Chapter 2: Literature survey

Many heat exchanger configurations have been discussed extensively in literature. STHEs are basic configurations that are used and consequently, many aspects thereof have been studied by researchers. Emanating from the research overview, the following relevant topics are discussed in this chapter:

- Heat exchangers and CFD
- The use of CFD on STHEs
- Pressure drop correlations
- Turbulence modelling
- Multiphase modelling

Standard shell-and-tube configurations have been investigated by various authors, generally commenting on the effects on heat transfer and pressure drop. Other authors have derived pressure drop correlations for STHEs, either by empirical or analytical means. After this discussion on pressure drop, studies that have been done on heat transfer and pressure drop within ideal tube banks, with and without modifications, will follow. Within the CFD environment, turbulence is arduous to solve explicitly, thus turbulence modelling, possibly the largest part of fluid mechanics and CFD, will be discussed. The survey will be concluded with several authors' contribution to multiphase modelling within different conditions and conclusions, gathered from the literature that was cited.

2.1. Heat exchangers and CFD in general

Throughout all the studies that were examined, several CFD software packages were used to simulate and solve various problems. Of these packages, FLUENT has the most widespread use, but according to De Wet (2012), the package that is used for this study namely STAR-CCM+, has similar computational capabilities and a better graphical user interface. Other packages that are utilized by researchers include OpenFOAM, PHOENICS, FLOTRACS, CFX, FIDAP, ADINA, CFD2000, POLYFLOW and STAR-CD, which is an earlier version of STAR-CCM+ (Aslam Bhutta *et al.*, 2012:1; Fernandes *et al.*, 2007:825; Hansen, 2008:1).

Investigations into and simulations of various types of heat exchangers are available in literature where the previously mentioned CFD software packages were used. The following are some of the configurations that fall outside the scope of the current study: plate-type heat exchangers by Kanaris *et al.* (2006:8) and Fernandes *et al.* (2007:833), helical tube heat exchangers by Kumar *et al.* (2006:4403) and Jayakumar *et al.* (2008:12), finned tube heat exchangers by Sridhara *et al.* (2005:5), Mcilwain (2010:169) and Yun and Lee (2000:2529), dimple plate heat exchangers by Witry *et al.* (2003:6) and a heat pipe heat exchanger by Saber *et al.* (2010:7). A summary of a broad range of heat exchanger models is presented by Aslam Bhutta *et al.* (2012:1).

The following summaries are of investigations into shell-and-tube construction of which the configuration falls outside the scope of the present study, but is recommended as reference for future studies.

Dong *et al.* (2008:660) studied the fluid, temperature and pressure distribution that occur within a rod-baffle STHE. The CFD code FLUENT was used to simulate a simplified model, resulting in a 10% deviation of stream velocity compared to experimental values. A much larger deviation of up to 85% was observed for fluid velocity values perpendicular to the main stream direction. Heat transfer correlations and values for important flow quantities were compared with other research centres' experiments and were found to be accurate.

Jafari Nasr and Shafeghat (2008:1332) investigated the flow patterns inside a helical-baffled STHE. Certain geometrical configurations were varied, of which baffle inclination angle and spacing were key. A comparative single-segmental baffle STHE was also modelled by using CFD. The greatest advantage of the helical-baffled STHE is the uniformity of shell-side flow direction. Recirculation zones, occurring behind baffles in segmental-baffled STHEs, are non-existent with helical baffles. The setup with a baffle inclination angle of 40 degrees was measured to have the best pressure drop and heat transfer performance.

Wang, Q. *et al.* (2009:1214) numerically investigated the thermal fields and flow structures in combined multiple shell pass STHEs with continuous baffles (CMSP-STHEs). A comparison between the CMSP-STHE and a standard single-segmental STHE was made by using the CFD code FLUENT. Concerning heat transfer and pressure drop for various mass flows, the CMSP-STHE performs better in comparison to the single-segmental STHE.

The performance of STHE with continuous helical baffles was compared with that of middle-overlapped helical baffles (Zhang *et al*., 2009:5381). Three helical baffle angles were simulated for both configurations and the 40° middle-overlapped baffle configuration performed the best, having the highest pressure drop to heat transfer ratio.

It is clear that numerical investigation into the pressure drop and heat transfer characteristics of heat exchangers can be accomplished effectively. Following this general overview of the use of CFD with regard to heat exchangers, STHEs of relevant construction will be discussed in greater detail.

2.2. Shell-and-tube heat exchangers

Research into STHEs can be divided into three topics: the effect of fluid properties, optimisation of geometrical parameters and the comparison of different configurations.

2.2.1. The effect of fluid properties

Karlsson and Vamling (2005:713) investigated the effects of fluid properties by studying the flow field and condensation rate for pure R22 gas, and a zeotropic mixture of R32 and R134a within STHEs. CFD predicted the tube bank penetration of both fluids differently. Experimental data for flow of the zeotropic mixture contradicted a vertical flow assumption that was indicated by CFD calculations. It was discovered that the bundle-shell clearance had little effect on the flow field of the zeotropic mixture, but not on the flow field of R22.

Velasco *et al.* (2008:961) investigated the effect of a guillotine tube breach in an STHE. Particle image velocimetry (PIV) data were obtained experimentally for velocity and turbulence parameters, and compared with results of CFD simulations of a particle laden jet at different Reynolds numbers. They established that the jet followed a semi-parabolic path when it evolves from oblique cross-flow to an axial-flow configuration. Jet penetration and the Reynolds number increased, with similar tendencies that were observed for turbulence intensity close to the breach. Jet turbulence and velocity were increased by neighbouring tubes and followed the tube profiles closely, reducing penetration, but increasing main flow stream entrainment.

Cebrucean *et al.* (2009:28) modelled heat transfer and pressure drop in STHEs with single-segmental baffles by using FLUENT. The model consisted of nine compartments with a baffle-cut of 25%. Fluid properties were kept constant and were found to have minimal effect on heat transfer when compared to experimental results.

Valuable conclusions can be drawn from these studies. The versatility and robustness of an STHE is shown. Although these studies are not directly relevant to the problem at hand, flow properties are shown to affect the performance of heat exchangers as well as the accuracy of numerical simulations. Other than simulating STHEs by means of CFD, the study presented by Velasco *et al.* (2008:961) has indirect relevance also due to the possibility of tube breaches. Entrainment of process/tube-side fluid can change fluid properties and add to the difficulties that have already been observed in the present setup, although Cebrucean *et al.* (2009:28) stated controversially that fluid properties have little effect on heat exchanger performance. These results should be considered for future studies.

2.2.2. Optimisation

The following studies focus on initial conditions and the influence of geometry (some of the results comment on heat transfer, although it is not within the scope of the present study; it is, however, of relevance, giving an indication of the bundle velocity).

Sparrow and Reifschneider (1986:1628) researched the effect of baffle spacing on heat and mass transfer and pressure drop. The per tube, per row and per compartment transfer coefficients were obtained by naphthalene sublimation and compared with the Tinker and Bell-Delaware Project design methods. The research showed that the spacing greatly affects the transfer coefficients in the inflow window of the compartment with larger spacing, having the greater influence. In the cross-flow zone and the row adjacent to the baffle in the outflow zone, the coefficients were larger for smaller baffle spacing. Compartment transfer averages were unaffected by the spacing changes.

