

## Abstract

The effect of the flow geometry parameters on pressure drop and flow rate for turbulent flow in a rectangular duct with an array of plates aligned at angles with positive and negative manner to main flow direction have been investigated numerically and experimentally. The plates were installed so that each plate pair acts as periodically interrupted divergent and convergent short channels in a rectangular channel. Different geometrical parameters were varied and the flow was numerically studied solving the incompressible Navier-Stokes equations, using upwind difference scheme. The numerical results were compared with the experimental work for similar flow geometries. The experimental and theoretical results showed good agreement. New

## An Experimental and Numerical Study of the Flow in a Rectangular Duct with an Array of Plates Aligned at Angles to the Flow

Hussain Al Madani, Teoman Ayhan,  
Taj O. Mohieldin, Omar Al Abbasi  
University of Bahrain,  
Mechanical Engineering Department, Bahrain

pressure drop/flow rate measurements were coupled with a previously reported pressure drop coefficient correlation equation for an array of plates aligned at angles with positive and negative manner to the main flow direction in a rectangular duct. The proposed empirical correlations were considered to be applicable within the range of Reynolds number  $3000 < Re < 20000$  for air, and Prandtl number of 0.7. The analysis showed that different arrays of plate configurations results were obtained, depending on the optimal design criteria imposed on the flow.

**Keywords:** array of plates, rectangular duct, static mixer, periodically interrupted flow.

## 1. Introduction

The performance of heat exchanger for single-phase flow can be improved by many enhancement techniques. The flow geometries are dominant on the heat transfer surfaces. Special flow geometries for heat transfer surfaces bring about the transfer enhancement by establishing a higher heat transfer coefficient. Flow interruption created in flow passages at periodic intervals is a popular means of enhancing heat transfer in compact heat exchangers. In the last half decade, forced flow over an array of plates aligned at angles to the flow has received increasing attention for applications such as various industrial compact heat exchangers (Jacobi and Shah 1995), automotive radiators (Oliet *et al* 2007) and air conditioning (Saman and Ali Zadah). Fluid flow over an array of plates aligned at angles to the flow in the rectangular channels results in a pressure drop, which is often used as criterion to optimally design heat transfer systems like heat exchangers (Kotcioglu *et al* 1998). Literature survey reveals numerous experimental and numerical studies aspiring to estimate the pressure drop for flow over the array of plates configurations aligned with the fluid flows in a rectangular channel (Lee 1986). There are only a limited number of studies that explore the role of orientation angle  $\theta$  (Ayhan and Al-Madani 2005).

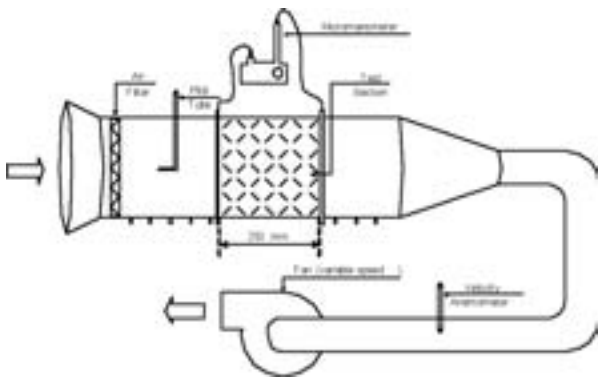
Very few incorporate or accurately represent the effect of flow mixing in the buffer zone, and further, none of these studies have examined duct aspect ratio  $W/H$ , vertical spacing to the plate length ratio  $S_T/L_p$ , longitudinal spacing to the plate length ratio  $S_L/L_p$  and the ratio of the plate length to the effective channel length  $L_p/L_{eff}$ . at high air velocity where flow unsteadiness plays an important role.

The purposes of the present study are as follows: first, to carry out a numerical analysis of the effect of the flow geometry parameters on pressure drop and flow rates for turbulent flow in a rectangular duct with an array of plates aligned at angles with positive and negative manner to main flow direction. Second to conduct experiments to support theoretical findings. Several dimensionless parameters of the flow geometry are used for the experimental setup as well as the numerical study.

