Scan to know paper details and author's profile

CFD Analysis to Study Single Phase Steam Distribution in a Dividing Manifold for Higher Pressures: Effect of Distributor to Reduce Maldistribution

Arijit Ganguli & Mathpati C. S.

Ahmedabad University

ABSTRACT

Fluid distribution in various geometries like plate heat exchangers, spargers, fuel cells, solar collectors etc. is very important for improved performance of these equipment's. Parameters like the header to tube ratio and the ratio of areas of the headers is extremely important and has been studied by many researchers for a variety of equipment's. We consider a slightly different geometry from the conventional ones. The present case considers entry of steam (feed) at the center of top header (in the header tube assembly) and leaves from the center of the bottom header. The tubes are uniformally distributed along the top and bottom headers such that the central tube is exactly below the feed header. Experimental measurements were performed with air as a working fluid for the base geometry (existing geometry). For experimental measurements, inlet mass flowrates ($990 < Re < 10^6$) were varied for a fixed operating pressure (P = 1 atm) . Three dimensional (3D) CFD simulations were performed for the above geometry (all other conditions and working fluid remaining the same) and the predictions were validated against the experimental data. The model predictions were in a good agreement with the experimental data (~15% deviation). The validated model was used to perform 3D simulations ($1 \leq P \leq 40$ bars) and different inlet flowrates (i.e. Reynolds numbers in the range of $10^4 < Re < 3x10^6$) with steam as working fluid for the base geometry.

Kyewords: steam distribution, heat transfer, cfd, extent of non-uniformity, distributor design, mal-distribution.

Classification: FOR Code: 091399

Language: English

London Journals Press

LJP Copyright ID: 392821
Print ISSN: 2631-8474
Online ISSN: 2631-8482

London Journal of Engineering Research

Volume 20 | Issue 1 | Compilation 1.0

CFD Analysis to Study Single Phase Steam Distribution in a Dividing Manifold for Higher Pressures: Effect of Distributor to Reduce Maldistribution

Arijit Ganguli[α] & Mathpati C. S.[σ]

ABSTRACT

Fluid distribution in various geometries like plate heat exchangers, spargers, fuel cells, solar collectors etc. is very important for improved performance of these equipment's. Parameters like the header to tube ratio and the ratio of areas of the headers is extremely important and has been studied by many researchers for a variety of equipment's. We consider a slightly different geometry from the conventional ones. The present case considers entry of steam (feed) at the center of top header (in the header tube assembly) and leaves from the center of the bottom header. The tubes are uniformly distributed along the top and bottom headers such that the central tube is exactly below the feed header. Experimental measurements were performed with air as a working fluid for the base geometry (existing geometry). For experimental measurements, inlet mass flowrates ($990 < Re < 10^6$) were varied for a fixed operating pressure (P = 1 atm) . Three dimensional (3D) CFD simulations were performed for the above geometry (all other conditions and working fluid remaining the same) and the predictions were validated against the experimental data. The model predictions were in a good agreement with the experimental data (~15% deviation). The validated model was used to perform 3D simulations ($1 \le P \le 40$ bars) and different inlet flowrates (i.e. Reynolds numbers in the range of $10^4<Re<3x10^6$) with steam as working fluid for the base geometry. Extent of non-uniformity (ENU) was studied for these cases as a function of Reynolds number for the pressure range considered. It was observed that the %ENU increased with increase in Reynolds number as well as pressures. Different distributor configurations were tried and the distributor plate with least %ENU was selected. 3D simulations have been performed for different Reynolds numbers (in the range of $10^4<Re<3x10^6$) and pressures (1 bar < P < 40 bar). Significant decrease in %ENU <5% is observed due to the distributor plate.

Keywords: steam distribution, heat transfer, cfd, extent of non-uniformity, distributor design, mal-distribution.

Authorα: School of Engineering and Applied Sciences, Ahmedabad University, Navrangpura, Ahmedabad, 380009, India.

σ: Department of Chemical Engineering, Institute of Chemical Technology, Matunga, Mumbai - 400019, India.

