NUMERICAL ANALYSIS FOR TWO PHASE FLOW DISTRIBUTION HEADERS IN HEAT EXCHANGERS

[1]A.Gogulakrishnanan, [2]M.Arun Pranesh, [3]M.Vengatesh, [4]M.Mareeswaran
[1,2,3] Assistant Professor, Department of Mechanical Engineering, Kathir College of Engineering, Coimbatore.
[4] PG Scholar, Jai Shriram Group of institutions

ABSTRACT

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 based on the assumption that the fluid distribute equally to all lateral pipes. In practical situation the flow is not uniform and equal in all lateral pipes. Mal distribution of flow in heat exchangers significantly affects their performance. Non-uniform flow distribution from header to the branch pipes in a flow system will lead to 25% decrease in effectiveness of a cross flow heat exchanger. Mal distribution of flow in the header is influenced by the geometric parameters and operating conditions of the header. The flow distribution among the branch pipes of dividing flow header system is analyzed for two phase flow condition. In the two phase flow condition, the effect of change in geometric cross sectional shape of the header (circular, square), inlet flow velocities are varied to find the flow mal distribution through the lateral pipes are investigated with the use of Computational Fluid Dynamics software.

Key words: Heat exchangers; Mal distribution; Cross sectional shape; two phase flow

INTRODUCTION

Flow distribution headers are an integral part of many engineering applications such as heat exchangers, air conditioning systems etc. These headers involve redistributing the internal flow stream into several passages. They are designed based on the common assumption that through these distribution manifolds, flow is distributed uniformly. In practical situations, transition can occur anywhere along the flow length inducing flow mal distribution and a reduction in the desired performance of the equipment. The heat exchanger headers are not an exception and their design is based on the assumption of equal flow rate in all the channels. But in reality, the flow distribution is non-uniform due to number of influencing variables like area ratio, lateral pipe resistance, No of lateral pipes, pitch distance etc., and this affects the thermal and hydraulic performances of the heat exchanger. The basic design of heat exchangers normally has two fluids of different temperatures separated by some conducting medium. The flowing inside the tubes and the other fluid flowing around the tubes. The design of these heat exchangers, for the sake of compactness and also for its functioning necessitates a branching of the internal fluid flow into several parts. The uniform flow through each branch without any flow mal distribution is desired for uniform temperature variation over the different branches.

Flow Mal distribution:

Flow mal distribution is defined as non-uniform distribution of the mass flow rate on one or both fluid sides in any of the heat exchanger ports and/or in the heat exchanger core. Flow mal distribution can be induced by

1. Heat exchanger geometry
2. Heat exchanger operating conditions

Geometry Induced Flow Mal distribution

One class of flow mal distribution, which is a result of geometrically non ideal fluid flow passages or non ideal heat exchanger inlet/outlet header design, is referred to as geometry induced flow mal distribution. This is peculiar to a particular heat exchanger in question and cannot be influenced significantly by modifying operating conditions. The most important causes of flow non-uniformities can be divided roughly into three main groups of mal distribution effects:

i. Gross flow mal distribution
ii. Passage to Passage flow mal distribution
iii. Manifold induced flow mal distribution

Gross Flow Mal Distribution

The major feature of the flow mal distribution is that non-uniform flow occurs at the macroscopic level due to poor header design or blockage of some flow passages. The gross flow mal distribution does not depend on the local heat transfer surface geometry. This class of flow mal distribution may cause a significant increase in the heat exchanger pressure drop and some reduction in heat transfer rate.

Passage To Passage Flow Mal Distribution

Compact heat exchangers with uninterrupted flow passages, while designed for non-fouling applications, are highly susceptible to passage to passage flow mal distribution. That is because the neighboring passages are geometrically never identical, due to the

ISSN 2229-5518

manufacturing conditions and variations are precise when small dimensions are involved. Since differently sized and shaped passages exhibit different flow resistances, this phenomenon usually causes a slight reduction in pressure drop, while the reduction in heat transfer rate may be significantly compared to that for average size passages.

Manifold Induced Flow Mal Distribution

Manifolds are integral parts in heat exchangers due to construction features. It is common and attached separately in many other applications. In the Plate heat exchangers, the fluid enters and exits the manifold laterally and the flow within the core axially. In other applications, the fluid enters and exits the core axially or a combination of axial and lateral entry and exit. The manifolds are of two basic types: dividing flow and combining flow. In dividing flow manifolds, fluid enters laterally and exits the manifold axially (through lateral pipe axis).

