

STUDY OF TURBULENT FLOW IN RECTANGULAR CHANNEL WITH INCLINED BAFFLES

M. Nasr

A. Abdel-Fattah

W. A. El-Askary

mostafanasr@yahoo.com

ashourabdefatah@yahoo.com

wageeh_elaskary@yahoo.com

*Mechanical Power Engineering Department, Faculty of Engineering,
Menoufiya University, Shebin El-Kom, EGYPT*

ABSTRACT

Experimental and numerical studies investigate the pressure recovery coefficient along the wall of a rectangular channel fitted by inclined-perforated and solid baffles. Two baffles of same overall size are used in the experiment. The upstream baffle is attached to the top surface, while the position, orientation, and the shape of the other baffle are varied. Different inflow Reynolds number for this study is tested and ranged between 71,000 and 122,500. Experimental results show that the pressure recovery distribution is strongly depended on the position, orientation, and geometry of the second baffle plate. The pressure drop goes down with an increase in the Reynolds number, but its value depends on the arrangement of baffles. The pressure drop is much higher for two inclined baffles (one perforated on the upper wall and one solid on the lower wall). For the sake of numerical simulation, the turbulent governing equations are solved by a control volume-based finite difference method and using the standard as well as the RNG $k-\varepsilon$ turbulence model associated with wall function to describe the turbulent structure. A modification including the effect of streamline curvature is considered in the turbulence model. The velocity and pressure terms of momentum equations are solved by SIMPLE (semi-implicit method for pressure-linked equation) method. Satisfied comparisons are achieved to verify the turbulence model used. The results show that the wall pressure recovery coefficient is strongly affected by the geometry and location of baffles in the duct. Also, the comparisons of the numerical results given by using turbulence models indicate the quality of the $k-\varepsilon$ including the streamline curvature in the turbulence model.

البحث يشمل دراسة عملية و نظرية لمعامل إسترجاع الضغط علي طول جدار لمجرى ذات مقطع مستطيل و مثبت داخله عوائق صلبة مثقبة أو مصمتة و تميل بزواوية ثابتة علي جدار المجرى. العائق الأول مثبت بالسطح العلوي للمجرى بينما يتغير شكل و مكان و ترتيب العائق الثاني. تمت الدراسة في ظروف سريران مختلفه في رقم الرينولدز (من 71000 إلى 122500) عند الدخول. أظهرت النتائج المعملية أن معامل إسترجاع الضغط يتأثر تأثير شديد بمكان و ترتيب و شكل العائق الثاني. و لقد اتضح أن الفقد في الضغط ينخفض بزيادة رقم رينولدز في حين تتأثر قيمته علي حسب ترتيب العائق الثاني. و أظهرت النتائج أيضا أن الفقد في الضغط يكون كبير في حالة استخدام عائقين الأول مثقب علي السطح العلوي و العائق الثاني مصمت علي السطح السفلي.

تمت محاكاة النتائج العملية باستخدام نماذج عديدة مختلفة يتم تطبيقها خلال نماذج (Standard $k-\varepsilon$)، (RNG)، و كذلك نموذج ($k-\varepsilon$) المحسن بالأخذ في الاعتبار تقوس خطوط الأنسياب. أظهرت مقارنة النتائج النظرية بالعملية توافق في الأداء في الحالات التي تمت دراستها باستخدام النماذج العددية، و لقد أظهرت النتائج مدى تحسن أداء النموذج ($k-\varepsilon$) المحسن بالأخذ في الاعتبار تقوس خطوط الأنسياب. و هذا التطبيق يساعد في فهم طبيعة السريان في المجارى في حالة وجود عوائق بأشكال مختلفة.

Keywords: Rectangular channel flow; Baffle; Inclined baffle; Pressure drop

1. INTRODUCTION

The use of baffled type channels is one of the commonly used passive heat transfer enhancement strategies in single-phase internal flow. This passive heat-transfer enhancement strategy has been used for various types of industrial applications such as heat exchanger, furnace design, nuclear reactor electronic cooling devices, thermal regenerators, internal cooling system of gas turbine blades and turbomachines.

