EXPERIMENTAL AND NUMERICAL INVESTIGATION FOR THE AERODYNAMICS OF A LINEAR COMPRESSOR CASCADE

Saeed A. El-Shahat1 Hesham M. El-Batsh2 Ali M.A. Attia3
Benha University, Egypt Benha University, Egypt Benha University, Egypt

ABSTRACT

In this study, a linear compressor cascade was installed in an open-loop wind tunnel and experimental measurements were acquired at Reynolds number of $2.87 \times 10^5$ based on blade chord and inlet flow velocity. To gain a better understanding for the flow features in axial compressors the static pressure distribution was measured on the endwall employing static pressure tabs connected to digital micro-manometers. The three-dimensional flow field was measured using a calibrated five-hole pressure probe connected with a data acquisition system. The flow features have been measured including 3-D flow velocity, static pressure, and total pressure through the flow passage between the blades. To investigate the loss mechanism through the cascade the total pressure loss coefficient was calculated from the measurements. The flow was further investigated using computational fluid dynamics (CFD) calculations. Flow governing equations representing continuity, momentum and an appropriate turbulence models were solved. The results obtained here demonstrated a good agreement between the experimental and computational results. Separation lines and flow vortices were predicted and measured through this study which enhanced understanding the loss mechanism in compressor cascades. The flow characteristics showed the regions of corner separation where the velocity has been reduced and also the total pressure and the flow was reversed at the corner region which may cause flow blockage.

1. Introduction and Literature Review

Air compression has significant effect on the weight and efficiency of aero-engines or gas turbines. The compressor makes up about 50% of the gas turbine engine, so that improvements in this component significantly influence the whole engine. Within the last decade successive improvements in compressor aerodynamics were accomplished. High-capacity CFD tools allowed to simulate and analyze complex flow phenomena within research and the design process [1]. Since the pressure rise in the compressor is given by the thermodynamic cycle, the number of stages should decrease for a required overall pressure rise. This demand leads to aerodynamically highly loaded blading with high diffusion factors.

The flow field inside modern high pressure compressors with high blade loading and small aspect ratio blades is highly three-dimensional and unsteady by nature where various flow phenomena occur and interact, especially at the endwall regions and the blade suction side are particularly affected. Flow separation is an important parameter that deteriorate compressor performance. It decreases efficiency, lowers static pressure rise capability and contributes to instability in compressors. With the demand of high pressure ratio per stage, prevention of flow separation becomes challenging objective for both the

1 Demonstrator, Mechanical Engineering Department, Benha Faculty of Engineering, Benha University, saeedanwar2012@yahoo.com
2 Prof. Mechanical Engineering Department, Benha Faculty of Engineering, Benha University, helbatsh@bhit.bu.edu.eg
3 Lecturer, Mechanical Engineering Department, Benha Faculty of Engineering, Benha University, ali.attia@bhit.bu.edu.eg
researchers and designers. Endwall boundary layer separation, horseshoe vortex, corner vortex, tip vortex, endwall crossflow, and passage vortex are secondary flow components in the cascade. These secondary flows extract energy from the fluid and increase the flow instability. Therefore, controlling the secondary flows is of crucial importance in order to improve the aerodynamic performance of compressors.

Three-dimensional separations in blade passages, especially for the endwall corner regions, are common in compressors, which may cause significant aerodynamic blockage and losses production, and can even affect the compressor stability [2],[3],[4]. There are two basic factors which affect the formation of three-dimensional separations in compressor blade passages: (i) the adverse pressure gradient in the streamwise direction and (ii) the secondary flow effects throughout blade passages [5],[6]. Number of studies have discussed the importance of three-dimensional flows in axial compressors where 3D separations were clearly evident on the suction surface / endwall corner of both stator as well as rotor blades[7],[8],[9],[10],[11],[12]. Although the deleterious consequences of 3D separations have been identified by these and other authors, effective management and control of these effects has been very difficult to achieve. This is perhaps primarily because the nature and characteristics of 3D separations in turbo machines are not clearly understood, nor are the mechanisms and factors that influence their growth and ultimate size [13].