## 2. Experimental Setup Equipment

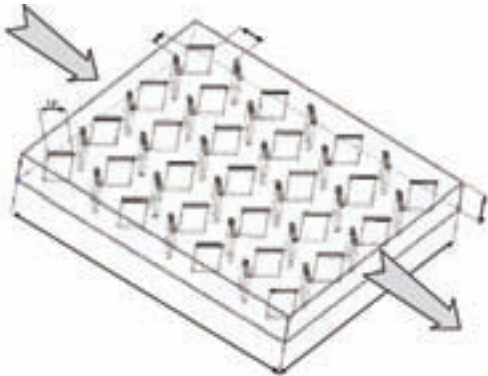
The main features of the test rig are shown schematically in *Figure 1*. Air was used as a working fluid. Experimental setup mainly consists of flow entrance section, flow development section, test section, flow exit section and data acqui-

sition equipment. Experimental setup and test section are fabricated completely from Plexiglas. The test section consist of array of plates inside channel where the fluid flows and mixes and pressure ports outside the channel facilitate pressure measurements at a different locations, in *Figure 2*. The local pressures are measured with calibrated micromonometer. In order to measure the total pressure drop for the test section, circumferential pressure taps are attached upstream and downstream of the test section, as shown in *Figure 2*. The pressure drop between the inlet and the exit of the test section is measured by electronic calibrated electronic micromonometer.



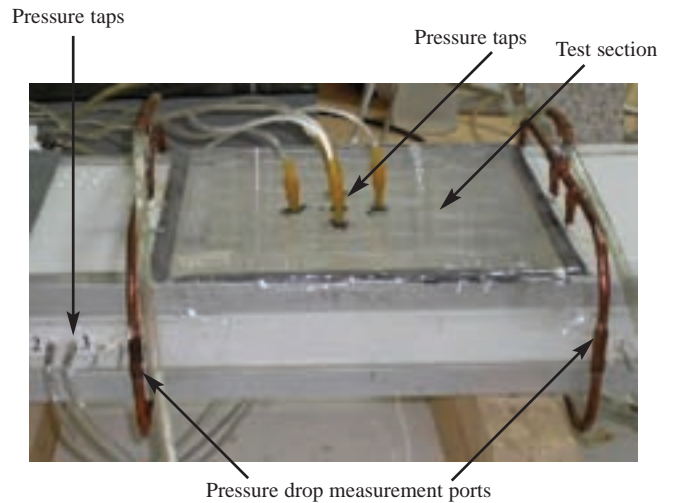
*Figure 1. Schematic lay out of the experimental setup.*

Local velocities are measured with a pitot tube in the inlet section of the setup and mean velocity at the inlet is calculated. And also mean velocities are measured by using anemometer at the exit section of the set up to compare the error on mass flow rates of air in the channel.



*Figure 2. Geometry and placement of array of plates in the rectangular duct.*

The test section used was rectangular channel that contained a plate array, its geometry is presented in *Figures 2* and *3*. The channel section was 195 mm wide, 70 mm high and 180 mm long. The channel walls were made of 5mm thick Plexiglas to facilitate the flow visualization tests. The rectangular plate with a thickness of 4 mm was made from Plexiglas (*Figure 4*).



*Figure 4. View of the test section.*

The variable parameters of the plate array were the relative transverse pitch ( $X_T = S_T/L_P = 0.60, 0.65, 0.75, 0.8, 1.0$  and  $1.18$  where  $S_T$  is the transverse distance between the entering edge of plate pairs and  $L_P$  is the length of the rectangular plate), the relative longitudinal pitch ( $X_L = S_L/L_P = 0.60, 0.65, 0.75, 0.8, 1.0$  and  $1.18$  where  $S_L$  is the longitudinal distance between the adjacent plates).

