EXPERIMENTAL AND NUMERICAL STUDIES OF FLOW IN A LOGARITHMIC SPIRAL CURVED DIFFUSER

Mohamed S. Mohamed, Berge Djebedjian and Magdy Abou Rayan

Mechanical Power Engineering Department,
Faculty of Engineering, Mansoura University,
El-Mansoura 35516, Egypt

ABSTRACT

The incompressible turbulent flow inside a curved diffuser of rectangular cross section has been analyzed using the time-averaged Navier-Stokes equations. The numerical solutions of the elliptic partial differential equations written for a body fitted coordinate have been obtained using a turbulence model. The standard k-ε turbulence model and a modified k-ε model proposed by Myong (1997) which takes into account the extra rate of strain produced by curvature are used in this paper.

Measurements for developing turbulent flow in a logarithmic spiral curved diffuser have been made. Mean velocities, static and total pressures, have been measured using a three-hole probe.

The experimental results are compared to the numerical solution results. It is appeared from the comparison that a reasonable agreement is noticed for the modified k-ε model while less accurate results are observed for the standard k-ε model. This is in accordance with the well-known fact that the accuracy of the standard k-ε turbulence model decreases under the condition of strong adverse pressure gradient or strong streamline curvature.

Experiments show that, there is no flow reversal in any part of the diffuser. Streamwise bulk flow is seen to shift toward the concave wall side in the downstream half of the diffuser, under the influence of centrifugal force. The static pressure recovery coefficient, on both convex and concave, are 0.34 and 0.40, respectively.

INTRODUCTION

Curved diffusers are essential components in many fluids handling systems. They are used in such applications to turn and decelerate the flow (to convert the dynamic pressure into static pressure) simultaneously. Depending on the application, they have been designed in many different shapes and sizes. S-shaped diffusing ducts are used as intake ducts for aircraft engines and interconnecting ducting between components of gas turbine engines. Part bend or 90° curved diffusers are used in wind tunnels, ducting systems, plumes, draft tubes, etc. The use of such diffusers mainly depends either on the specific design of the machine or on the space limitation where compactness is desired or both. The flow characteristics in these diffusers are strongly dependent on the outlet-inlet area ratio (AR), centerline length-throat width ratio (L/W1), inlet velocity and turbulence intensity profiles, and the turning angle (Δβ). Strong secondary motions are generated due to unbalance of centrifugal force and radial pressure gradient, causing non-uniform flow distribution and increasing losses.

Fox and Kline (1962) have systematically investigated the flow regimes for curved diffuser passages. They have also presented experimental results showing the effect of the gross geometry of the curved diffusers on the performance of the diffuser and flow regimes. They developed maps of flow regimes for stalled and unstalled curved diffusers with turning angles ranging from 0° to 90° in steps of 10°, with a circular arc centerline and linear area distribution. The data of Fox and Kline (1962) are commonly used for diffuser design. Sagi and Johnston (1967) extended their work and developed a design procedure for two-dimensional curved diffusers having an area ratio between 1.5 and 2.1, L/W1 = 4-10 and Δβ = 30-90°. Parsons and Hill (1973) have shown that modification of the method of Stanitz (1953) gave better agreement between theorectical and experimental results for curved diffusers by incorporating streamline curvature effects. Friberg and Dormoy (1978) studied experimentally the flow behavior in a 90° curved diffuser (AR = 2) at different Mach numbers. They were also able to establish a semi-empirical theoretical model based on the hypothesis of an apparent coefficient of viscosity to predict the wall pressure distribution.