Correspondingly, there are various studies concerning the three-dimensional separated flow in an axial compressor stator have been reported. The flow in a stator with high loading and low aspect ratio was investigated and a stalled region with high aerodynamic loss in the corner of the suction surface and the hub wall was reported [14]. Also, the flow and loss mechanisms in a single-stage low speed axial flow compressor stator Large separated areas were observed at both near the hub and near the casing [15]. A detailed flow measurements in an isolated subsonic compressor stator at various blade loadings were performed and Corner separations both near the hub and near the casing were observed for some cases [16]. The steady RANS analysis was performed by [17], [18] and their numerical results were incorporated with the experimental results of [16] in order to advance the understanding of the basic mechanism of compressor hub corner stall. Regions of three-dimensional separations have been identified as an inherent flow feature of the corner formed by the suction surface and endwall of axial compressors is known as corner separations and it has been identified as the main source of total pressure loss and compressor efficiency reduction [19]. These separations contribute greatly to passage blockage, which effectively places a limit on the loading and static pressure rise achievable by the compressor. In addition, the subsequent mixing of the flow in the separated region with the main passage flow may lead to a considerable total pressure loss and a consequent reduction in compressor efficiency [20]. The work described in the present paper has two main objectives. The first objective is to improve our understanding of the development of flow features in a linear compressor cascade. The second objective is to test the ability of the current CFD codes with Reynolds -Averaged Navier-Stokes (RANS) turbulence modelling to accurately compute these flow features. Therefore, experimental and numerical investigations have carried out in order to examine the three-dimensional flow field in a linear compressor cascade.

2. Experimental Facility and Procedures

2.1 Wind Tunnel and Cascade

In the current work, an experimental setup has been built in the aerodynamic laboratory of the Mechanical Engineering Department, Benha Faculty of Engineering, Benha University, Egypt. An open-loop low-speed wind tunnel has been constructed to provide uniform air flow of velocity range from 25 to 50 m/sec. Then a test section containing a linear cascade of axial compressor equipped with instrumented blades for measuring purposes is coupled to the constructed wind tunnel as shown in Figure 1. The constructed wind tunnel is composed of centrifugal blower, settling chamber, contraction cone, and the test section. The flow delivered from a 10 HP centrifugal blower which is controlled by a flow obstruction cone at the suction side. The settling chamber consists of perforated plate, honeycomb, and five screens of diameters 5, 3, 2, 1.2, and 0.8 mm respectively that have been carefully designed to provide uniform flow with low turbulence intensity, as the centrifugal blower exit flow neither uniform nor steady. The design of settling chamber has been constructed based on previous publications [21], [22]. A smooth contraction section is constructed to provide exit dimensions appropriate to the inlet of the test section which has rectangular cross-section with 408×185mm, and contraction ratio of 4.8. To achieve smooth air motion and prevent boundary layer separation in the contraction section, a curvature surfaces of 5th order polynomial have constructed based on [23].
The test section contains a linear cascade of six NACA 65-009 blades. For purpose of flow measurements at the passage between two blades different holes have been carefully manufactured. The notation used in describing this subsonic compressor cascade is shown in Figure 2 and the corresponding values of different parameters are summarized in Table 1. The flow is measured at inlet velocity of \( U_\infty = 30.0 \pm 0.3 \) m/s with Reynolds number \( Re_c = 2.87 \times 10^5 \) calculated based on blade chord.

![Figure 1: Test rig configuration](image)

![Figure 2: Notations for the axial compressor cascade](image)

### 2.2 Measuring Instrumentations

Endwall pressure measurements were performed using a set of differential pressure digital micro-manometers (Model HD 755) of range \( \pm 3447 \) Pascal with resolution of 1 Pascal. For that purpose a grid of 66 pressure taps of 1.7 mm diameter copper tubing embedded into the endwall surface with a port of 0.5 mm opening to the flow on the endwall surface have been opened as shown in Figure 3. All
manometers have been tested together against repeatability and abilities to give approximately the same reading for the specific pressure value. There was high degree of repeatability and ability to read the same value (the digital micro-manometer accuracy estimated to be 0.3% of reading). Since the measured flow is turbulent, the average value of each static pressure value was considered during one minute.

Figure 3: Locations of endwall static tabs  

In order to measure the flow downstream of the cascade, L-Shaped 5-hole pressure probe coupled with data acquisition system designed for that purpose has been used. This measurement DAQ system (ATX sensor module) manufactured by Aeroprobe international company in USA can measure up to sampling rate of 10000 sample/s so, it is very sensitive and accurate. It can be connected to 3 or 5 or 7 or 12 hole-probe because there are 19 holes on its inlet as shown in Figure 5. In this system the analog pressure data from 5-hole probe is converted into digital data via DAQ system and the analysis is performed via Aeroacquire software to obtain the velocity vectors, total and static pressures. The flow field throughout the passage have been measured on 16 planes of points perpendicular to the axial chord at 9 positions through the pitchwise direction making a grid of 144 points through X-Y plane as shown in Fig.4. At each point 11 discrete readings have been collected through Z-direction, with small interval near the endwall surface because of being in the boundary layer region and large interval as we go through spanwise in Z-direction. For inflow mean velocity of 30 m/sec, a sampling rate of 10 sample/sec to collect 100 readings for each measuring value. A set of averaged values exported by the DAQ like the static and total pressures, 3-D velocity components as well as the velocity magnitude. Each value was time averaged to record the corresponding local mean value and the calculated local turbulence intensity (standard deviation).

