EXPERIMENTAL AND NUMERICAL STUDY ON THE IMPROVEMENT OF UNIFORMITY FLOW FOR THREE-LATERAL DIVIDING MANIFOLD

Jafar M. Hassan[1], Thamer A. Mohamed[2], Wahid S. Mohammed[1], Wissam H. Alawee[1]
[1]Department of Mechanical Engineering, University of Technology, Baghdad, Iraq
[2]Department of Civil Engineering, Universiti putra Malaysia, Serdang, Malaysia
Email: wissam_772005@yahoo.com

ABSTRACT

The flow distribution in manifolds is highly dependent on inlet pressure, configuration and total inlet flow to the manifold. The flow from a manifold has many applications and in various field of engineering such as civil engineering, mechanical engineering and chemical engineering. In this study, experimental and numerical models were employed to study the uniformity of the flow distribution from manifold with various configurations. The physical model consists of main manifold with uniform longitudinal section having diameter of (10.16 cm), three laterals with (5.08) cm diameter and spacing of (22 cm). Different inlet flows were tested and the values of these flows are (576, 738 and 997 L/min). A manifold with tapered longitudinal section having inlet diameters of 10.16 cm (4 in) and dead end diameter of 5.08 cm (2 in) with the same above later specifications and flow rates was tested for its uniformity too. The percentage absolute mean deviation for manifold with uniform diameter was found to be (25%) while its value for the manifold with non-uniform diameter was found to be (14%). This result confirms the efficiency of the non-uniform in fluid distribution.

Keywords: *manifold, uniform flow, flow distribution.*

INTRODUCTION

Flow in manifold is of great importance in many industrial processes when it is necessary to distribute a large fluid stream into several parallel streams and then to collect them into one discharge stream. Manifolds can usually be categorized into one of the following types: dividing, combining, parallel, and reverse flow manifolds [1]. Parallel and reverse flow manifolds are those which combine dividing and combining flow manifolds and are most commonly used in plate heat exchangers. In a parallel flow manifold, the flow directions in dividing and combining flow headers are the same which is generally referred as a Z-manifold. In a reverse flow manifold, the flow directions are opposite is referred as a U-manifold. A uniform flow distribution requirement is a very common issue in many engineering circumstances because it has significant influence on the performance of fluidic devices such as plate-type heat exchangers, a variety of piping system, heat sinks for cooling of electronic devices, fuel cells, chemical reactors, solar thermal collectors, flow distribution systems in treatment plant, the piping system of pumping stations. Therefore, for most applications, the goal of manifold design is to achieve the uniform flow distribution through all of the lateral exit ports [2].

A great number experimental, analytical and numerical studies deal with flow in manifold. The flow in distribution manifold has been studied by several investigators [2–6]. For instance, Bajura [2] developed the general theoretical model for investigation of the performance of single-phase flow distribution for both intake and exhaust manifolds. Bajura and Jones [3] extended the previous model and prediction for the flow rates and the pressures in the headers of dividing, combining, reverse and parallel manifold configurations. Majumdar [4] developed a mathematical model with one-dimensional elliptic solution procedure for predicting flows in dividing and combining flow manifolds. Bassiouny and Martin [5,6] presented an analytical solution for the prediction of flow and pressure distribution in both intake and exhaust conduits of heat exchanger for both types flow (U-type and Z-type). A great number of experimental and numerical studies covered the affect of design parameters on flow distribution in manifold. Choi et al. [7,8] studied numerically the effect of the Reynolds number and the width ratio on the flow distribution in manifolds of a liquid cooling module for electronic packaging. Kim et al. [9] investigated numerically the effects of header shapes and the Reynolds number on the flow distribution in a parallel flow manifold of a liquid cooling module for electronic packaging, for three different header geometries (i.e., rectangular, triangular, and trapezoidal) with the Z-type flow direction. Jiao et al. [10] investigated experimentally the effect of the inlet pipe diameter, the first header's diameter of equivalent area and the second header's diameter of equivalent area on the flow maldistribution in plate-