Repeated ribs or turbulators have been used as the promoters of turbulence to enhance the heat transfer to the flow of coolants in a channel. These roughness elements break the laminar sub-layer of the flow. The heat transfer is enhanced as well as the pressure drop, an important parameter in the analysis of the overall performance of such flows. Investigations have been conducted to predict the effect of the number of ribbed walls on heat transfer and friction characteristics. Rib turbulators are widely used in gas

turbine cooling. These turbulators are installed on the two opposite sides of the cooling channel. In some cases, rib turbulators are casted on one side or four sides of the cooling channel. In spite of turbine blade internal cooling has been widely considered in the past studies, there are other applications such as heat exchangers, electronic equipment and nuclear reactors and may use the results of enhanced internal cooling in channels with one, two or four rib-roughened walls

Like jet impingement, ribs, and other heat transfer enhancement techniques, insertion of baffles in heat transfer devices is popular to promote better mixing of the coolant and increase cooling performance [1]. Applications of the inclined baffles may be in the large land based modern gas turbines, in which it has become a growing trend to increase the temperature of the combustion product to increase the specific thrust and to reduce the specific fuel consumption. Such a high temperature is far above the allowable temperature of super alloys and thermal barrier coatings (TBC) used in gas turbine blades. In gas turbine, usually air is used in the interior side of the blade to maintain the blade at the proper working temperature. Also in a number of other engineering and industrial applications such as, air-cooled solar collectors, laser curtain seals, labyrinth shaft seals, compact heat exchangers, and microelectronics, air is preferred as a coolant for its lightweight. However, due to the very low thermal conductivity internal cooling with gases is less effective than cooling with liquids. There are several techniques available to enhance the heat transfer coefficient of gases in internal cooling. The most commonly used technique for internal cooling enhancement is the placement of periodic ribs. Ribs are generally mounted on the heat transfer surface, which disturbs the boundary layer growth and enhances the heat transfer between the surface and the fluid. These ribs are small and do not disturb the core flow and therefore, the turbulence enhancement and boundary layer break down are mostly localized near the heat transfer surface. Ribs provide excellent cooling enhancement with a comparatively low penalty in the pressure drop increase for moderate Reynolds numbers, see Dutta and Dutta [1].

Impingement cooling as a second method uses high velocity jets to cool the surface of interest. However, often a large region needs to be cooled and multiple jets are required. Multiple jets get deflected in the presence of cross flow developed by upstream spent-jets.

The third one is a common internal cooling enhancement technique in which the placement of internal flow swirls, tape twistors, or baffles is used. The swirl insert and tape twister techniques create a significant amount of bulk flow disturbance, and the

pressure drop penalties are much higher compared to the gain in heat transfer coefficient. Inclined solid baffles may be considered as a combination of ribs and channel inserts. The baffles are big enough to disturb the core flow, but like ribs, they are mounted on or near the heat transfer surface and can be periodic in nature. Perforations in inclined baffles create a multiple jet impingement condition and thus create a situation where all three major heat transfer coefficient enhancement techniques work in unison. Baffles also create bulk flow disturbance, but unlike tapes or swirls, baffles are discrete objects. Therefore, the flow disturbance created by baffles may be localized, but more intense. Usually the baffle plate is attached to the thermally active surface to augment heat transfer by providing additional fin-like surface area for heat transfer and better mixing [1].

In the past, experimental results were published with baffle plates perpendicular to the flow direction. Among important studies, Berner et al. [2] obtained mean velocity and turbulence results in flow over baffles; and Habib et al. [3] investigated heat transfer and flow over perpendicular baffles of different heights. But these works mainly emphasized on baffles that were perpendicular to the flow direction and for that reason penalties (friction factor) were higher than the improvements (heat transfer augmentation). However, it is possible to obtain enhanced heat transfer with comparably less frictional head loss by inserting inclined baffles in the flow path. These baffles are big enough to disturb the core flow, but like ribs, they are mounted on or near the heat transfer surface. Moreover, inclined perforated baffles contain circular holes, which facilitate jet impingement toward the heat transfer surface. Hence by utilizing inclined perforated baffles, the three major heat transfer augmentation techniques can be combined for effective cooling.