The plate length is varied between 20 mm and 25 mm. Plate angle  $\theta$ , is defined with respect to flow direction, which is varied from  $5^\circ$  to  $35^\circ$  with  $5^\circ$  intervals and  $45^\circ$ . Characteristics of configuration investigated in the present study can be obtained in *Table 1*.

**TABLE 1. Geometrical characteristics for the flow in rectangular duct with the array of plates aligned at angles to the flow ( $N=49$  and  $t=2$  mm)**

Type	$\theta$ (rad)	$S_L/L_P$	$S_T/L_P$	$L_P/L_{eff}$	$\beta$
A	0.087	0.605	0.202	0.100	0.827
B	0.174	1.000	0.400	0.101	0.825
C	0.262	1.176	0.588	0.069	0.881
D	0.349	0.800	0.480	0.101	0.825
E	0.436	1.000	0.500	0.081	0.860
F	0.523	0.652	0.435	0.093	0.839
G	0.785	0.750	0.750	0.081	0.860

### 3. Experimental Uncertainty

The uncertainty calculation method used involves calculating derivatives of the desired variable with respect to individual experimental quantities and applying known uncertainties (Mc Clintock and Klin 1953).

Experimental uncertainties in the Reynolds number, and friction factor were estimated by the above procedure described in (Mc Clintock and Kline 1953). The mean uncertainties were found to be  $\pm 2.5\%$  in the Reynolds number,  $\pm 4\%$  in the friction factor.

### Derivation of Dimensionless Parameters

The problem analyzed in this paper is schematically pictured in **Figure 2** and **Figure 3**. The obstacle in the form of a fin-pair (of two mm thickness) is placed inside the channel, in such a way that periodically interrupted enlarged and contracted channel flow domains can be established. Thus, each plate pairs acts as periodically interrupted divergent and convergent short channels in the main channel flow (**Figure 2**).

The configuration studied can be specified by the following parameters: duct aspect ratio  $W/H$ , vertical spacing to the plate length ratio  $S_T/L_p$ , longitudinal spacing to the plate length ratio  $S_L/L_p$ , plate thickness to plate aspect ratio  $t/L_p$ , plate length to test section length ratio  $L_p/L_{eff}$ , and orientation angle of the plates  $\theta$ .

The total pressure drop  $\Delta P_T$  through the duct with the plate of array consists from three components namely pressure drop due to friction, Pressure drop due to inertia losses through the plate of array and pressure drop due to entrance/exit losses.

In order to calculate the pressure coefficient for the flow domain, the maximum mean velocity can be defined in the array of plates as:

$$V_{in}HW\rho_{in} = A_c V_{max}\rho_{in}$$

for isothermal and incompressible flow, therefore;

$$V_{max} = V_{in} \left( \frac{HW}{A_c} \right)$$

$$A_c = WH - N_L H L_p \sin \theta$$

where  $N_L$  is the plate number for per row.

#### Pressure coefficient for the plate of arrays CP

In determining the pressure coefficients, the resistance due to friction on the duct walls and plate surfaces and resistance due to changes in momentum we taken in to consideration.

$$C_p = \frac{\Delta P_N}{\frac{1}{2}\rho V_{max}^2 \left( \frac{L_{eff}}{D_{equ}} \right)}$$

where  $D_{equ}$  is the equivalent diameter of flow geometry therefore,

$$D_{equ} = \frac{V - NV_p}{2N(t + L_p)H + 2H L_{eff} + 2W L_{eff} - 2N t L_p}$$

where  $V$  is the volume of flow domain ( $V=HWL_{eff}$ ),  $V_p$  is the plate volume ( $VP=tL_pH$ ) and  $N$  is the total plate number in the flow domain.

