19th Australasian Fluid Mechanics Conference
Melbourne, Australia
8-11 December 2014

Design of a Test Rig to Improve Thermal Design Approach for Evaporators for Organic Rankine Cycle Power Plant

S. Siwach and M. Jermy
Department of Mechanical Engineering, University of Canterbury
Private Bag 4800, Christchurch 8140, New Zealand

Abstract

The need for increasing the efficiency of energy production systems and harnessing renewable resources of energy has necessitated an increase in efficiency of all the components involved in a power generation system. The transfer of heat from the resource to the motive/working fluid is accomplished by heat exchangers. The heat exchangers have a major role to play in both the economics and operational performance of a power plant in the form of being a major part of capital cost and having a bearing on performance and efficiency of entire plant. A better understanding of the functioning of heat exchangers will translate directly into increasing the efficiency of entire power plant and reducing initial and maintenance costs, by providing us correctly sized heat exchangers with higher thermal performance without compromising on pressure drop aspect. In this project, the endeavour is to seek & validate (with modifications, if required) heat transfer models for an accurate description/analysis of heat transfer process occurring in heat exchangers (HEs) of common types (viz. plate, shell and tube) and then validate them against both CFD and experimental data, so as to lend credence to both the CFD simulations and also the theoretical model(s) (selected ones).

Introduction

Challenges for a vaporizer in Duty

The vaporizer is an essential and critical component in a power generation system. It is where the motive/working fluid undergoes change of phase to power the prime rotor (expander) and generate electricity or produce required mechanical work.

It has been estimated that more than half of the heat exchangers employed in process industries involve two-phase flow on the shell-side [26], and yet the two-phase flow patterns in cross-flow have received much less attention than in-pipe two-phase flow patterns. A few of the studies are by Noghrekar et.al. (1998), Jensen et.al. (1996, 1997) [10, 16-18]. Also, shell-side pressure drop is mechanistically different; as the pipe flow pressure drops are due to wall friction, whereas shell-side pressure drops are due to flow separation and re-attachment phenomena [4].

Void Fraction

Prediction of void fraction inside vaporizers is of utmost importance if we want an accurate prediction of local heat transfer coefficient due to the fact that heat transfer mechanisms, and hence the correlations required for prediction, change as the flow pattern changes with increasing void fraction value [5].

There are 3 main types of flow models that can be used for prediction of void fraction values, and their use depends on the particular application. These are: a) Homogeneous flow model, b) In-tube flow model, c) Separated flow model

Several investigators have proposed void fraction correlations, e.g. Schrage et.al. (1988), Dowlati et.al. (1990) and Feenstra et.al. (2000); while Ishihara et.al. (1980), Xu et.al.(1998) and Simovic et.al. (2007) have proposed methods for frictional pressure drop [26, 4, 9, 15, 23].

Also, most of the studies done to develop two-phase void fraction prediction models used adiabatic two-phase flows [2], which is a situation totally different from actual operating conditions, where the vapor is generated on the tubes and thermo-hydraulic parameters keep changing in both vertical and horizontal directions inside a tube bundle.

Construction Requirements

The vaporizer has to have a thermally efficient and easy to clean, yet robust construction. The goal is to gain a deeper understanding of the influence of mechanical (geometrical) parameters (tube surface characteristics, tube pitch, tube diameters, and tube layout) and operational parameters (temperatures and pressures of fluids, scaling of tubes) on the heat transfer during phase change process. A significant constraint to designing of a heat exchanger with high heat transfer efficiency is that the mechanical design should be simple enough to lend itself to periodic cleaning procedures and within reasonable cost.

Thermal Design

The thermal design of the vaporizers is a complex issue due to the complex physics behind the boiling process. Boiling is a complex process in which mass, momentum and energy transfer (single- and two- phase) involving a solid wall, liquid and vapor are tightly coupled [21]. There are a number of factors that affect the boiling process and the mechanism and extent of influence is not fully understood due to a lack of well-established mechanistic models and the lack of computational resources to simulate the phase change, boiling process on large geometries.