Dutta and Dutta [1] reported the enhancement of heat transfer with inclined solid and perforated baffles. In that study, the effects of baffle size, position, and orientation were studied for internal cooling heat transfer augmentation. Later on, a number of research groups have utilized the perforated baffle concept for internal cooling augmentation both experimentally (Ko and Anand [4]; Ziolkowska et al. [5]) and numerically (Yang and Hwang [6]; Yilmaz [7]; Tsay et al. [8]). In those studies, different aspect ratio channels and different porosity baffles were used; see Dutta and Hossain [9].

Several publications have addressed the state of the art review of turbine blade cooling and the analysis of flow with two opposite ribbed walls. The effects of flow Reynolds number and rib geometry (rib height, rib spacing, rib angle-of-attack, and rib configuration) on heat transfer and pressure drop in

the fully developed region of uniformly heated square channels have been investigated [10, 11]. Further study of the combined effects of rib geometry and channel aspect ratio on the local values of heat transfer and pressure drop was also reported [12–14]. The results show that the angled ribs provide a better heat transfer performance than transverse ribs, and the lower aspect ratio (AR) channels perform better than the higher aspect ratio channels.

The heat/mass transfer analogy has been applied to study detailed heat transfer distribution in two-pass and three-pass rectangular channels [15–19]. The square channel with parallel and crossed arrays of cross cut and beveled discrete ribs has been studied [20, 21]. Also, the effect of the equally segmented ribs arranged in both aligned and staggered arrays on two opposite walls of a square channel has been reported [22].

A paper presenting results of an experimental study of heat transfer and friction in rectangular ducts with baffles (solid or perforated) attached to one of the broad walls was published by Karwa et al. [23]. They used boundary conditions correspond closely to those found in solar air heaters. The friction factor for the solid baffles is found to be 9.6–11.1 times of the smooth duct. Performance comparison with the smooth duct at equal pumping power shows that the baffles with the highest open area ratio give the best performance.

Recently, experimental study attempted to explore the local heat transfer in rectangular channel with baffles was carried out by Lin [24]. Lin [24] analyzed the experimental results of baffles with different heights and bores in the event of different Reynolds numbers and heating quantities. He found that; apart from increasing the perturbation of flow field, the channel's flow field with baffles, which is similar to a backward-facing step flow field, is very helpful to heat transfer.

In a more recent publication, a detailed study was conducted for different height of entry channel to uniformly heated rectangular channel with two different angles turned flow, and with one wall mounted baffle [25]. Kurtbas [25] concluded that the pressure drop and heat transfer rate are strongly increased with the presence of a baffle and turned flow compared with a straight duct.

In the present work experimental and numerical studies of pressure recovery behavior of isothermal-turbulent flow in a rectangular channel is presented for different configuration, positions, and orientations of inclined baffles attached to the upper and lower surfaces. The study also includes the effect of the inflow Reynolds number on the pressure recovery.

2. EXPERIMENTAL SETUP

The experiments were conducted in the heat engine laboratory of the Faculty of Engineering, Menoufiya University. Figure 1 (a) shows a simple schematic view of the experimental setup. Filtered air to the test facility was supplied from two-parallel connected screw compressors. The maximum delivery pressure from the compressors is 15 bar. Each of the two compressors is capable of delivering $0.054\text{m}^3/\text{s}$ compressed air to a large air reservoir from which the air passes through a control valve in the main pipeline of 10cm diameter. Therefore, the inflow discharge to the working section is manually regulated. A settling pipe of 4 hydraulic-diameter length is installed followed by a duct converter, smoothly converting the circular cross-section to a rectangular cross-section duct of the working section; see El-Askary and Nasr [26]. The test section used in the present paper is manufactured from steel. Its cross-sectional area is $W \times H = 12\text{cm} \times 5\text{cm}$ (internal dimensions) and length 100 cm.

Both (steel) solid and perforated baffles are of the same overall size (length $L=29$ cm, width $W=12$ cm and thickness $t=3$ mm) and utilized in the present study. The schematic view of the perforated baffles is given in Fig. 1 (b). The leading edges of the baffles are kept sharp to reduce the flow disturbance by the protruding edge. All perforated baffles have uniform holes of diameter $d=1\text{cm}$. A total of three baffles, two perforated baffles and one solid are used. The summary of these baffles dimensions is presented in Table (1). In all arrangement, constant inclination angle of five degrees is maintained constant for both plates. This small inclination angle keeps these plates streamlined with the flow, and hence they avoid major flow blockage [9].