fin heat exchanger. Wen et al. [11] investigated flow characteristics in the entrance region of plate-fin heat exchanger by means of particle image velocimetry (PIV). Tong et al. [12] investigated numerically the strategies capable of perfecting manifold design to achieve the same rate of mass outflow through each of the exit ports of a distribution manifold. Pan Zeng et al. [13] performed a three-dimensional computational fluid dynamics (CFD) model to calculate the velocity distribution among multiple parallel microchannels with triangle manifolds. The effect of channel width and channel spacing on flow distribution among microchannels with U-shape rectangular manifolds has been investigated by Mathew et al [14]. Andrew and Sparrow [15] present a method to investigate the effect of geometric shape of the exit ports on mass flow rate uniformity effusing from a distribution manifold; three candidate exit-port geometries were considered: (a) an array of discrete slots, (b) an array of discrete circular apertures, and (c) a single continuous longitudinal rectangular slot. In order to have a valid comparison of the impacts of these individual geometries, the total exit areas were made identical. Dharaiya et al. [16] studied numerically the effect of tapered header configuration to reduce flow maldistribution in minichannels and microchannels. Tong et al. [17] applied a logic-based systematic method of designing manifold systems to achieve flow rate uniformity among the channels that interconnect a distribution manifold and a collection manifold. The method was based on tailoring the flow resistance of the individual channels to achieve equal pressure drops for all the channels. The tailoring of the flow resistance was accomplished by the use of gate-valve-like obstructions.

In general all previous studies for manifolds with different application have shown that typical manifold design does not give a uniform flow distribution at outlets. Therefore, the objective of the study was to predict the flow distribution through each outlet for circular cross-section manifold with three laterals and to develop an optimized tapered cross-section header design having a better flow distribution through outlets.

METHODOLOGY

A. Experimental setup

The line diagram of the experimental setup is shown in Figure 1. Water is the test fluid. The experimental setup consists of water tank with over flow, steel support, a pump, an, a sump, number of valves to set the required flow rate through the setup and two dividing manifold, the first manifold with uniform longitudinal section and the second manifold with tapered longitudinal section (optimal taper shape from numerical section). The rig was assembled at a selected site in the hydraulic laboratory of Department of Civil Engineering, University Putra Malaysia. The water tank is rested on 3m high steel elevated frame. At the outlet of the each branch pipe, a shallow tank with cross section 150 cm x 150 cm is used to collect the water flowing from the branch pipes as shown in Figure 2. The water from the branch pipes is measured using 50 liters capacity rectangular tank. A constant head was ensured during the experiments and as a result constant flow rate from branch pipes were obtained. Six uniformly spaced piezometers were installed along the pipe to monitor the pressure head at the branch pipes. The spacing was 22 cm. The dimensions of two configuration manifold are shown in Figure 3 and Table 1. The manifolds have been fabricated with acrylic material to ensure the developed flow and good visibility of flow pattern. The branch pipes junctions are at right angles with header. The difference between two models only lies in the header configuration.

