A Review on Applications of Computational Fluid Dynamics of Various Warm Interchanger

Prabhat Kaushal[1], Dr. Manish Gangil[2]

[1]PhD Research Scholar, RKDF University, Airport Road, Gandhi Nagar, Bhopal-462035, India
[2]Professor, RKDF University, Airport Road, Gandhi Nagar, Bhopal-462035, India

ABSTRACT

This literature review focuses on the applications of Computational Fluid Dynamics (CFD) in the field of Warm Interchangers. It has been found that CFD has been employed for the following areas of study in various types of Warm Interchanger: fluid flow misdistribution, fouling, pressure drop and thermal analysis in the designand optimization phase. The quality of the solutions obtained from these simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behavior and performance of a wide variety of Warm Interchanger.

Keywords –Computational fluid dynamics (CFD), Warm Interchanger, LMTD, NTU, Fouling, pressure drop, Thermal Analysis, Design Optimization.

INTRODUCTION

Warm Interchangerplay a significant role in the operation of many systems such as power plants, process industries and heat recovery units. Its inevitable need has necessitated work on efficient and reliable designs leading towards optimum sharein the overall system performance. The Log Mean Temperature Difference (LMTD) method and the number of heat transfer units (NTU) method have been used for heat exchanger design [1]. These methods have some shortcomings associated with them i.e. iterative in nature and need of a prototype to implement the design. Due to these reasons, these methods are time consuming as well as expensive especially for largescale models. However, economical access to powerful micro processorshas paved the way for evolvement of Computational Fluid Dynamics (CFD) during the design phase.CFD is a science that can be helpful for studying fluid flow, heat transfer, chemical reactions etc.By solving mathematical equations with the help ofnumerical analysis. It is equally helpful in designing a Warm Interchanger system from scratch as wellas in troubleshooting/optimization by suggesting design modifications. CFD employs a very simple principle of resolving the entire system in small cells or grids and applying governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature gradients, flow parameters and the like in a shorter time at a lower cost because of reduced required experimental work [2].

The results obtained with the CFD are of acceptable quality [3]. In the current work, various problems encountered in the design of Warm Interchangers and their solutions with the help of CFD have been reviewed. This workcan serve as a ready reference for application of CFD in design of various types of Warm Interchangers.

Types of Warm Interchangers

There are innumerable types of Warm Interchangers in use including but not limited to plate type Warm Interchanger (PHE), shell-and-tube Warm Interchanger, vertical mantle Warm Interchanger and micro Warm Interchanger and a broad classification of Warm Interchangers based on their construction.[4]

Applications of CFD in various aspects of Warm Interchangers

Some of the commercial CFD codes in use are FLUENT, CFX, STAR CD, FIDAP, ADINA, CFD2000, PHOENICS and others [5]. Using these, extensive work has been carried out on the various aspects of Warm Interchangers with primary focus on optimization through study of thermal performance, pressure drop, fouling and fluid misdistribution. Ithas listed various possible models that might be utilized in the CFD analysis and carried out the simulations on a PHE with different corrugations to show the possibility of application of CFD on various problems encountered in Warm Interchangers.

In the subsequent sections, various issues related to Warm Interchangers have been presented under different headings. Under each heading, exchangers have been discussed in order of their types.

Flow misdistribution

Non-uniformity in fluid flow is one of the primary reasons resulting in a poorheat exchanger performance. This may be attributed to improper designof inlet/outlet port and header configuration, distributor construction and plate corrugations [6]. To understand the fluid flow misdistribution, [7] studied the characteristics of vertical mantle heat exchangeremployed ina solar water heater. It studied the characteristics of vertical mantle heat exchangeremployed ina solar water heater. A three block model was considered in the analysis which revealed that fluid distribution along the mantle is affected by recirculation produced due to buoyancy force in the mantle in case of high and low temperature inputs respectively. A local Nusslets Rayleighcorrelation was worked out, for which x is the distance from top of annulus.

