Applied Mechanics and Materials Vols. 465-466 (2014) pp 608-612
Online available since 2013/Dec/19 at www.scientific.net

doi:10.4028/www.scientific.net/AMM.465-466.608

Flow Separation Prediction in a Single-Phase Flow in an Inline Tube Bundles

Azmahani Sadikin[1,a] and Norasikin Mat Isa[1,b]

[1]Department of Plant and Automotive Engineering, Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Malaysia

[a]azmah@uthm.edu.my, [b]sikin@uthm.edu.my

Keywords: Flow separation, heat exchanger, in-line bundle

Abstract. The vertical single-phase flow was studied on the shell side of a horizontal tube bundle. In the present study, CFX version 14.0 from ANSYS was used to predict the flow regimes in the two tube bundles; i.e. the 19 mm and 38 mm arranged in an in-line configuration with a pitch to diameter ratio of 1.32. The simulations were undertaken to inform on how the fluid flowed within the tube passages in different tube bundle diameter that gives different gaps between the tubes, where the fluid must pass. The results show that the maximum gaps between the tubes have no clear effect to the flow where the flow separation and re-attachment and the average velocity is the same when increasing the tube bundle. This is consistent with other published data.

Introduction

This study was initiated to support previous studies of kettle reboilers [1-8]. Reboilers are widely used in the process industry for vapour generation. Some developments of horizontal steam generators for nuclear power plants are based on the kettle reboiler design. The kettle reboiler is a shell and tube type heat exchanger usually consisting of a tube bundle arranged on a square-in-line pitch enclosed in a shell for easy cleaning. It also contains a vertical oriented weir of sufficient height to ensure liquid covers the bundle. The heating medium, usually steam, flows in the tubes while the liquid to be partially vapourised is on the shell side. The liquid is usually below the boiling temperature at the bottom-most portion of the bundle. It is heated by natural convection and then by subcooled and saturated boiling as it moves from the bottom to the top. The extent of each regime depends upon the composition of fluid as well as parameters affecting performance, such as type and volume of liquid, operating pressure, heat flux and geometrical parameters. The separation and re-attachment phenonema in the heat exchanger affect the pressure drop and flow pattern in the heat exchangers. Many researchers have constructed flow regime maps to improve the design of shell and tube heat exchangers. Most of these maps were based on visual observations and they were constructed using the maximum superficial gas velocity on the x-axis and the maximum superficial liquid velocity on the y-axis. Some were constructed using more objective methods, e.g. void fraction transients. Grant and Chisholm [6] used visual observations to study the flow regimes of vertical air-water flow across horizontal tube bundles in a segmental baffled heat exchanger consisting of 39 tubes, 19 mm in outside diameter, arranged in an in-line configuration with a pitch to diameter ratio of 1.25. Upward flow could be described as either bubbly, intermittent, or spray flow, whereas downward flow could be described as bubbly, stratified and stratified-spray or spray flow. McNeil et al. [7] suggest that the flow in heat exchanger is said to be in two regions, the separated flow region and the attached flow region. The separated flow region contains the flow between the separation and re-attachment points. The attached flow region contains the flow between the re-attachment and the separation points. The mechanistic model was deduced for each region. The frictional pressure drop is shown to depend on a liquid layer located on the upper portion of the tubes at low gas velocity and on acceleration effects at high gas velocity. Many has observed the flow in heat exchanger in a tube bundle less than 20 mm, but few has reported flow in bundle bigger than 20 mm. The objective of this paper is to investigate the effect of tube diameter to the flow separation and re-attachment in the tube bundle at different gap between the tubes.

Computational Fluid Dynamics

The flow in a tube passage is assumed to be symmetrical because the geometry and physical conditions causing it are symmetrical and because the flow in any passage between the tubes is likely to be the same as that in any other. So, in the simulations, only a symmetrical half of a flow passage between the tubes is used. The flow is simulated over ten tubes in the flow direction to ensure fully developed flow is achieved. The tube bundles were created in DesignModeler. Two dimensional models for the three bundles were produced in CFX-PRE for the symmetrical half of the water-only bundles. The boundary conditions for the tube bundles are shown in Fig. 1. The tubes were set to solid surfaces with no slip and the east, west, front and back surfaces set to the symmetrical boundary condition. The opening boundary condition at the top of the bundle was set to atmospheric pressure and the inlet boundary was set to a normal velocity of 6 m/s. An inflation layer of 1.0 mm thickness and containing 16 layers with an expansion factor of 1.3 was inserted between the tube walls and the bulk fluid to capture the effects near the wall. The simulation was run until the residual of the pressure and velocities were less than 0.00001.