The velocity within the manifold, parallel to the manifold axis, varies from the inlet velocity to zero value. Conversely, in combining flow manifolds, fluid enters axially and exits at the end of the manifold laterally, with the velocity within the exit manifold varying from zero to the outlet velocity. When interconnected by lateral branches, these manifolds result in parallel and reverse flow systems. Because the inlet and outlet manifolds have the same effective diameter pipes connected by the lateral branches, this construction has built in inherent flow non-uniformity and the difference in mass flow rate distribution through the outlets.

Operating Condition Induced Flow Mal distribution

Operating conditions such as temperature level, temperature differences, multiphase flow conditions etc., inevitably influence thermo physical properties (viscosity, density) and or process characteristics of the exchanger fluids, which in turn may cause various flow mal distributions. Two phase flow heat exchangers are particularly sensitive to mal distribution because of the natural tendency of the phase separation.

Types of Headers In Flow Distribution Systems

Headers commonly used in flow distribution systems can be classified as dividing and combining flow headers. 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.

DESIGN AND PROCEDURE FOR CARRYING OUT THE ANALYSIS

CFD Model

The three dimensional model for analysis was created using GAMBIT modeling software and meshed. The internal diameter of the header is 17mm. The dimensions of lateral pipe are 16×1.03×125 mm respectively. The analysis is done for varying the cross sectional shape of the header (circular, Square).The Following parameters are square header pipe dimensions

S.No.	Description	Dimensions
1.	Header Dimension	17×17×415 mm
2.	Number of pipes	10
3.	Pitch distance	32mm
4.	Dimensions of lateral pipe	16×1.03×125 mm

Governing Equations

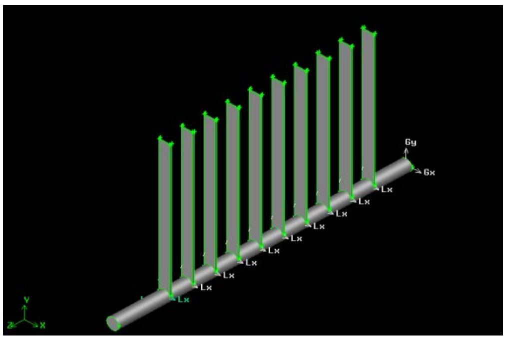
The governing flow equations are obtained by the application of physical principle. The most important equations are those governing the fluid dynamics namely:
1. Mass is Conserved (Continuity equation)
2. Newton's second law, F=ma (Momentum equation)

Continuity Equation

The integral form of the continuity equation:

$$\frac{\partial}{\partial t}\iiint \rho dv + \iint \rho V.ds = 0$$

The equations in the conservative and non conservative f



1. Non-conservative form

$$\frac{D\rho}{Dt} + \rho\Delta.V = 0$$

2. Conservative form

The physical principle of momentum equation is

F = ma (Newton's second law)

Body forces, which act directly on the volumetric mass of the fluid element. These forces "act at a distance"; example are gravitational, electric, and magnetic forces. The equation for conservation and Non-conservation form are given below

I. Non-conservative form

X-component

$$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x}+\frac{\partial\tau xx}{\partial x}+\frac{\partial\tau yx}{\partial y}+\frac{\partial\tau zx}{\partial z}+\rho f_X$$

Y-component

$$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y}+\frac{\partial\tau xy}{\partial x}+\frac{\partial\tau yy}{\partial y}+\frac{\partial\tau zy}{\partial z}+\rho f_Y)$$

Z-component

$$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial z}+\frac{\partial\tau xz}{\partial x}+\frac{\partial\tau yz}{\partial y}+\frac{\partial\tau zz}{\partial z}+\rho f_Z)$$

Turbulence Model

The turbulence model used in this analysis is standard k-ε model. It is basically a two equation model based on model transport equations for the turbulence kinetic energy (***k***) and its dissipation rate (**ε**).