In the United Kingdom, Pekdemir *et al.* (1993:10) studied the shell-side flow distribution of STHEs. Cross-flow velocities were obtained by using a neutral-density particle tracking technique. All measurements were taken in the fifth of nine baffle compartments for Reynolds numbers, ranging from 301-2177. Results concerning velocity were quantified as follows: i) An increase in baffle spacing increased the cross-flow velocity; ii) velocity decreased as the flow moved away from the central vertical flow plane; iii) the flow of fluid onto the downstream baffle generated areas with high velocities; and iv) in contrast, the velocity near the upstream baffle and shell wall was the lowest.

Roetzel and Lee (1993:3771) investigated leakage in segmental-baffled STHEs experimentally, also by means of the axial dispersion model. The effects of leakage in the heat exchangers were observed by varying the stream flow direction, tube and shell-side flow rate and the influence of baffle spacing and baffle-shell clearance. The study found that the shell-side Peclet number is only dependent on the baffle-shell clearance and baffle spacing. The authors also mentioned that open literature correlations for heat transfer can be used with confidence.

Li and Kottke (1998a:348) studied the heat transfer characteristics of the first compartment in an experimental STHE. Heat transfer was established to be higher near the inlet nozzle region and the circumferential transfer coefficients had greater axial directional variation.

Li and Kottke (1998b:433) investigated the effect of leakage on the heat transfer coefficient and pressure drop of single-segmental-baffled STHEs. An absorption mass transfer technique was used to visualise the transfer characteristics and measured pressure distributions were converted to visualize the shell-side flow distribution. They established that as the magnitude of leakage streams increased, the heat transfer was reduced.

Li and Kottke (1998c:1311) also investigated the effect of baffle spacing on the same variables by the same means. The study showed increased heat transfer for increased baffle spacing at constant Reynolds numbers. These results can be attributed to a reduction in the baffle-shell leakage.

Roetzel and Balzereit (2000:1038) examined the assumption of one-dimensional flow in STHEs. A residence-time measurement technique was used to determine the Peclet number for several baffle spacing, baffle-shell leakage and axial plug-flow Reynolds number configurations. The Peclet number is a measure of axial dispersion of the flow field, which can vary from infinite (one-dimensional/axial flow) to zero (total three-dimensional dispersion within the shell). The Peclet number was established to vary in proportion to the Reynolds number and inversely to the other parameters.

Wang, S. *et al.* (2009:2438) increased heat transfer by adding shell-side sealer strips to decrease the ineffective flow through the baffle-shell clearance. The shell-side heat transfer, overall coefficient of heat transfer and exergy efficiency was increased. The researchers found that the negative result of increase in pressure drop that was brought on by the strips was outweighed by the positive result of increase in heat flux that was provided and thus benefited energy conservation measures.

Ozden and Tari (2010:1014) examined the effect of baffle-cut, baffle spacing and shell diameter on heat transfer and pressure drop by means of CFD. The effects of baffle number and spacing variation were visualised. The results were compared with the outlet temperature, heat transfer and pressure drop values, obtained from implementing the initial conditions in the Kern and Bell-Delaware methods, and were found to correlate well with the latter method.

The articles that are presented on experimental optimisation deliver crucial results on the presence of low-velocity recirculation zones behind baffles are confirmed, highlighting the need to focus on the same areas in the current study. Reynolds numbers are discussed by Roetzel and Balzereit (2000:1038) as well as and Roetzel and Lee (1993:3771): the increase in Reynolds number is seen to have an increasing effect on the Peclet number, whereas the mass flow rate is found to have no effect. A decrease in baffle spacing increases bundle velocity, dispersing, decreasing and varying closer to the shell surface. Leakage was found to be decreased by a decrease in baffle spacing, resulting in lesser axial dispersion of the flow. By increasing baffle spacing and adding sealing strips, the heat transfer, which is coupled to velocity, is increased.

Several articles on numerical optimisation of STHEs are presented below.

Saffar-Avval and Damangir (1995:2506) developed a correlation to determine the optimum baffle spacing in several STHE configurations, accompanying the recommendations in the Heat exchanger design handbook no. 4 (Bell *et al.,*1983b). The researchers used an optimization program to calculate the spacing and number of the sealing strips that were required. Correlation sets were presented for the determination of baffle spacing.

In a dual article study, Prithiviraj and Andrews (1998a:799) developed a procedure to determine the mean velocity fields within an STHE in the first part. In the calculation, the tubes were modelled by the method of distributed resistances, combined with volumetric porosities and surface permeabilities. The results for velocity correlated well with other authors' data.

The second part of the study of Prithiviraj and Andrews (1998b:817) was devoted to the heat transfer characteristics of the same heat exchanger. An enthalpy balance was set up for the tube-side fluid, from where the shell-side equations were coupled to the tube-side equations. The results that were obtained from the model correlated well with data from the Bell-Delaware method. Generated results differed by a maximum of 15% from experimental results.

Stevanović *et al.* (2001:1105) developed a three-dimensional numerical model to iteratively design STHEs, given a pressure drop requirement. Once the pressure drop requirement was implemented, geometry and thermo-hydraulic characteristics of the heat exchanger were optimised. Turbulence models were also studied and the results were compared with experimental data.

Soltan *et al.* (2004:2810) developed a computer-based algorithm for the determination of the optimum baffle spacing for single-segmental STHEs in order to decrease capital and operating cost.

Patel and Rao (2010:1425) discussed an optimisation algorithm for STHEs, called particle swarm optimization, which can be used to determine the minimum annual operating cost. Design parameters that can be optimised in heat exchangers are baffle spacing and shell-and-tube diameter, as well as rotated triangular and square tube layouts and pitches.

Prithiviraj and Andrews (2010:15) developed a three-dimensional numerical model for the prediction of shell-side flow fields. Comparison of the numerical model with an experimental setup and three other calculation methods resulted in their model outperforming the Kern, Donohue and Bell methods, having a maximum pressure drop error percentage of 8,3% for 12 different cases. The Reynolds number, baffle-cut percentage, inlet nozzle diameter, number of cross passes and mass flow rate were varied for the 12 cases.

Although optimisation of the final configuration that has been chosen falls outside the scope of the study, the articles that are discussed can be of great value for future studies. It gives an indication of how geometric parameterization can be used to achieve optimum heat transfer, velocity distributions and sediment transportation.

2.2.3. Configuration comparisons

Different configurations and the comparison of the differences in terms of flow characteristics are highly relevant topics for discussion. A limited amount of literature has been published; the research is discussed below.

In another paper published by Li and Kottke (1999:3521), the heat and mass transfer characteristics of a disc-and-doughnut-baffled STHE was discussed. The authors found that the disc-and-doughnut configuration had a higher heat transfer/pressure drop ratio than a single-segmental baffle unit.

Various disc-and-doughnut configurations were investigated to determine STHE thermo-hydraulic performances. Krishnan and Kumar (1994:624) found that modified disc-and-doughnut baffles resulted in the following: i) Improved heat transfer per unit pressure drop characteristics in comparison to conventional disc-and-doughnut baffles; ii) fewer sharp edges that damage the tubes; and iii) better tube support, leading to a decrease in vibration damage.