Detailed measurements of the streamwise and secondary velocities, wall static pressure, and turbulence intensities for
low speed turbulent flow on a 40° turn C-diffuser, \( L/W_1 = 3 \), \( AR = 1.32 \), \( AS = 1.5 \) have been made by McMillan (1982). He observed two counter-rotating vortical secondary motions between the parallel walls, which dominated the flow behavior. Sajaniak et al. (1982) reported that, for a given curved diffuser, the pressure recovery increases with an increase in the Reynolds number and decreases with an increase in the turning angle. Sullerey et al. (1983) reported slightly higher pressure recovery for an equivalent straight diffuser (\( \Delta \beta = 0° \)) in comparison to a curved diffuser of \( AR = 1.56 \), \( AS = 0.8 \) and \( \Delta \beta = 55° \). They attributed this to the enhanced growth of the boundary layer along the convex wall which reduces the effective area ratio. Rojas et al. (1983) have generated extensive data for laminar and turbulent flows in S- and C-shaped curved diffusers having an area ratio of 1.5 and rectangular cross-section. They used Laser Doppler Anemometry (LDA) to measure the three components of velocity and \( UV \) cross-correlation (for the turbulent case) along with the wall pressure distribution and the pressure recovery. Detailed flow investigation within a 90° curved diffuser (\( AR = 2.0 \), \( AS = 6.0 \)) has also been reported by Majumdar et al. (1998). They observed secondary flows, which are not very strong (maximum 6 percent of the inlet velocity). They also found complex three-dimensional flow characteristics in the downstream side of the diffuser.

Miller (1976) made studies in curved diffusers that have high area ratios (\( AR > 2.0 \)) and compiled the results in his book. After Miller, Agrawal and Singh (1991) were the first to focus on the flow in a large area ratio curved diffuser. They carried out flow visualization studies in a 90° large area ratio curved diffuser with an elliptical centerline. Detailed flow characteristics of the same diffuser have been described by Majumdar et al. (1992). The performance evaluation showed comparatively low pressure recovery and higher losses in comparison to conical diffusers of equivalent area ratio.

Taking a lead from the work of Awai et al. (1986), Majumdar et al. (1996) obtained improved performance with the 10° splitter vanes compared to the 0° and 5° cases (\( C_P = 42\% \) for 10° and 28% for 0° and 5°).

The logarithmic spiral curved diffuser represents one form of the radial diffuser cascade, which is more complex in design and analysis than the other vaned diffusers. Sakurai (1971, 1972 and 1975) considered such geometry. He concentrated on obtaining overall diffuser performance and boundary layer parameters experimentally and modeled the passage flow by means of a potential flow core and an integral boundary layer without coupling between them. Further detailed investigations on the flow field for this diffuser are needed.

The objectives of the present study are to investigate the characteristics of the flow field and to compare experimental results with the predicted numerical results using the standard k-\( \varepsilon \) model and the proposed modification of Myong (1997) to the k-\( \varepsilon \) turbulence model.

### NOMENCLATURE

- \( AR \): area ratio, \( W_2/W_1 \)
- \( AS \): aspect ratio, \( H/W_1 \)
- \( C_1, C_2 \): constants in the k-\( \varepsilon \) model
- \( C_\mu \): eddy viscosity coefficient
- \( C_P \): static pressure recovery coefficient
- \( D_h \): hydraulic diameter, \([ 2W_1H/(W_1 + H) ]\), (m)
- \( H \): depth of the diffuser, (m)
- \( k \): turbulent kinetic energy, \((m^2/s^2)\)
- \( L \): centerline length of the curved diffuser, (m)
- \( p \): turbulence energy generation rate, \((m^2/s^2)\)
- \( p_a \): static pressure, (Pa)
- \( p_{av} \): average inlet static pressure, (Pa)
- \( r \): arbitrary radius, (m)
- \( r_i \): inner radius of the diffuser cascade, (m)
- \( Re \): Reynolds number, \( \rho U_D W / \mu \)
- \( s \): distance from the diffuser inlet (along the centerline), (m)
- \( U \): mean streamwise velocity component, (m/s)
- \( U_2 \): average inlet velocity, (m/s)
- \( V \): mean transverse velocity component, (m/s)
- \( W \): width of measured section, (m)
- \( W_1 \): width of diffuser at inlet, (m)
- \( W_2 \): width of diffuser at exit, (m)
- \( y \): transverse distance measured from convex to concave wall, (m)
- \( \alpha \): vane inlet angle, (deg)
- \( \Delta \beta \): angle of turn, (deg)
- \( \varepsilon \): dissipation rate, \((m^2/s^3)\)
- \( \mu \): laminar viscosity of fluid, (Pa.s)
- \( \mu_{eff} \): effective viscosity, (Pa.s)
- \( \mu_t \): turbulent viscosity, (Pa.s)
- \( \rho \): density of fluid, (kg/m³)
- \( \sigma_k, \sigma_\varepsilon \): Turbulent Prandtl numbers in k and \( \varepsilon \) transport equations
- \( \phi \): angular coordinate, (deg)