Figure 5: DAQ (ATX sensor module and its inlet tubes) with five-hole probe
3. Computational Procedures

In the current work, the commercial CFD package (ANSYS Fluent 14) has been used to predict the flow characteristics in 3-D domain that simulates compressor cascade. The computational work includes the grid generation and boundary conditions, the governing equations and finally the turbulence model selection and the numerical technique.

3.1 Problem Description

Three dimensions steady-state calculations were performed to obtain the complex flow structure through the linear compressor cascade. Since the flow considered here has Mach number less than 0.3 (M=0.1), the compressibility effect is negligible. The 3-D computational domain includes the entire blade height. Periodic boundaries were considered to represent the periodic flow through the blade passages. To be able to compare these results with the experimental results, the inlet and the exit of the computation domain were placed at 4.13 axial chords upstream from leading edge and 2.16 axial chords downstream from trailing edge of the blade, respectively.

3.2 Grid Generation

The multi-block structured method was used to ensure the grid quality, as well as the matching periodicity strategy. Three-dimensional grid is used in this work to distribute the cells in appropriate way. The computational domain is divided into multi-blocks structured grid. The grid for each block is generated individually with total 15 blocks in the blade passage. The grid topology is such that O-grid type is used close to the blade surface which is required to solve the boundary layer. The grid generated in a cross-section plane and then the two-dimensional grid is copied in the direction of blade height. The multi-block mesh on the endwall is shown in Figure 6. The number of cell used in the computational domain is 4983000 and it is adequate for solving the computational.

3.3 Boundary Conditions

The inlet velocity profile is obtained experimentally with the help of 5-hole coupled with the DAQ system previously described as shown in Fig.7. The incidence angle is specified for the inlet boundary (i=4), whereas the static pressure is used for the outlet boundary. From the inflow velocity measurements, a fixed value of turbulence intensity of 0.768% for the inlet boundary and the turbulent length scale is 17.82 mm based on that the turbulent length scale (L)=0.07*hydraulic diameter. Furthermore, non-slip condition is adopted for all of the solid walls (endwall and blade surfaces). Periodic conditions are imposed along the pitchwise boundaries.

3.4 Numerical Technique

The pressure-velocity coupling is handled by the SIMPLE algorithm. A diffuser flow and NASA rotor 64 transonic flow with Spalart-Allmaras turbulence model have numerically investigated, and the corner separation have been over simulated [24]. Efforts have been done to improve the capability of the S-A turbulence model to predict the corner separation of axial compressors [25], [26]. Consequently, the turbulence model Spalart-Allmaras was employed in the computational study and compared with the measured results of the experiment.

4.5 Governing Equations

Fluid flow characteristics are described by the conservation of mass (Continuity equation) and momentum (Navier-Stokes equations). For turbulent flow, Reynolds averaging procedure is commonly used. Then, the governing equations are called Reynolds-Averaged Navier-Stokes equations. For incompressible flow neglecting external forces they are given by:

\[
\frac{\partial \bar{u}_i}{\partial x_i} = 0
\]  

Equation 1

\[
\rho \frac{\partial}{\partial x_j} (u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left[ \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right] - \rho \bar{u}_i \bar{u}_j \right)
\]

Equation 2
Where the velocities $u^i$ are time-averaged and consists of mean values $\overline{u_i}$ and fluctuating values $u'_i$, and $\rho u'_i u'_j$ are the Reynolds stresses which are calculated using turbulence models. Boussinesq introduced eddy-viscosity approach by considering that Reynolds stresses are proportional to the mean velocity gradient and he used a constant which is called eddy or turbulent viscosity. Turbulence models which follow this approach are called eddy viscosity models. These models are classified according to the number of differential equations which are required to calculated turbulent viscosity $\mu_t$. 