Mohammadi *et al.* (2009:1135) simulated the flow through an STHE, focussing on the effects of leakage and baffle orientation. Heat transfer and pressure drop values at several Reynolds numbers were simulated for vertical and horizontal single-segmental baffles as well as ratios of these parameters for the horizontal/vertical ratio that were presented. The horizontal baffle configuration performed the best.

Yongqing *et al.* (2011:53) studied the heat transfer and pressure drop characteristics of STHEs with segmental, rod- and innovative H-type baffles. The three configurations were tested at various flow rates, using both experimental and numerical methods (FLUENT). The heat transfer coefficients were shown to have a maximum error of 10.89%, with accuracy increasing with an increase in mass flow rate. The performance of the H-type baffle was average in comparison to the other configurations, having a higher heat transfer/pressure drop ratio than the rod-baffle. CFD was deemed successful in modelling the different heat exchanger phenomena.

Disc-and-doughnut-baffled STHEs are shown to be possibly one of the more effective configurations. The heat transfer of the configuration is not necessarily increased, but the flow field is disturbed less, resulting in a better heat transfer/pressure drop ratio. The use of different types of disc-and-doughnut baffles is a viable option for even better performance and should be considered for future study, although, Yongqing *et al.* (2011:53) observed that the single-segmental configuration might still yield better heat transfer results.

The studies that were presented in this section show the extent of and deficiencies in research on STHEs in open literature. There have only been limited studies done on STHEs specifically directed towards different configurations and the comparison between models. Conclusions drawn from the section reveal clear focus areas for future work and give insight into the possible outcome of the present study.

2.3. Heat exchanger pressure drop

An examination into the pressure drop characteristics of STHEs is presented with a discussion of the experimental/numerical performance and the development of pressure drop correlations.

2.3.1. Experimental and numerical performance

The pressure drop characteristics of consecutive baffle compartments were studied by Sparrow and Reifschneider (1986:1628). Pressure-tapped tubes were placed at the inflow, outflow and centre of each of six baffle compartments, excluding the outflow compartment. Graphical representation of the measurements fell well within expectations of greater pressure drop between the inflow-centre measurement points, as opposed to the centre-outflow points. The decrease in baffle spacing resulted in a decrease in pressure drop per compartment, although the total pressure drop increased for smaller baffle spacing. At smaller inter-baffle spacing ranges, the experimental results compared better with the Bell-Delaware-based calculations, while the opposite is true for the Tinker-based calculations.

Pekdemir *et al.* (1993:10) also studied the pressure drop characteristics of an STHE, making use of pressure-tapped instruments. The results were the following: i) Reynolds numbers have a strong influence; ii) a lower Reynolds number unifies fluid distribution; iii) pressure drop increases with increasing baffle spacing and Reynolds numbers; iv) pressure drop is more prominent in the central flow plane than bypass regions; and v) radial pressure drop differences decrease with increasing baffle spacing.

In two related studies, Li and Kottke (1998b:433) established that their pressure drop measurements correlated well with the values that were calculated by the VDI-Wärmeatlas (cited by Li and Kottke, 1998b:433), although they were only valid for Reynolds numbers above 3000. The authors concluded that an increase in baffle spacing decreases the total overall pressure drop. In the alternative paper (1998c:1311), the effect of baffle-shell leakage had a positive influence on the overall pressure drop, also validated by the VDI-Wärmeatlas calculations (cited by Li and Kottke, 1998c:1311). Their study that was conducted on disc-and-doughnut baffles (1999:3521) resulted in an improvement in pressure drop of 55% and a greater heat transfer/pressure drop ratio compared to a similar single-segmental configuration.

Prithiviraj and Andrews (1998a:799; 1998b:828) studied single-segmental STHEs in three dimensions. The pressure drop correlations that were used for the study were developed by Zukauskas and Rehme (cited by Prithiviraj and Andrews, 1998a:805) for cross-flow and axial-flow, respectively. Experimental and numerical pressure drop values were obtained and shown to have a minimum and maximum variation of 9% and 24%. The authors varied the number of cross-flow passes, nozzle diameters, tube layouts, mass flow rate and baffle-cut percentage. In all, the pressure drop correlations were shown to be sound. Decreasing baffle-cut percentage and nozzle diameter had an increasing effect on pressure drop, while an increase in mass flow rate and number of passes had a similar effect. Leakages, on the other hand, decreased the total pressure drop.

Cebrucean *et al.* (2009:28) modelled the pressure drop in STHEs with single-segmental baffles. The comparison of the study with previous work showed slight but significant differences in the accuracy of pressure drop values, mainly due to differences in viscosity.

Wang, S. *et al.* (2009:2438) investigated the effect of sealer strips to combat baffle-shell leakage flows. The pressure drop in all experimental setups were measured to be 44% to 49% higher with the addition of the strips, but the exergy coefficient for the heat exchanger increased with this addition.

Ozden and Tari (2010:1014) simulated a simple STHE thoroughly. The authors calculated values of pressure drop and Nusselt numbers for the configuration from the work of Kapale and Chand (2006:610) as well as the Bell-Delaware and Kern methods. These values were then compared with the results of the simulations by using several turbulence models and discretization schemes. Numerical pressure drop values were compared with analytical values and were discovered to vary inconsistently with the chosen baffle spacing and baffle-cut percentage – the smallest spacing had the smallest percentage differences.

In a study on rod-, segmental and H-type baffles, Yongqing *et al.* (2011:53) used various flow rates to determine the pressure drop over STHEs. The CFD results were shown to have a maximum error of 9,34%, with accuracy increasing with the increase in mass flow rate. H-type baffles performed adequately, with segmental baffles having the best performance in terms of the heat transfer/pressure drop ratio.

Various factors have an influence on the pressure drop of STHEs. The following geometric parameters are indirectly proportional to pressure drop: baffle spacing, leakage, baffle-cut percentage and inlet/outlet nozzle diameter, whilst the Reynolds number and number of passes are directly proportional. One can also consider the effect of fluid properties on the pressure drop. Disc-and-doughnut baffles can result in a possible 55% decrease in pressure drop, while sealer strips can increase the pressure drop by 49%. Determining pressure drop by means of the Bell-Delaware method is more accurate at a smaller baffle spacing and higher flow rate, with possible errors of up to 25%. This method is one of the fundamental and most accurate ways of determining pressure drop and is discussed below, along with others.

2.3.2. Pressure drop correlations

Donohue (1949:2511) developed heat transfer and pressure drop correlations for un-baffled, segmentally-baffled and disc-and-doughnut-baffled STHEs. The equations were developed by using experimental data. The author stated that disc-and-doughnut-baffled heat exchangers would have better heat transfer than a single-segmental configuration if compared at flow rates yielding similar pressure drops.