The turbulence kinetic energy ***k***, and its rate of dissipation, **ε** is obtained from the following transport equationsIn these equations, $\boldsymbol{G}_{\boldsymbol{k}}$ represents the generation of turbulence kinetic energy due to the mean velocity gradients, $\boldsymbol{G_b}$ is the generation of turbulence kinetic energy due to buoyancy, $\boldsymbol{Y_M}$ represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and, **ε** respectively. S_k and S_ε are user-defined source terms. The values for the constant given in the analysis are,

$C_{1\varepsilon}$=1.42, C2ε =1.68

Mixture Model Theory overview

The mixture model is a simplified multiphase model that can be used to model multiphase flows where the phases move at different velocities, But assume local equilibrium over short spatial length scales. The coupling between the phases should be strong. It can also be used to model homogeneous multiphase flows with very strong coupling and the phases moving at the same velocity. In addition, the mixture model can be used to calculate non-Newtonian viscosity. The mixture model can model n phases (fluid or particulate) by solving the momentum, continuity, and energy equations for the mixture, the volume fraction equations for the secondary phases, and algebraic expressions for the relative velocities. Typical applications include sedimentation, cyclone separators, particle-laden flows with low loading, and bubbly flows where the gas volume fraction remains low. The mixture model is a good substitute for the full Eulerian multiphase model in several cases. A full multiphase model may not be feasible when there is a wide distribution of the particulate phase or when the inter phase laws are unknown or their reliability can be questioned. A simpler model like the mixture model can perform as well as a full multiphase model while solving a smaller number of variables than the full multiphase model.

***Continuity* Equation**

The continuity equation for the mixture is

$$\frac{\partial}{\partial t}(\boldsymbol{\rho}_m)+\nabla.(\boldsymbol{\rho}_m v_m)=0$$

Momentum Equation

The momentum equation for the mixture can be obtained by summing the individual Momentum equations for all phases. It can be expressed as

$$\frac{\partial}{\partial t}(\boldsymbol{\rho}_m v_m)+\nabla.(\boldsymbol{\rho}_m v_m v_m)$$

$$=-\nabla p+[\mu_m(\nabla v_m+\nabla v^T{}_m)]+\boldsymbol{\rho}_m g+F+\nabla.\left(\sum_k^n \alpha k\rho kvd\right)$$

Where n is the number of phases, F is a body force, and µm is the viscosity of the mixture:

Mesh Characteristics

The arrangement of discrete points throughout the flow field is simply called a grid. The matter of grid generation is significant consideration in CFD. In this work, the structured grid has been designed after many trails.

Grid Independent Analysis

In order to establish grid independence 2 different grids were used. The coarser grid had 60,000 cells and a finer grid had 2, 00,000 cells. Tetrahedral mesh element was used as meshing element for both the cases. The main goal of the study is to investigate the flow rate distribution through the lateral pipes. The maximum difference in flow rates between the two cases is 13.2%. The 2, 00,000 cell case gives better mass balance result than 60,000 cell case. Hence 2L case has been adopted for the rest of the study.

Mesh characteristics

S.No	Volume mesh type	Tetrahedral mesh
1	Spacing	2.3
2	Growth ratio	1.2

Solver Parameters

3DDP-It means that three dimensional fluid domain model is solved by double precision solver and gives the results accurately. Steady state method-The fluid flow parameters are do not change with time. So the steady state solver was selected for do the analysis. Implicit method-By definition, an implicit approach is one where the unknowns must be obtained by means of simultaneous solution of the difference equations applied at all grid points arrayed at a given time level. Pressure-velocity coupling- The SIMPLE (Semi-Implicit Method For Pressure-Linked Equations) algorithm gives a method of calculating pressure and velocities. The method is iterative and when other scalars are coupled to the momentum equations, the calculation needs to be done sequentially.

Discretization technique- For Tetrahedral grids, the flow is never aligned with the grid, so the more accurate results obtained by using the second-order Discretization. In this work start with the first-order scheme and then switch to the second-order scheme after a few iterations and continue the calculation to convergence. 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}.

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

Boundary Conditions		
Inlet	Velocity inlet	Vair-22.048 Vw-.027m/s
Outlet	Pressure outlet	Atm pressure.

METHODOLOGY

Problem Definition

In flow headers it has been found out that mal distribution is very closely related to the geometrical and operating conditions of the header. Studies reveal that to geometrical parameters such as, cross sectional shape of the header, lateral pipe spacing, lateral pipe height, No of pipes, inlet flow velocity etc. But, fabricating the setup and conducting several experiments is a tedious task. So the experiments have been performed for two combination of influencing parameters and are validated with CFD software. The results are found to have close correlation. Based on this concept a Numerical study of flow distribution for Dividing Headers has to be carried out and the degree to which all these parameters affect flow distribution have to be found out.