The experimental work was done through two cases. The first case is done without baffles. The second case is done with different baffle arrangements, see Table 1.

The experimental study was only aimed to measure the static wall-pressure distributions along the lower wall of the duct. The wall static pressure was measured through 1 mm diam. pressure taps on the wall along the center plane of duct. The uncertainties in measuring wall pressure led to an uncertainty in the wall static pressure coefficient of ± 0.009 . The measurements of the static pressure distribution were conducted using multi-tube water manometer. The inflow Reynolds number was determined from the measured flow rate (Q) through a calibrated orifice meter fitted with a mercury U-tube manometer with an accuracy of 1.5 %.

The wall pressure distributions are sampled in the form of wall pressure recovery coefficient defined as

$$C_p = \frac{\bar{p} - \bar{p}_{ref}}{0.5 \rho U_{in}^2}, \text{ where } \bar{p}_{ref} \text{ is the atmospheric}$$

pressure and $U_{in} = \frac{Q}{W \times H}$ is the average velocity at entrance [26].

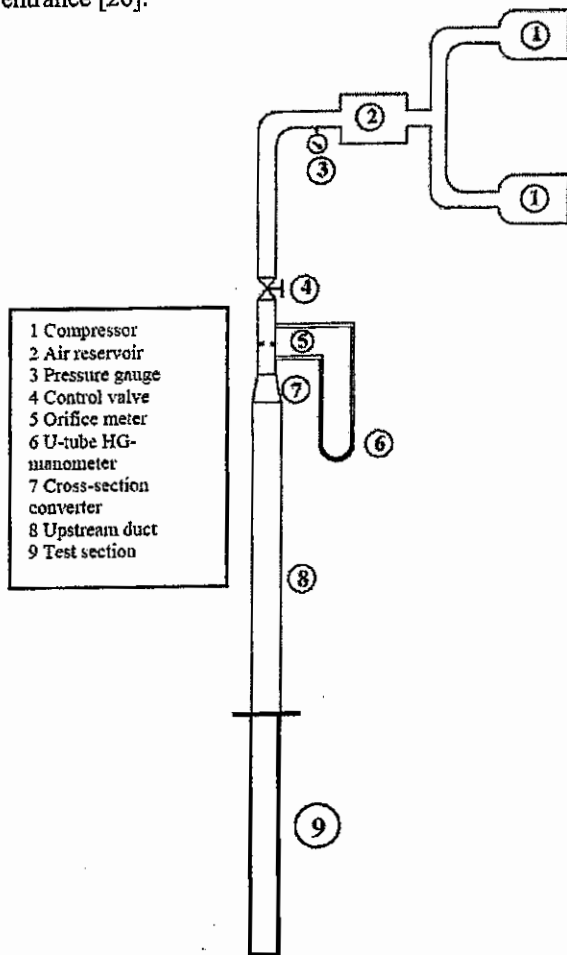


Fig. 1.a Schematic diagram of the experimental setup

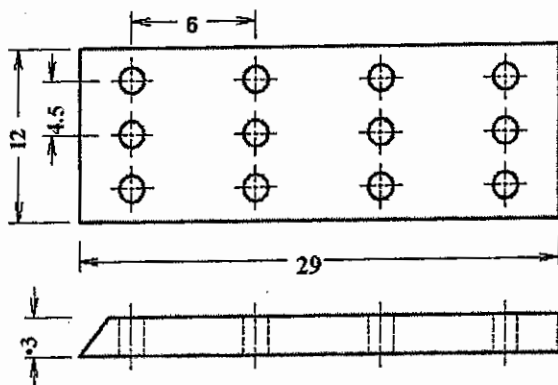


Fig. 1.b a perforated baffle plate (dimensions in cm.)