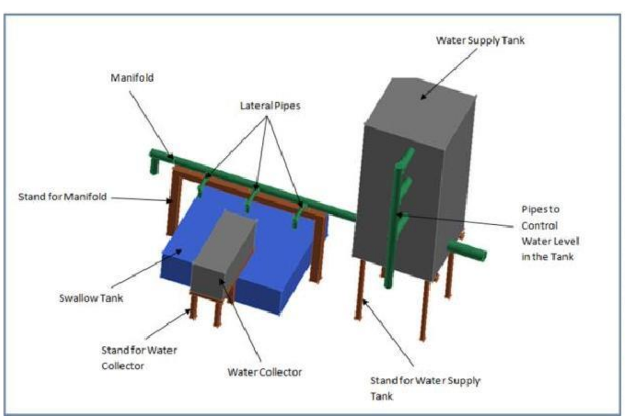

Figure 1: The diagram experimental setup

Figure 2: Photography of the experimental rig

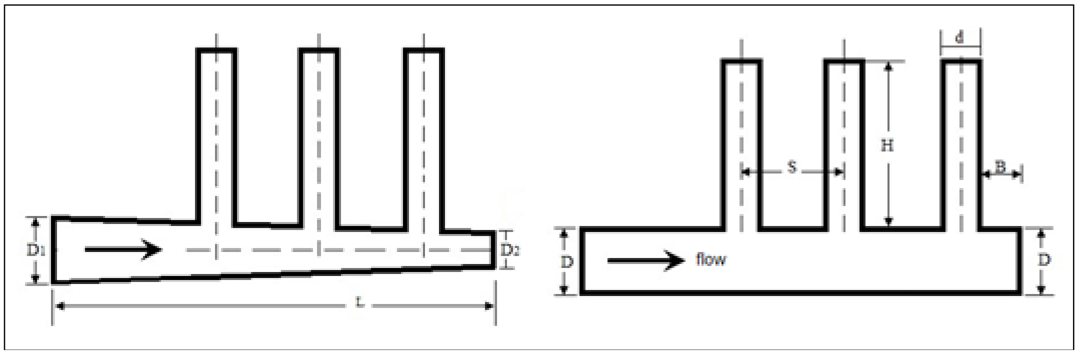

Figure 3: Manifolds used for conducting experiment

Table 1: The dimensions of two manifold configurations

Manifold Diameter, D	**Lateral Diameter, d**	**Manifold Length, L**	**lateral Length, H**	**Space between laterals, S**	**Distance after last lateral, B**	**Dead end diameter, D_2**
10.16 cm	50.08 cm	110 cm	22 cm	22 cm	11 cm	10 cm – 5.0 cm

B. CFD model

In the CFD analysis, a model of the manifold with uniform longitudinal section was prepared. The schematic of the geometry used in the analysis is as shown in Figure 3. Later, the simulation was performed to develop a manifold design to achieve nearly uniform flow distribution through the outlets. The geometry of manifold with tapered longitudinal section is shown in Figure 3. The manifold diameter ratio (D_1/D_2) is varied parametrically to estimate the optimal tapered ratio and uniform flow distribution. In the present problem, the fluid flow is three-dimensional; that is, all three possible velocity components (x, y, and z) exist, and all three components depend on the three coordinates of Cartesian geometry. The statement of the governing equations for the fluid flow being considered here amounts to writing a set of four partial differential equations.

Conservation of mass:

$$\frac{\partial \mathrm{u}}{\partial \mathrm{x}}+\frac{\partial \mathrm{v}}{\partial \mathrm{y}}+\frac{\partial \mathrm{w}}{\partial \mathrm{z}}=0 \tag{1}$$

x-momentum

$$\mathrm{P}\left[\frac{\partial}{\partial \mathrm{x}}(u^2)+\frac{\partial}{\partial \mathrm{y}}(uv)+\frac{\partial}{\partial \mathrm{z}}(uw)\right]=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial u}{\partial z}\right) \tag{2}$$

y- momentum

$$\mathrm{P}\left[\frac{\partial}{\partial \mathrm{x}}(vu)+\frac{\partial}{\partial \mathrm{y}}(v^2)+\frac{\partial}{\partial \mathrm{z}}(\mathrm{vw})\right]=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial v}{\partial z}\right) \tag{3}$$

z-momentum

$$\mathrm{P}\left[\frac{\partial}{\partial \mathrm{x}}(wu)+\frac{\partial}{\partial \mathrm{y}}(wv)+\frac{\partial}{\partial \mathrm{z}}(\mathrm{w}^2)\right]=-\frac{\partial p}{\partial z}+\frac{\partial}{\partial y}\left(\mu_{eff}\frac{\partial w}{\partial y}\right)+\frac{\partial}{\partial x}\left(\mu_{eff}\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial w}{\partial z}\right) \tag{4}$$

Where u, v, w are the velocity components in three dimensions respectively, ρ is the fluid density and the effective viscosity, μ_{eff}, is defined as $=\mu+\mu_{eff}$. The turbulent viscosity is depends on the selected turbulence model as well as on the specific application. in the present study the Realizable k–e model was chosen for application here[15][18]. The simulation of two geometries was done using a commercial CFD software FLUENT. The design, meshing and

boundary definition of the geometries were done using the presolver software, GAMBIT. Tet/Hybrid T-grid scheme was used for the mesh generation [16]. The grid elements in each geometrical model were approximately 2,000,000 elements. Grid independence test was carried out to determine the best mesh spacing for the geometrical model. the solutions are considered to be converged when all of the residuals for the continuity and momentum equations are less than or equal to 10-5 . The boundary conditions used for the simulation are shown in Table 2.

