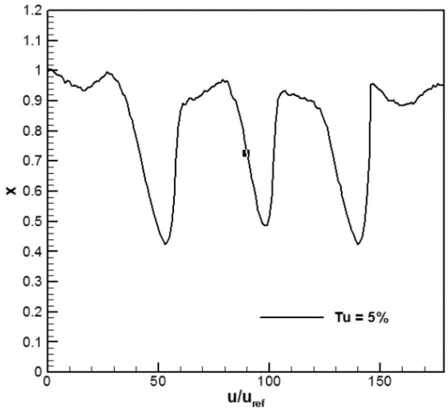


Fig. 11 : $^{u}/_{u_{ref}}$ diagram ($\frac{x}{d} = 0/5$, Tu = 5% , I =25).

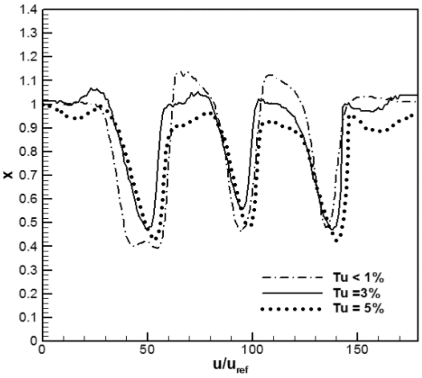


Fig.12 : u/u_{ref} diagram ($\frac{x}{d} = 0/5$, Tu = 3% , Tu = 5% ,Tu < 1% , I =25).

8- Results;

1- According to the diagrams, the velocity on convex surface increases and the velocity decreases under the impact of concave surfaces.

2- By decrease in the impact angle of the fluid, the convex surface shows less influence in increasing the velocity of the fluid.

3- By increase in turbulence of inlet flow up to 3 percent, the effect of convex surface of blades decreases; however, the velocity is still higher than reference velocity under the effect of convex surface.

4- By increase in turbulence in inlet flow to 5 percent, the effects of concave surface continues its decreasing process as far as the velocity of the two cascade blades becomes less than reference velocity.

5- The process of reduction in velocity by increase in turbulences causes flow momentum decrease to overcome the existing pressure gradients between compressor layers that in turn, causes stall phenomena.

List of symbols:

Re	Reynolds number
Tu%	Percent of turbulence
Ū	m/s Velocity , Time average
U	m/s Horizontal flow velocity category
Ú	m/s Turbulence category of horizontal velocity
V	m/s Vertical category of flow velocity
V’	m/s Turbulence category of vertical velocity
W	m/s Category of perpendicular on flow velocity plane
W’	m/s turbulence perpendicular on velocity plane
U_{ref}	m/s velocity of inlet flow
U_{rms}	$(m/s)^2$ average square root of velocity fluctuations
X	Distance of blade escape edge
d	Length of blade chord

REFERENCES

[1] Sanieinejad, M., 2009. Fundamentals of Turbulent Flows and Turbulence Modeling”, Daneshnegar Publisher, 978-964-2927-35-337, Tehran, (InPersian).

[2] Akbari, S.A., A. Sedaghat, A.R. Azimian, 2007. Numerical Study of the Effect of electromagnetic on Flow Separation Control on Airfoil “The 7th Conference of Iranian Aerospace Society (InPersian).

[3] Lasse, G. and N.S. Niles, 2009. Large Eddy Simulation of an Airfoil in Turbulent Inflow”, The 4th PhD Seminar on Wind Energy in Europe: Book of Abstracts, No. 4, The European Academy of Wind Energy, Germany.

[4] Foroozesh, F., A.B. Khoshnevis and M. Vahidi, 2013. Experimental Investigation of Flow Turbulence Effects on the Flow Wake Parameters around a NACA0012 Airfoil“, Fluid Mechanics & Aerodynamics Journal, 2(2): 83-93, (In Persian).

[5] Khoshnevis, A.B., M. Vahidi, 2013. Experimental Investigation of Characteristics of Wake Related to Linear Compressor Cascade with Turbulence Promoters“, Fluid Mechanics & Aerodynamics Journal, 2(2): 83-93, (In Persian).

[6] Swalwell, S.K., J. Sheridan, W.H. Melbourne, 2004. “The Effect of Turbulence Intensity on Performance of NACA4421 Airfoil Section”, The 42nd AIAA Aerospace Sci. Meeting and Exhibit.

[7] Swalwell, S.K., J. Sheridan and W.H. Melbourne, 2001. The Effect of Turbulence Intensity on Stall of the NACA0021 Airfoil", The 14th Australasian Fluid Mechanics Conference, Adelaide Univ., Adelaide, Australia.
[8] Tutar, M., I. Celik and I. Yavuz, 2006. Modeling of Effect of Inflow Turbulence on Large Eddy Simulation of Bluff Body Flows", Mathematical and Computational Application., 11(3): 225-234.
[9] saha, A.K., K. Muralidhar and G. Biswas, 2000. Experimental Study of Flow Past a Square Cylinder at High Reynolds Numbers", Experiments in Fluids, 29(4): 553-563.
[10] Shad Aram, A., M. Azimifard and N. Rostami, 2007. Experimental Study of Flow Characteristics in Near Wake of Square Cylinders", J. Mech. And Aero., 3(4): 13-23.