Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879

Heat and Flow Analysis of a Chilled Water Storage System using Computational Fluid Dynamics

Open Access

Ubaidullah Selamat[1], Kahar Osman[1,*], Arul Hisham A. Rahim[2]

[1] Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia
[2] AHAR Consultants PLT, 1-2, 2nd Floor, Jalan Cecawi 6/33, Kota Damansara, 47810 Petaling Jaya, Selangor, Malaysia

ARTICLE INFO

Article history:
Received 15 August 2018
Received in revised form 21 November 2018
Accepted 2 December 2018
Available online 11 May 2019

Keywords:
Water cooled thermal energy storage system, Thermocline thickness analysis

ABSTRACT

Thermal energy storage cooling system has been used to reduce peak power consumption of air conditioning system in buildings. Low energy cost during night time is utilized to power water chiller to chill water and stores in tank rather than running the chiller directly during daytime at higher tariff rate. This study embarks on Computational Fluid Dynamics (CFD) approach to analyse heat distribution and fluid flow behaviour within a chilled water storage system. Three cases are simulated which are; tank without diffuser, diffuser without tank and tank with diffuser. The results are compared with designer's design calculation for validation purposes. Simulated thermocline thickness is approximately 1.3m and 0.22m/s for nozzle outlet velocity. This study provides better understanding of velocity vectors, magnitudes, and temperature distribution across the tank compared to analytical data. Conclusively, our results show good agreement with analytical data. This study will become validation basis for our future tank and diffuser design optimization.

Copyright © 2019 PENERBIT AKADEMIA BARU - All rights reserved

1. Introduction

Thermal energy storage (TES) systems is used to reduce electricity consumption in air conditioning (AC) system. Instead of continuously turn on the chiller during the day with high priced electricity cost, the TES system cools water in tank during nighttime with lower tariff. In a CWS system, water from storage tank acts as the cooling fluid for the air conditioning system. During peak hours, building's chiller system is turned off or is run at part-load whilst the cooling load is satisfied by circulating the chilled water from TES tank. TES system's chiller is operated during nighttime with cheaper electricity rate compared to peak hour rate (daytime). In addition, lower night time ambient temperature provides more efficient cooling tower operating conditions. During that period, warm water in the tank, resulting from daytime usage, is chilled using chiller for an estimated 8 hours charging process. The chilled water is slowly diffused from lower part of the tank through bottom diffuser with nozzle facing downward to minimize temperature mixing of warm and chilled water. A

* *Corresponding author.*
E-mail address: kaharosman@utm.my (Kahar Osman)

water layer of temperature difference known as thermocline occurs during both charging and discharging processes. A good CWS system diffuser design is to minimize mixing of the thermocline as much as possible. Chiller is then turned off upon completion of charging (when the chilled water has filled-up uniformly in the tank). During the discharging process, chilled tank water from the tank is pumped out and flows through the air conditioning piping system inside the buildings. Simultaneously, warm water from the air conditioning systems fills the tank through the top diffuser with nozzle facing upward. The thermocline will slowly moves downward as warm water fills the tank. Figure 1 shows the schematic and flowchart of a CWS TES system.

In general, TES systems are categorized as; heat storage material type, storage cycle frequency, delivery scheme, heat storage mechanism and operating temperature range-based TES systems. Common TES systems include chilled water, coil pipe, packed bed cold, ice slurry and gas hydrate cool storage systems, are categorized depending on operating temperature type TES system [1]. In this study, a chilled water (CWS) based TES system is analyzed using Computational Fluid Dynamics (CFD).