In a similar exchanger, [8] observed presence of mixed flow in the mantle close to inlet by studying the flow pattern inside tank and mantle of the exchanger.In [9] established that 2D analysis is sufficient to identify the effect of the corrugations shape, but to investigate on the significance of the corrugation orientation, a 3D analysis is also mandatory. For the purpose, a numerical model was constructed and simulation performed between two corrugated plates of a PHE keeping Re and Pr 4482 and 6.62 respectively. It was observed that recirculation bubbles are formed, which acts as a hindrance to the steady flow. [9] Studied a double sine-chevron PHE for a range of corrugation angles (i.e. 30 to 85) and channel aspect ratios (i.e. 0.38 - 0.76). The shape factor and the Tortuosity coefficient were found to depend on the corrugation angle and channel aspect ratios respectively. An increase in Tortuosity coefficient and Kozeny's coefficient was observed with the increase in the channel aspect ratio which eventually decreased with the increase in chevron plate angle. It was deduced that shape factor is almost independent of channel aspect ratios and inversely proportional to chevron plate angles.

Louvered find and flat tube Warm Interchangers have widespread application in automotive and air conditioning industry. [10] carried a study for 2D and 3D simulation. The authors observed, for lower Re (i.e. 78.6), air makes a thin boundary

Fouling

Degradation of the working fluid because of high temperatures or interaction with the exchanger walls results in production of by products which get deposited on the walls and reduce the coefficient of heat transfer. Frequent periodic maintenance may be required if extreme cases of fouling are encountered. [11] through simulation for temperature profiles of milk inside a PHE pointed out weak regions formed due to fouling by Beta-lectoglobulin, which primarily relies on the difference of temperature between the plate and the working media. [12] Investigated the impact of corrugations and their position on fouling rates for asimilarPHE used for the pasteurization of milk. An alternate design of a compact Warm Interchanger was proposed in an effort to establish the possibility of using unconventional Warm Interchangers in the food processing industry. It was found that area requirements are decreased for the same amount of heat transfer along with a reduction in deposition rate by 92.6%.

In addition to reinstating the fact fouling rate depends on wall temperatures, [13] studied the fouling rates with respect to flow velocities. As the Re of flowing media is increased, a drastic increase in the fouling rate is encountered with a deposition of a little more than 1 g for Re 1700 and almost 22 g for Re 3700 after a given time period. Thus, through the research, it was established that basic computation may be usedfor a variety of geometry and biochemical products.