Flow domain void ratio is defined as  $\beta = \frac{V - NV_p}{V}$

The Buckingham  $\pi$  theorem was used to derive the non-dimensional groups governing total pressure drop of an array of plates. Since the pressure drop is dependent upon the free stream velocity, plate geometries and void ratio

$$\Delta P_N = \Delta P_N (V_{max}, L_p, Leff, D_{eq}, k, \mu, \rho, \beta, \theta, t, S_L \text{ and } S_T)$$

### Computational Physical Model

#### Physical and Mathematical Formulation

In the present study, the numerical simulation deals with turbulent flow regime only. In the flow geometry, the governing conservation equations for turbulent flow and the appropriate boundary conditions will be presented for FLUENT software. The analysis was based on the following assumptions: (1) The fluid properties are constant; (2) the flow is steady, turbulent, and periodically fully developed; (3) the body forces term is neglected. Regarding the boundary conditions, the no-slip condition was used along the walls and plates. The present code utilizes the segregated method which employs SIMPLEC algorithm with second order scheme for the integration of momentum equation. The computations were performed on unstructured grids for all cases.

The final computation domain is shown in **Figure 2**. The fluid flow can be described by the following equations with Cartesian tensor notation:

The conservation equations for steady two dimensional incompressible laminar and turbulent flows may be written as:

- Continuity:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

- x - component momentum:

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) - \frac{\partial}{\partial x} (\overline{u'u'}) - \frac{\partial}{\partial y} (\overline{u'v'})$$

- y - component momentum:

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) - \frac{\partial}{\partial y} (\overline{v'v'}) - \frac{\partial}{\partial x} (\overline{u'v'})$$

- Energy:

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \frac{k}{\rho C_p} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) - \frac{\partial}{\partial x} (\overline{T'u'}) - \frac{\partial}{\partial y} (\overline{T'v'})$$

where the overbars are the time average quantities for Turbulent flow.

### 4. Results and Discussions

Flow visualization tests were used to explain the large pressure drop due to the various flow geometries. In the Hella-Shaw apparatus, the working fluid was water typical flow pattern for laminar flow is shown in **Figure 5**.

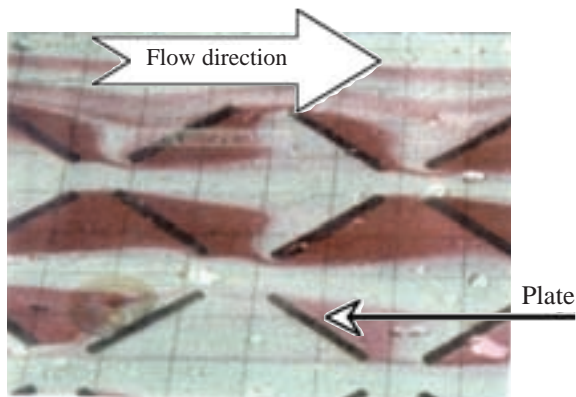


Figure 5. Flow visualization for a typical flow domain

Similar flow patterns were observed for the same flow geometries with FLUENT software solutions. The typical flow patterns as obtained from the FLUENT software at a  $45^\circ$  and 3 m/s inlet velocity for air are presented in Figure 6.

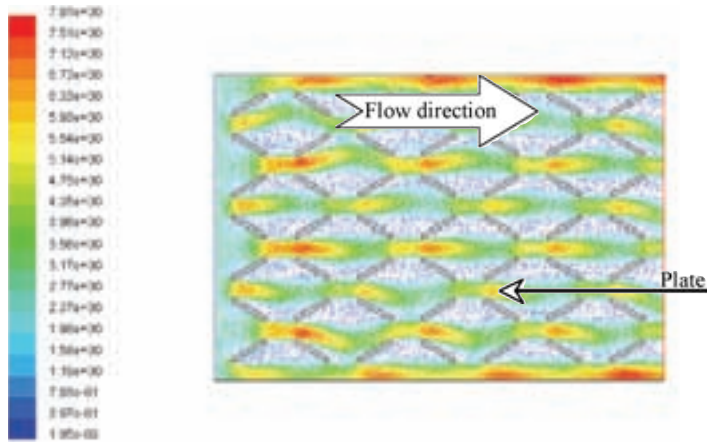


Figure 6. Flow pattern for a typical flow geometry

In case where the plate angles are greater than  $5^\circ$ , a back-flow region forms just downstream of diverging section, indicating the occurrence of the flow separation and the presence of a recirculating zone which can be seen in both Figures 6 and 7. In addition, there is a region situated between diverging and converging channels which called a buffer region, where the velocity is too low to enable to visualize the flow direction.