The pressure drop and heat transfer characteristics of several studies on STHEs were investigated by Emerson (1963:668). An investigation into an integral method of determining the pressure drop across STHEs with segmental baffles was done and found to be accurate for simple geometries only. In order to account for the increase in complex geometry, an analytic approach had to be developed to account for various internal flow paths and respective effects. For the integral approach, the author reported on Short (cited by Emerson, 1963:668), who meticulously varied single parameters of STHEs to obtain an experimental correlation for pressure drop. From an analytical viewpoint, several authors were cited, discussing correlations that were developed experimentally for rectangular and cylindrical tube banks, bundle-shell bypass flow (cross-flow bypass), baffle window flow, baffle-shell leakage and stream velocities within a baffle compartment.

Gaddis and Gnielinski (1997:159) improved on the equations of the Bell-Delaware method, found in the Heat exchanger design handbook no. 3 (Bell *et al*., 1983a). Total pressure drop was determined by evaluating several equations for each section of the heat exchanger, namely in- and outlets, end cross-flow regions, cross-flow regions and window sections. The effects of bundle bypass streams and baffle leakage were also incorporated into the equations. Comparison of the theoretical prediction with experimental measurements gave inconclusive results. In two-thirds of the experimental values that were determined, the pressure drop fell within 35% of that which was predicted by the present method, which translates into possible difficulties with its application. With the wide range of heat exchanger configurations available, design issues will arise if the heat exchanger does not fall within certain geometrical ratios. Consequently, the authors prescribed a validity range within which the equations will be valid.

A second report on the Bell-Delaware method for STHEs was presented by Serna and Jimènez (2005:550). The work that was presented showed little deviation from the original intent of the Bell-Delaware method, but it formulated a relation to determine the shell-side pressure drop in terms of the available heat transfer area.

Kapale and Chand (2006:610) developed a geometrical model to predict theoretically the pressure loss due to inlet and outlet nozzles and baffle compartments in an STHE. The results for Reynolds numbers, ranging from 10^3 to 10^5, show a better correspondence with experimental results than that of other models, having errors of up to 4%.

In a study that was conducted on tube surface configurations, Hosseini *et al.* (2007:1008) developed pressure drop correlations similar to the Bell-Delaware method. Smooth, corrugated and micro-finned tubes were tested; the latter had the largest pressure drop and smallest error.

From the work that has been discussed on pressure drop correlations, two studies are suitable and will consequently be used in the current study. The work of Gaddis and Gnielinski (1997:159) as well as Kapale and Chand (2006:610) were chosen because of the ease of their implementation and accuracy. One should, however, note that certain geometric limits are given for the models, which could affect the precision with which predictions are made. The other models that were discussed have implementation difficulties and uncertainties, or were unavailable for implementation.

As a final remark, an important and relevant study presented by Ozden and Tari (2010:1014) used the Bell-Delaware method to validate the results of the CFD simulations. The correlations that will be used in the present study are both based on this method and are thus deemed sufficient.

2.4. Tube bundles

A discussion on tube bundles is presented in order to give insight into the effect of changes in configurations. Two changes can be implemented, namely orientation and surface finish.

Sparrow and Kang (1985:350) studied the heat transfer and pressure drop characteristics of finned tubes in a rotated triangular array. The fin configurations were varied, according to fin placement, thickness and tip shape. The addition of fins had the tendency to increase pressure drop compared to a plain arrangement, with the exclusion of a backward-facing, contoured fin tip. It was determined that fin thickness had little effect on pressure drop. If the diameter of plain tubes was increased to yield the same pressure drop as finned tubes, the heat transfer of the latter would be significantly higher.

Faghri and Rao (1987:372) studied the heat transfer and flow characteristics of an inline tube array with and without longitudinal fins. Several cases that were tested with varying Reynolds and Prandtl numbers correlated well with published experimental data. Longitudinal finned tubes have lesser heat transfer and similar pressure drop characteristics compared to plain tubes.

Yanez-Moreno and Sparrow (1987:1995) obtained pressure drop and flow distributions experimentally for square and rotated triangular tube arrangements at various yaw angles. The oil lampblack technique was used for flow visualisation and pressure-tapped tubes were placed throughout the array. Nusselt number correlations for tube rows (depth-wise) were obtained at various Reynolds numbers for both configurations for angled flows. The inline arrangement had better heat transfer and pressure drop performance, accentuated by the flow inclination angle. For a fully developed flow regime, pressure drop was found to have a linear relationship with the yaw angle.

Achenbach (1991:207) studied the flow characteristics of rough and smooth tubes in an inline arrangement at high Reynolds numbers. The author clarified the effect of roughness by noting the point of fluid separation, finding that flow over a rougher surface attained separation earlier. An increase in surface roughness at high (trans-critical) Reynolds numbers broadened the wake and fluid-tube separation occurred earlier, resulting in less pressure drop due to smoother streamlines; the inline arrangement had better performance than that of the rotated array. Tube static pressure, skin friction and heat transfer coefficients were presented; it was found that heat transfer for a rough bundle was higher and that the efficiency increased with increase in Reynolds numbers.

Nishimura *et al.* (1993:563) studied the flow and mass transfer characteristics of tube banks in turbulent transitional flow. Deep within the tube bank, vortex shedding was observed for both a rotated and inline square arrangement, where vortex initiation moved upstream with increasing Reynolds numbers; it was noted that the rotated array exhibited quicker transition characteristics.

Wilson and Bassiouny (2000:14) developed a calculation procedure for heat transfer and pressure drop over inline and rotated tube banks in a flow channel for laminar and turbulent flow. Friction factors and Nusselt numbers were presented for the two banks under consideration at various cross-flow Reynolds numbers. For inline and rotated arrangements, it was determined that an increase in pitch increased pressure drop, with optimum heat transfer at a diameter/pitch ratio of 3 and 1.5 respectively. The dependence of these results does, however, not take into account the effects of downstream tube rows.

Iwaki *et al.* (2004:363) used PIV and CFD to study the cross-flow structure over vertical tube bundles. The study was aimed at measuring the wake, turbulent structures and velocity distributions throughout a rotated and inline tube arrangement. For inline and rotated tube bundles in channel flow, the results were quite clear. Particles tended to follow the linear path through the bundle for the inline array, but were diverted onto the following tube in the rotated array, following a sinusoidal pattern through the tube bank. Higher heat transfer and pressure drop can be expected for the latter arrangement.

Khan *et al.* (2006:4838) used an analytical approach to determine the heat transfer from tube banks in cross-flow for rotated triangular and square arrangements. The values that were obtained correlated well with other literature: Zukauskasas well as Zukauskas and Ulinskas (both cited by Khan *et al.,* 2006:4837). The authors concluded that generally, the rotated arrangement had better heat transfer and pressure drop characteristics.

Hosseini *et al.* (2007:1008) experimented on STHEs with different tube surface configurations. Three tube configurations were chosen, namely smooth, corrugated and micro-fin copper tubes. Nusselt numbers for micro-finned tubes were the largest due to the greater surface area. Corrugated tubes have lower Nusselt numbers than smooth tubes. The experimental work correlated well with theory at higher Reynolds and Prandtl number products.

Ibrahim and Gomaa (2009:2158) studied the characteristics of an elliptical tube bank in cross-flow for a rotated arrangement. The tube major axis/minor axis ratios, flow inclination angles and Reynolds numbers were varied in numerical models and verified with experimental results. By increasing the angle of attack, the heat transfer to pressure drop ratio was increased greatly, but the maximum thermo-hydraulic performance was attained at flows that were parallel to the major axis and vice versa.