### EXPERIMENTAL APPARATUS AND PROCEDURE

The schematic layout of the experimental apparatus is shown in Fig. 1. Air is used as the working fluid. System air is drawn into the inlet, through the contraction and the test section, expanded in the diffuser section, propelled through the fan and exhausted through the diffuser/silencer to the atmosphere. The main tunnel air is supplied to the test diffuser at a maximum air velocity of approximately 62 m/s with free stream turbulence intensity \( \sqrt{\frac{\mu^2}{U}} \) less than 0.5%. The Reynolds number based on the inlet hydraulic diameter of the diffuser is 1.0x10⁶. The control panel of the wind tunnel consists of a variable frequency controller and a remote speed control device. The air speed in the test section can be controlled from the control panel of the wind tunnel using a
The error in measuring the velocity is ±2 percent.

The test section consists of two curved side walls (convex and concave) and the other walls, top and bottom, are flat (i.e. parallel). The curved side walls have a log-spiral profile, which extends from an inner radius of 2.29 m to an outer radius of 3.21 m of the radial diffuser, giving radius ratio \((r_o/r_i)\) of 1.4 from inlet to exit. The diffuser centerline logarithmic profile is described in polar coordinates \((r, \varphi)\) by the equation:

\[
\varphi = \frac{1}{\tan \alpha_i} \ln \frac{r}{r_i} \quad \ldots \ldots (1)
\]

where:
- \(\varphi\) is the angular coordinate.
- \(\alpha_i\) is the vane inlet angle.
- \(r_i\) is the inner radius of the diffuser cascade.

The test diffuser has a constant height (the distance between parallel walls of the diffuser) of 30 cm and is made from galvanized sheet with smooth inner surfaces. The static pressure distribution is measured using pressure taps located along the centerline (mid-way between the upper and lower parallel diffuser walls). Fourteen taps are placed on both curved side walls. The tubes extending from these taps are connected to a digital micro-manometer for pressure record. The error in measurement of pressure is in the order of ±0.1 mm.

At the top wall of the test diffuser, seven slots (equals to the number of test sections) are shaped. Each slot has a width slightly larger than the diameter of the probe holder of the hot wire and extends from the convex towards concave walls. During measurements these slots which are not in use are sealed. A traversing unit is fixed in the slots and the probe holder is accommodated in it.

The diffuser was made with vane inlet angle \(\alpha_i\) equal to 30°, which was judged from the vanless diffuser tests to be the optimum angle, (Reddy and Kar, 1971). The angle of opening of spiral was 10°, which gives maximum performance, Sakurai (1975). The test diffuser cross section is rectangular, with inlet width, \(W_1\), of 21.4 cm, a constant height of 30 mm while the exit width, \(W_{exit}\), is 30 cm and the diffuser centerline length, \(L_c\), is about 197 cm. Therefore, the area ratio \(AR\) (area of exit section to area of inlet section = \(W_{exit}/W_1\)) is 1.4 and length ratio, \(L_r = L_c/W_1\), is 9.2. The diffuser geometry and nomenclatures are shown in Fig. 2.

The test diffuser has a constant height (the distance between parallel walls of the diffuser) of 30 cm and is made from galvanized sheet with smooth inner surfaces.
The three-hole probe employed in this investigation is shown in Fig. 3. The probe is calibrated in a special wind tunnel using a special mechanism to measure the angle of flow and the corresponding pressures from the right, center, and left holes. These pressures are used to calculate the coefficients of pressure, velocity, and direction needed when the probe is used in fixed direction method measurements.