Table 2: Boundary condition for two manifolds

	Reynolds number	Inlet flowrate L/min	Inlet water temp. °C	Outlet gage pressure
Test,1	$10x10^4$	579	20	Zero
Test,2	$15x10^4$	739	20	Zero
Test,3	$20x10^4$	997	20	Zero

RESULTS AND DISCUSSIONS

A . Numerical results: A numerical model was developed in this study to:

1. Determine the flow distribution and pressure drop at the parallel pipes and to validation the result with the data obtained from experimental setup.
2. Determine the optimum design of the tapered manifold that can give uniform water distribution through changing the diameter ratio (D_1/D_2) parametrically.

CFD simulation was first performed on manifold with uniform longitudinal section having circular diameter of 10.16 cm (4 in) and straight flow with outlets of constant cross-sectional areas. The axial momentum would progressively decrease. This would give rise to the static pressure from the entrance to the manifold dead end. Such an increase in static pressure should favour a higher efflux through the downstream outflows. Figure 4 represents the static pressure contour for circular cross-section manifold with Reynolds number (Re=150,000). It can be clearly seen from Figure 4 that the pressure along the manifold is increasing which resulting in non-uniformity flow. The flow velocity decreases in the flow direction due to fluid loss into laterals creating momentum deficiency along the dividing flow header, as shown in Figure 5.

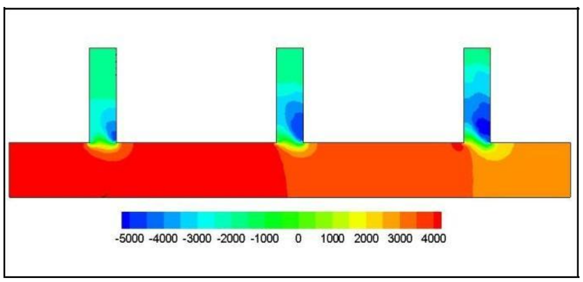

Figure 4: Pressure contour for flow in manifold with uniform longitudinal section

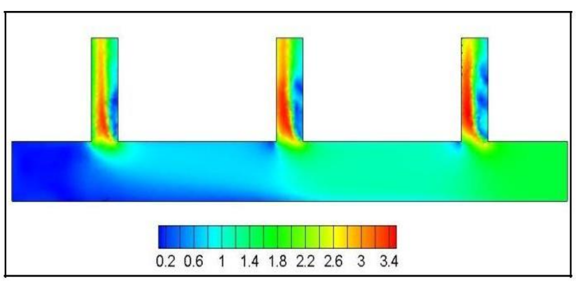

Figure 5: velocity contour for flow in manifold with uniform longitudinal section

For counting the flow distribution among the parallel tubes, the dimensionless parameters, β_i and Φ are used for evaluating the flow distribution. Their definitions are given as following [16, 19]:

$$\beta_i = \frac{Q_i}{\sum_{i=1}^{n} Q_i / N} \quad (5)$$

$$\Phi = \sqrt{\frac{\sum_{i=1}^{n}(1-\beta_i)^2}{N}} \quad (6)$$

where Φ the non-uniformity, denotes the flow ratio for ith pipe, represents volume flow rate for ith pipe (m^3/s), and N is the number of total pipes in the parallel flow. The smaller value of Φ indicates better distribution. For this reason, the minimum value of non-uniformity coefficient will give the optimum configuration for the taper manifold.
From Figure 6, the values of non-uniformities (Φ) of tapered manifold are 0.250, 0.226, 0.222, 0.20, 0.14 and 0.182 at diameter ratio of 1, 1.1, 1.25, 1.42, 1.66, 2, and 2.5, respectively. The (Φ) values are lower than circular cross-section manifold with corresponding value of 0.25 as shown in Figure 6. Then the dead end diameter is reduced gradually from 10.16 cm (4 in) to 5.08 cm (2 in), the flow distribution is generally improved. The non-uniformity (Φ) decreases until it reaches a minimum value (optimal design) then it start to increase although the diameter ratio was increased too as shown in figure. Figures 7 and 8 shows the pressure and velocity contour respectively for tapered manifold for optimum tapered ratio (D_1/D_2 =2). The pressure and velocity along the manifold was found to be nearly uniform which resulted in a better flow distribution through the outlets.