Pressure Drop

Every fluid undergoes some pressure loss while flowing in the Warm Interchanger. This phenomenon is mainly affected by the choice of design of core or the matrix and the fluid distribution components i.e. Inlet/outlet ports, headers, nozzles, ducts and etc. [14]. This pressure loss particularly because of flow distribution devices seriously affects the heat transfer rate of the exchanger. [15] performed a CFD simulation on the tubes of a Warm Interchanger used in closed-wet cooling towers. Pressure drop was found to depend on the tube configurations and water to air ratio. Predicted pressure loss coefficient was found inversely proportional to transverse pitch but was is in direct relationship with water to air mass flow rate. Moreover, pressure loss was seen to increase by 17% with the interference of upstream air, but a reduced value was obtained when this air is bypassed. [16] observed the pressure to be maximum at the separation plate of hot and cold streams in the vertical plane of a cross flow air to air Warm Interchanger. The cold air pressure in the tube was found to be highest at the inlet that decreased gradually by 202 Pa due to friction with the tube inner wall. [17] considered a rod baffle shell and tube Warm Interchanger. A periodic flow unit duct model was used for examining the pressure drop. Simulations were carried out using water as the working fluid at 300 K while for the experimental setup, laser Doppler anemometry was opted to compute flow velocities. Pressure drop was found to have an inverse relation- ship with the baffle pitch assuming flow velocity to be constant whereas it was seen to increase with the rise in fluid velocity at constant baffle pitch. The results concluded that the periodic duct unit model can define the flow and pressure characteristics more precisely than other models. Similarly, [18] established that amongst small round hole plate, plum blossom plate, rectangle hole plate, big round hole plate, rod baffle and ting baffle, rod baffle is the most reliable and suitable supporting structure with a minimum pressure drop. [19] stated the pressure drop of a combined multiple shell-pass shell-and-tube heat exchanger (CMSP-STHX) to be lower than shell-and-tube Warm Interchanger (SG-STHX) with segmental baffles, for the same overall heat transfer, by almost 13%. [20] studied the effect of baffle cut and location on an exchanger having 660 tubes while neglecting the effect of leakage flows. Carrying the study forward, in [21] undertook another simulation with the leakage flows accounted for as well. The employed exchanger was in accordance with the TEMA standards with 76 plane tubes. Two different baffle orientations were considered and three different fluids i.e. air, water and engine oil used to evaluate the effect of viscosity for a Pr in the range of 0.7 - 206. The CFD results established that a horizontal baffle exchanger has a 20% higher heat transfer coefficient with 250% greater pressure drop than the one with vertical baffles. [22] simulated the pressure drop in an S- shaped finned micro channel Warm Interchanger which was found 6 - 7 times lower than the ordinary zigzagged finned exchanger. An experimental study was performed to investigate the performance (pressure drop and Nu characteristic) of the two different fins respectively. A four to five times lesser pressure drop was observed experimentally in the micro channel Warm Interchanger while keeping the Nu 40% lower. Pr was established to be independent in the range 0.75 - 2.2 from Nu and pressure drop factor, provided Re varied between 3.0 103 - 2.0 104. Considering the results of [23] proposed to replace the zigzag channels with airfoil type finned channels for a printed circuit board Warm Interchanger carrying supercritical carbon dioxide. The new design reduced the pressure drop while giving same rate of heat transfer as the conventional design.

Thermal analysis

The very purpose of aWarm Interchanger is to enhance the heat transfer between two fluids. This reduces the energy requirements and helps make a process more efficient both in terms of production and economics. With the help of CFD analysis, researchers have approached thermal characteristics of Warm Interchangers from two points of view .

Influence on thermal parameters

A basic strategy to evaluate an exchanger's performance is through the study of parameters like Nusselt Number, Dean's number, Prandtl number, Friction factor, Colburn factor etcandtheir impact on the overall heat transfer coefficient of the Warm Interchanger [24]. It considered a cross flow air to air tube type Warm Interchanger and observed the relation between heat transfer and Dean's Number keeping the flow rate constant in the outer region. An increase in heat transfer from 800 to 900 W/m^2 was observed with a rise in inner De from 3500 to 6000. For the outer De, an increase from 500 to 1000 yielded an increase from600 to 950 W/m^2. It was assumed that heat lost occurred through convection from the top surface of the exchanger only whereas all radiation losses were neglected in the study. Due to symmetry in geometrical construction, a section of Warm Interchanger was considered for CFD analysis. [25] presented different correlations for Nu, Re and Pr and discussed the fluid entry effect on average Nu for stirred yogurt during cooling in a PHE. Her- scheleBulkley model was employed and numerical analysis carried out by first considering and then ignoring the temperature effect on the viscosity. A comparative analysis with correlations present in past literature concluded that Nu experiences an increase with an increase in the Re. The study made it evident that for yoghurt simulation, CPU time can be greatly reduced when the influence of temperature is ignored. Similarly, [26], focused on establishing the significance of CFD as a method to study the corrugated type PHE with stainless steel plates and a herringbone structure for single pass counter current flow. The temperature distribution was shown to remain constant in the y-direction i.e. along the width of the channel. The authorestab- lished that Nu/Nuo increases with Re while friction factor decreases for trapezoidal corrugation, which is greater than sinusoidal corrugation for same Re. A good agreement was found with the results presented by [27]. Carrying their study forward, determined a general suitable mechanism of designing a PHE with undulated surfaces. The results were expressed in the form of Nu and friction factor for the optimization function. These were compared with the literature and optimal characteristics were proposed for an array of Re for two values of weighing factor. Through the CFD model, it was observed that the when the distance between channel plates reduces the positive performance of PHE increases.