![Figure 3. Three-hole probe](image)

A probe traversing unit having two axes traversing mechanism is employed to position and report the location of the probe at any point in the test section horizontal plane. The device is driven manually via two handwheels precise and high helix aluminum lead screws running in polyurethane nuts. Two precision potentiometers are secured to the drive screws to report the location of the probe in each axis. The voltage signal from each potentiometer is transmitted to the digital display in a cabinet meter. The probe can be traversed from the convex to the concave wall of the test section in steps of 2.25 mm. The error in measuring the transverse distance is ±0.1 mm.

**MATHEMATICAL AND PHYSICAL MODELS**

**Governing Equations:**

The present work is based on the numerical solutions of the two-dimensional form of the time-averaged Navier-Stokes equations. Turbulent viscosity is defined by the high Reynolds number version of the $k-\varepsilon$ model of turbulence.

For a variable $\phi$, where $\phi$ may be unity (mass conservation), $U$ and $V$ (momentum conservation), $k$ (turbulence energy), and $\varepsilon$ (Energy dissipation), the conservative form of the $\phi$-conservation equation for the steady two-dimensional incompressible flow in Cartesian system is:

$$
\frac{\partial}{\partial x} \left( \rho U \phi \right) + \frac{\partial}{\partial y} \left( \rho V \phi \right) = \frac{\partial}{\partial x} \left( \Gamma_\phi \frac{\partial \phi}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma_\phi \frac{\partial \phi}{\partial y} \right) + S_\phi
$$

$$
\text{Equation } \phi \quad \Gamma_\phi \quad S_\phi
\begin{align*}
\text{Continuity} & & 1 & 0 & 0 \\
U \text{ momentum} & & \mu_{eff} & -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial U}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial V}{\partial x} \right) \\
V \text{ momentum} & & \mu_{eff} & -\frac{\partial p}{\partial y} + \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial V}{\partial y} \right) + \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial U}{\partial y} \right) \\
Turbulence \ Energy & & \mu + \frac{\mu_t}{\sigma_k} & \rho P - \rho \varepsilon \\
Energy \ Dissipation & & \mu + \frac{\mu_t}{\sigma_\varepsilon} & C_1 \frac{\rho \varepsilon}{k} - C_2 \frac{\rho \varepsilon^2}{k} + M_p \frac{\rho \varepsilon^2}{k}
\end{align*}
$$

Table 1. Equations solved, exchange coefficients and source terms in general transport equation (2)

<table>
<thead>
<tr>
<th>(C_\mu)</th>
<th>(C_1)</th>
<th>(C_2)</th>
<th>(\sigma_k)</th>
<th>(\sigma_\varepsilon)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

Table 2. Values for the constants used in turbulence modeling, Launder and Spalding (1974)

The isotropic effective viscosity is given by:

$$
\mu_{eff} = \mu + \mu_t = \mu + C_\mu \rho \frac{k^2}{\varepsilon} \quad \text{...... (3)}
$$

The turbulence energy generation rate $P$, which occurs in both $k$ and $\varepsilon$ equations, is given by:

$$
P = \frac{\mu_t}{\rho} \left[ \left( \frac{\partial U}{\partial x} \right)^2 + 2 \left( \frac{\partial U}{\partial y} \right)^2 + 2 \left( \frac{\partial V}{\partial y} \right)^2 \right] \quad \text{...... (4)}
$$

Myong (1997) proposed the following expression to take into account the extra rates of strain resulting from the curvature, rotation, flow separation, and other effects:

$$
M_p = \frac{k^2}{\varepsilon} \left[ 2 \frac{\partial U}{\partial y} \frac{\partial V}{\partial x} + \left( \frac{\partial U}{\partial x} \right)^2 + \left( \frac{\partial V}{\partial y} \right)^2 \right] \quad \text{...... (5)}
$$
Numerical Solution Procedure

The closure provided by these equations is completed by the specification of boundary conditions over the whole perimeter of the solution domain.

For the numerical solution of Eqs. (2), a staggered grid system is used. The discretization equations are obtained via a finite difference method. The iterative solution of the momentum and continuity equations is accomplished via the SIMPLER algorithm, Patankar (1980). The treatment of the momentum equations at wall boundaries has been achieved by the wall function.

