ISSN (Print) : 0974-6846
Indian Journal of Science and Technology, *Vol 9(48), DOI: 10.17485/ijst/2016/v9i48/108486, December 2016* ISSN (Online) : 0974-5645

A Comparative Study and Flow Analysis of Multiple Branch Pipe Flow Header used in Tube Heat Exchangers

A. Ravinthiran*, N. Sankar, P. Vijaya Rajan and Priyadarshi Dutt

Sri Sairam Engineering College, Chennai, India; ravinthiran24@gmail.com, sankar.mech@sairam.edu.in,
vijayarajan.mech@sairam.edu.in, priyadarshi.mech@sairam.edu.in

Abstract

Objectives: The aim of this paper is to increase the performance of heat exchangers. A flow header having number of small multiple branch pipes are commonly used in boilers and heat exchangers. In the beginning, the headers were designed to give equal flow distribution. But experimentally, it is not equal. In this paper, the uneven distribution of flow in the heat exchangers which affects the performance is rectified. **Analysis Method:** In this project, the flow distribution among the branch pipes of flow header is analyzed for two different geometrical flow nature (square and circular). The flow distribution are analysed with the use of CFD software. After analyzing the square and circular header tubes, the result values are compared with Osakabae's laboratory experimental results. **Result Extraction:** Based on the result values, it is found that header shape will determine and by choosing the better shapes the performance of the Heat Exchanger can be increased and the fuel consumption can be reduced and can increase the total performance of the Boiler. **Applications:** The uniform flow distributions in the flow header definitely increase the performance of Heat exchangers.

Keywords: Branch Pipes, Flow Header and Heat Exchanger, Headers, Non-Uniform

1. Introduction

Flow headers are vital part of many engineering systems like heat exchangers, air conditioning systems etc. The main work of the header is to redistribute the flow into several passages the headers were designed to give equal flow distribution. But experimentally, it is not equal because of flow stress that occurs, anywhere along the length, which results in un-even flow, which so obviously reduces the performance of the heat exchanger[1]. The design of the heat exchanger is to give equal flow in all the ranch pipes. But in practical, the flow is non-uniform because of some parameters like area, branch pipe properties, No of branch pipes, pitch distance etc., and this affects the performances of the heat exchanger[2].

Types of headers in flow Distribution Systems: Headers commonly used in flow distribution systems can be classified as dividing and combining flow headers. These headers are illustrated in Figure 1. In a dividing flow header the main fluid stream is decelerated due to loss of fluid through the lateral pipes. Therefore, pressure will rise in the direction of flow if the effects of friction are less. The frictional effects would cause a decrease of pressure in the direction of flow. Therefore, the possibility exists for obtaining a uniform pressure by suitable adjustment of the flow parameters so that the pressure regain due to the flow distribution balances the pressure losses due to friction[3].

1.1 CFD Model

Using GAMBIT the 3D model for analysis was created and meshed. The internal diameter of the header is 20mm. The dimensions of lateral pipe are 18×1.07×136 mm. The

---

**Author for correspondence*

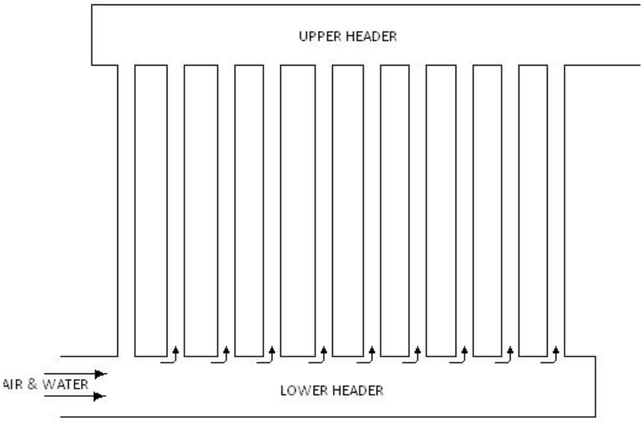

Figure 1. Header-shell and tube heat exchanger.

analysis is made for both square and circular header. The three dimensional model created in gambit is shown in Figure 2 and 3.

2. Existing Conditions

- Cross sectional area of tube (circular header): 0.095 m^2
- Cross sectional area of tube (square header): 0.119 m^2
- No. of lateral tubes: 10 tubes
- Material used: aluminum
- Pressure within the Header inlet: 50-100 KPa
- The inlet mass flux of air/water mixture were 70–165 kg/m2 s
- Inlet air and water velocity: 0.05 m/s and 0.06 m/s respectively

3. Solver Parameters

The solver parameters selected to solve the fluid domain using FLUENT software shown in Table 1.