PHE performance can also be increased by decreasing the channel aspect ratio and increasing the angle of the attack [28].It used depth-averaged flow and energy equations to perform CFD simulation. Equations were limited to 2D from 3D but friction factor and heat transfer through plates was accounted. A comparison was made between 2D simulation and their results with the 3D model of five unlike corrugation angles and lengths. The 2D simulation results presented a relatively good summary of pressure loss and temperature variation with respect to corrugation angle and length. 30 more different geom- etries were simulated using fast 2D approach and it was observed infrared thermography to investigate the heat transfer coefficients of a similar PHE. The surface mapping of the exchanger revealed presence of minimum heat transfer coefficient at the point of intersection of two meshes and lines of local maximum parallel to the length of corrugations. A CDF analysis was performed that demonstrated a better convergence mode for RSM-EASM turbu- lence model. The authors pointed out that despite the possible under estimation of heat transfer coefficient found via CFD, applying these results in Warm Interchanger design would only result in a better exchanger performance than predicted; thus, the significance and possible use of CFD in heat exchanger design remains non-detrimental and unquestionable. [29] modeled a matrix Warm Interchanger using porous medium for heat transfer. It had researched on the heat transfer and fluid flow characteristics in actual micro heat exchangers. The authors, utilizing the results of that study stated a correlation between Re and the Nu number for the matrix Warm Interchanger implying porous media.

To evaluate the uncertainties in the coefficient of heat transfer, the KlineeMcClintock method of uncertainty calculation was used which was found to be 2.7%. [30] evaluated the Nu and friction factor for three types of fin and tube Warm Interchangers i.e. with slit fins, longitudinal fins and plain fins. Study was carried on to observe the difference between laminar and turbulent heat transfer for larger diameter of tubes and comparatively larger number of rows of tubes.

In [31] employed the KAIST Helium test loop system and the CFD techniques to calculate the thermal parameters at the inlet and outlet of a Printed Circuit Board Warm Interchanger. The intrinsic properties of PCHE were analyzed previously by [32] using CO_2 as a working fluid. presented correlations for global fanning factor, global Nu and Local Nu were proposed and it was found that the local Nu found through the CFD simulations is best to carry out the analysis: [33], it adopted the porous medium and full scale geometry techniques to analyze the heat transfer in elliptical tube staggered Warm Interchanger. The results were quantified in form of graphs of Nu with fluid velocity and the tube number. An increase in the Nu/Nuref by a factor of 0.2 was observed with an increase in velocity whereas it was concluded that the first two tubes playeda primary role in the thermal characteristics of the exchanger.A study of the f and j was undertaken by [34] for an offset and wavy fins compact plate fin Warm Interchanger. f and j were seen to increase with increasing height to fin ratio while for the same height to fin ratio, f and j were found to be higher for smaller Re. Owing to under estimation of results, it was concluded that all results should to be multiplied by 1.6.

Design optimization

Ever sinceits advent, CFD has proven to be an effective tool in the design optimization of the Warm Interchangers by studying thermal properties. It has been employed to study different modifications [35], compare results and present the best possible combination of variables to ensure optimum performance.In discussed the effect of inlet positions of the mantle on the thermal behavior for a vertical mantle Warm Interchanger used in solar domestic hot water systems. By assuming the symmetry along the mid plane, only half of the model was considered for CFD simulation to reduce the computational time. It was concluded that it is more efficient to have the inlet located at the top for a high inlet-temperature to the mantle while for low temperature inlet, moving the inlet down yields better performance. In another study following year, the authors extended the same work and concluded that the heat transfer rate for the exchanger is dependent more on inlet temperature of the mantle as compared to the temperature of the core tank [36].