RESULTS AND DISCUSSION

The measurements are obtained at the mid-plane between the top and bottom parallel walls at various downstream stations. The stations are distributed from the inlet to exit of the tested diffuser. A three holes probe is used. The experimental results for the mean flow characteristics at different inlet Reynolds; $Re = 4.86 \times 10^5$, $6.48 \times 10^5$, $8.1 \times 10^5$ and $10^6$; are presented for the mean flow velocity components in the streamwise and transverse directions. The local wall static pressure recovery coefficient along middle height of both curved side walls is indicated. Comparison between the experimental results, numerical results of the original model and the modified model are presented.

It should be mentioned here that the flow inside the diffuser is affected by several competing mechanisms such as, potential-like flow asymmetric area expansion driven flow, pressure-driven secondary flow, and growing boundary layers. At a certain position in the diffuser, one or more of these mechanisms may dominate, complement or counter each other to give the resultant flow. In the vicinity of the curved side walls, the secondary flows are generated due to the unbalance between the centrifugal force and the pressure force.

Mean Flow Velocity

The longitudinal and transverse mean velocities ($U$, $V$) are presented in Figures 4 and 5 for inlet Reynolds number ($Re = 4.86 \times 10^5$). Fig. 4 shows the dimensionless longitudinal mean velocity profiles at six downstream measuring stations ($s/W_1 = 1.54, 3.07, 4.62, 6.13, 7.67, 9.2$). The flow at the diffuser inlet is still developing with thin boundary layer near the curved walls. As a result of higher pressure (due to centrifugal forces), thin boundary layer on curved walls lead to potential-like flow situation in which the flow is seen to accelerate near the convex side wall, in order to balance the slower flow near the concave side wall. The figure shows that the locus of maximum longitudinal mean velocity moves away from the convex wall towards the concave one as the flow moves in the downstream direction. The figure also indicates that, at all sections out side the boundary layer region, the main flow velocity decreases from convex wall in the direction of the concave wall; as required by the inviscid flow theory. It can be noted that at the two exit sections of ($s/W_1 = 7.67$ and $9.2$) the bulk flow shifts to the concave wall and the thickness of development boundary layer on the convex wall is larger than that developed on the concave wall. This is consistent with the effect of concave curvature, which acts to increase the turbulence mixing, and leads to increased velocity close to the wall. In general, it is seen from the figure that as the flow moves downstream, the longitudinal mean velocity decreases due to the increment of the cross-sectional diffuser area, the main flow velocity profiles are non-uniform due to the curvature effects.

The transverse mean velocity profiles at the same measuring stations are presented in Fig. 5. The positive sign implies movement of flow from the convex wall to the concave one and negative sign shows the reverse direction. It is obvious that all curves of $V/U_0$, in the figure, the values of
V/U_o are very small in comparison with U/U_o values. Positive transverse mean velocity profiles are observed in the first three measuring stations (s/W_1 = 1.54, 3.07 and 4.62) while it had negative values in the last three measuring stations (s/W_1 = 6.13, 7.67 and 9.2). The positive sign means movement of flow from the convex wall to the concave one while the negative sign implies the reverse direction.

**Wall Pressure Recovery**

The streamwise variation of the local static pressure recovery coefficient \( C_p = (p - p_m)/(\rho U_o^2/2) \) along the middle height of both curved walls (i.e., convex and concave) are presented in Fig. 6. The figure indicates that the wall pressure coefficient increases continuously with the downstream distance due to the continuous increase in the diffuser cross-sectional area. It is seen from the figure that the static pressure coefficient \( C_p \) along concave wall is greater than the corresponding value along the convex wall at any downstream distance due to the centrifugal force.

**Effect of Inlet Reynolds Number**

The effect of the inlet Reynolds number on the longitudinal mean velocity, transverse mean velocity and pressure recovery is investigated. It was found that, the effect of Reynolds number on these flow parameters (in the range considered) is negligible.