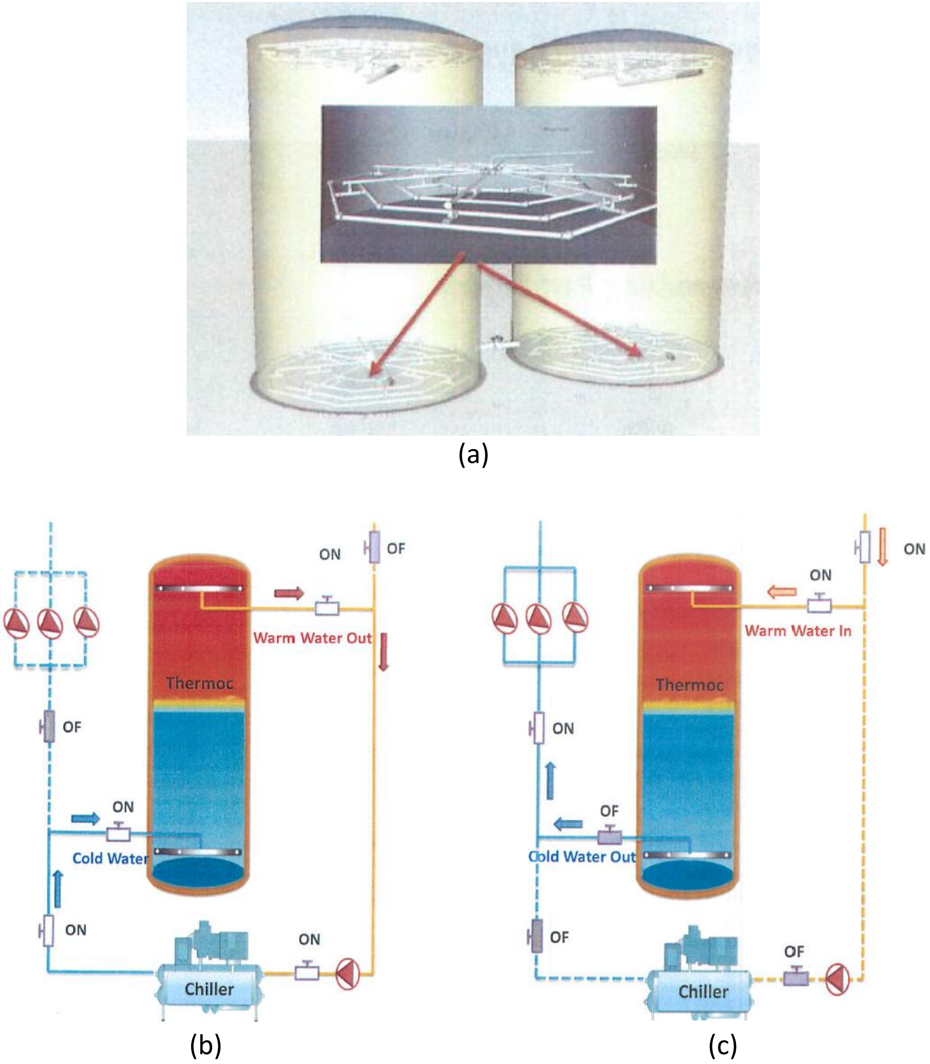


Fig.1. (a) Water tank and diffuser (b) system flowchart during charging (c) discharging

In a CWS system, fundamental tank design optimum ratio has been studied extensively. Parametric study on stratified chilled water storage system justified that tank height to diameter ratio up to a value of 3 improves performance of the system. In contrast, thermoclines degrades due to ambient heat transfer, thermal diffusion, axial wall conduction as well as thermal mixing [2]. Design of a TES system should facilitate normal operating temperature of -40°C to 600°C and chosen materials should be thermally, chemically and mechanically stable within the temperature range. In addition, sliding coil around the tank wall also affects the thermal stratification of thermal storage especially in seasonal TES systems. Low temperature zone will appear in the middle of the tank during winter season [4].

One of the methods to optimize TES systems is Unit Commitment (UC) model. In the model, heat load, electricity price, outdoor temperature and heat generation unit characteristics are analyzed to yield minimum system operating cost, heat generation reduction and storage utilisation strategy [5]. For an ice storage system, enhancing heat transfer in phase change material (PCM) maximizes the energy stored. It can be accomplished using parametric study to obtain ideal configuration of tube-in-tank latent heat (LHTES) system [6]. Besides, heat transfer performance enhancement of PCM can also be achieved using nanoparticles [14]. Design of TES remains challenging as good TES systems are sensitive to fluctuating economics and environmental boundary conditions [7]. Nelson *et al.,* [2] experimentally tested effects of aspect ratio, flow rates, initial temperature difference and thickness insulation on mixing coefficient of stratified chilled-water tanks. The height to diameter ratios of more than 3 show insignificant performance improvement. Larger height to wall thickness ratio results in good thermal stratification and does not improve much for ratios higher than 200 for any tank. Besides, the percent cold recoverable (PCR) during discharge increases with increasing initial temperature difference, aspect ratio and flow rate. Charge and discharge efficiencies increase with increasing flow rate and initial temperature difference [8].