It stated that nature of the fluid, operating conditions and geometry play a major role in defining the performance of a micro channel Warm Interchanger. CFD analysis was coupled with the analytical and multi-objective genetic algorithms methods respectively to determine the optimum design and the results obtained from the two approaches were compared. The latter was found more useful as it did not only help find the optimum dimensions but the shape of the optimum design structure too. The performance of the exchanger was found greatly influenced by the aspect ratio, defined as ratio of height to the width of a channel. In case of a variable volume of micro channels, for an increase in the aspect ratio from 10 to 100, the pressure drop and the heat flux experienced a steep drop initially attaining a stable value at 100. However, the heat transfer rate increased slightly and stabilized at an overall aspect ratio of 30. On the contrary, for a constant volume micro channel, pressure loss and the heat transfer tends to increase with increasing aspect ratio whereas; the heat flux experiences a drop. In the same year [37] performed CFD simulation to study the characteristics of a helix Warm Interchanger. Baffle geometry and position of the nozzle for inlet and exit configurations were primarily considered together with three different helix angles measured relative to radial axis. It was established that the much desired plug flow is observed at the outer region of the exchanger which increases with the helix angle. A comparison between the results of the pressure drop obtained through ABB Lummus Heat Transfer, Bloomfield, New Jersey correlations and simulation shows a good agreement; however more effort is required to estimate the impact of nozzle losses on helix changer performance.

In [38] developed a contact free and fluid autonomous method for measuring convective coefficient of heat transfer of a PHE. The method relied on experimental data of the outside body temperature subjected to periodic heating, measured through infrared camera, and on the Warm Interchanger wall. The pattern of the heat transfer coefficient was found to be very distinct where heat transfer was least i.e. at the intersecting points of the corrugated body and lines of maximum value adjacent to the corrugation. The whole plate was analyzed (PHE) to measure the macro scale flow pattern and no misdistribution was accounted. [39] evaluated the impact of exchanger design on decomposition behavior of sulphur trioxide in bayonet type Warm Interchanger. They concluded that low decomposition is achieved at temperatures below 1000 K, and sufficient above1073 K. Considering these results, to predict a better design, thermal modeling was undertaken to evaluate the decomposition process. The impact of cylindrical, spherical, cubical and hollow pellets arranged inside the packed bed was studied on the overall exchanger performance. Pellets were composed of silicon with platinum catalyst while the decomposer was made of silicon carbide and quartz. A comparative analysis of the decomposition percentage of SO3 for the packed bed and the porous mediaapproach was executed. Results yielded that spherical pelletsprovide the maximum decomposition percentage, in this case, 61%for 5 ml/min. For shell and tube Warm Interchanger [40] performed the comparison of combined multiple shell-pass shell and tube type Warm Interchanger with segmented baffles using CFD simulation. The comparative analysis of the results obtained from simulation of the two distinct models revealed that under similar mass and heat transfer rate with same pressure drop in the shell side the overall heat transfer rate of CMSP-STHX is 5.6% higher than the SG-STHX. [41] performed the CFD simulation in a shell and tube Warm Interchanger to estimate the effect of baffle spacing, cut and shell diameter on the heat transfer and pressure drop characteristics. To evaluate the performance, different baffles and turbulence models were examined. In order to determine the best turbulence model a comparison of Bell Delaware method and Kern method [42] were carried out and two important conclusions were deduced: Kern method always underestimates the heat transfer coefficient while Bell Delaware [43] method underestimates the heat transfer rate of varying spaced baffles, showing good agreement with the heat transfer characteristics of equally spaced baffles. It was further established that a baffle cut of 25% gives superior results for heat transfer. [44] Considereda shell and tube type Warm Interchanger based on the PCM Module for the cooling purposes. A modular type arrangement was used for the exchanger with several modules arranged over one another consisting of air spacers between two successive modules. Transient and steady state CFD modeling was carried for a single module and two consecutive air spacers found.