EXPERIMENTAL WORK

Results Extraction Procedure

In FLUENT the conservation equations of mass and momentum were solved using finite volume method and a second order upwind differential scheme was applied for the approximation other convective terms. This analysis done by using coupled solver to solve continuity and momentum equations. The basic equations such as turbulent equation if the flow is turbulent were to be chosen and the values of the boundaries such as velocity inlet at the header, operating pressure etc. were specified according to each case. The fluid (water-liquid) was selected and the fluid properties such as viscosity, density were defined. The solution control parameters were adjusted and the solver was initialized and iterated till the solution was converged. With the use of surface integral option in the fluent the volume flow rate of each pipe outlet is calculated directly. Then the mal distribution value of the header is calculated by

$$m = \frac{m_i - m_{av}}{m_{av}}$$

Where m_i is the Mass flow rate through 1st pipe and m_{av} is the flow in one lateral pipe for a completely uniform distribution

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.

Experimental Setup Proposed By Osakabe [1] and the Cfd Model

ISSN 2229-5518

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 a interval of 130mm. The entrance length between the header inlet and the first branch pipe was 600mm. The length of the branch pipe was 1000mm. For two phase analysis water and air are supplied at velocities of 0.054m/s and 0.04m/s respectively. In the CFD analysis, a similar model as that of the experimental setup with exact dimensions was used. A schematic diagram of the CFD model is shown in fig.6.1. All the experimental conditions were imposed without any change.

Extraction of The Results:

Using the fluxes option in the software the mass flow rate through each pipe is directly calculated. From this value the dimensionless volumetric flow rate through each pipe are calculated. Then the results are compared with experimental results and it is shown in figure. The similar procedure is applied to validate the CFD results when the system operating at two phase condition and the results are shown in the Fig.

Validation of Two Phase Flow Distribution Results

For two phase analysis water and air are supplied at velocities of 0.054m/s and 0.04m/s respectively. The experimental and CFD results for distribution of water through the each lateral pipe are compared. A well established step by step CFD procedure was obtained for analyzing any type of flow system. The inlet condition of liquid velocity is 0.54m/s and V_{sg} =0.04m/s. The same procedure is validated against the published experimental results. And a good agreement is obtained between the two results with an acceptable error limit. Hence this procedure can be used for analyzing the flow system of a different header configuration and operating at different operating conditions.

Two-Phase Flow Results

Numerical analysis is done for dividing header having 17 mm diameter (D) with the branch pipes of dimensions 16×1.03×125 mm respectively. The effect of flow distribution when varying the geometrical shape of the dividing header (circular, Square) is discussed.

Flow Distribution in Circular Header

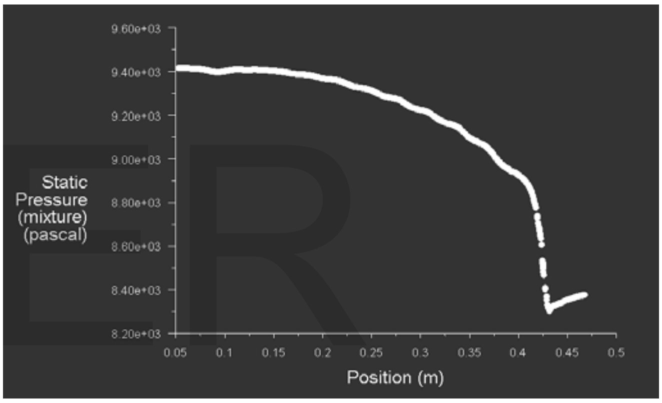
In the circular header the water flow rate is maximum at 10^{th} pipe and minimum at 1^{st} pipe. From first to 9^{th} pipe the flow rate gradually increased. Nearly 38% of total outlet flow goes through the last pipe. The air flow distribution also shown in fig. it is in reversed manner of water flow distribution. Only minimum amount of air will flow through last pipe. The amount of non-uniformity of distribution of flow is characterized using a parameter called mal distribution (m).

$$m = \frac{m_i - m_{av}}{m_{av}}$$

The mal distribution value 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.

Flow Distribution in Square Header

In the square header the water flow rate is maximum at 10^{th} pipe and minimum at 1^{st} pipe. From first to 9^{th} pipe the flow rate gradually increased. Nearly 25% of total outlet flow goes through the last pipe.