Hasan and Siren (2011:644) experimentally investigated the flow over a single oval tube for various major axis/minor axis ratios. The heat transfer coefficients were compared to that of circular tubes and were determined to be less at higher Reynolds numbers. On the other hand, the drag coefficients of oval tubes were less, which resulted in better heat transfer/pressure drop ratios.

The results of tube configurations are shown to be unanimous. Rotated arrays have higher heat-transfer rates due to a more complex flow path. This, in turn, results in higher pressure drops compared to those for inline arrays. Tube pitch increases the pressure drop, but an optimum balance between pressure drop and heat transfer can be achieved. Lacking a fully developed flow regime can have a negative impact on the accuracy of pressure drop correlations, as many correlations are based on a developed flow field. The addition of shaped, profiled or roughened surface configurations can greatly increase the heat transfer rates of tube bundles without greatly increasing the pressure drop. This addition should be reconsidered, taking into account the effects of fouling, which is an important consideration in the current study.

2.5. Turbulence modelling

Turbulence modelling constitutes a large, if not the greatest, part of CFD. The following section briefly discusses the background of the models and continues to demonstrate their versatility as well as shortcomings. Factors that need to be considered when evaluating the effectiveness of different turbulence models are accuracy, stability, solution time, flexibility and resource intensiveness. The literature also gives other insights into selection considerations, namely application suitability, flow pattern/geometry, boundary values and performance.

2.5.1. Turbulence modelling – an overview

According to CFD Online (2011) and Chung (2002:683), turbulence modelling can be split into four categories:

- Reynolds-averaged Navier-Stokes (RANS) models
- Large-eddy simulation (LES) models
- Detached-eddy simulation (DES) models
- Direct numerical simulation (DNS) models

These modelling methods differ in the extent to which the turbulent structures/scales in the flow are solved. When using DNS, the total length and time scale of the turbulence is solved from the largest to the finest Kolmogorov scale, proportional to the kinematic viscosity and indirectly proportional to the turbulent dissipation rate $(v^3/\varepsilon)^{0.25}$. This requires a very fine grid resolution $(N \propto Re^{2.25})$ as well as large computational resources (Chung, 2002:703). DNS is not available in STAR-CCM+.

LES has more realistic resource requirements, although it still is computationally intensive. The defining factor of the LES method is the direct solution of larger eddies and the modelling of smaller ones. The grid size is less complex than what is required for DNS, thus eddies are modelled if they fall in between the grid, while those that are able to be captured are solved by filtered Navier-Stokes equations. DES is a combination of LES and simpler RANS models; the hybrid approach solves the turbulent structures in the main flow field and uses RANS modelling techniques to solve the intricate near-wall regions (Chung, 2002:703). According to the STAR-CCM+ User Guide v.6.06 (2011:2751), both of the methods are inherently time consuming, which immediately eliminates them.

In the development of the RANS models, the Navier-Stokes equations are split into a fluctuating component and an average component. The effect of separating the equations results in an extra term in the momentum transport equation, called the Reynolds stress term (STAR-CCM+ User Guide v.6.06, 2011:2737).

Due to the complexity of the problem at hand, RANS models are less resource intensive and thus more applicable to this study. The models are designed for closure of the time-averaged Navier-Stokes equations where unknown variables (Reynolds stresses) are introduced in the momentum transport equation (STAR-CCM+ User Guide v.6.06, 2011:2737). RANS methods are divided into three constituents: linear eddy viscosity models, non-linear eddy viscosity models and Reynolds stress models (RSM) (Chung, 2002:703; CFD Online, 2011).

2.5.2. Reynolds-averaged Navier-Stokes models

The following are mostly linear eddy viscosity models of which the degree indicates the number of additional differential/transport equations to be solved:

a) Zero-equation models
   - Prandtl's mixing length model
   - Cebeci-Smith model
   - Baldwin-Lomax model
   - Turbulent heat flux vector

b) One-equation models
   - Spalart-Allmaras model
   - Prandtl's one-equation model
   - Baldwin-Brath model

c) Two-equation models
   - Standard k-ε model
   - Realizable k-ε model
   - RNG k-ε model
   - Wilcox's k-ω model
   - Wilcox's modified k-ω model
   - SST k-ω model

The cubic k-ε model and the explicit algebraic Reynolds stress model are two non-linear eddy viscosity models, unavailable in STAR-CCM+ (CFD Online, 2012).

Of these turbulence models that have been mentioned, four RANS turbulence modelling options are available in STAR-CCM+:

- Spalart-Allmaras models
- K-ω models
- Reynolds stress transport models
- K-ε models

Spalart-Allmaras model

The Spalart-Allmaras turbulence model is a one-equation turbulence model which solves the transport equation for turbulent viscosity (CFD Online, 2011). The use of the model gained great acceptance in the aerospace industry for its straightforward CFD implementation and reasonably accurate solution of flow over an airfoil fuselage and other aerospace-related external-flow applications. Areas where the model has inadequate performance are plane and round jet, or generally jet-like, free-shear regions or complex recirculation flows, rendering it impractical for the present study (STAR-CCM+ User Guide v.6.06, 2011:2780).

K-ω model

The k-ω model is a two-equation model in which transport equations for turbulent kinetic energy and the specific dissipation rate ω (dissipation rate per unit turbulent kinetic energy) are introduced for closure. The k-ω model is well suited for boundary layer flows under adverse pressure gradients and the base equations do not have to be modified for the boundary region or viscous sublayer. No wall distance calculations are required by the model and history effects such as diffusion and convection of turbulent energy are captured. A disadvantage of the k-ω model is the boundary layer calculation's sensitivity to the specific dissipation rate that occurs in the free stream, resulting in the inlet boundary's choice of ω to be critical. The model is used as an alternative to the Spalart-Allmaras model for aerospace applications, also rendering it impractical for the present study (STAR-CCM+ User Guide v.6.06, 2011:2876).

Reynolds stress model

The Reynolds stress model is a complex seven-equation model which has extensive computational requirements. The seven equations that are solved are the transport equations for the components of the specific Reynolds stress tensor. Six are for the symmetric Reynolds stress tensor in three dimensions and an additional equation is for the turbulent dissipation rate ε. The complexity of the model grants reliable use in highly anisotropic flow situations such as swirling motion, secondary flows in ducts, streamline curvature and rapid changes in strain rate. This model would have been a viable option for the present study, but the RSM equations are numerically stiff by nature, increasing the computational time even further, resulting in its elimination from the current study (STAR-CCM+ User Guide v.6.06, 2011:2919).