I. INTRODUCTION

Uniformity in flow distribution in any geometry (sparger, different manifolds: dividing, combining, parallel and reverse) needs proper balance between the pressure recovery and the frictional pressure drop (Acrivos et al., 1959). Thus, for a given mass flow rate, the flow distribution depends upon header diameter, number and tube geometry (diameter of pipes, diameter of holes, pitch, etc.). During the past 50 years, although flow distribution in plate and frame exchangers, PEMFCs (Proton Exchange Membrane Fuel Cells) have received some attention, other applications of pipeline networks which are used in practice (air distribution in diffuser system of aerobic biological treatment, steam distribution in passive decay heat removal systems, etc.) have received comparatively less attention. Such arrangements are found in industrial equipment's (heat exchangers) where the flow is distributed from a main header to a number of tubes. Substantial research has been reported on this problem by researchers over the past few decades (Acrivos et al., 1959; Bassiouny and Martin, 1984a,b; Choi et al. 1993a,b; Jiao et al., 2003; Lalot et al., 1999; Majumdar, 1980; Mohan et al. 2004; Mueller and Chiou, 1988; Pigford et al., 1983; Wen and Li, 2004; Zang and Li, 2003). In such cases, the sudden changes in flow direction make the pressure rise in the top header and fall in the bottom header (Gandhi et al. 2011). This pressure distribution leads to non-uniformity in the flow pattern and the flow in the channels will consequently increase in the direction of the inflow. Bassiouny and Martin, (1984 a, b) derived expressions for predicting the velocity distribution, flow distribution, and pressure drop in such systems. The quantification of mal-distribution was done on the value of 'm^2' (where 'm' is a parameter which depends on the diameter ratio and area ratio of the larger pipe (the header) and the individual tubes as in a pipe tube assembly) for all the three derivables. For uniform flow distribution, the value of m^2 should be 1. They concluded that for large positive or negative values of m^2 the flow through some channels were practically absent giving rise to extensive mal-distribution. Lalot et al. (1999) carried out extensive investigations on the change in flow distributions in electrical heaters, heat exchangers and condensers. They found that the efficiency of the equipment depended enormously on the flow distribution. Further, they concluded that the flow distribution improves with an increase in pressure drop. Lalot et al. (1999) also recommended that placing a perforated plate as internal in the top header. They concluded that placing internal results in an increase in drag coefficient which decrease the maximum velocity by half. A detailed review of the work on flow distribution can be found in Gandhi et al. (2011). Literature review depicted that all the work reported in the literature has been carried out with air or water as working fluid. In certain applications (Nuclear safety) vapors may also be used as working fluids. Gandhi et al. (2011) have carried out extensive simulations and experimental work for different geometries where the inlet is from the middle rather than from the sides as in conventional headers. Gandhi et al. (2011) considered different combinations of header diameter to tube diameter ratio's variation of tube pitch and variation of number of tubes. Different configurations were considered to achieve reduction in non-uniformity. Gandhi et al. (2011) concluded that the flow distribution can be improved without internals just by changing the magnitude of header and tube diameters.

Gandhi et al. (2011) have carried out simulations at high pressures and eliminated the middle tube for achievement of uniform flow distribution. However, in some situations where the geometry is such that the middle tube is exactly beneath the feed pipe then the use of internals may help in achieving near uniform distribution. The present study has been carried out to see if by introducing a distributor as an internal one can reduce mal-distribution and what would be the pressure drop in that case. In the present study, a geometry similar to the one of Gandhi et al. (2011) is considered (with flow from the center of the header and distribution in the tubes). The study is an attempt to see the effect of addition of internals to above mentioned geometry. Steam as working fluid at pressures (in the range $1\text{atm} \leq P \leq 40\text{atm}$) and Reynolds number (in the range of $10^4 < Re < 3x10^6$) have been considered. CFD simulations have been performed for two configurations: The original existing geometry and the geometry with a distribution plate. The pressure distribution, velocity distribution and flow patterns have been presented. The configuration with distributor has been seen to perform well and reduction in mal-distribution upto 90% has been obtained. However, an increase in pressure drop upto 0.15 bar is observed for higher pressures (40 bar).

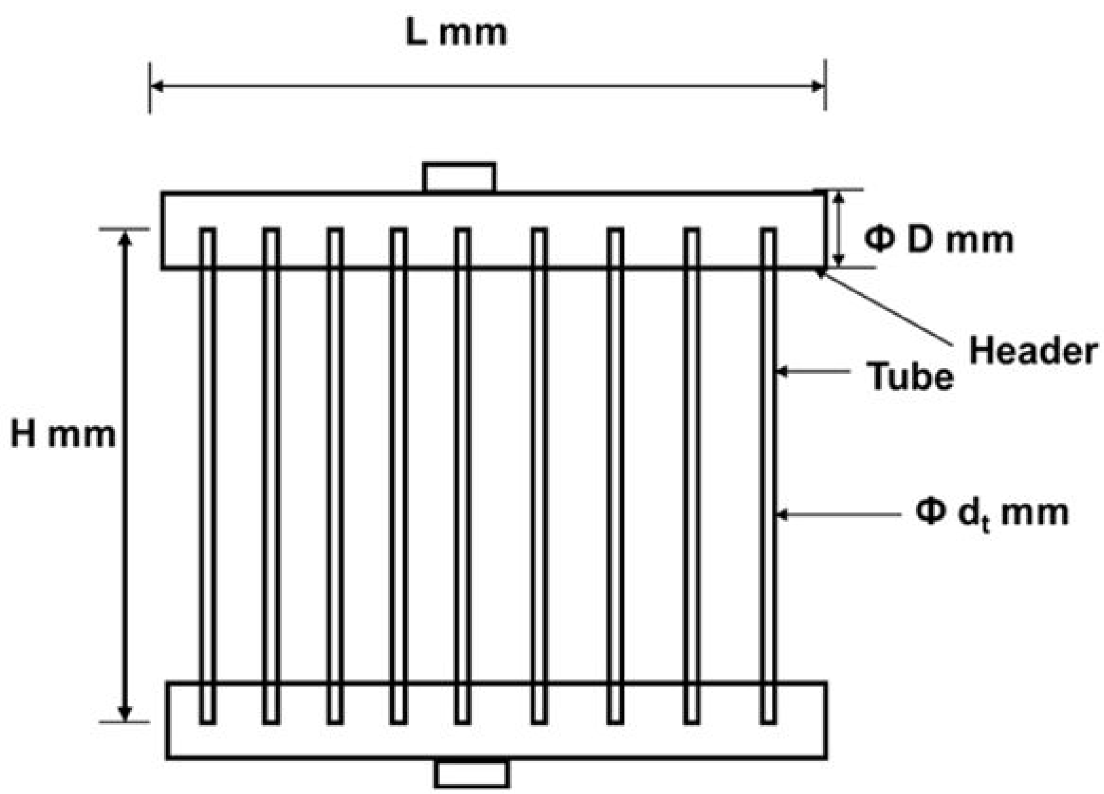
While the work of Gandhi et al. (2011) is a comprehensive work the mechanical issues related to installation might be challenging and investigation on distributors in configuration A should be considered to understand the advantages of such installations.