Table 1 Different baffle configurations used in the experiments, see Fig. (2)

Case study	C (cm)	Baffles	Attachment of Baffle to surfaces	Fixed dimensions (cm)
1	29	1 perforated and 1 solid	one perforated up and one solid down	A=1.25, B=2.5
2	44.5	1 perforated and 1 solid	one perforated up and one solid down	A=1.25, B=2.5
3	60	1 perforated and 1 solid	one perforated up and one solid down	A=1.25, B=2.5
4	29	2 perforated	one on the top surface and the other on the down surface	A=1.25, B=2.5
5	44.5	2 perforated	one on the top surface and the other on the down surface	A=1.25, B=2.5
6	60	2 perforated	one on the top surface and the other on the down surface	A=1.25, B=2.5
7	29	2 perforated	Top surface	A=1.25, B=2.5
8	44.5	2 perforated	Top surface	A=1.25, B=2.5
9	60	2 perforated	Top surface	A=1.25, B=2.5

3. MATHEMATICAL MODEL

The physical model used in this study is shown in Figure 2, where two-dimensional incompressible-turbulent flow is considered. In the present study, it is assumed that the fluid (air) is incompressible and Newtonian with temperature-independent fluid properties.

Based on the characteristics scales of H (the duct cross-section height) and U_{in} (the inflow bulk velocity), the dimensionless variables are defined as follows:

$$x = \frac{\bar{x}}{H}, \quad y = \frac{\bar{y}}{H}, \quad k = \frac{\bar{k}}{U_{in}^2}, \quad u = \frac{\bar{u}}{U_{in}}$$

$$v = \frac{\bar{v}}{U_{in}}, \quad p = \frac{\bar{p}}{0.5 \rho U_{in}^2}, \quad \varepsilon = \frac{\bar{\varepsilon} H}{U_{in}^3},$$

where, the over bar represents the dimensional quantities. According to the above assumptions and dimensionless variables, the dimensionless governing equations are expressed as the following:

Continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

Momentum equations:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[\left(\frac{1}{Re} + \frac{1}{Re_t} \right) \frac{\partial u}{\partial x} \right] + \frac{\partial}{\partial y} \left[\left(\frac{1}{Re} + \frac{1}{Re_t} \right) \frac{\partial u}{\partial y} \right] + \frac{\partial}{\partial x} \left(\frac{1}{Re_t} \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial x} \left(\frac{1}{Re_t} \frac{\partial v}{\partial y} \right) \quad (2)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left[\left(\frac{1}{Re} + \frac{1}{Re_t} \right) \frac{\partial v}{\partial x} \right] + \frac{\partial}{\partial y} \left[\left(\frac{1}{Re} + \frac{1}{Re_t} \right) \frac{\partial v}{\partial y} \right] + \frac{\partial}{\partial x} \left(\frac{1}{Re_t} \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial y} \left(\frac{1}{Re_t} \frac{\partial v}{\partial x} \right) \quad (3)$$

The dimensionless equations for standard k-ε model are written as:

$$u \frac{\partial k}{\partial x} + v \frac{\partial k}{\partial y} = \frac{\partial}{\partial x} \left[\left(\frac{1}{Re} + \frac{1}{\sigma_k Re_t} \right) \frac{\partial k}{\partial x} \right] + \frac{\partial}{\partial y} \left[\left(\frac{1}{Re} + \frac{1}{\sigma_k Re_t} \right) \frac{\partial k}{\partial y} \right] + G - \varepsilon \quad (4)$$

$$u \frac{\partial \varepsilon}{\partial x} + v \frac{\partial \varepsilon}{\partial y} = \frac{\partial}{\partial x} \left[\left(\frac{1}{Re} + \frac{1}{\sigma_\varepsilon Re_t} \right) \frac{\partial \varepsilon}{\partial x} \right] + \frac{\partial}{\partial y} \left[\left(\frac{1}{Re} + \frac{1}{\sigma_\varepsilon Re_t} \right) \frac{\partial \varepsilon}{\partial y} \right] + c_1 \frac{\varepsilon}{k} G - c_2 \frac{\varepsilon^2}{k} \quad (5)$$

Where G is the rate of production of turbulent kinetic energy and is given by:

$$G = \frac{2 S^2}{Re_t} \quad (6)$$

and

$$S^2 = \frac{1}{2} \left[\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right)^2 + 2 \left(\frac{\partial u}{\partial x} \right)^2 + 2 \left(\frac{\partial v}{\partial y} \right)^2 \right] \quad (7)$$

The empirical values of the model constants are reported as: $c_1=1.44$, $c_2 = 1.92$, $c_\mu =0.09$, $\sigma_k=1$ and $\sigma_\varepsilon=1.3$.