CONCLUSION

Conventional methods used for the design and development of Warm Interchange rare largely tedious and expensive in today's competitive market. CFD has emerged as a cost effective alternative and it provides speedy solution to Warm Interchanger design and optimization. Easily accessible general purpose CFD commercial software's can fulfill the requirements of CFD analysis of various types of Warm Interchangers including but not limited to Plate, Shell and Tube, Vertical Mantle, Compact and Printed Circuit Board Exchangers. These are flexible enough to accommodate any kind of analysis requirement ranging from prediction of fluid flow behavior to complete heat exchanger design and optimization involving a wide range of turbulence models and integrating schemes avail- able in CFD softwares. Of these, the k ε turbulence model has been most widely employed for Warm Interchanger design optimiza- tion. The simulations generally yield results within good agreement with the experimental studies ranging from 2% to 10% while in some exceptional cases, vary up to 36%. In case of large deviations, user defined sub routines specific to the design problem may become necessary. Dependability and reliability of CFD results has reached a point making it an integral part of all design processes, leading towards eliminating the need of prototyping.

REFERENCES

[1]. V. Kumar, S. Saini, M. Sharma, K.D.P. Nigam, Pressure drop and heat transfer study in tube-in-tube helical heat exchanger, Chem. Eng. Sci.61 (2006)4403e4416.
[2]. A.G. Kanaris, A.A. Mouza, S.V. Paras, Flow and heat transfer prediction in a corrugated plate Warm Interchanger using a CFD code, Chem. Eng. Technol. 8 (2006) 923e930.
[3]. Y. Wang, Q. Dong, M. Liu, Characteristics of fluid flow and heat transfer in shellside of Warm Interchangers with longitudinal flow of shellside fluid with different supporting structures, in: International Conference on Power Engineering2007, October 23e27, Hangzhou, China.
[4]. K. Kritikos, C. Albanakis, D.Missirlis, Z. Vlahostergios, A.Goulas, P. Storm, Investigation of the thermal efficiency of a staggered elliptic-tube Warm Interchanger for aero engine applications, Appl. Therm Eng. 30 (2010) 134e142.
[5]. S. Jun, V.M.Puri, 3D milk-fouling model of plate heat exchangers using computational fluid dynamics, Int. J. Dairy Tech. 84 (4) (2005).
[6]. Y. Wang, Q.W. Dong, M.S. Liu, D. Wang, Numerical study on plate-fin Warm Interchanger with plain fins and serrated fins at low Reynolds number, Chem. Eng. Technol. 32 (8) (2009) 1219e1226.
[7]. J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, P.K. Vijayan, R. Bhoi, Experimental and CFD estimation of heat transfer in helically coiled Warm Interchangers, Chem. Eng. Res.Des 86 (2008) 221e232.
[8]. A.M. Gustafsson, L. Westerlund, G. Hellström, CFD-modeling of natural convection in a groundwater-filled borehole Warm Interchanger, Appl. Therm Eng.30 (2010) 683e691.
[9]. C. Habchi, T. Lemenand, D.D. Valle, H.Peerhossaini, Turbulent mixing and residence time distribution in novel multifunctional heat exchan- gersereactors, Chem. Eng. Process 49 (2010) 1066e1075.
[10]. E. Ozden, I. Tari, Shell side CFD analysis of a small shell-and-tube Warm Interchanger, Energy Conversion Manage. 51 (2010) 1004e1014.
[11]. K.L. Wasewar, S. Hargunani, P. Atluri, N. Kumar, Simulation of flow distribu-ion in the header of plate-fin Warm Interchangers, Chem. Eng. Tech. 30 (10) (2007) 1340e1346.
[12]. L.Z. Zhang, Flow maldistribution and thermal performance deterioration ina cross-flow air to air Warm Interchanger with plate-fin cores, Int. J. Heat MassTransf 52 (2009) 4500e4509.
[13]. S. Knudsen, S. Furbo, Thermal stratification in vertical mantle Warm Interchangers with application to solar domestic hot-water systems, Appl. Energy 78 (2004)257e272.
[14]. S. Freund, S. Kabelac, Investigation of local heat transfer coefficients in plate Warm Interchangers with temperature oscillation IR thermography and CFD, Int. J. Heat Mass Transf 53 (2010) 3764e3781.
[15]. B. Sundén, Computational fluid dynamics in research and design of Warm Interchangers, Heat Transf Eng. 28 (11) (2007) 898e910.
[16]. K. Grijspeerdt, B. Hazarika, D.Vucinic, Application of computational fluid dynamics to model the