II. PROBLEM DEFINITION

A tube header assembly with top and bottom header having an inlet and outlet respectively and tubes vertically connected has been considered for the present investigation. The pitch between the tube centers is considered to be L=1.125H the header length being L. The height of the tubes is considered to be H, such that D = 0.1875H. The fluid enters at certain pressure (in range $1 \leq P \leq 40$ atm) from the inlet and departs at the outlet. Due to a high diameter ratio between the tube header and the tubes the position of the inlet poses a problem. The problem mainly demands minimum maldistribution while keeping pressure drop minimal. The inlet being located at the center first the flow distribution for the current geometry is investigated. On understanding the extent of maldistribution, a distributor design in the top header has been planned.

III. GEOMETRY AND GRID DETAILS

The geometry chosen for this system is a tube-header assembly consists of the top and the bottom headers which are assumed to have diameters D while the tubes have diameters d_t. The ratio d_t/D is assumed as 0.2. The inlet and outlet diameters are identical to the header diameters. Two configurations with different internals have been chosen. All the configurations with grids have been shown in Figure 1.



A three dimensional grid has been considered in the study. A tetrahedral mesh has been created both for header and tube assembly with and without distributor. For the geometry without the distributor the mesh size is 187731 tetrahedral cells with fine cells inside the tubes and walls and uniform mesh at the headers. The grid independency for the geometry without the distributor was carried out with three different grids namely 187731, 268338 and 496273 and the centreline axial