Due to the pressure rise at the end of the diverging channel and pressure drop at the neighboring converging channel, the velocities were high enough to create a flow against the main flow direction. The typical velocity field in this flow domain is presented in Figure 7.

The typical pressure distribution along the main and flow direction and total pressure drop in this flow domain is presented in Figure 8. And also pressure distributions through the main flow direction and in the buffer regions (the flow against the main flow directions) are presented in Figure 9. It can be seen that, down stream of main flow direction mixing effects in the buffer regions becomes more uniform.

Flow through an array of plates exerts a force on each plate. The total force acting on each plate consists from two components; the skin friction force and the drag force. The arrangement of the plates cause pressure variation at both inlet and exit of the array of plates in the test section. This variation could be attributed to the blockage effect.

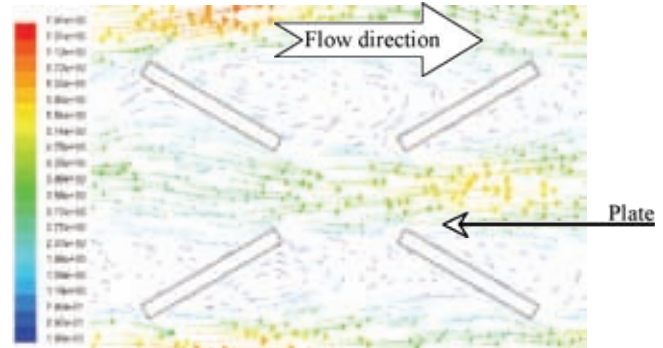


Figure 7. Close up to the buffer region (mixing zone and recirculating zone) for the flow domain.

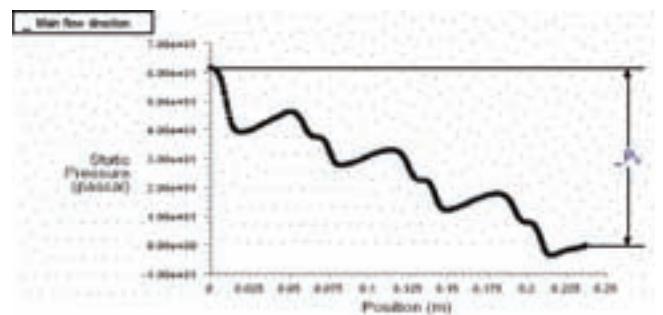
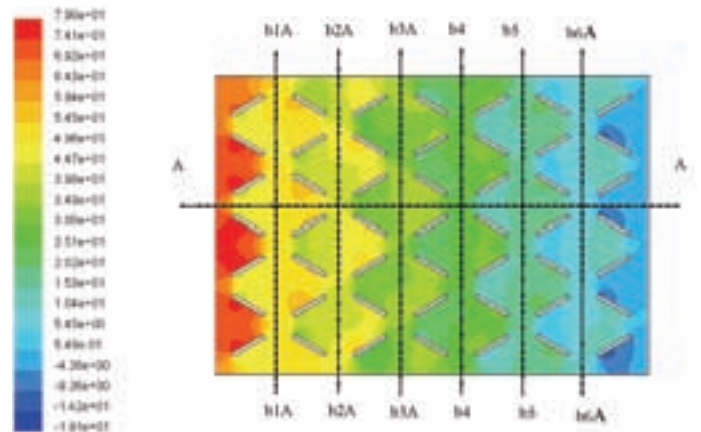


Figure 8. Pressure distribution along main flow direction second order scheme.