hydrodynamics of plate Warm Interchangers for milk processing, J. Food Eng. 57 (2003) 237e242.

[17]. L.J. Shah, G.L. Morrison, M. Behnia, Characteristics of vertical mantle Warm Interchangers for solar water heaters, Solar Energy 67 (1e3) (1999) 79e91.

[18]. S. Knudsen, G.L. Morrison, M. Behnia, S. Furbo, Analysis of the flow structure and heat transfer in a vertical mantle Warm Interchanger, Solar Energy 78 (2005)281e289.

[19]. C.S. Fernandes, R.P. Dias, J.M. Nobrega, J.M. Maia, Laminar flow in chevron- type plate Warm Interchangers: CFD analysis of tortuosity, shape factor and fric- tion factor, Chem. Eng. Process 46 (2007) 825e833.

[20]. T. Perro, D. Clodic, Thermal-hydraulic CFD study in louvered fin-and-flat-tubeWarmInterchangers, Int. J. Refrigeration 27 (2004) 422e432.

[21]. C. T'Joen, H. Huisseune, H.Canière, H.J. Steeman, A.Willockx, M.D. Paepe, Interaction between mean flow and thermo-hydraulic behavior in inclined louvered fins, Int. J. Heat Mass Transf 54 (2011) 826e837.

[22]. D.A. Yashar, H.H. Cho, Air-Side Velocity Distribution in Finned-Tube Warm Interchangers. National Institute of Standards and Technology, United States Department of Commerce, 2007.<http://www.fire.nist.gov/bfrlpubs/build08/PDF/b08002.pdf> (accessed on 23.03.11.).

[23]. Z. Zhang, Y.Z. Li, CFD simulation on inlet configuration of plate-fin Warm Interchangers, Cryogenics 43 (2003) 673e678.

[24]. J. Wen, Y.Z. Li, Study of flow distribution and its improvement on the header of plate-fin Warm Interchanger, Cryogenics 44 (2004) 823e831.

[25]. M. Yataghene, J. Pruvost, F. Fayolle, J. Legrand, CFD analysis of the flow pattern and local shear rate in a scraped surface Warm Interchanger, Chem. Eng. Process47 (2008) 1550e1561.

[26]. A.G. Kanaris, A.A. Mouza, S.V. Paras, Flow and heat transfer in narrow channels with corrugated walls, A CFD Code application, Chem. Eng. Res.Des 83 (2005)460e468.

[27]. W.W. Focke, P.G. Knibbe, Flow visualization in parallel-plate ducts with corrugated walls, J. Fluid Mech. 165 (1986) 73e77.

[28]. M.L. Kim, Y. Lee, B.W. Kim, D.H. Lee, S.W. Song, CFD modeling of shell-and-tube Warm Interchanger header for uniform distribution among tubes, Korean J. Chem. Eng. 26 (2009) 359e363.

[29]. R.Y. Miura, F.C.C. Galeazzo, C.C. Tadini, J.A.W. Gut, the effect of flowarrangement on the pressure drops of plate Warm Interchangers, Chem. Eng. Sci. 63 (2008) 5386e5393.