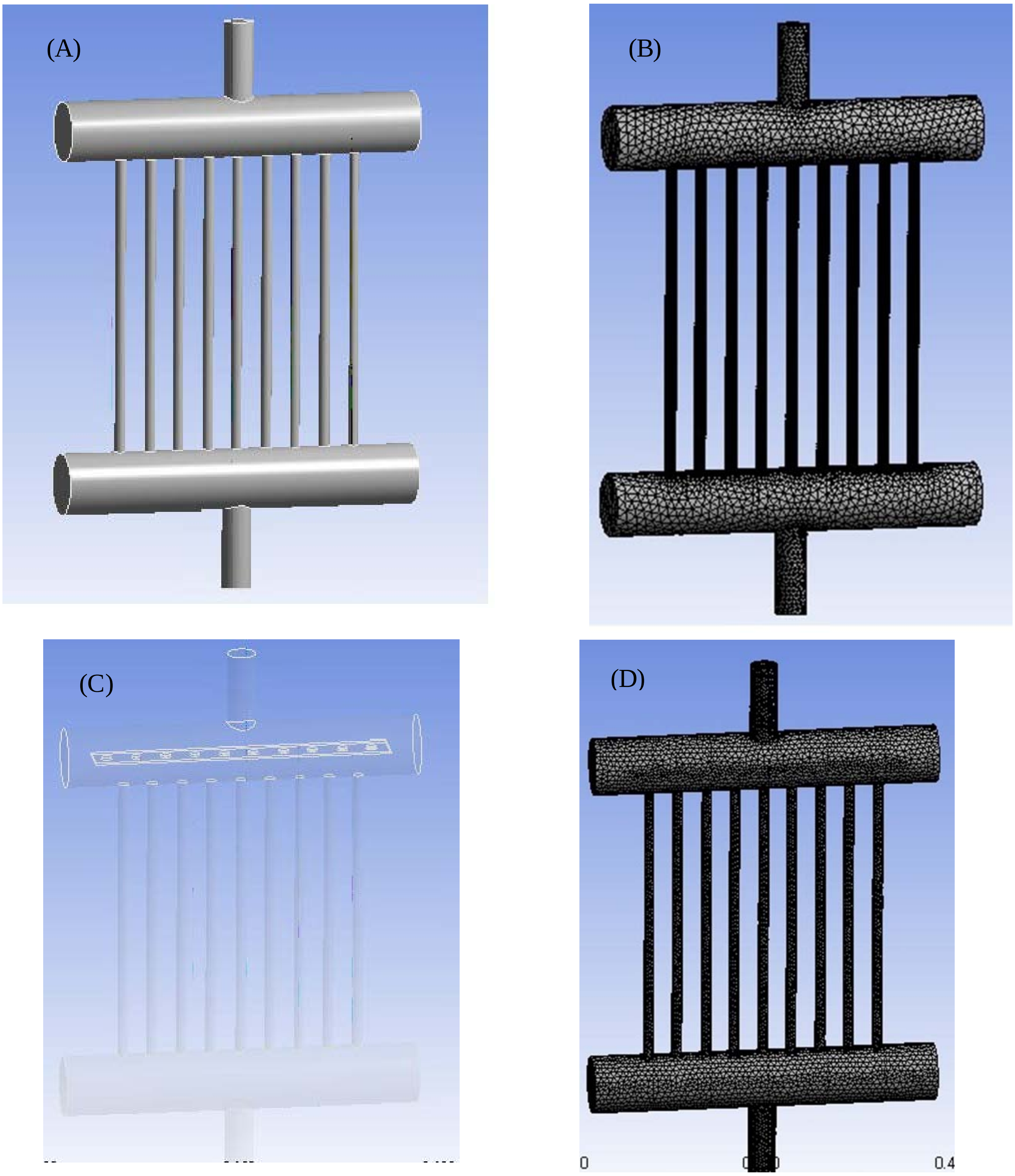


Figure 2: Geometry and mesh of header tube assemblies (A) Base case geometry (B) Mesh for the base case (C) Geometry of the assembly with newly designed distributor (D) Mesh for the header-tube assembly with distributor

velocity was checked. Since the difference between the magnitude of centerline axial velocity was 3% (Figure 3) for the 268338 and 496273 grids, 268338 grids was selected for investigation. Similarly, different grids were selected for the configurations for distributor and header tube assembly. Three different grids namely 261995, 348821 and 522166 and the centerline axial velocity was checked. Since deviation was nearly 2% (Figure 3) between 348821 and 522166 grid assemblies 348821 was selected for simulations.

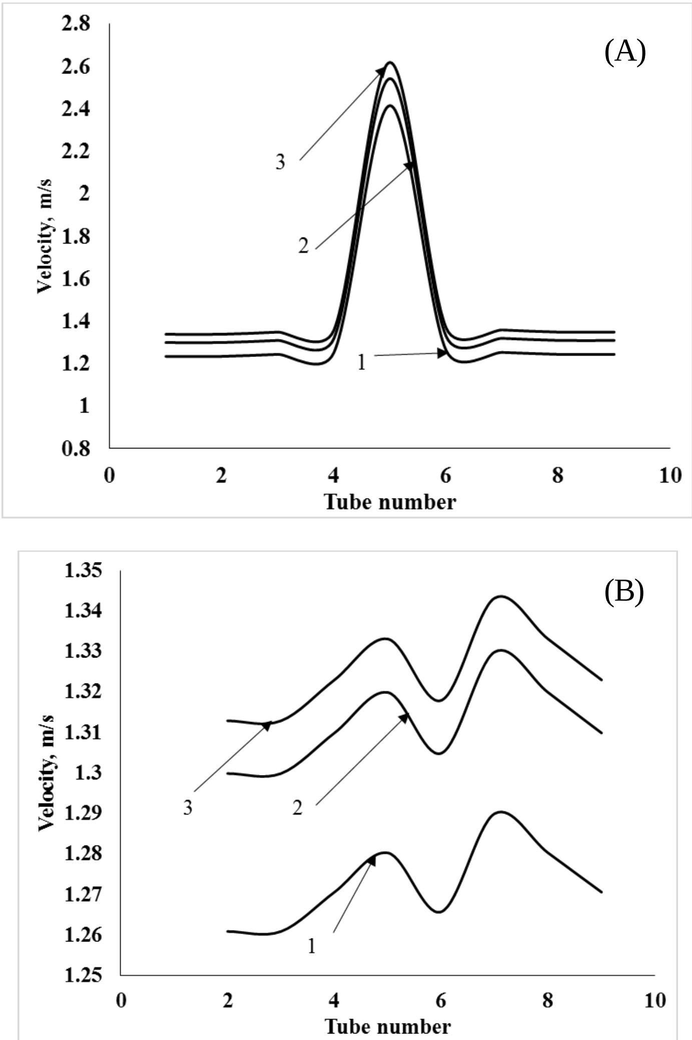


Figure 3: Grid sensitivity (A) Grid sensitivity for base case. Re = 704000 P = 40 atm 1. 187731 2. 268338 3. 496273 (B) Grid sensitivity for geometry with distributor Re = 704000 P = 40 atm 1. 261995 2. 348821 3. 522166

IV. EXPERIMENTAL WORK

The experimental set-up of Gandhi et al. 2011 has been utilized for carrying out experimental measurements. The geometry used is 9 tube geometry similar to Figure 1. Air has been used as working fluid at atmospheric pressure. Experiments were performed for a set of Reynolds numbers (in the range 35,600 < Re < 68,500). The air is passed through a filter to remove any particulate matter present in air. Filtered air is compressed with the help of reciprocating compressor. The air then passes through a rotameter and enters the test section. The air flowrate is maintained by the needle valve. The pressure in the measuring valve is measured by various measuring taps provided in the top header with the help of a U-tube manometer filled with mercury as the manometric fluid. Pressure taps are also used to measure pressures at the outlet of the tubes while mass flowmeters were used to measure the flow rate of air past each tube. All the experiments have been performed thrice and reproducibility has been found to be within 8-10%.