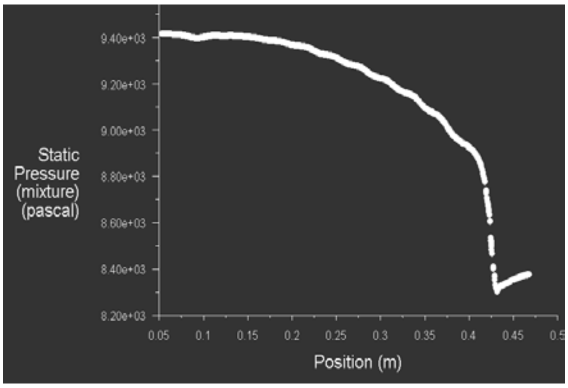


Pressure Distribution inside the Header

The predicted static pressure values along this line were plotted in graph. The graph plotted for the line drawn at the center of the header model**.** 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 mal distribution value for first pipe is -0.2. The negative sign indicates there is an under flow. And only 0.8 times of actual flow rate will go through it. The last pipe vale is 1.5. That means there is an over flow on the particular pipe, of amount of 1.5times greater than the actual flow rate. From the mal distribution values it is found that the square header shall gives the more uniform distribution when comparing the circular header.

Because it is observed that the mal distribution values are almost reduced to 50% when using the square header instead of circular header.

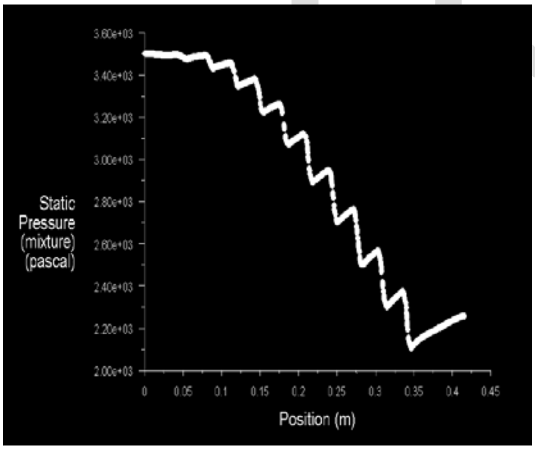
ISSN 2229-5518



Square

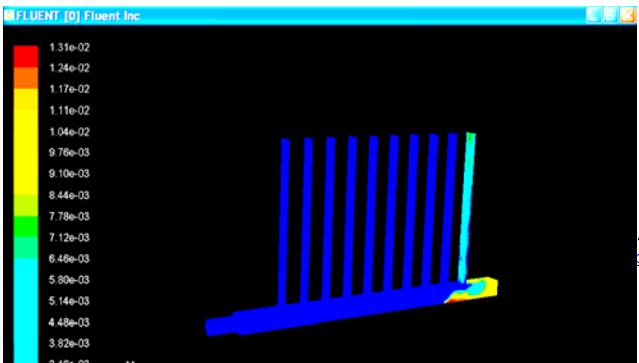
Header mass flow distribution

This figure shows the pressure distribution inside the square and circular header respectively. Figure explains the phenomenon of pressure recovery inside the header. The decrease in pressure in the initial portion of the curves indicates the pressure drop due to the flow resistance of the pipes.



Square Headers Pressure flow distribution

The net effect of pressure recovery and pressure drop causes a comparatively non-uniform pressure distribution in circular header. This picture of pressure variation cannot be brought out by an experimental analysis due to the limitation in placing pressure sensors.



Mass Flow Distribution in Square Header

The figure shows the volume fraction of water inside the header. It also reflects the mass flow rate graph values. Because in the 10^{th} pipe only have more distribution of water. In the contour diagram also the maximum value for volume fraction observed at 10^{th} pipe when compare to other pipes. In the analysis we also observe that when changing the shape it only effects the flow distribution but it not affects the distribution pattern.

CONCLUSION AND FUTURE WORK

Header with a square cross section will give a uniform flow distribution compared to other shape. In the analysis we also observe that when changing the shape it only effects the flow distribution but it not affects the distribution pattern. When increasing the air velocity the water will flow through the rear end of the header. The results are analyzed to generate guidelines for header design. So the header shape is the important parameter which effects the flow distribution. The effect of variations on the flow distribution in a combined header needs to be studied.

REFERENCES

1) R.C. Sachdeva, fundamental of engineering heat and mass transfer 4th edition
2) T. Kulkarni, C.W. Bullard, K. Cho, Header design tradeoffs in microchannel evaporators, Appl. Therm. Eng. 24 (2004) 759–776.
3) R.L. Webb, K. Chung, Two-phase flow distribution in tubes of parallel flow heat exchangers, Heat Transfer Eng. 26 (2004) 3–18.

Pressure distribution inside the quare eader	**Pipe number**	**Circular**	**square**