3ddp-It means that three dimensional fluid domain model is solved by double precision solver and gives the results accurately.

The standard K-ε model is valid only for fully turbulent flows. In our fluid flows domain the Re>4000. So the K-ε model is selected. The standard K-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (K) and its dissipation rate (ε). The model transport equation for K is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart[4].

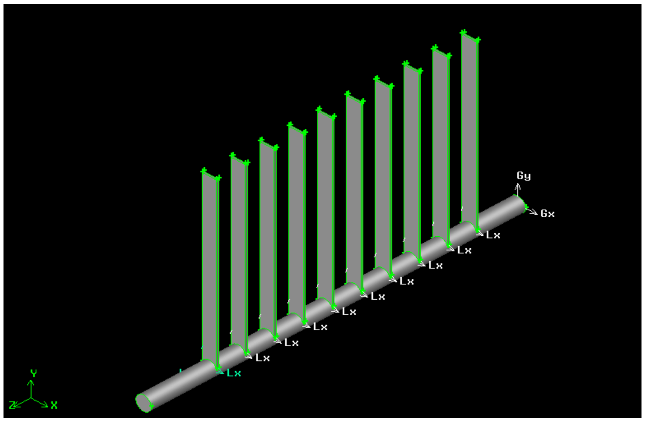

Figure 2. Three dimensional model of circular header.

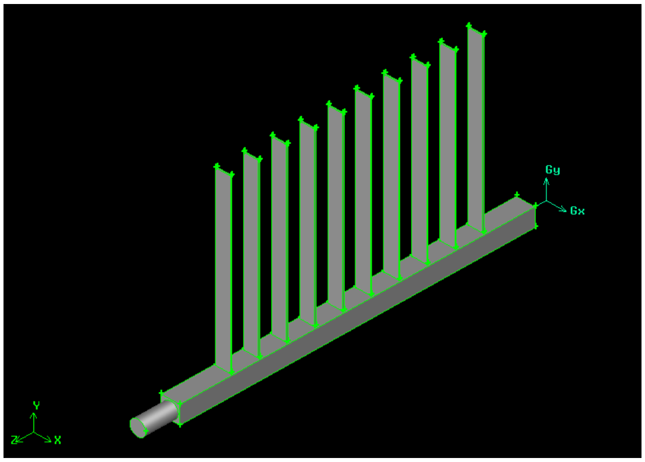

Figure 3. Three dimensional model of square header.

Drag coefficient law- The Schiller and Naumann model is the method to specify the fluid-fluid drag function. Here air-water distribution is analyzed and Schiller and Naumann model is used to specify the drag between the two fluids.

Thus the mass error, velocity errors as well as K and ε errors as measured by the residuals of their equations, being summed for all the grid nodes and normalized by their inlet values, were all below 10^{-6}.

4. Boundary Conditions

The boundary conditions imposed on the CFD model are given below. In two phase flow condition the left side of the header model is taken as inlet where water and air are supplied at a specified velocity. The outlets of the lateral pipes of the model are specified as pressure outlet. In this analysis water is taken as a primary phase and air is taken

Table 1. Solver parameters.

Parameters	Two phase flow
Type of solver	3ddp- pressure based -implicit-steady state solver
Material	Primary phase-Water Secondary phase-Air
Boundary conditions	Inlet-velocity inlet &volume fraction of secondary phase. Outlet-Pressure outlet
Pressure –velocity coupling	SIMPLE
Multiphase model	Mixture model
Mixture parameter	Slip velocity
Drag coefficient law	Schiller -Naumann
Discretization	Pressure - PRESTO Others-Second order upwind scheme
Turbulence model	Standard k-є
Residual absolute criteria	10^{-6}
Mesh size	14,00,000

as secondary phase so we have to mention the volume fraction of the secondary phase in the boundary condition. The values are given below

5. Experimental Setup Proposed by Osakabe and the CFD Model

The experimental apparatus consisted of a header, four lateral pipes and separators which were made of transparent acrylic material for observation of the flow pattern. The branch pipes are connected at an interval of 135mm. The entrance length between the header inlet and the first branch pipe was 650mm. The length of the branch pipe was 1 m. For two phase analysis water and air are supplied at velocities of 0.06m/s and 0.05m/s respectively[5].