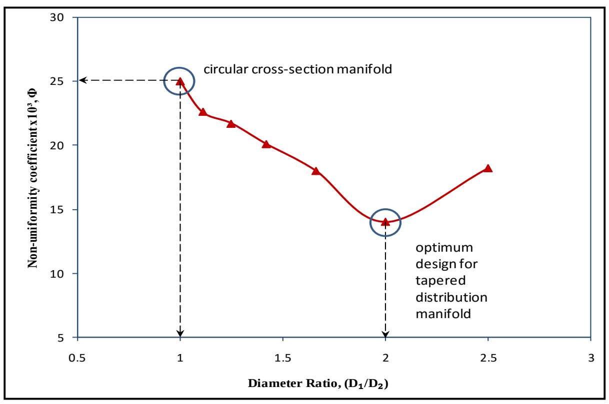

Figure 6: the non uniformity coefficient (Φ) for different diameter ratio

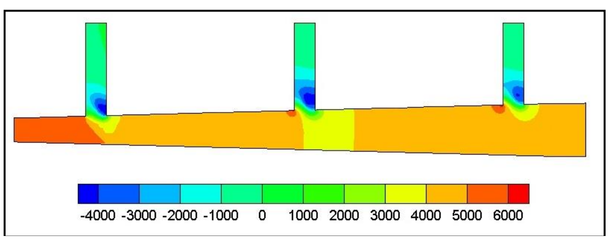

Figure 7: Pressure contour for flow in manifold with tapered longitudinal section

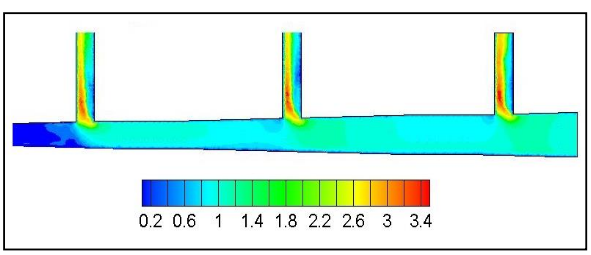

Figure 8: velocity contour for flow in manifold with uniform longitudinal section

B. Experimental results

Figure 10 shows the flow distribution plots for manifold with uniform longitudinal section for three values of Reynolds

number (100,000, 150,000, and 200,000). The flow through the first outlet was found to be very small compared with the last outlet as shown by pressure contours (Figure 4). Uniform flow distribution through the manifold with tapered longitudinal section can be achieved using the design obtain from the numerical model. Figure 10 represents the flow distribution from manifold with tapered longitudinal section having inlet diameters of 10.16 cm (4 inch) and dead diameter of 5.08 cm (2 in). The improvement of flow distribution through the outlets is compared to that obtain from circular cross-section manifold as shown in Figures 9 and 10.