2.5.3. K-ε model

The remaining and chosen family of models for the simulation was the k-ε model, a semi-empirical, two-equation model. Transport equations are solved for k, the turbulent kinetic energy and ε, the turbulent dissipation rate. It is the most widely used model for basic industrial uses and many derived forms of the model exist for different applications (Aslam Bhutta *et al.*, 2012:1; STAR-CCM+ User Guide v.6.06, 2011:2808). It allows a balance between accuracy, computational requirements and robustness for normal and complex flows that lends itself to recirculation, including or excluding heat transfer, but has difficulty in simulating flows with large pressure gradients (CFD Online, 2012). STAR-CCM+ gives the user a choice between the following seven k-ε models, of which the first two are the only available models that have been modified for multiphase flow:

- Standard k-ε (multiphase)
- Standard two-layer k-ε (multiphase)
- Standard low Reynolds number k-ε
- Realizable k-ε
- Realizable two-layer k-ε
- Abe-Kondoh-Nagano low Reynolds number k-ε
- $\overline{v^2} - f$ low Reynolds number k-ε

In order to determine the most suitable model, the criteria in the preceding sections will be applied, along with references from literature.

Low Reynolds number models are specifically designed to explicitly solve the flow in the viscous sublayer. The standard low Reynolds number and Abe-Kondoh-Nagano models use damping functions to model the coefficients C_μ, $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ as functions of turbulent Reynolds numbers and, in most circumstances, wall distance. On the other hand, the $\overline{v^2} - f$ low Reynolds number model does not apply a damping function or wall distance approach, but models near-surface flows automatically through use of a velocity scale and an elliptic relaxation function. The three low Reynolds number models are specifically suited for use in the simulation of low Reynolds number flows (STAR-CCM+ User Guide v.6.06, 2011:2843). By eliminating these from the available options, the choice remains between the standard and realizable models that are used with either a wall function or a two-layer approach.

2.5.4. Standard k-ε model

The standard k-ε model in STAR-CCM+ is a combination of two developments, namely equations, proposed by Jones and Launder, and coefficients, proposed by Launder and Sharma (both cited by STAR-CCM+ User Guide v.6.06, 2011:2810). This combination is confirmed to incorporate the most significant developments.

The two-layer approach allows the k-ε model to be applied within the viscous sublayer of the flow (see Chapter 3.5). Rodi (cited by STAR-CCM+ User Guide v.6.06, 2011:2809) developed the method to be a substitute for the standard low Reynolds number damping function approach. The computational domain of the boundary is divided into two sublayers: one adjacent to and one near the wall. Values of turbulent dissipation and turbulent viscosity, ε and μ_t, are described as functions of the wall distance in the wall-adjacent layer. The near-wall layer links the adjacent layer with the free stream, blending the value of ε smoothly between layers, whereas the value of turbulent kinetic energy is solved for the entire domain.

Lai and Yang (1997:575) investigated the influence of ring devices on turbulence in developing and fully developed flow. The standard and three other low Reynolds number k-ε models were selected for the study. None of the turbulence models that were used could describe the induced suppression accurately, although the low Reynolds number models predicted it to some extent. Turbulence enhancement was predicted by the standard model's employment of wall functions.

Lakehal and Rodi (1997:65) studied flow past a square cylinder in fully developed channel flow. The standard k-ε model was used in the simulation, along with different boundary-layer solution approaches. By comparing the performance of the wall functions and the two-layer approach with LES data, the authors concluded that the latter surpassed the former in its ability to predict the separation and reattachment points accurately. The complex converging-diverging vortex structures could also not be described accurately enough. One drawback that was observed with the two-layer approach was its inclination to significantly increase the computational time that was required for convergence.

Murakami (1997:1) discussed the difficulties that turbulence models have in predicting the flow around bluff bodies. The author states specifically that the standard k-ε model has the tendency to over-predict the production of turbulent kinetic energy.

Eight k-ε models were selected by Costa *et al.* (1999:4391) to evaluate their accuracy in simulating mixed convection in internal flows. Once again, the Jones and Launder as well as Launder and Sharma models formed part of the selection (cited by Costa *et al.,* 1999:4397). All the models predicted the main characteristics of the flow field adequately, but both models overpredicted the near-wall turbulent diffusion by an approximate order of magnitude. The authors mentioned that the latter model had a tendency to overpredict turbulent diffusion and heat transfer in forced or natural convection flows. However, if applied, the Yap correction greatly improved the accuracy. Overall, the model of Nagano and Hishida (cited by Costa *et al.,* 1999:4406) had the best performance, while models that used wall functions had accuracy difficulties.

Lew *et al.* (2001:209) observed instability of the standard model, which is inherent to convection-diffusion-reaction equations. The instability was said to arise from interaction of discretization errors and certain non-linear characteristics.

Stevanovic *et al.* (2001:1091) investigated the capability of the standard k-ε model to predict shell-side flow and heat transfer characteristics of STHEs. The model was shown to overpredict heat transfer and pressure drop when compared to a modified k-ε model, specifically designed for fluctuating turbulent dissipation rates which are encountered in recirculating flows.

Han *et al.* (1996:2717) examined the use of the standard k-ε model in modelling the rapid distortional flows in diesel engine cylinders. The standard model had problems in predicting the turbulent length scale accurately, but modifications to the length scale improved the results at various running speeds.

The effectiveness of the standard k-ε model in the prediction of flow in blood pumps was investigated by Song *et al.* (2003:935). The authors stated that the standard k-ε model is outperformed by the standard k-ω model because of its inability to simulate near-wall flows with standard wall functions accurately.

Both models that were used in the description of the standard k-ε model utilized in STAR-CCM+, along with two other low Reynolds number k-ε models, were implemented for the determination of the heat transfer characteristics of supercritical carbon dioxide (Dang and Hihara, 2004:748). The authors investigated the validity of conventional turbulence models and observed that the Jones and Launder model (cited by Dang and Hihara, 2004:748) performed the best.

Dong *et al.* (2008:651) simulated longitudinal, steady-state flow effectively in a rod-baffle STHE with the standard k-ε model. Two heat transfer and pressure drop correlations were used to validate the heat transfer and pressure drop values that were obtained in the simulation; errors of less than 17% were observed for both parameters. On the other hand, the influence of the shell, inlet and outlet, among others, were not taken into account, which may affect the results negatively.

Kim *et al.* (2008:1293) studied the accuracy of different turbulence models in mixed convection. The standard, Abe-Kondoh-Nagano and $\overline{v^2} - f$ were the applicable k-ε models. All three models were singled out for their ability to predict the heat transfer, velocity and friction coefficient characteristics of buoyancy-driven flows well, but the standard model performed the best. It was mentioned that the accuracy of the standard model could be attributed to opposing inaccuracies in the eddy viscosity and damping functions. Wall-bounded flow predictions were also noted to be inaccurate.

Although some of the conditions for and applications of the turbulence models were not specifically relevant, many articles still question the validity of the standard model in the respective situations. Consequently, focus will be shifted to the application of the realizable k-ε model in order to determine its suitability for the current study.

2.5.5. Realizable k-ε model

According to Shih *et al.* (1995:231), who are the originators of the realizable k-ε model, the difference between the standard and realizable dissipation equations lies in the turbulence source term. Within the realizable model, the Reynolds stress terms are omitted, resulting in greater robustness when the model is used in conjunction with second-order closure schemes, due to better numerical behaviour of the mean strain rate compared to the behaviour of the Reynolds stress terms. The authors believe that the turbulent vortex stretching and dissipation terms of the model are described in a better way.