Leschziner and Rodi [27] incorporated the effects of streamline curvature on c_μ in the form

$$c_\mu = \text{Max} \left\{ 0.025, \frac{0.09}{\left[1 + 0.57 \frac{k^2}{\varepsilon^2} \left(\frac{\partial V_{tot}}{\partial n} + \frac{V_{tot}}{R_c} \right) \frac{V_{tot}}{R_c} \right]} \right\} \quad (8)$$

Where, $V_{tot} = \sqrt{u^2 + v^2}$ is the total velocity.

The curvature radius is

$$\frac{1}{R_c} = \frac{uv \left(\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right) + v^2 \frac{\partial u}{\partial y} - u^2 \frac{\partial v}{\partial x}}{(u^2 + v^2)^{1.5}} \quad (9)$$

The dimensionless form of dissipation rate for renormalization group model (RNG model) is written as [28]:

$$u \frac{\partial \varepsilon}{\partial x} + v \frac{\partial \varepsilon}{\partial y} = \frac{\partial}{\partial x} \left[\left(\frac{1}{Re} + \frac{1}{\sigma_\varepsilon Re_t} \right) \frac{\partial \varepsilon}{\partial x} \right] + \frac{\partial}{\partial y} \left[\left(\frac{1}{Re} + \frac{1}{\sigma_\varepsilon Re_t} \right) \frac{\partial \varepsilon}{\partial y} \right] + \frac{\varepsilon}{k} \left[c_1 - \frac{\eta \left(1 - \frac{\eta}{\eta_0} \right)}{1 + \beta \eta^3} \right] G - c_2 \frac{\varepsilon^2}{k} \quad (10)$$

$$\text{where } \eta = \sqrt{c_\mu^{-1} \frac{G}{\rho \varepsilon}}$$

The values of the model constants are taken as: $c_\mu = 0.085$, $c_1 = 1.44$, $c_2 = 1.68$, $\sigma_k = 1.39$, $\sigma_\varepsilon = 1.39$, $\eta_0 = 4.38$ and $\beta = 0.012$

The eddy viscosity μ_t is given by the relation

$$\mu_t = c_\mu \frac{\rho k^2}{\varepsilon} \quad (11)$$

The boundary conditions for the above set of governing equations are as follows:

a) Inlet boundary

The uniform velocity profiles and turbulent kinetic energy and its dissipation rate are imposed at the inlet boundary, i.e., $\bar{u}_i = U_m$, $\bar{k}_i = 0.01 U_m^2$ and $\bar{\varepsilon}_i = \frac{c_\mu \bar{k}_i^{1.5}}{0.1H}$.

b) Wall boundary

At the wall, the no slip boundary condition is imposed, $\bar{u} = \bar{v} = 0$ and using the wall function suggested by Launder and Spalding [29] to compute the near wall velocity. The law of the wall is used to

modify the flow parameters \bar{u} , \bar{v} , \bar{k} and $\bar{\epsilon}$ at the nearest grid point.

In the porous region, the contribution of turbulence in a porous region is neglected by setting the turbulent contribution to viscosity equal to zero. In addition, the generation of turbulence will be set to zero in the medium.

c) Exit boundary

A zero gradient condition is employed for the outlet boundary. Although this boundary condition is strictly valid only when the flow is fully developed, it is also permissible for sufficient downstream from the region of interest, i.e., $\frac{\partial \phi}{\partial x} = 0$ and

$$\phi = \bar{u} = \bar{v} = \bar{k} \text{ and } \bar{\epsilon}$$

4. SOLUTION PROCEDURE

The mathematical models described above consist of a set of differential equations subject to appropriate boundary conditions. To provide the algebraic form of the governing equations, a fully staggered grid system is adopted for the velocity components and the scalar variables. These equations are discretized using a control volume finite difference method (CVFDM). The numerical solution in the present work is accomplished using Semi- Implicit Method for Pressure Linked Equation (SIMPLE) utilized by Patankar [30]. The velocity component u is calculated at the east and west faces of the main control volumes from the solution of the axial momentum equation. Similarly, the velocity component v at the north and south faces is calculated. When the pressure correction equation is solved, then the