[30]. F.C.C. Galeazzo, R.Y. Miura, J.A.W. Gut, C.C. Tadini, Experimental and numerical heat transfer in a plate Warm Interchanger, Chem. Eng. Sci. 61 (2008) 7133e7138.

[31]. A.S. Bansode, S. Patel, T.R. Kumar, B. Muralidhar, T. Sundararajan, S.K. Das, Numerical simulation of effects of flow maldistribution on heat and mass transfer in a PEM fuel cell stack, Heat Mass Transf 43 (2007) 1037e1047.

[32]. M.V.D. Bonis, G. Ruocco, Conjugate fluid flow and kinetics modeling for Warm Interchanger fouling simulation, Int. J. Therm Sci. 48 (2009) 2006e2012.

[33]. R.K. Shah, D.P.Sekulic, Fundamentals ofWarm Interchanger Design, first ed.Wiley, New York, 2003, 378e381.

[34]. G. Gan, S.B. Riffat, L. Shao, CFD modelling of pressure loss across tube bundles of a Warm Interchanger for closed-wet cooling towers, Int. J. Ambient Energy 21 (2000) 77e84.

[35]. V. Kumar, D. Gangacharyulu, P.M. Rao, R.S. Barve, CFD analysis of cross flow air to air tube type Warm Interchanger (2003).<http://www.cham.co.uk/PUC/PUC_Melbourne/papers/Paper18_Kumar.pdf> (accessed on 23.04.11.).

[36]. Q.W. Dong, Y.Q. Wang, M.S. Liu, Numerical and experimental investigation of shell side characteristics for ROD baffle Warm Interchanger, Appl. Therm Eng. 28 (2008) 651e660.

[37]. Q. Wang, Q. Chen, G. Chen, M. Zeng, Numerical investigation on combined multiple shell-pass shell- and-tubeWarm Interchanger with continuous helical baffles, Int. J. Heat Mass Transf 52 (2009) 1214e1222.

[38]. K. Mohammadi, W. Heidemann, H. M€uller-Steinhagen, Numerical investigationof the effect of baffle orientation and baffle cut on heat transfer andpressure drop of a shell and tube heat exchanger, in: Proceedings of theInternational Heat Transfer Conference IHTC-13, 2006, Sydney.

[39]. K. Mohammadi, W. Heidemann, H. Müller-Steinhagen, Numerical investigationof the effect of baffle orientation on heat transfer and pressure drop ina shell and tube heat exchanger with leakage flows, Heat Transf Eng. 30(2009) 1123e1135.

[40]. T.L. Ngo, Y. Kato, K. Nikitin, T. Ishizuka, Heat transfer and pressure dropcorrelations of micro-channel Warm Interchanger with S-shaped and zigzag finsfor carbon dioxide cycles, Exp. Therm Fluid Sci. 32 (2007)

560e570.

[41]. T. Ishizuka, Y. Kato, Y. Muto, K. Nikitin, N.L. Tri, H. Hashimoto, Thermalhydrauliccharacteristics of a printed circuit heat exchanger in a supercriticalCO2 loop, in: Proceedings of the 11th International TopicalMeeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), 2005, pp.218e232.

[42]. N. Tsuzuki, Y. Kato, T. Ishizuka, High performance printed circuit heatexchanger, Appl. Therm Eng. 27 (2007) 1702e1707.[43] D.E. Kim, M.H. Kim, J.E. Cha, S.O. Kim, Numerical investigation on thermal hydraulicperformance of new printed circuit heat exchanger model, Nucl.Eng. Des 238 (2008) 3269e3276.

[43]. Y.C. Tsai, F.B. Liu, P.T. Shen, Investigations of the pressure drop and flowdistribution in a chevron- type plate heat exchanger, Int. Commun. Heat MassTransf 36 (2009) 574e578.

[44]. V.M. Kays, A.L. London, Compact Heat Exchanger, third ed. McGraw-Hill,NewYork, 1984.