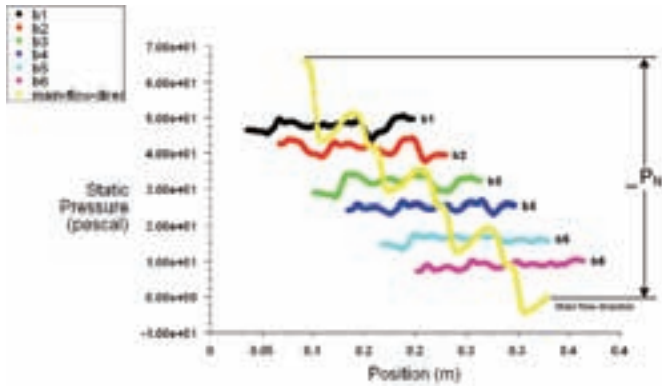


Figure 9. Pressure distribution through flow geometry.

Figures 8 and 9 show the pressure variation along the flow direction including blockage effects at the inlet and exit. These results are deduced from a numerical solution of Navier-stokes equations using FLUENT software.

Experimental observation of the same behavior of the flow can be seen in Figure 10 which shows a similar trend of the pressure variation.

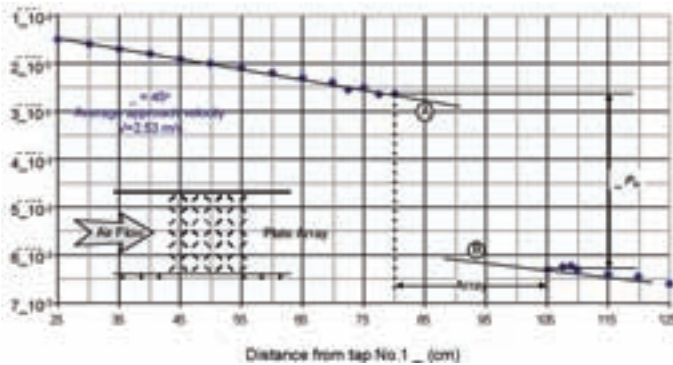


Figure 10. The graph of pressure drop along the duct from the inlet up to the reducer.

A pressure drop against main flow direction and transverse direction for the quartet located in the centre of the test section is observed as shown in Figure 11. Similar behavior is observed in the numerical solution shown in Figure 9.

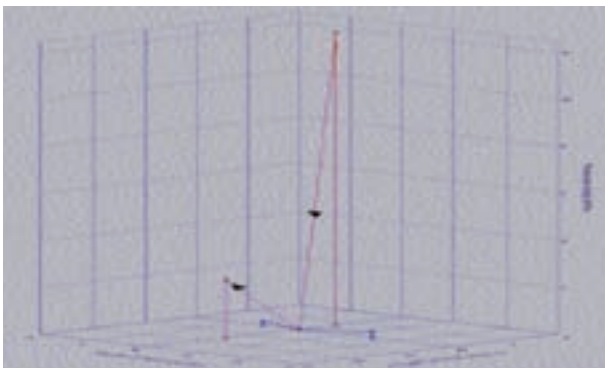


Figure 11. Three dimensional views of the pressure losses for the test location  $\theta = 30^\circ$   
 $V=2.52551$  (m/s)  $\Delta P=73.3$  Pa

Pressure drop against main flow velocity for different plate angles is shown in Figure 12.

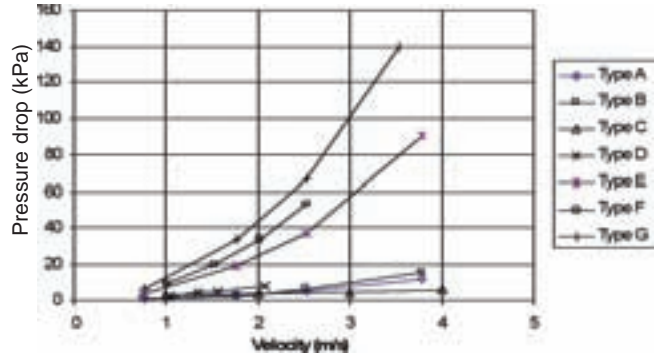


Figure 12. Maximum Velocity against the Pressure Drop.