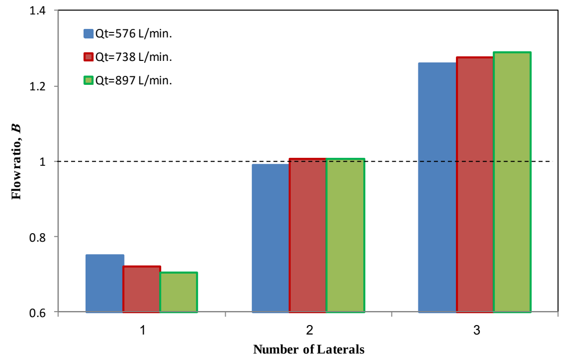

Figure 9: Flow distribution plot for manifold with uniform longitudinal section

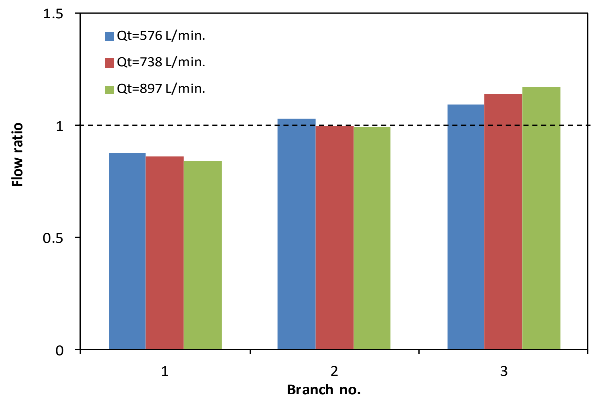

Figure 10: Flow distribution plot for manifold with tapered longitudinal section

The non-uniformity flow coefficient (Φ) was taken as a parameter to quantify the uniformity in flow distribution through the manifold outlet. The Φ can be defined using Equation (6). Flow distribution through the outlets is better at lower (Φ) values. Table 3 shows the non-uniformity coefficient for circular and tapered manifold cross-sections from the values of the Reynolds number, 100,000, 150,000, and 200,000. It can be seen that the flow distribution was severe in case of manifold of circular cross-section. The flow was evenly distributed for the manifold with tapered cross-section.

Table 3: Shows the non-uniformity coefficient for circular and tapered manifold cross-sections

Manifold cross section	The non-uniformity flow coefficient (Φ)	
	Circular	**Taper**
Re=100,000	0.249	0.140
Re=150,000	0.250	0.140
Re=200,000	0.253	0.142

Figures 11 and 12 show that the flow rate fraction of each outlet (which is the rates of outlet to the total flow rate in the manifold). For circular cross-section manifold, the result shows a non-uniform flow smallest flow rate occurred in found the outlet closest to manifold inlet and the highest flow rate found in the last manifold outlet. Let the respective outlets be numbered as (1) which are the first outlet while the last is outlet (3). The discharge from outlet (1) is lower by 37% than the outlet (3). While for the tapered cross-section manifold, the percentage is reduced from 37% to 26%.

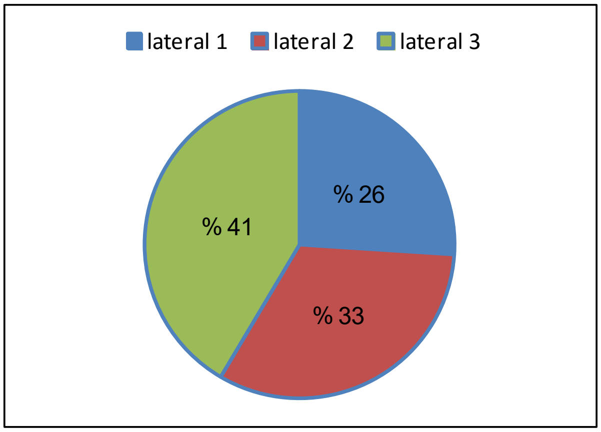

*Figure 11: Flow rate fraction Percentage for manifold of taper cross-section (Re=150,000, (Q=738 L/m**in)***

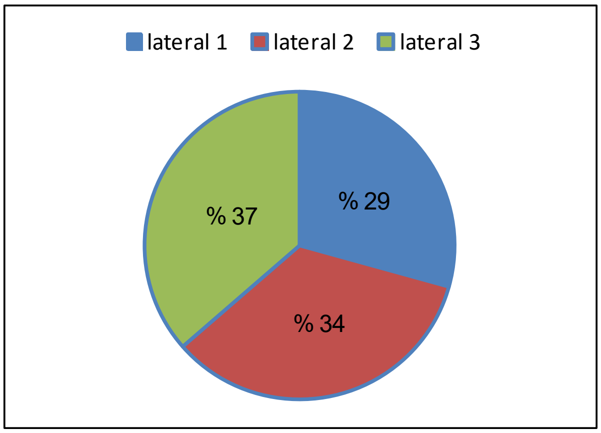

Figure 12: Flow rate fraction Percentage for manifold of circular cross-section (Re=150,000, (Q=738 L/m

VALIDATION OF NUMERICAL RESULTS BY EXPERIMENTAL TESTS

Experimental tests for flow distribution from two manifolds with different configurations have been conducted. The numerical simulation results obtain by using FLUENT which is a CFD package. The experimental test was conduct to measure the flow rate at the 3 outlets. The accuracy of the solution from the FLUENT$^{@}$ CFD package in flow field calculation of the manifolds system is used to determine the optimal design. If the solution from the FLUENT$^{@}$ CFD code cannot reproduce the actual performance of the manifold, this mean that configuration for the taper distribution manifold is not optimum. The first task is thus to demonstrate the accuracy of the numerical solution.