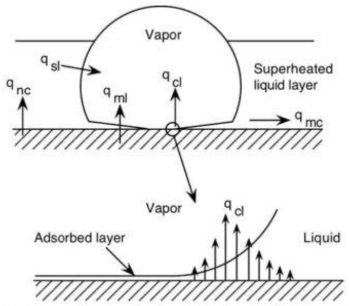
The current study endeavours to validate the theoretical (empirical) models with the CFD analysis (ANSYS Fluent) and experimental observations. The comparison of results coming out of the three different approaches will provide for a credible and logical way to test new models and validate their accuracy. The lab being set up will serve to provide a basis for further research into the design of heat exchangers.

Bubble Dynamics

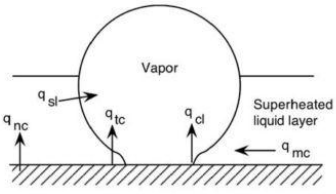
The dynamics of bubble generation and departure are still being investigated and are not fully understood. There are three main models/hypotheses for heat transfer bubble generation and departure process:

- The "Transient Conduction Model" (Han & Griffith(1965), Mikic & Rohsenow(1969))
- "Microlayer Heat Transfer Model" (Snyder & Edwards(1956), Moore & Mesler(1961), Hendricks & Sharp(1964), Cooper & Lloyd(1969))
- "Contact-line Heat Transfer Model" (Stephan & Hammer(1994))

There are a variety of methods available for micro-measurements of heat transfer during bubble generation and departure: Liquid Crystals, Micro-heater Array (constant temperature), Micro heat-flux sensor's array, Infrared Cameras (local heat flux is obtained by performing energy balance on each individual pixel), Interferometric techniques (Judd & Hwang, [19])



a. Heat transfer mechanisms during bubble growth.



b. Heat transfer mechanisms during bubble departure.

Figure 1. Various heat transfer mechanisms during bubble formation and departure, [21]

In recent experimental investigations the mechanisms of heat transfer during bubbling events under pool boiling conditions have been studied on micro-scales and all the different mechanisms have been properly delineated and their relative contributions measured and presented.

The dominant mechanisms have been found to be transient conduction and micro-convection. e.g. [21] states that the dominant heat transfer mechanisms are transient conduction and micro-convection (bubble agitation) while the microlayer evaporation and contact line heat transfer have a less than 25% contribution. Myers et.al. (2005) have put forward findings similar to [20], also presented results limiting the contribution of microlayer evaporation to maximum of 28.8% (in agreement with [19]), and negligible contribution by contact line heat transfer mechanism; and the contribution of micro-convection was observed to increase as the wall temperature increased, while transient conduction is more dominant at lower surface temperatures.

The main mechanisms are therefore transient conduction and micro-convection with their relative dominance a function of surface temperature; transient conduction dominating at low surface temperatures and micro-convection dominating at high surface temperatures. Another significant finding is that transient conduction starts well before the bubble departure, which is in complete contrast to the usual definition of transient conduction in boiling literature.

Empirical Modeling vs. Mechanistic Modeling

Originally, the process of heat exchanger design has been based on empirical correlations and formulas developed by a number of researchers based on experimental data and observations and using coefficients/exponents for data fitting. It is an effective technique for designing equipment with similarities in geometry and operating conditions during the experiments, but it fails in being a universal method and also does not represent the extent and manner of influence of different factors that affect the final thermal performance of heat exchangers.

Basic Sizing Calculations

The heat exchangers constitute the major portion of capital cost required to set up a power plant. Also, the heat exchangers are the components having a large amount of over-design built into them due to following factors:

- Consideration of future fouling issues
- Current design methods not being highly accurate thus necessitating a significant over-design, 15-20%. As a consequence of many uncertainties in the predictive models for heat transfer in flooded-type evaporators, safety margins taken for the thermal design of heat exchangers are quite large, and result in an overly conservative design of vaporizers [9, 3, 13].