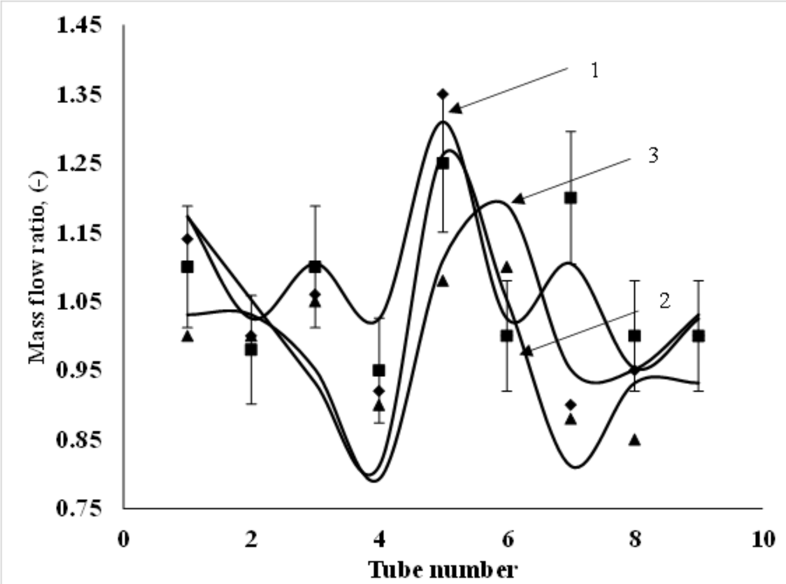


Figure 4: Variation of mass flow ratio with tube number for air as working fluid 1. Re = 995 2. Re =69700 3. Re = 177000 Symbols represent experimental points while lines denote CFD simulations

V. GOVERNING EQUATIONS

The basic governing equations of continuity and momentum in Cartesian co-ordinates have been used. The k-ε turbulence model has been used for modeling the turbulence. The commercial software Ansys FLUENT 17 has been used.

VI. METHOD OF SOLUTION

All the computational work has been carried out using the commercial software Ansys FLUENT 17. Second Order Upwind discretization scheme was used for the pressure, velocity, k and ε equations. All the discretized equations were solved in a segregated manner with the PISO (Pressure Implicit with Splitting of Operators) algorithm. In the present work, steady simulations were performed. The under-relaxation are set to 0.3 for pressure, 1 for density and body forces, 0.7 for momentum and 0.8 for turbulence parameters. The turbulence boundary conditions have been formulated similar to Gandhi et al. 2011. All the solutions were considered to be fully converged when the sum of residuals was below 10^{-4}. All the computations have been performed on an intel i-7,machine with a quad-core processor with 2 GB RAM, 2.4 GHz processor speed.

VII. RESULTS AND DISCUSSION

In this section, we first present the pressure and velocity distribution both qualitatively and quantitatively. By incorporating proper internals the distributions are revisited.

7.1 Model Validation

The model predictions were compared with the experimental measurements as the ratio of individual mass flow rate to inlet mass flow rate for the existing (base) geometry (as shown in Figure 4). The trends of mass flow rate higher in the middle tube while lower in other tubes was shown both in experiments and the model. However, the model under-predicts mass flow ratios for higher Reynolds numbers with a deviation of 12-14%. This is partly attributed to experimental uncertainty in measurements (8%).

7.2 Flow patterns

In this section we try to understand the effect of the newly designed distributor on the flow distribution and pressure drop.

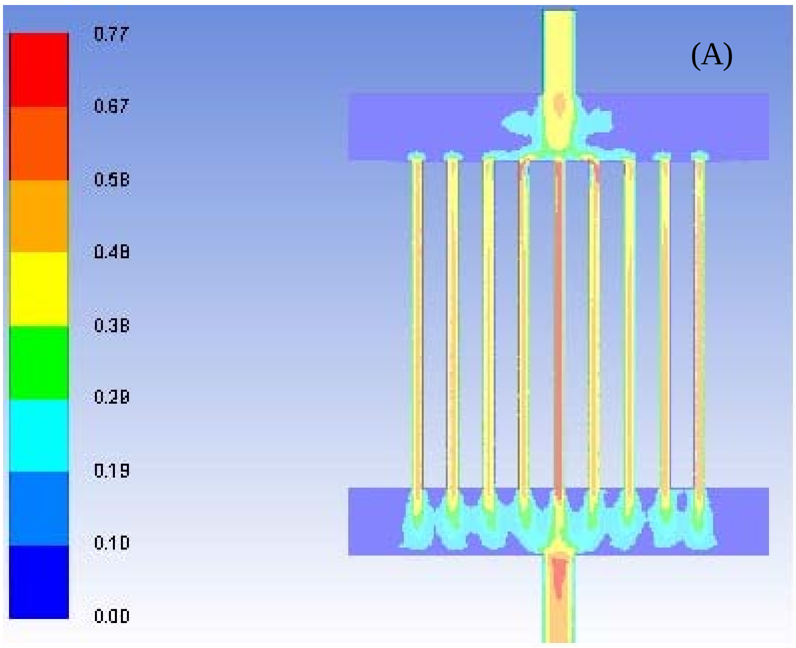
7.2.1 Without distributor

Figure 5 shows the velocity contours for two different Reynolds numbers (Re = 3.58×10^4 and Re= 6.75×10^5) for P = 40 atm. It can be observed (as shown inthe figure), that the majority of the fluid passes through the centreline of the tubes. Further, the flow with higher velocity (~ range) passes through the central tube while the fluid gets distributed through other tubes at lower velocities in both cases.