velocities and the pressure fields are corrected. To complete iteration, the turbulent kinetic energy and energy dissipation rate are solved successively. The discretization equations are solved by the line by line procedure, which is a combination of Gauss- Seidel and tridiagonal matrix algorithm in the streamwise direction. The tridiagonal matrix algorithm (TDMA) is used to solve a set of discretization equations in the cross- stream direction. Relaxation factors are employed to promote smooth convergence of the discretized equations. The relaxation factors are controlled to be 0.75, 0.75, 0.5, 0.5, and 0.5 for u , v , p , k and ϵ , respectively. The turbulent viscosity is also under-relaxed at a value of 0.5. The converged criterion in this study is based on the successive changes in variables. All field variables are monitored, and the following condition is used to declare convergence:

$$\text{MAX} \left| \frac{\phi_{i,j}^n - \phi_{i,j}^{n-1}}{\phi_{i,j}^n} \right| \leq 10^{-4} \quad (12)$$

In addition, the ratio of the difference between the inlet mass flow rate and the outlet mass flow rate to the inlet mass flow rate is also examined. Convergence is declared if the relative mass imbalance is less than 10^{-3} and Equation (12) are satisfied simultaneously. To verify the algorithm, numerical tests are performed to ensure that the solution is grid algorithm.

The grid points are distributed uniformly over the computational domain. A 350×150 grid points are placed in the computational domain after different tests of grid dependence, see Fig.2 (d).