The pressure coefficient  $C_p$  was found to be independent of Reynolds Number  $Re$  for each plate angles. This is seen clearly in Figure 13.

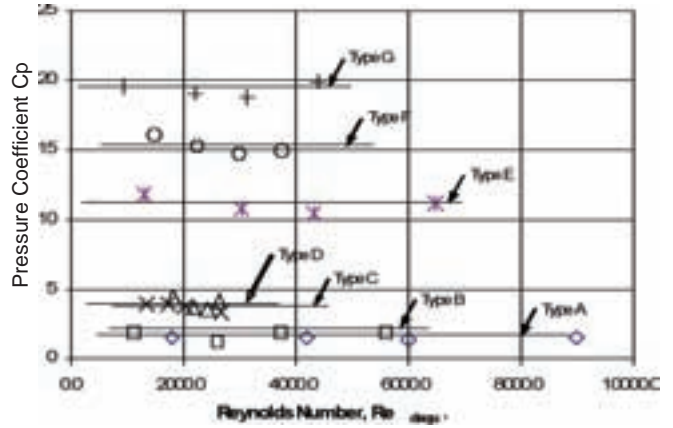


Figure 13. Pressure Coefficient versus Reynolds Number.

Flow domain void ratio is kept constant for each type but for different Reynolds Number. Flow domain void ratios are varied from 0.825 to 0.88 for all ranges of Reynolds Number.

### Concluding remarks

Although compact heat exchangers with different configuration of plate arrays offer many advantages compared to the conventional heat exchangers, their main drawback is the absence of a general design method. The proposed flow geometries is promising techniques to enhance their effectiveness because of the secondary flow developments against the main flow direction. In this respect, FLUENT simulation allows computation of various flow geometries, and study the effects of various design configuration on the flow characteristics.

The present experimental and theoretical studies demonstrate that an array of plates aligned at positive and negative angles in-line to the stream wise direction in rectangular duct brings about significant increase in the total pressure drop and develops secondary flows against the main flow direction, when the plate angles are increased.

The pressure drop coefficient depends on plate angles not Reynolds number.

The results of this study, a part from enhancing physical understanding of the flow inside plate arrays, can also contribute to the formulation of design equation for flow resistance.

### References

- [1] "Pressure drop characteristics and flow visualization of an array of plates aligned at angles to the flow in rectangular duct", T. Ayhan, and H. Al-Madani, the 14<sup>th</sup> International Conference on Thermal Engineering and Thermogrammetry (THERMO), Budapest, Hungary, (2005).
- [2] "Heat transfer surface enhancement through the use of longitudinal vortices: a review of recent progress", A.M. Jacobi, and R.K. Shah, Exp. Therm. Fluid. Sci., Vol.11, pp. 295-309, (1995).
- [3] "Heat transfer and flow in a structure rectangular channel with Wing-Type Vortex Generator", I. Kotcioglu, T. Ayhan, , H. Olgun, , and B. Ayhan, Tr.J. of Engineering and Environmental Science, Vol.22, pp. 185-198, (1998).
- [4] "Heat transfer and pressure drop characteristics of an array of plates aligned at angles to the flow in a rectangular duct",Y.N. Lee, Int.J.Heat and Mass Transfer, Vol.29, No.10, pp. 1553-1563, (1986).
- [5] "Describing uncertainties single-sample experiments", F.A. McClintock, and S.J. Kline, ASME, Mech.Eng. Vol.75, pp. 3-8, (1953).
- [6] "Parametric study on automotive radiators", Oliet, C., Oliva,A., J. Castro, and C.D. Prez-Segarra, Applied Thermal Engineering, Vol.27, Issue 11-12, pp. 2033-2043, (2007).