7.2.2 With Distributor

Figure 6 shows the velocity contours for two different Reynolds numbers (Re = 3.58×10^4 and Re= 6.75×10^5) for P = 40 atm but with distributor. Clearly, the contours show the uniformity in distribution.

A small analysis for the present case with the parameter developed analytically by Bassiouny and Martin (1984 a,b) (‘m^2’) shows important observation. The fraction of mean velocity going into the tubes and ratio of the areas of the top and bottom headers are the most important in determining the distribution. The parameter takes negative and positive values depending on the above two factors. For values greater than 1, the pressure drop in the channels



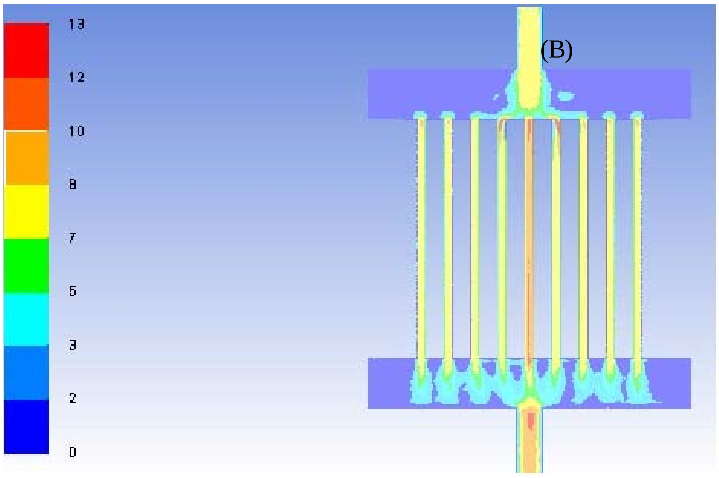


Figure 5: Velocity contours for two different Reynolds numbers for without distributor case and P = 40 bar (A) Re = 14100 (B) Re = 141000

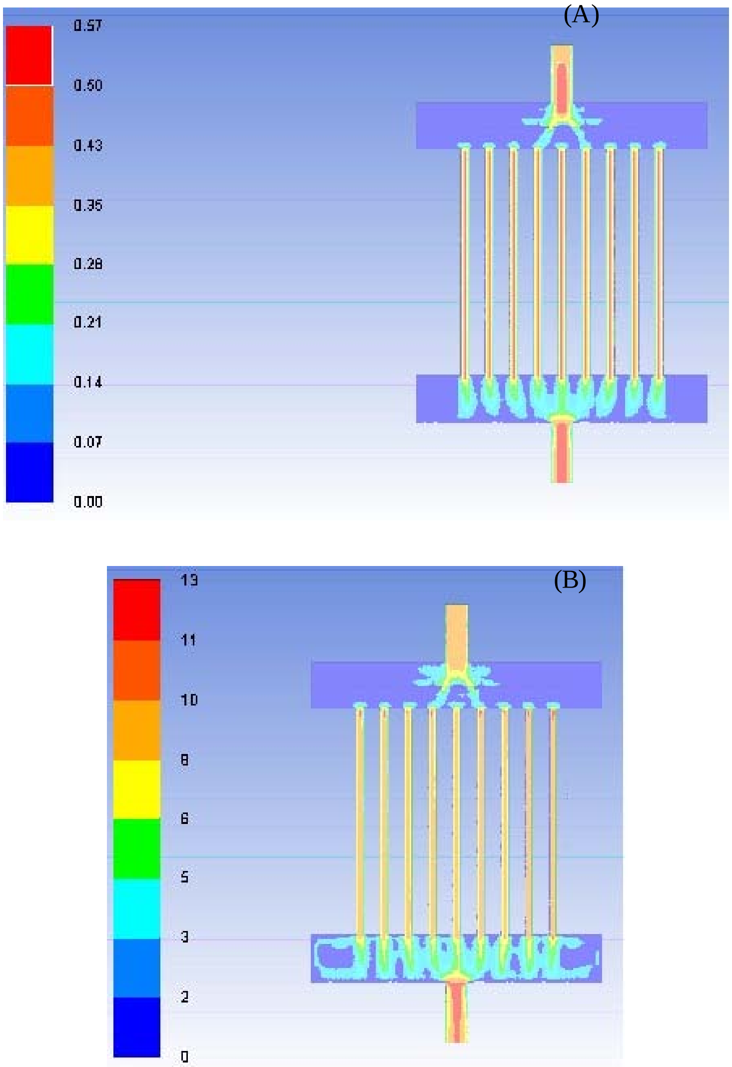


Figure 6: Velocity contours for two different Reynolds numbers for with distributor case and P = 40 bar (A) Re = 423000 (B) Re = 2110000

increases as per the analysis of Bassiouny and Martin (1984a). The velocity distribution also reaches uniformity when the values of 'm^2' nears 1. In the present work, we design a distributor in such a way that the area ratio of header to tube are kept constant and the fraction of mean velocity going to the tubes are varied by placing a distributor as an internal.