For example, Fig. 7 shows the comparison between the longitudinal mean velocity for four Reynolds numbers; \( Re = 4.86 \times 10^5 \), 6.48 \times 10^5, 8.10 \times 10^5 and 1.00 \times 10^6; at s/W_1 = 7.67. In general, the trend of the velocity profile is quite similar for the considered range of Reynolds number.

**Numerical Results**

A 2-D computer program fitted with the standard k-\( \varepsilon \) model and Myong modification is employed in this investigation. Comparison between the predicted \( U \) and \( V \) by the two models and the experimental data is performed for \( Re = 4.86 \times 10^5 \). The results presented in this study have been obtained using (130x30) grid system. The grid is not uniform but become more dense in the regions where the flow variables change rapidly near the walls. Successive iterations are performed until the summation of the residuals of the solved variables becomes less than 10^-4.

Fig. 8 and Fig. 9 show the experimental dimensionless \( U \) and \( V \) results compared with the predicted results by the standard k-\( \varepsilon \) model and Myong model for flow at inlet Reynolds number of \( Re = 4.86 \times 10^5 \).

Generally, Fig. 8 shows that the experimental \( U \) velocity profiles decrease from convex to concave wall while the predicted ones are almost flat in most of the test sections and there is velocity gradients near the diffuser walls. Myong modification predicted higher \( U \) velocity values than standard k-\( \varepsilon \) model. The agreement between the experimental and predicted profiles is good within the experimental accuracy.

The predicted dimensionless transverse velocity profiles predicted, by both models, are higher than the experimental one in most of the section. Good agreement is obtained by the modified model at all sections except that one at section s/W_1 = 7.67.

Fig. 10 shows the velocity vectors for the predicted flow at inlet Reynolds number \( Re = 4.86 \times 10^5 \). Fig. 11 presents the isolines of the local pressure recovery coefficient predicted by standard k-\( \varepsilon \) model for the same flow. Fig. 11 indicates that the predicted pressure recovery coefficient has higher values on the concave wall than those on the convex wall. The predicted \( C_p \) is higher than the experimental one. It has the values of 0.42 and 0.36 on the concave and convex walls at the diffuser exit, as shown in Fig. 11.
CONCLUSIONS

The turbulent flow inside a single passage with logarithmic spiral profile for the vaned radial diffuser cascade is experimentally investigated and numerically modeled. In this investigation two 2D computer programs are used with standard k-ε turbulence model and the other is a modified k-ε model proposed by Myong (1997) which takes into account the extra rate of strain produced by curvature. Based on the comparison between the experimental, numerical data (with original k-ε model) and numerical data (with the modified model) the following conclusions are offered:

1- The longitudinal mean velocity profile is non-uniform and decreases as the diffuser cross-sectional area increases in the downstream direction.

2- The transverse mean velocity profile changes sign from positive to negative as the flow moves downstream. This means that the streamwise bulk flow is seen to shift toward...
the concave wall side in the downstream half of the diffuser, under the influence of centrifugal force.

3- The wall static pressure increases continuously on both the convex and concave walls as a result of diffusion, and the static pressure coefficient $C_p$ along concave wall is greater than the corresponding value along the convex wall at any downstream distance. Both models predict slightly higher values of $C_p$.

4- The effect of the inlet Reynolds number on the flow characteristics, in the range considered, is negligible.

5- The standard $k-\varepsilon$ model captures the main feature of the present experimental results. However, discrepancies in the $V$ velocity component are observed. This is in complete agreement with the well-known fact that the accuracy of the standard $k-\varepsilon$ turbulence model decreases under the condition of strong adverse pressure gradient or strong streamline curvature.

6- Reasonable agreement is noticed for the modified $k-\varepsilon$ of Myong (1997). The modified model predicts correctly the transverse velocity component at most test sections.

ACKNOWLEDGMENT

The authors would like to express their thanks and gratitude for the technicians of Fluid Mechanics Laboratory (Mansoura University) for their continuous effort and support. The authors are grateful to Eng. Mohamed Arafat for his effort during the construction of the test diffuser.

REFERENCES