Challenges towards development of more efficient & accurately sized heat exchangers:

- Experimental investigation on industrial sized full-scale heat exchangers is prohibitively expensive
- There have been developed some highly accurate predictive models of mechanistic type also (in addition to numerous empirical correlations available), but they are limited by their requirement of the knowledge of local thermo-hydraulic conditions which are generally not available [9].
- The design of a heat exchanger needs to keep in mind both the heat transfer performance and the accompanying pressure drop, and it is seen mostly that the steps required to increase heat transfer performance lead to higher pressure drop.

Modelling to Select Configuration of Tubes

Tube layout pattern has a significant bearing on both the heat transfer performance and pressure drop characteristics of the heat exchanger. The first step towards effective use of a theoretical model is to be able to do a row-wise simulation of the vaporizer, as the correlations required change with change in quality (void fraction) of the working fluid. This is a coarse form of discretization which can be made finer once a particular theoretical model is finalized for use. A code has been developed using MATLAB that does a row-wise simulation of the vaporizer, and predicts local heat transfer coefficient on every row of the vaporizer. The code first develops a tubesheet model based on the tubes layout pattern and diameter specified by the user. The calculated heat duty is then displayed and user is asked to specify the desired heat duty, and the program alters the tube length to achieve the required heat duty. It is capable of predicting thermal conditions of the vaporizing fluid in case of a vaporizer after every row of the tube bundle. The program can also be used to run off-design simulations and check the compensations required as the operating conditions change.

Sensitivity to Scaling and Fouling

Scaling and fouling are inevitable in geothermal power generation due to the large amounts of dissolved salts and solids present in the geothermal brine.

This plays a crucial role in the life cycle of a heat exchanger. The geothermal power plant needs to produce a constant power output to be fed into the grid, which translates into fixed amount of heat duty required from the exchangers. The drop in heat duty of HE with fouling build up needs to be compensated by varying operating conditions, mainly working fluid mass flow rate.

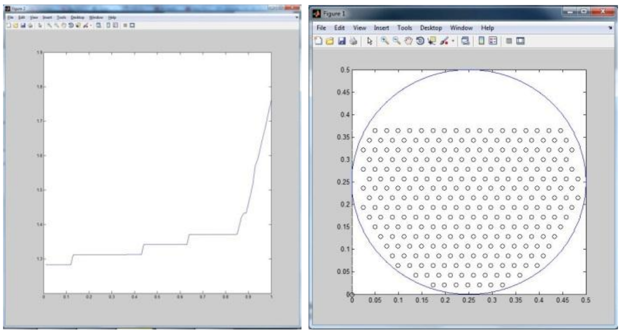


Figure 2. Results of MATLAB code, showing effect on mass flow rate (kg/sec, Y-axis) with fouling build up (mm, X-axis) and tubesheet design

The graph is for a vaporizer with Aprin model used to calculate the local heat transfer coefficient and Feenstra-Weaver-Judd method used to calculate the void fraction.