The case with distributer have been reported in the present work with 'm^2' values of 0.97.

$$m^2 = \frac{2-\beta^*}{2-\beta}\left(\frac{A}{A^*}\right)^2 \tag{1}$$

where, β^* is the average velocity ratio at the inlet and is the average velocity ratio at the outlet, A is the cross sectional area at the inlet while A^* is the cross-sectional area at the outlet and A_c is the cross sectional area of the tubes. The internal distributor is designed as follows: The ratio of hole diameter in the distributor to the tube diameter is kept equal to 1. The pitch between the holes are varied uniformly such that the hole lies between the spacing between two consecutive tubes of the distributor. The extent of non-uniformity is defined similar to Gandhi et al. (2011) as is given as

$$ENU\ (E_i) = \frac{m_{avg} - m_i}{m_{avg}} \tag{2}$$

wherem_i is the mass flow rate of individual tubes while m_{avg} is the average mass flowrate. The ENU reduces from 80% while with B reduces to 10% for pressure of 40 bar (Figure 7). This holds for three different pressures for which simulations have been done (P = 1 bar; P = 10bar and P = 40 bar). Properties of steam is given in Table 1. Important aspect to understand is that the maldistribution reduces to 5% or less for all the pressures. The only aspect is the increase in pressure drop due to the addition of the distributor.

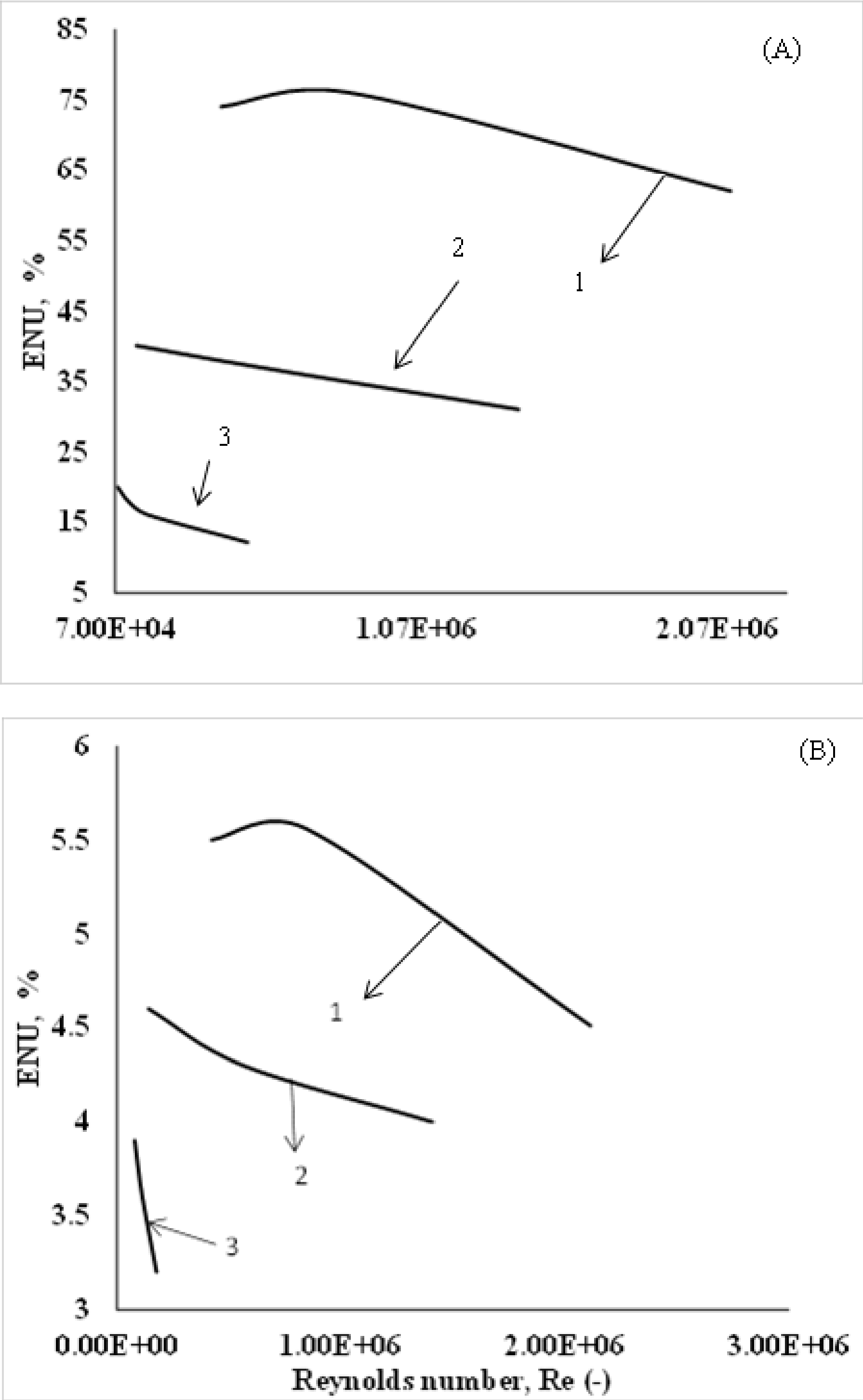


Figure 7: Pressure drop variation as a function of Reynolds number (A) Without Distributor (B) With distributor