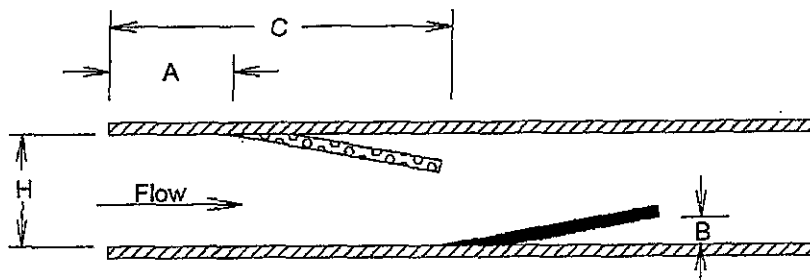


Fig.2 (a) Channel with one perforated baffle up and one solid baffle down

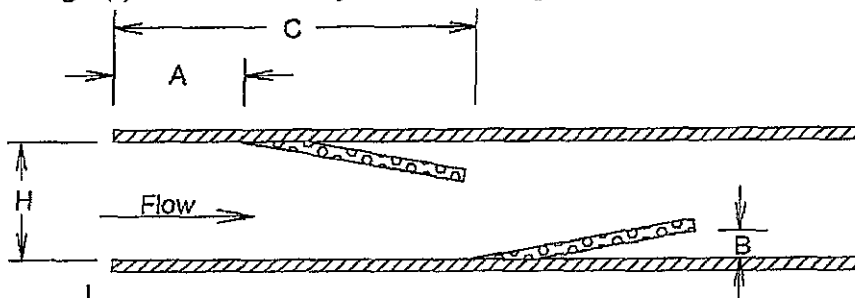


Fig.2 (b) Channel with one perforated baffle up and one perforated baffle down

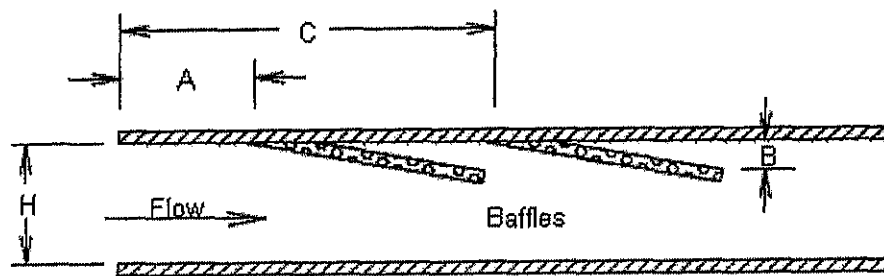


Fig. 2 (c) Channel with two perforated baffles on up wall

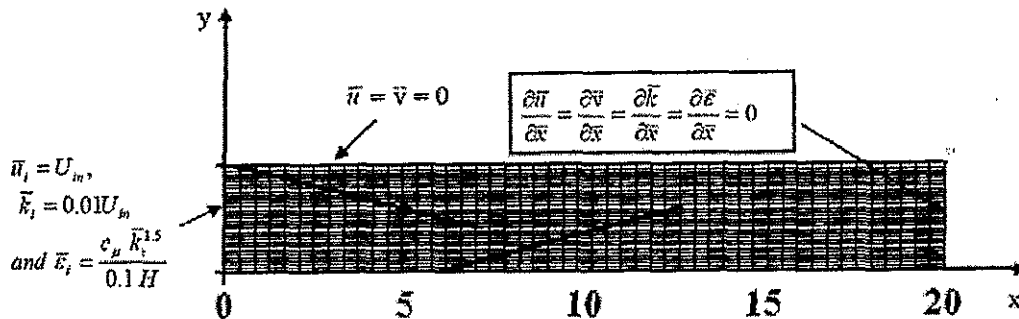


Fig. 2 (d) Grid generated to the channel with two inclined baffles

5. RESULTS AND DISCUSSION

The performance of the various discussed $k-\varepsilon$ models is first tested. The local pressure recovery coefficient C_p along the lower wall of the duct with the presence of one perforated baffle on the upper wall and another one on the lower wall (case 4, see Table 1) is presented in Fig. 3 for all tested turbulence models. As noticed the standard and the improved streamline curvature $k-\varepsilon$ models capture the shape of the pressure distribution. However, the improved $k-\varepsilon$ model of Leschziner and Rodi [27] adequately fits the experimental data, so it will be used for all other simulation cases.

Figure 4 shows the pressure coefficient on the bottom wall of a smooth channel without any installed baffles with dimensionless downstream distance x/H for different values of Reynolds number using $k-\varepsilon$ model with correction of Leschziner and Rodi [27]. The trend of pressure generally decreases in the downstream direction. As Reynolds number increases, the pressure coefficient decreases due to the friction losses increasing.

Figure 5 represents the relation between the pressure recovery coefficient C_p and the dimensionless downstream distance x/H at bottom wall of the channel with two inclined baffles of 5° inclined angle. One baffle is perforated and located on the upper wall at dimensionless distance $A/H = 0.25$ from the inlet of channel and a second baffle is solid and located on the bottom wall at different dimensionless distances, $C/H = 5.8, 8.9$ and 12 from the inlet of channel (cases 1, 2 and 3, see Table 1). The plots

show the effect of Reynolds number on the pressure recovery coefficient. Three different values of channel Reynolds number 71,000, 100,000 and 122,500 are considered, respectively. From Figure 5(a), it can be seen that the pressure decreases with downstream distance until the location of the second baffle. This is due to the acceleration of flow near the bottom wall with the downstream distance. It falls down suddenly behind the baffle to minimum value and then increases again reaching the reattachment point and it slowly increases to reach the atmospheric pressure at the duct exit. It may be expected to be an effect of a recirculation zone generated behind the baffle. Also from the Figure, it is seen that the pressure coefficient decreases by increasing Reynolds number. This is because the hydraulic losses are decreasing with increasing Reynolds number. The increment of the pressure recovery coefficient decreases in the front of the second baffle by changing its location in downstream direction $C/H = 8.9$. When the location of second baffle is at $C/H = 12$, the effect of the first baffle is appearances at the begging at $x/H = 6$. The pressure in this zone decreases with downstream due to the hydraulic losses. But the pressure recovery coefficient increases after the second baffle and this may be due to the presence of the recirculation zone behind the second baffle, see Figure 5 (c).

Figure 6 shows the relation between the pressure recovery coefficient and the dimensionless downstream distance x/H at bottom wall of the channel with two inclined perforated baffles with the same inclined angle of 5 degrees. One perforated baffle is located on the upper wall at a distance