When compared to the standard k-ε model for flows with a high mean shear rate and large separation zones, the standard model underperformed by overpredicting the eddy viscosity. It also showed difficulty in the determination of the appropriate turbulent length scale. The realizable model did not exhibit these difficulties. In rotating homogeneous shear flows, boundary-free shear flows, channel and boundary layer flows as well as backward-facing step (BFS) flows, the realizable model outperformed the standard model (Shih *et al.,* 1995:231).

Along with the modification of the transport equation for ε, the turbulent viscosity coefficient C_μ was no longer described as a constant, but by an equation. It stemmed from a limitation of the standard model when solving the normal Reynolds stresses, namely $u'_i u'_i$. This alteration alleviated certain difficulties that were experienced within the mathematical description of normal stresses, observed within turbulence structures (realizability constraints). The coefficient variability compared well with experimental observations (Shih *et al.,* 1995:231).

Brennan (2003:16) simulated centrifugal separators by using FLUENT. The author tested the software's ability to simulate gravity-driven, free surface, open-channel flow by using the realizable and standard k-ε, k-ω and the Reynolds stress turbulence models. Results of the simulations showed that the k-ω model predicted the flow behaviour (velocity profile and free surface height) within the law of the wall region the best, whereas the RSM and realizable models had accuracy difficulties because of the wall function approach, used by these models, which overpredicted turbulent viscosity.

Bartosiewicz *et al.* (2005:70) investigated the applicability of six different turbulence models on the prediction of flow characteristics within supersonic ejectors. Once again, the standard and realizable models were among the chosen turbulence models. Both models failed to predict the shock phase, strength and mean line of pressure recovery accurately.

Kim *et al.* (2005:493) studied the influence of different wall-treatment options and turbulence models on simulations of BFS flows. The realizable and standard k-ε models were two of the six models that were applied. Results from the study showed that the realizable k-ε with two-layer model gave the most accurate correlation with experimental data and, compared to the wall function approach, the two-layer model predicted the velocity distribution the best.

In a study by Van Maele and Merci (2006:138), the standard and realizable models were used to simulate different buoyancy-driven flows. The buoyancy production of turbulent kinetic energy was addressed by the simple gradient-diffusion hypotheses (SGDH) and general gradient diffusion hypotheses (GGDH). An axisymmetric turbulent buoyant plume and a plane turbulent buoyant wall plume are the two cases that were tested by using FLUENT. The numerical results were verified with experimental data and established to be accurate, delivering better results with the GGDH and providing better distinction between the effects of the standard and realizable turbulence models. The SGDH was unable to capture the effects of buoyancy and the realizable turbulence model was deemed to perform the best.

Ye and Gui (2006:10) numerically simulated stagnation flows that were observed in the spray painting process. Their aim was to observe the effect that different wall treatment options have on turbulent wall-bounded flow. The standard, realizable and Reynolds Stress models were tested in conjunction with standard, enhanced and two-layer wall treatment options that are available in FLUENT. The realizable model with enhanced wall treatment predicted the simulation reliably when compared with experimental data. The two-layered approach had difficulty in predicting the velocity and turbulent kinetic energy close to the wall, while the enhanced treatment was insensitive to the near-wall grid spacing.

In a study on mixed, natural and forced convection, Cable *et al.* (2007:1) studied the validity of several turbulence modelling options. The realizable k-ε model was included in the study and had the best correlation with LES and DES data, but the study was deemed to be slightly ineffective due to incomplete boundary conditions.

Researchers in Sweden studied the air flow, turbulence intensity and thermal fields inside a packaging facility by using different k-ε models (Rohdin and Moshfegh, 2007:3882). Both the standard and realizable models exhibited negligible difference with the experimental data.

Temperature differences of all models were within the measuring device's error margin and velocity differences were double the margin.

Jayakumar *et al.* (2008:12) used the realizable k-ε turbulence model with standard wall functions for the simulation of a helical coil heat exchanger. The results of the simulation correlated well with experimental data and fell within the range of experimental error, but the authors doubted the suitability of the model to simulate helical flows due to the strong centrifugal and torsional flow characteristics that are intrinsic to the heat exchanger.

Chevron-type plate heat exchangers were studied by Tsai *et al*. (2009:574). The authors investigated the use of CFD for the prediction of pressure drop and complex flow patterns. The performance of different turbulence models were investigated with the realizable k-ε model, generating pressure drop results 20% lower than those of experiments, but capturing flow structures adequately.

Marzouk and Huckaby (2010:12) studied swirling gas-particle flows by using different k-ε models and particle-parcel relationships. The setup consisted of coaxial vertical pipes, containing primary particle-laden air flow and secondary swirling annular air flow. The researchers established that the standard model delivered the best results, based on the mean gas-phase velocities, while the realizable model, in comparison, had difficulty in predicting the correct radial velocity, which could be attributed to the model's focus on flows with a high mean strain rate.

Ozden and Tari (2010:1014) studied STHEs, investigating different turbulence models' abilities to predict the flow fields accurately. The average Nusselt number and pressure drop values of the simulations were compared with the values predicted by the Kern and Bell-Delaware methods for accuracy. The Spalart-Allmaras, standard and realizable models were tested with first- and second-order discretization schemes. The realizable model with second order discretization had the best correlation with analytical results.

Chamoli *et al.* (2011:18) studied the heat transfer characteristics of a square tube bank with circular profile fins by using the standard and realizable model to obtain Nusselt numbers. Swirling fluid motion, as well as the detachment and reattachment of fluid around the circular fins, were successfully captured by the realizable model, resulting in the best available agreement with experimental data.

In a study on corrugated plate heat exchangers, Gherasim *et al.* (2011:1499) focussed on the ability of two equation turbulence models to predict the complex flow and heat transfer characteristics accurately. The realizable model was included in the investigation, along with three types of wall functions that are available in FLUENT. The realizable model with two-layer wall functions was shown to have the best performance. Numerical data for the friction factor and Nusselt number were compared with experimental data for various Reynolds numbers, of which the turbulent regime showed maximum and average errors of 3,4% and 2,5% for the friction factor and 15,1% and 8,6% for the Nusselt number.

The articles that were discussed in this section highlight the advantages and disadvantages of the standard and realizable k-ε models in various circumstances, evaluated on the criteria that had been mentioned initially. It is important to note that the correct choice of model for the application is crucial for accurate results. A clear indication of the superiority of the realizable model can be seen in the various relevant situations and applications that were presented, but the following articles have to be mentioned specifically.

The article published by Ozden and Tari (2010:1014) gives a clear indication of the relevance and supremacy of the realizable model in simulating STHEs. The model is proven to outperform the standard k-ε and other models with reasonable stability, within adequate computational time and with the greatest accuracy. The modelling was performed on a geometry that is similar to that in the current study, further justifying its selection.

A second argument can be made when considering the studies of Kim *et al.* (2005:493) and Chamoli *et al.* (2011:18). Both studies were on geometries that represent similar elements in the present study – tube banks in cross-flow and BFS. Flow in an STHE is continuously built up of tubes in cross-flow, followed by flow over and around a baffle (similar to the recirculating flow of a BFS), into cross-flow again. This is an indication that the realizable k-ε model is suited and a good choice for the current geometry.