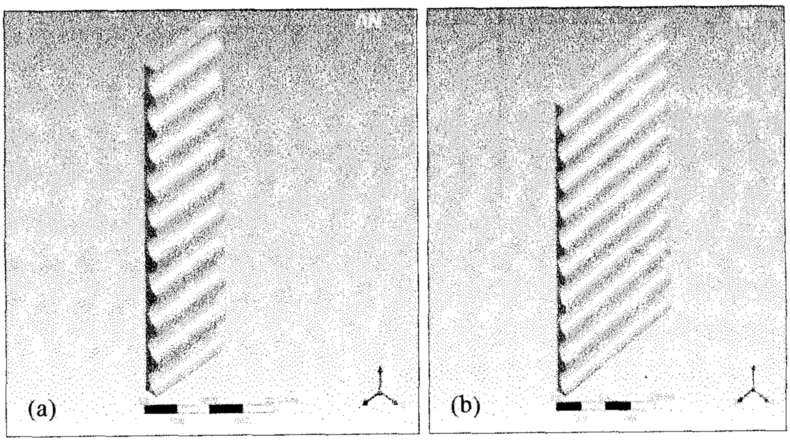

Fig. 1: The model (a) 38 mm in diameter in-line bundle (b) 19 mm in diameter in-line bundle

Grid Independency Study

In computational fluid dynamics analysis, accuracy of the results is controlled by the selection of the mesh density as finer mesh produces more accurate results but requires more computer time for solving the problem. To this point, simple investigation has been conducted to determine the acceptable mesh division without compromising accuracy of the results. Therefore, a grid independence study was carried out for two meshes for each tube bundles. In 38 mm inline tube bundle, two mesh configurations of 1,100,000 and 3,200,000 cells were conducted. In 19 mm inline tube bundle, two mesh configurations of 1,300,000 and 3,500,000 cells were made. The tube pitch pressure of each bundle for each mesh configurations were analysed. The results show there is no significant difference between the two mesh configurations as all lines of both configurations are almost overlapped. These indicate, using finer mesh does not improve the model prediction. Thus, meshing with lower number of mesh cells does not sacrifices the solution accuracy. Since the Central Processing Unit (CPU) time increases exponentially with the number of grids, the lower mesh cells, 1,100,000 and 1,300,000 were chosen for 38 mm in-line tube bundle and 19 mm in-line tube bundl respectively. Less mesh cells reduce CPU time during CFD simulation which permits a significant number of cases to be run. The parameters and boundary conditions for the models are shown in Table 1.

Table 1: Boundary condition

Geometry	Tube diameter	38 mm	19 mm
	Pitch	50 mm square pitch array	25 mm square pitch array
	Pitch to diameter ratio, P/D	1.32	1.32
	Number of tubes	10	10
	Tube length	150 mm	
	Tubes arrangment	In-line square	In-line square
	Working fluid	Water	
Domain	Domain type	Fluid domain	
	Water temperature	25°C	
	Turbulence model	Shear Stress Transport (SST)	
	Wall function	Automatic	
	Reference pressure	1 atm	
	Buoyancy option	Non-Buoyant	
	Domain motion	Stationary	
	Heat transfer model	None	
	Turbulence wall functions	Automatic	
	Reaction or combustion model	None	
	Thermal radiation model option	None	
Boundary condition			
Inlet	Flow regime option	Subsonic	
	Mass and momentum option	Normal speed	
	Normal speed	6 m/s	
Outlet	Flow regime option	Subsonic	
	Mass and momentum option	Static pressure	
	Relative pressure	0 Pa	
	Flow direction	Normal to boundary condition	
	Turbulence option	High intensity	
Symmetry	Boundary type	Symmetry	
Wall	Solid wall	No slip is applied between the fluid and solid	
Solver	2-Dimensional, steady state, axisymmetric		
	Advection Scheme Option	High resolution	
	Timescale control	Auto timescale	
	Maximum number of iterations	100	
	Residual type	RMS	
	Residual target	0.00001	

Results and Discussion

Fig. 2 (a) shows the velocity vector in 38 mm in-line bundles. There are two regions of flow that are clearly shown, the main flow and circulation zones. As the fluid flows past the tubes, separation occurs when the wall shear stress is zero. This results in separation bubbles behind the tubes in which some of the fluid is actually flowing upstream, against the direction of the main flow. The flow forms a circulation between the tubes due to low pressures in the separated wake regions, as shown in Fig. 3(a). The separation points occur when the wall shear stress is zero where separation occurs at separation angle, $\theta_S = 110°$ and re-attachment at re-attachment angle, $\theta_R = 51°$.

The vector velocity in the 19 mm in-line bundle is shown in Fig. 2 (b). After the first few tubes, the flow path is fully developed, so that what occurs in one tube pitch is repeated in the others. The main stream has a high velocity due to the area reduction and friction causes re-circulation to occur in the gaps between the tubes due to low pressure in the separated wake regions. There is a clear similarity between the 38 and 19 mm in-line flow fields, as seen in Fig. 2. The flow begins at the minimum gap between the tubes and decelerates as a potential flow until it separates at θ_S, where a wake is formed to the rear of the tubes. The flow is re-attached at θ_R . The separation point occur at $\theta_S = 107°$. The flow is re-attach at the maximum main flow area at $\theta_R = 52°$ as shown in Fig. 3. This happens at all tube in fully developed flow.