Consideration and Selection of Different Models to be used for Thermal Design

There are available quite a few models for thermal design of heat exchangers and the choice of the model depends on the duty type of the heat exchanger, i.e. preheater, vaporizer, superheater or a combination of any and all of these duties. For applications without any phase change we refer to Bell-Delaware or Kern method for thermal design of the heat exchanger. The Kern method gives conservative results and is only suitable for preliminary sizing. The Bell-Delaware method is a very detailed method and is usually very accurate in estimating the shell-side heat transfer coefficient and pressure drop for commonly used shell-side geometric arrangements. [24] If the exchanger is to be used as a vaporizer then we need to refer to correlations pertaining to nucleate boiling, convective boiling and models providing a combination of the two as the volume fraction of vapor phase increases to one as it rises towards the top-most tube row. The nucleate boiling models are an area of intense investigation in a quest to achieve a fully mechanistic model in place of the empirical and semi-empirical models available in literature. The problem in achieving a mechanistic model is a not yet fully understood boiling phenomenon and its complexity which renders the mechanistic approach too complex [12] and resource (computational power) hungry to be used as a design method. The currently available empirical models can be divided into two broad categories: reduced-pressure based correlations which predict the boiling heat transfer from macroscopic heat perspective, and thermophysical properties based correlations which are developed on the basis of the microscopic heat transfer mechanisms [11]. A promising model for predicting heat transfer in a vaporizer is the Aprin et.al. model [2, 3]. For the heat exchangers working as superheaters, forced convection models can be used as thermal design tools. The difference between Aprin model's approach and the previous approaches is that it recognizes different flow regimes and calculates different Reynolds and Prandtl numbers according the void fraction value, and uses different approaches to calculate the local heat transfer coefficient.

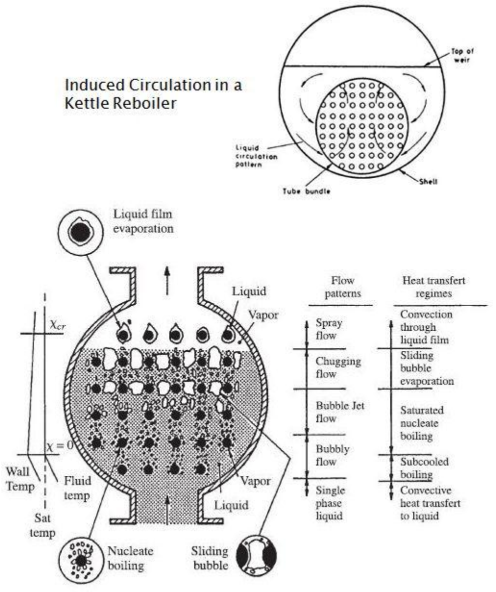


Figure 3. representation of the different flow regimes that can occur in a vaporizer (Collier & Thome, 1994)

A different approach has been put forward by Chu [12], which takes into account the microphysics of bubble dynamics to calculate the total boiling heat flux. The contributions taken are – latent heat by bubbles (q_{LH}), transient conduction (q_{CON}), heat transferred by natural convection (q_{NC}), represented as:

$$q_{tot} = (q_{LH} \cdot \tau_g + q_{CON} \cdot \tau_w) f + q_{NC}$$

Here, the τ_g is the bubble growth period, and τ_w is the bubble waiting period, and *'f'* denotes the bubble departure frequency.

Another limiting factor for the application of the models available is the fact that they require a detailed knowledge of the internal geometry of the HE which is often not available due to confidentiality or IPR (intellectual property rights) issues. This creates need for models that can provide us with the knowledge of outlet conditions of the fluids involved with inlet conditions and limited information of internal geometry as inputs. One of such approaches was presented by Vera-Garcia et.al. [8]. Thus, the models for designing can be divided into categories based on the user as well, for consumers who want to know if their HE is performing optimally and efficiently by comparing the actual outlet conditions to the ones predicted by theoretical models and ones for designers who have to design a heat exchanger from the start for a prescribed target duty.

Design Calculations for Test Rig and Uncertainty Analysis for Test-Rig

The design of the test-rig must ensure a measurement of local heat transfer coefficient, which is possible by recording and mapping the working fluid and hot fluid temperature gradient [14]. This requires placement of thermocouples at various positions inside the HE being used for experiments. This customization of the heat exchanger is both complicated and costly and puts a restraint on the use of different heat exchanger geometries. The proposed test rig will initially have instrumentation to vary the inlet conditions in terms of temperature, pressure and mass flow rate, and to measure and record inlet and outlet conditions. Further down the line it will be upgraded to be able to provide us with the information about the void fraction values. There are a few techniques available to measure the void fraction values: a) Gamma densitometer, b) Resistance void probe, c) Optical probe, d) Laser two phase detection and e) High speed photography.