The computed and experimental flow rate distribution per-outlet for Q_{total}= 750 l/min (Re=1500000) are shown in Figures 13 and 14 respectively. It can be clearly seen show that the different in flow rates between computed and measured are acceptable and therefore the validity of present numerical solution is evident.

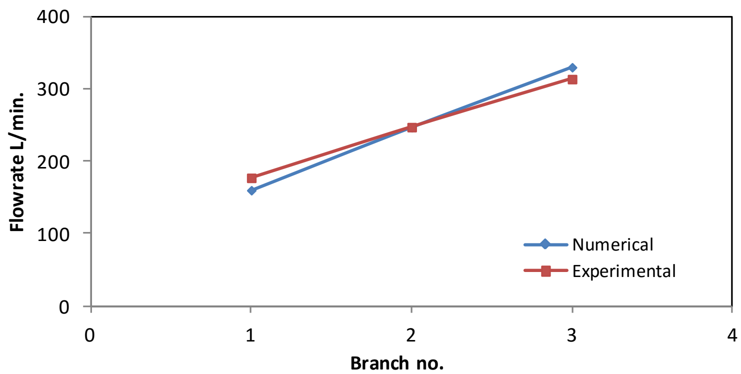

Figure 13: Flow distribution per outlet for manifold with uniform longitudinal section (Re=150,000, Q=738 l/m)

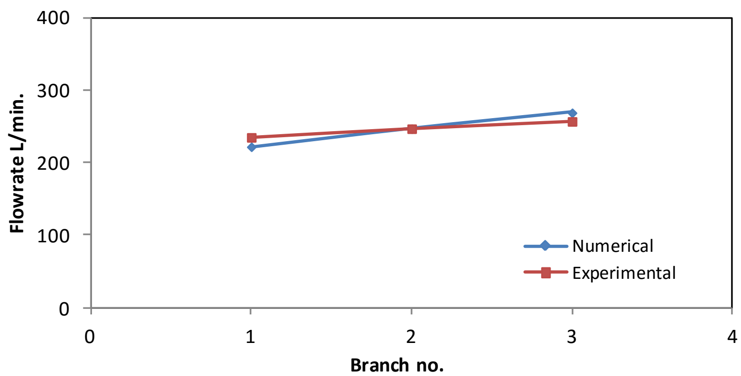

Figure 14: flow distribution per outlet for manifold with taper longitudinal section (Re=150,000, Q=738 L/m)

CONCLUSIONS

The main aim of this investigation is to evaluate the hydraulic parameters of manifold in order to use it in the design to get the same outflow rate from manifold outlets. The CFD simulation and experimental data from different outlets and configurations namely; circular and tapered cross-sections were compared. Severe maldistribution was found for flow rates from manifold outlets with circular cross-section whereas the flow rates from manifold outlets with tapered cross-section were found nearly uniform (almost equally distributed). A numerical model was used to predict the flow rate from each manifold lateral for three different Reynolds numbers (i.e., 100,000, 150,000 and 200,000) and the results were found with the same trend compared with experimental data. It is found that Reynolds number have slight effect on the uniformity of the mass effusion from the manifold outlets, so it can be concluded that the flow distribution from manifolds laterals is independent on Reynolds number.

ACKNOWLEDGMENTS

The work was carried out in hydraulic lab of Department of Civil Engineering, University putra Malaysia (UPM). This work has been a part of the project supported by Ministry of Higher Education and Scientific Research, Iraq.