As a final remark, De Wet (2011) confirmed in verbal communication with the author the model's selection, weighing accuracy, computational requirements and suitability.

2.6. Multiphase modelling

It is of great importance to discern if the simulation of multiphase fluids can be achieved successfully. The following papers discuss the modelling of multiphase fluids in various respects.

Tu *et al.* (1998:238) studied gas-particle flows for several particle diameters in an inline tube array. A RANS turbulence model, along with the Eulerian approach, was used to resolve the flow paths and concentration build up, which compared well with experimental laser Doppler anemometry results. Phase velocities and particle concentrations were described accurately, although different flow characteristics for both phases were observed at various particle sizes.

The influence of increasing the number of Eulerian phases in a mixture was simulated by Ibsen *et al.* (2000:11). The study was done on laboratory scale, circulating fluidized beds with several particle sizes that were introduced into the simulation. Performance was deemed adequate when compared with experimental data, but exhibited discrepancies during experiments with high-density particles.

Brennan (2003:16) studied multiphase flows within gravity sluices. The author tested two models' ability to predict the phase interfaces: the volume of fluid (VOF) method and the algebraic slip mixture model. The latter performed the best by solving the free surface position with sharper contours, accompanied with grid adaptation.

Frank *et al.* (2004:9) used CFX-5 to simulate gas-liquid vertical pipe flow. The Eulerian framework was effectively used to simulate the multiphase slug flow. Experimental validation of the CFD model was deemed to be successful.

Zhao and Fernando (2008:20) modelled the scour of sand below a sagging pipeline. The researchers used FLUENT to replicate the transport of sediment from under a pipeline downstream to a formed mound. The sagging of the pipeline increased the turbulent intensity enough to lift the surface sediment, increasing erosion. Overall, the agreement with laboratory data was satisfactory and the flow characteristics of both phases were captured effectively by means of RANS turbulence models.

Turbulent sediment-laden channel flow was simulated by Cantero *et al.* (2009:28). An Eulerian-Eulerian approach was used to solve flow stratification, turbulence quantities in the main stream and near-wall flows, as well as the mean flow velocities. Several configurations were successfully simulated, observing variables such as settling velocity and wall turbulence structures.

Pascual *et al.* (2009:5161) tested the applicability of CFD to the design of scraping devices in a scraped heat exchanger crystallizer. Two scraper geometries were investigated to determine the flow paths of particles, both numerically and experimentally, by using dye injection during the investigation. Simulation and experimental results were compared in order to determine the extent of the scraper's ability to direct particles into the bulk of the main fluid stream. Good correlations between experimental and numerical results were found.

Marzouk and Huckaby (2010:12) used particle-parcel relationships to simulate swirling gas-particle flow in circular pipes. The method was deemed to be successful due to the simplifying nature thereof, reducing computational time and cost.

The heat transfer characteristics of particles in gas flows were studied by De Bellis and Catalano (2012:1016). By using FLUENT, the authors employed a mixed Eulerian-Lagrangian model to predict the heat transfer of the particles in suspension. RANS turbulence models were used and he study resulted in an effective description of the heat transfer characteristics.

The cited literature provides guidance on several aspects of multiphase simulation that could be vital.

Transient multiphase simulations are intrinsically difficult with respect to their stability, combined with the time consuming nature thereof. Several studies have shown the ability of available CFD software to solve complex flow structures successfully with simple RANS turbulence models in lieu of DES, LES or DNS. High particle concentrations can be problematic, leading to further instability. These difficulties can be addressed by means of high under-relaxation factors and small time steps.

Particle sizes can also influence the flow patterns that are generated in the simulation; it is thus important to simulate the actual size range in order to have an accurate representation of the phase concentrations within the model.

It was further established that the use of the correct multiphase model is essential for proper results. The Eulerian model was determined to be best suited to provide information on the distribution of phase concentrations (Zhao and Fernando, 2008:20). It is also the only model that is able to simulate the interaction between fluids and solids as well as solving for the concentration of the different phases.

As a final comment on this section, it is noted that current and specifically relevant literature is difficult to obtain. Great reliance will thus be placed on the experience of experts in the field.

2.7. Conclusion

This chapter described the areas of research that are necessary for the successful execution and completion of the study. A thorough investigation was done and the following conclusions can be drawn.

Little has been done and published in open literature concerning the experimental evaluation and use of CFD to simulate STHEs. Even less work has been done on the comparison of configurations with a fixed mass/velocity flow rate. This deficiency, however, highlights the value of and need for the present study.

Many authors have used the Bell-Delaware approach to determine or validate the pressure drop (and heat transfer) characteristics of heat exchangers numerically. The calculations were shown to be sound and accurate for various types of heat exchanger configurations if certain geometrical parameters fall within the model boundaries. Based on the Bell-Delaware method, the work of Gaddis and Gnielinski (1997:159) as well as Kapale and Chand (2006:610) will be used to verify the pressure drop that has been determined from the CFD results in the present study.

The articles that were presented on tube bundles provided insight into the effectiveness of certain configurations. Some notable conclusions are that rotated arrangements allow higher heat transfer rates at the cost of increased pressure drop, whereas inline arrangements offer lower pressure drops at similar heat transfer rates. This inclination can be taken advantage of, dependent on the pressure drop and heat transfer requirements. Compact tube bundles increase heat transfer and pressure drop and tube shapes and geometrical modifications can be used successfully to increase the effectiveness of the tube bank.

Studies concerning turbulence modelling shed light on one specific model due to its applicability to all aspects of flow that have been encountered in this study. The realizable k-ε model was used by several authors to simulate flow within heat exchangers, flows prone to recirculation and, in some cases, particulate flows. Several authors commented on the model's inability to simulate certain flows accurately, but none of these are applicable to heat exchangers specifically. On the other hand, with specific reference to the article by Ozden and Tari (2010:1014), the realizable model was deemed to have the best performance in predicting shell-side flow. The developers (Shih *et al.,* 1995:238) also commented on the specific applicability and transcendence of the standard model in varying flow situations.

Another turbulence model was observed to be viable. The RNG k-ε model was shown by several authors to have similar performance to the realizable model. It has yet to be implemented in STAR-CCM+, but De Wet (2011) queries the reason for its absence.

The effect of wall treatment is important. The literature that was presented shows the influence of different wall treatment options on accuracy, stability and time, and many articles illustrate the superiority of a two-layer wall treatment in lieu of a wall function or low Reynolds number approach.

Multiphase simulations were shown to be effective in predicting the behaviour of sediment in recirculating flows and sedimentation. The Eulerian approach (as opposed to the Lagrangian or other approaches) will be used to describe the sedimentation of particulates in the heat exchanger. Concentrations of particles in certain areas of the heat exchanger are necessary and not the particular flow paths of single particles, as described by the Lagrangian approach.

From the literature that was presented, it is shown that CFD can be employed effectively to determine the flow characteristics of heat exchangers in steady-state and transient multiphase simulations. Now that the different concepts mentioned at the beginning of the chapter have been discussed by noting the use thereof within literature and conclusions drawn from the research, it is necessary to address the theory behind these concepts.