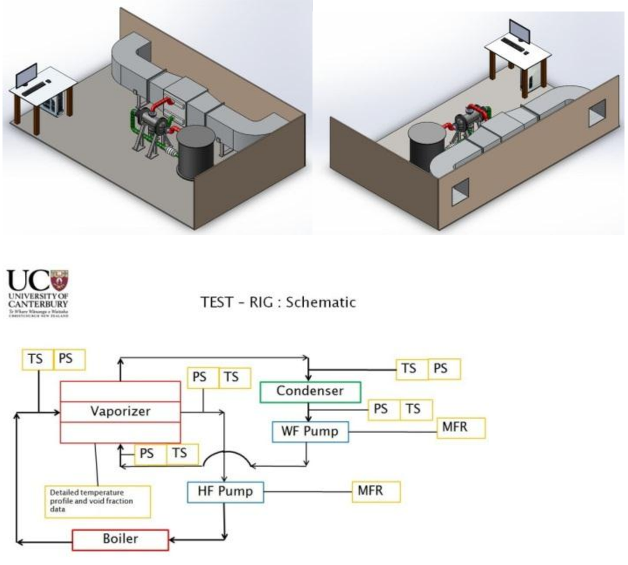


Figure 4. Pics of the proposed lab set up are shown below, along with a schematic

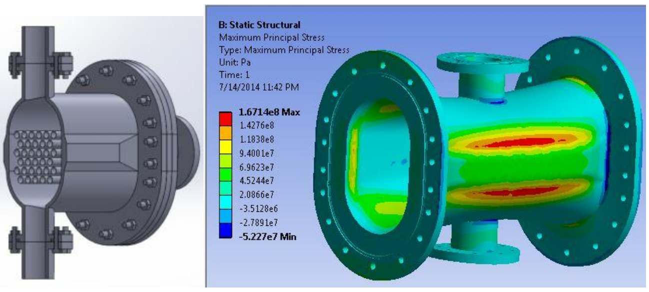


Figure 5. Showing cut-section of the vaporizer to be used in the test rig and pressure test for the central core with a mesh of 1.17 million elements at pressure of 16 bar

CFD Results

The CFD analysis of vaporizers is complicated. The modelling of phase change process is an inherently complex process, and the sheer complexity of geometry and huge sizes of vaporizers make the simulation process intensely computational resources hungry. It is necessitated because of the simple fact that the use of theoretical models can predict deficiencies in design but it is unable to pin point the location and factors of weaknesses [24].

CFD simulations are a necessity to improve upon existing designs and test new designs for any industrial equipment due to the fact that it is economically non-viable to manufacture full scale prototypes of all the conceivable designs and so CFD forms a filter mechanism at a fraction of a cost of actual manufacture and testing to narrow down on a few final designs that can be then manufactured and tested upon. There are a number of papers and literature present for single phase heat transfer [24, 25, 6, 7, 1] but only a few on the phase change in large geometries such as vaporizers. There is a significant leap in both complexity and computational resources' requirement between single phase and multi-phase simulations. The CFD simulations can also be used to check flow maldistribution issues and effects of baffle cut and baffle spacing [24].

Below are shown a few plots from single phase heat transfer simulations for non-baffled and baffled STHEs:

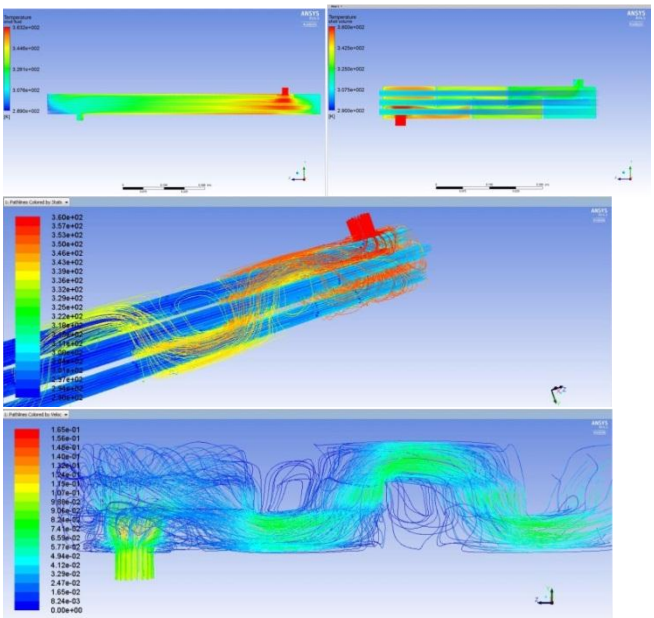


Figure 6. (a) Non-baffled STHE, (b) Baffled STHE – Clipped Volume Render, (c) Temperature Pathlines, Baffled STHE, (d) Velocity Pathlines, Baffled STHE

Conclusion

The modelling of phase change process in Ansys FLUENT is underway with promising results. However, the key lies in validation of CFD results and that is where the lab (being set-up) plays a crucial and significant role. The project has a significant contribution to make due to the three pronged approach towards finalizing a model for heat exchanger design.

Acknowledgements

The authors would lie to thank the NZ Heavy Engineering Research Association for funding under MBIE contract HERX1201.

References

1) Ahmed K. N., Bashir, M. H., Hayat, N., Khan, A. R., Khan, S. & Muhammad M. A. B., CFD Applications in various heat exchangers design: A review. *Applied Thermal Engineering*, 2012. 32(2012): p. 1-12.

2) Aprin, L., Mercier, P. & Tadrist, L., Experimental analysis of local void fractions measurements for boiling hydrocarbons in complex geometries. *International Journal of Multiphase Flow*, 2007. 33(2007): p. 371-393.

3) Aprin, L., Mercier, P. & Tadrist, L., Local heat transfer coefficient for boiling of hydrocarbons in complex geometries: A new approach for heat transfer prediction in staggered tube bundle. *International Journal of Heat and Mass Transfer*, 2011. 54(2011): p. 4203-4219.

4) Bamardouf, K. H., McNeil, D.A. & Sadikin, A., A Mechanistic Analysis of Shell-side Two-phase flow in an Idealised in-line Tube Bundle. *International Journal of Multiphase Flow*, 2012. 45(May 2012): p. 53-69.

5) Bansal, P.K. & Browne M.W., Heat Transfer Characteristics of Boiling Phenomenon in Flooded Refrigerant Evaporators. *Applied Thermal Engineering*, 1999. 19(1999): p. 595-624.

6) Bhoi, R., Jayakumar, J.S., Mahajani, S.M., Mandal, J.C. & Vijayan, P.K., Experimental and CFD Estimation of Heat Transfer in Helically Coiled Heat Exchangers. *Chemical Engineering Research and Design*, 2008. 86: p. 221-232.

7) Bock, J., Jacobi, A.M., Liu, W., Ma, L. & Yang, J., A comparison of four numerical modelling approaches for enhanced shella nd tube heat exchangers with experimental validation. *Applied Thermal Engineering*, 2014. 65: p. 369-383.

8) Cabello, R., Garcia-Cascales, J.R., Gonzalvej-Macia, J., Llopis, R., Sanchez, D., Torella, E. & Vera-Garcia, F., A simplified model for shell-and-tube heat exchangers: Practical application. *Applied Thermal Engineering*, 2010. 30(2010): p. 1231-1241.

9) Chan, A.M.C., Dowlati, R. & Kawaji, M., Two-phase Crossflow and Boiling Heat Transfer in Horizontal Tube Bundles. *Journal of Heat Transfer*, 1996. 118(February 1996): p. 124-131.