6. Results and Validation of the CFD Procedure

Any CFD analysis needs its results to be validated for genuineness of the analysis procedures. This is due to the fact that, CFD if used with wrong inputs and boundary conditions can generate wrong results. The selection of computational models to be included in a particular analysis is also important. A step by step procedure for analyzing the flow system of a dividing header is obtained. This procedure has to be cross checked before going for real analysis. From the collected literatures, Osakabe have experimentally analyzed water flow distribution in a dividing header for two phase conditions. These experimental results are considered for validation of CFD procedure. The mass flow rate for each pipe is calculated, when the system operating at two phase condition and the results are shown in the Figure 5. Apart from this for two phase analysis water and air are supplied with 0.06m/s and 0.05m/s of velocity respectively. The experimental results and software (CFD) results for water and air flow through the each branch pipe are compared and shown in Figure 5.

7. Header Properties

7.1 Circular Header

Header Dimension :	20×20×430 mm
Number of pipes :	10
Pitch distance :	35mm
Dimensions of branch pipe :	18×1.07×136 mm

7.2 Square header

Header length :	430mm
Header diameter :	20mm
Number of pipes :	10
Pitch distance :	35mm
Dimensions of branch pipe:	18×1.07×136 mm

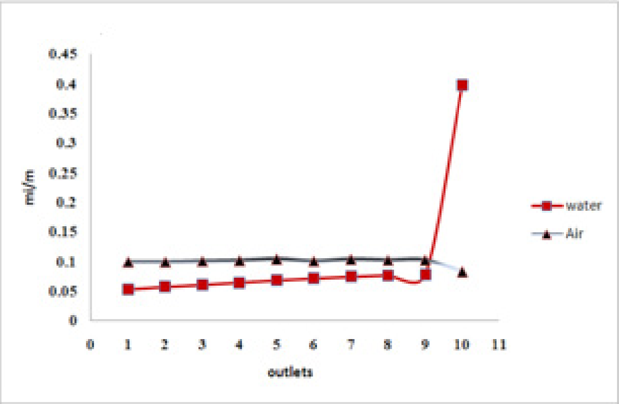

Figure 4. Circular header flow distribution.

7.3 Flow Distribution in Circular Header

In the circular header the water flow rate is maximum at 10th pipe and minimum at 1st pipe. From first to 9th pipe the flow rate gradually increased. Nearly 38% of total outlet flow goes through the last pipe. The air flow distribution also shows in Figure, it is in reversed manner of water flow distribution. Only minimum amount of air will flow through last pipe. The amount of non-uniformity of distribution of flow is characterized using a parameter called maldistribution[6].

From the maldistribution value we can able to find how much amount of water will over flow or under flow through the particular lateral pipe. The maldistribution values for first pipe is-0.48. The negative sign indicates there is an under flow. And only 0.52 times of actual flow rate will go through it. The last pipe vale is 3. That means there is an over flow on the particular pipe, of amount of 3times greater than the actual flow rate.

7.4 Flow Distribution in Square Header

In the square header the water flow rate is maximum at 10th pipe and minimum at 1st pipe. From first to 9th pipe the flow rate gradually increased. Nearly 25% of total outlet flow goes through the last pipe. The air flow distribution also shown in fig. it is in reversed manner of water flow distribution. Only minimum amount of air will flow through last pipe. The maldistribution value for first pipe is -0.2, the negative sign indicates there is an under flow, and only 0.8.

That means there is an over flow on the particular pipe, of amount of 1.5 times greater than the actual flow rate.

8. Comparison between Circular and Square Headers

In comparison, the water flow rate of the square header has less non-uniform distribution (Figure 7). From the values we can say square header is better than circular header in case of flow distribution. Also, it is observed that the uneven flow is reduced to a greater extent (almost 50%) of using the square header for circular header. In Figure 4 the flow of water and air in the circular header has been compared. In Figure 6, Flow Distribution of water both Circular and Square header is compared. Figure 8 shows the final result of the analysis that shows flow analysis in circular header.

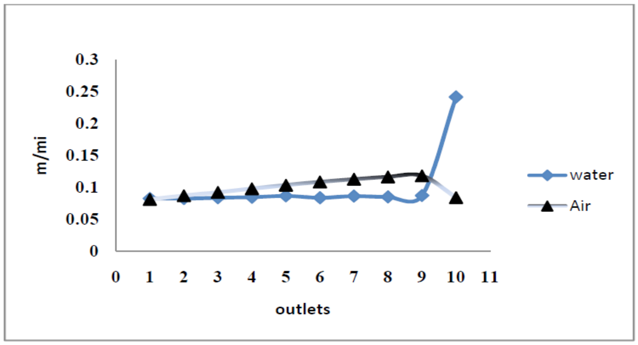

Figure 5. Square header flow distribution.