REFERENCE

[1] Gandhia, M. S. at al., 2012, "CFD simulation for steam distribution in header and tube assemblies" chemical engineering research and design Vol. 90, pp. 487-506.
[2] Bajura, R. A., 1971, "A model for flow distribution in manifolds," Journal of Engineering for Power-Transactions ASME, Vol. 93, No. 1, .June, pp. 7-12.
[3] Bajura, R. A. and E. H. Jones, 1976, "Flow distribution manifolds," Journal o f Fluids Engineering-Transactions

ASME, Vol. 98, No. 4, December, pp. 654-666

[4] Majumdar, A.K., 1980, "Mathematical modeling of flows in dividing and combining flow manifold" Appl. Math. Model, Vo. 4, pp.424–432.

[5] Bassiouny, M.K., Martin, H., 1984, "Flow distribution and pressure drop in plate heat exchangers-I U-type arrangement". Chem. Eng. Sci. Vo. 39, pp. 693–700.

[6] Bassiouny, M.K., Martin, H., 1984. "Flow distribution and pressure drop in plate heat exchangers-II Z-type arrangement". Chem. Eng. Sci. Vo. 39, pp. 701–704.

[7] Choi, S. H., Shin, S. and Cho, Y. I.,1993, "The effect of area ratio on the flow distribution in liquid cooling module manifolds for electronic packaging," Int. Comm. Heat Mass Transfer, Vol. 20, pp. 221-234.

[8] Choi, S. H., Shin, S. and Cho, Y. I., 1993, "The effect of Reynolds number and width ratio on the flow distribution of liquid cooling module manifolds for electronic packaging," Int. Comm. Heat Mass Transfer, Vol. 20, pp. 607-617.

[9] S. Kim, E. Choi, Y.I. Cho, 1995, "The effect of header shapes on the flow distribution in a manifold for electronic packaging applications", International Communications in Heat and Mass Transfer, Vo. 22, pp. 329-341.

[10] Jiao, R. Zhang, S. Jeong, 2003, "Experimental investigation of header configuration on flow mal-distribution in plate-fin heat exchanger", Applied Thermal Engineering, Vo. 23, pp. 1235-1246.

[11] Wen, J., Li, Y., Zhou, A., Ma, Y., 2006, "PIV investigations of flow patterns in the entrance configuration of plate-fin heat exchanger". Chin. J. Chem. Eng. Vo. 14, pp. 15–23.

[12] J.C.K. Tong, E.M. Sparrow, J.P. Abraham, 2009, "Geometric strategies for attainment of identical outflows through all of the exit ports of a distribution manifold in a manifold system", Applied Thermal Engineering, Vo. 29, pp. 3552-3560.

[13] Pan, M., Zeng, D., Tang, Y., & Chen, D., 2009, "CFD-based study of velocity distribution among multiple parallel microchannels", Journal of Computers, Vo. 4, pp. 1133-1138.

[14] Mathew, B., John, T. J., & Hegab, H., 2009, "Effect of manifold design on flow distribution in multichannel microfluidic devices", In 2009 ASME Fluids Engineering Division Summer Conference, Vol. 2, pp. 543–548.

[15] Andrew W. Chen, Ephraim M. Sparrow, 2009 "Effect of Exit-port Geometry on the Performance of a Flow

Distribution Manifold", Applied Thermal Engineering, Vol. 29, pp. 2689-2692.

[16] V. V. Dharaiya at el., 2009, "evaluation of a tapered header configuration to reduce flow maldistribution in minichannels and microch-annels" Proceedings of the ASME 2009 7th International Conference on Nanochannels, Microchannels and Mini-channels.

[17] J.C.K. Tong, E.M. Sparrow, J.P. Abraham, 2007, "Attainment of flowrate uniformity in the channels that link a distribution manifold to a collection manifold", J. Fluids Eng. Vo. 129, pp. 1186–1192.

[18] Andrew Chen and Ephraim M. Sparrow 2009, "Turbulence modeling for flow in a distribution manifold" International Journal of Heat and Mass Transfer, Vo. 52, pp. 1573–1581.

[19] C.C. Wang, K.S. Yang, J.S. Tsai, I.Y. Chen, 2011, "Characteristics of flow distribution in compact parallel flow heat exchangers, part I: typical inlet header", Appl. Therm. Eng. Vo. 31, pp. 3226-3234.