Table 1: Properties of steam at different pressures

	Properties		
Pressure, atm	**Density**	**viscosity**	**Diameter**
1	0.60	1.23E-05	0.029
10	5.2	1.50E-05	0.058
40	20.37	1.75E-05	0.058

VIII. CONCLUSIONS

Three dimensional (3D) CFD simulations have been carried out for header tube geometry where inlet and outlet are at the middle of the top and bottom headers. A new distributor has been designed for reduction of maldistribution in a combining-dividing assembly. 3D CFD simulations have been performed for the assembly with and without distributor for different Reynolds numbers. Representative results for system operating at 40 bar pressures show considerable reduction in non-uniformity. For high Reynolds numbers the results are even more encouraging since the flows show very good distribution with %ENU around 5%

High pressure drops may lead to back pressure for equipment's present at the upstream of the equipments (like conpressors etc.). Hence such designs (design with distributors) can only be used where the upstream pressure is higher than the pressure drop specifications so that no damage is made to the upstream equipments.

ACKNOWLEDGEMENT

One of the authors Arijit Ganguli would like to acknowledge the University Grants Comission for the fellowship provided during this research work and thankful to Prof A B Pandit for providing valuable comments for this article

Nomenclature

A is the cross sectional area at the inlet (m^2)

A_c is the cross sectional area of the tubes (m^2)

A^* is the cross-sectional area at the outlet (m^2)

D diameter of the header (m)

d_t tube diameter (m)

ENU extent of non-uniformity

g gravitational constant ($m\ s^{-2}$)

H height of the tube connecting the headers (m)

L header length (m)

m_i is the mass flow rate of individual tubes (kg/s)

m_{avg} is the average mass flowrate (kg/s)

m^2 analytical parameter (-)

n number of tubes (-)

P pressure (atm)

Greek symbols

β^* is the average velocity ratio at the inlet

ρ density of fluid ($kg\ m^{-3}$)

REFERENCES

1. Acrivos, B.D. Babcock, R.L. Pigford. Flow distribution in manifolds. Chem. Eng. Sci. 10, 112-124. 1959
2. M.K. Bassiouny, H. Martin. Flow distribution and pressure drop in plate heat exchangers-I U-type arrangement. Chem. Eng. Sci. 39, 693–700 1984a.
3. M.K. Bassiouny, H. Martin. Flow distribution and pressure drop in plate heat exchangers-II Z type arrangement. Chem. Eng. Sci. 39, 701-704 1984b.
4. S.H. Choi, S. Shin, Y.I. Cho The effects of the Re ynolds number and width ratio on the flow distribution in manifolds of liquid cooling modules for electronic packaging. Int. Comm. Heat. Mass Transfer 20, 607-617 1993a.
5. S.H. Choi, S. Shin, Y.I. Cho. The effect of area ratio on the flow distribution in liquid cooling module manifolds for electronic packaging. Int. Comm. Heat Mass Transfer 20, 221-234 1993b.
6. FLUENT 6.2.16, 2005. User's Manual to FLUENT 6.2.16. Fluent Inc. Centrera Resource Park, 10 Cavendish Court, Lebanon, USA.
7. M.S. Gandhi, A.A. Ganguli, J.B. Joshi, P.K. Vijayan, CFD simulation for steam distribution in header and tube assemblies. Chem. Eng. Res. Des. 90 (4), 487 – 506, 2011.
8. A. A. Ganguli, C. S. Mathpati, M. J. Sathe CFD analysis to study single phase steam distribution in a verticle tube bundle. Int. J. Sci. Prog. Res. 87 (31), 6-9, 2017
9. A. Jiao, R. Zhang, S. Jeong, Experimental investigation of header configuration on flow maldistribution in plate-fin heat exchanger. Appl. Therm. Eng. 23, 1235–1246, 2003.
10. S. Lalot, P. Florent, S.K. Lang, A.E. Bergles, Flow maldistribution in heat exchanger. Appl. Therm. Eng. 19, 847–863, 1999.

11. A.K. Majumdar, Mathematical modelling flows in dividing and combining flow manifold. Appl. Math. Model. 4, 424-432, 1980.
12. G. Mohan, B.P. Rao, S.K. Das, S. Pandiyan, N. Rajalakshmi, K.S. Dhathathreyan,. Analysis of flow maldistribution of fuel and oxidant in a PEMFC. Trans. ASME 126, 262-270, 2004.
13. A.C. Muller, J.P. Chiou, Review of various types of flow maldistribution in a heat exchangers. Heat Trans. Eng. 5, 36-50 1988.
14. R.L. Pigford, M. Ashraf, Y.D. Miron. Flow distribution in piping manifolds. Ind. Eng. Chem. Fundam. 22, 463-471 1983.
15. J. Wang, Theory of flow distribution in manifolds. Chem. Eng. J. 168, 1331–1345 2011.
16. J. Wen, Y. Li Study of flow distribution and its improvement on the header of plat fin heat exchanger. Cryogenics 44, 823-831, 2004.
17. Z. Zhang, Y. Li, CFD simulation on inlet configuration of plate in a heat exchangers. Cryogenics 43, 673-678, 2003.