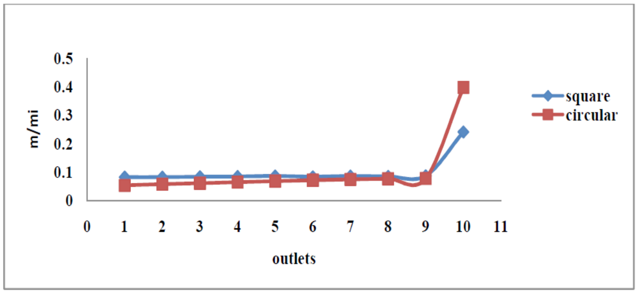

Figure 6. Flow distribution of water - circular and square.

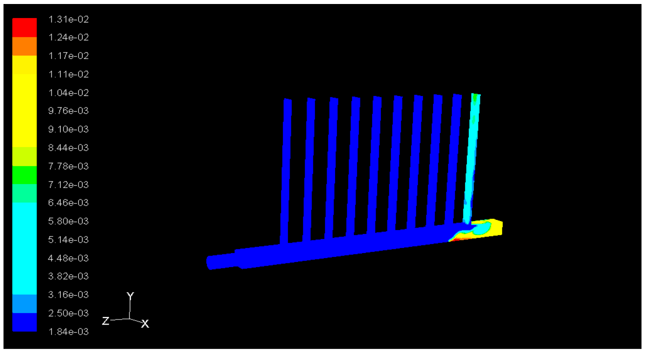

Figure 7. Flow analysis of square header.

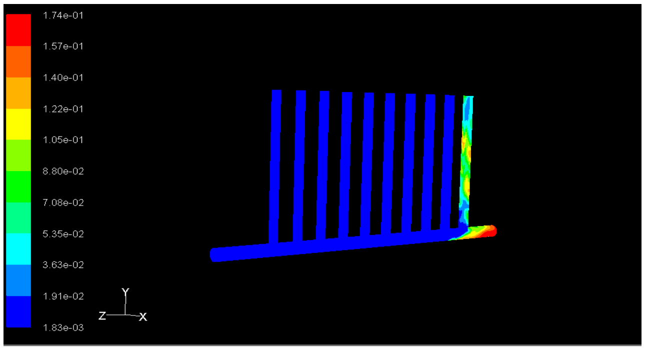

Figure 8. Flow analysis of circular header.

9. Conclusion

When comparing to all the other shapes square header gives the uniform flow distribution which pays way for the increase of performance of the heat exchanger. The geometrical parameters are responsible for the performance of the equipment. If the header is made based on this type of flow analysis various losses can be avoided (economic and material Losses).

10. References

1. Nae-Hyun Kim, Tae-Ryong Sin. Two-Phase Flow Distribution of Air–Water Annular Flow in a Parallel Flow Heat Exchanger, International Journal of Multiphase Flow. 2006; 32(12):1340–53.
2. Jun Kyoung Lee. Two-Phase Flow behavior Inside a Header Connected to Multiple Parallel Channels, Experimental Thermal and Fluid Science. 2009; 33(2):195–202.
3. Marchitto A, Devia F, Fossa M, Guglielmini G, Schenone C. Experiments on Two-Phase Flow Distribution Inside Parallel Channels of Compact Heat Exchangers, International Journal of Multiphase Flow**.** 2008; 34(2):128–44.
4. Habib MA, Ben-Mansour R, Said SAM, Al-Qahtani MS, JAl-Bagawi BJJ, Al-Mansour KM. Evaluation of Flow Maldistribution in Air-Cooled Heat Exchangers, Computers and Fluids. 2009; 38:677–90.
5. Masahiro Osakabe, Tomoyuki Hamada, Sachiyo Horiki. Water Flow Distribution in Horizontal Header Contaminated with Bubbles, International Journal of Multiphase Flow. 1999; 25(5):827–40.
6. Mohammad Ahmad, Georges Berthoud, Pierre Mercier. General Characteristics of Two-Phase Flow Distribution in a Compact Heat Exchanger, International Journal of Heat and Mass Transfer. 52(1-2):442–50.