![Figure 6: Grid distribution](image)
4. Results and Discussion

4.1 Static Pressure on Endwall:

The static pressure distribution on the endwall has been measured with the help of digital micro-manometers and 66 pressure tabs on endwall as shown in Figure 9. The static pressure coefficient is calculated from the local static pressure at the endwall by the relation

\[ C_{PS} = \frac{(P_{St} - P_{S\infty})}{(P_{t\infty} - P_{S\infty})} \tag{Equation 3} \]

where:
- \( P_{St} \) is the local measured static pressure
- \( P_{S\infty} \) and \( P_{t\infty} \) are the reference static and total pressures respectively

The received contours for endwall static pressure by numerical and experimental calculations are shown in Figure 8. The region where the flow is accelerated lies at the leading edge and has lowest static pressure coefficient. As shown in the figure, there is a good agreement between the numerical and experimental where the region that has low static pressure coefficient lies at the leading edge whereas the regions near the trailing edge have higher static pressure coefficient.
4.2 Local total pressure losses of the outlet flow:

The local total pressure loss coefficient $\omega$ was calculated from the equation

$$\omega = \frac{(P_{\infty} - P_1)}{(P_{\infty} - P_S)}$$

Where $P_1$ is the local measured total pressure

The contours of the exit total pressure loss coefficient at the cascade exit downstream of the leading edge at a distances of 0.091Ca, 0.182Ca, 0.273Ca, and 0.364Ca after the leading edge and perpendicular to the axial chord were measured and also calculated by numerical calculations. All the experimental measurements were taken at an incidence angle $i= 4^\circ$. So, at the mentioned incidence angle, the losses increase in the spanwise direction from mid-span to endwall as shown in Figures 9,10,11,12 where the shape in the left is the numerical calculations and the shape in the right is the experimental result. Generally, very good agreement were obtained between the experimental measurements and the numerical calculations. The agreement is not only obtained in the trend of the flow but also on the local values which demonstrate accurate measurements and appropriate CFD calculations. The total pressure loss coefficient increases in the boundary layer region of endwall and suction side. The passage vortex was obtained by both experimental and numerical technique.
4.3 Local total pressure losses through the cascade

Through the spanwise the local total pressure loss coefficient has been calculated from equation 4. The local total pressure loss coefficient is a sign of the effect of flow separation especially corner separation. The regions where $\omega$ is high is affected with the flow separation at these regions. Figures 13, 14, 15, 16 show the local total pressure loss coefficient at a plane 5, 7, 10, 20 mm through the spanwise and parallel to endwall. It is cleared that there is an increase in the total pressure loss coefficient at corner region. As we go through spanwise the effect of corner separation decreases and the total pressure loss coefficient decreases while at 5, 7 mm it was higher because we are in the boundary layer region and the effect of corner separation is still high affected.

a) Numerical calculations

b) Experimental measurements

Figure 11: Local Total pressure loss coefficient at Plane X2

Figure 12: Local Total pressure loss coefficient at Plane X1
Figure 13: Total Pressure Coefficient at Plane 5 mm from the endwall

Figure 14: Total Pressure Coefficient at Plane 7 mm from the endwall

Figure 15: Total pressure coefficient at plane 10 mm from the endwall
a) Numerical calculations  

b) Experimental measurements

**Figure 16: Total Pressure Coefficient at Plane 20 mm from the endwall**

Figure 17 shows contours of the static pressure coefficient the blade suction side and on the endwall. The limiting streamline are also drawn from the wall shear stress obtained from the numerical calculations. The streamlines indicate that the existence of separation lines and vortices on endwall and suction side. The flow separation and vortex generated in corner region may lead to flow blockage. The corner stall, with distinct regions of three-dimensional reversed flow, was no longer located in the endwall-suction junction of the blade passage. Instead, it was moved further into the passage and occupied a large area over the airfoil suction surface [27]. On the blade suction side near the hub, low momentum fluid from the blade pressure surface moves upstream along the corner, turns around and ends up as a (second) vortex. Consequently:

- A considerable blockage, decreasing the pressure rise capability of the compressor
- A source of high secondary losses, reducing the efficiency of the compressor and this is consistent with [28], [29].

**Figure 17: Numerical calculation of static pressure coefficient contours and stream lines**
5. Conclusions

Detailed and accurate measurements of 3D flow field through a linear compressor cascade has been performed. RANS numerical simulations were carried out using Spalart-Allmaras turbulence model. It was found through this study that very good agreement were obtained between the measurements and calculated flow field. Both techniques demonstrated total pressure loss coefficient increases as we entered in the boundary layer region of endwall and suction side and separation increases especially in the corner region because what is called corner separation. Flow separation and vortex generated especially in corner region is predicted numerically and measured experimentally which may lead to flow blockage because of separated and cross flow. As we go through spanwise the effect of corner separation decreases and the total pressure coefficient increases while at planes near endwall total pressure coefficient is lower because we are in the boundary layer region and the effect of corner separation is still high affected.

References