10) Chan, A.M.C., Kawaji M. & Noghrekar, G.R., Investigation of two-phase flow regimes in tube bundles under cross-flow conditions, *International Journal of Multiphase Flow*, 1998. 25(1999): p. 857-874.

11) Chen, T., Water-heated Pool Boiling of Different Refrigerants on the Outside Surface of a Smooth Horizontal Tube. *Journal of Heat Transfer*, 2012. 134(February 2012).

12) Chu, H. & Yu, B., A new comprehensive model for nucleate pool boiling heat transfer of pure liquid at low to high heat fluxes including CHF. *International Journal of Heat and Mass Transfer*, 2009. 52(2009): p. 4203-4210.

13) Das, M.K. & Swain, A., A review on saturated boiling of liquids on tube bundles. *Heat Mass Transfer*, 2013. DOI 10.1007/s00231-013-1257-1.

14) Eckels, S. & Gorgy, E., Local heat transfer coefficient for pool boiling of R-134a and R-123 on smooth and enhanced tubes. *International Journal of Heat and Mass Transfer*, 2012. 55(2012): p. 3021-3028.

15) Feenstra, P.A., Judd, R.L. & Weaver, D.S., An Improved Void Fraction Model for Two-phase Cross-flow in Horizontal Tube Bundles. *International Journal of Multiphase Flow*, 2000. 26(2000): p. 1851-1873.

16) Gebbie, J.G. & Jensen, M.K., Void Fraction Distributions in a Kettle Reboiler. *Experimental Thermal and Fluid Science*, 1997. 14(1997): p. 297-311.

17) Gebbie, J.G., Jensen, M.K. & Rahman, F.H., An Interfacial Friction Correlation for Shell-side Two-phase Cross-flow past Horizontal in-line and Staggered Tube Bundles. *International Journal of Multiphase Flow*, 1996. 22(1996): p. 753-766.

18) Hsu, J. T., Jensen, M.K. & Reinke, M.J., The influence of tube bundle geometry on cross-flow boiling heat transfer and pressure drop. *Experimental Thermal and Fluid Science*, 1989. 2(1989): p. 465-476.

19) Hwang, K.S. & Judd, R.L., A comprehensive model for nucleate pool boiling heat transfer including microlayer evaporation. *Journal of Heat Transfer*, 1976. November (1976): p. 623-629.

20) Kiger, K. & Moghaddam, S., Physical mechanisms of heat transfer during single bubble nucleate boiling of FC-72 under saturation conditions - I: Experimental Investigation, II: Theoretical Analysis. *International Journal of Heat and Mass Transfer*, 2009. 52(2009): p. 1284-1294,1295-1303.

21) Kim, J., Review of nucleate pool boiling heat transfer mechanisms. *International Journal of Multiphase Flow*, 2009. 35(2009): p. 1067-1076.

22) Kim, J., Kim, M.H. & Oh, B.D., Experimental study of pool temperature effects on nucleate pool boiling. *International Journal of Multiphase Flow*, 2006. 32(2006): p. 208-231.

23) Ockoljic, S., Simovic, Z.R. & Stevanovic, V.D., Interfacial Friction Correlations for the Two-phase flow across tube bundle. *International Journal of Multiphase Flow*, 2006. 33(2007): p. 217-226.

24) Ozden, E. & Tari, I., Shell side CFD analysis of small shell and tube heat exchanger. *Energy Conversion and Management*, 2010. 51: p. 1004-1014.

25) Pawar S. S. & Sunnapwar, V.K., Experimental and CFD Investigation of Convective Heat Transfer in Helically Coiled Tube Heat Exchanger. *Chemical Engineering Research and Design, 2014*. "Article In Press".

26) Tou, K.W., Tso, C.P. & Xu ,G.P., Hydrodynamics of two-phase flow in vertical up and down-flow across a horizontal tube bundle. *International Journal of Multiphase Flow*, 1998. 24(1998): p. 1317-1342.