Apart from diffuser optimization, another important factor to ensure efficient energy storage is heat transfer control. During operation, poor heat transfer between storage medium and ambient coolant such as air, significantly affects charging and discharging rate of TES systems. Uncontrollable energy losses severely affect performance of TES systems. A study [9] reported an annual heat loss of 421 MWh from Friedrichshafen district heating system (3017 MWh annual capacity) caused by high operating temperature, poor insulation, lengthy piping network, cyclic phase separation, tank container corrosion and vapor pressure leakage. Meanwhile during relaxation period (without external flow), walls of the container have a strong effect in destroying thermocline. Experiments show that better thermal stratification can be maintained using horizontal porous baffles [3]. In addition, flow separation at T-junctions on top and bottom diffuser can also been improved using vortex generator for a more uniform diffusing process [13]. Moreover, detailed investigation of flow pattern through mitre bend on a few joints of the diffusers are also worthwhile during diffuser design process [15].

Thermal energy storage system using CFD simulation such as dual medium thermocline TES systems has been studied [10]. For such medium, they justified that charging and discharging efficiencies decrease with time. However, inlet fluid temperature does not significantly affects performance. Recently, better medium for TES system with higher storage capacity per unit is studied which is paraffin wax-water nano emulsion [11]. William *et al.,* [12] did a comparative study using CFD on slotted pipe diffusers during charging process. They found clear relationship between inlet Richardson number based on slot hydraulic radius and thermal performance of both field data and simulations. In prior studies, turbulence has less effect than Richardson number. Instead, turbulence show significant effect at low Richardson number (on the order of 100) for slotted pipe diffuser.

This study provides a comparative study using CFD on a CWS storage system that is lacking in the literature. For this purpose, detailed analysis on heat and flow behavior at bottom diffuser of specific octagonal diffuser design is conducted.

2. Methodology

CFD simulations have been conducted to investigate flow characteristics within a thermal energy storage tank. Our results are compared with calculated flow velocities and temperature from analytical data and yield satisfactory agreement. For turbulence cases, the fluid is governed by:

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(v + v_t)\left(\frac{\partial U_i}{\partial x_j}\right)\right] \tag{1}$$

$$\frac{\partial U_j}{\partial x_j} = 0 \tag{2}$$

$$v_t = C_\mu \frac{k^2}{\varepsilon} \tag{3}$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = -\overline{u_i u_j}\frac{\partial U_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\frac{K_m}{\sigma_k}\frac{\partial k}{\partial x_j}\right) - \varepsilon \tag{4}$$

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = -C_{\varepsilon 1}\frac{\varepsilon}{k}\overline{u_i u_j}\frac{\partial U_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\frac{K_m}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) - C_{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{5}$$

Here, $C_\mu = 0.09$, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$

Three types of CFD simulation cases are investigated which are; 2D tank without diffuser (Case A), 3D diffuser without tank (Case B) and 3D tank and diffuser (Case C). This study commenced with some simplifications from suppliers design schematics to compensate relatively small nozzle compared to tank size which are total number of nozzles reduction and symmetrical assumption of the tank as shown in Figure 2. All results presented here are for charging process. Tank details and diffuser details are shown on Table 1 (a) and Table 1 (b) respectively.

Table 1
Simulation parameters
(a) 2D tank without diffuser (Case A)

Width	24.50 m
Height	25.42 m
Inlet temperature, T_1	5 °C
Outlet temperature, T_2	12 °C
Inlet velocity, V_1	6.12993×10^{-4} m/s

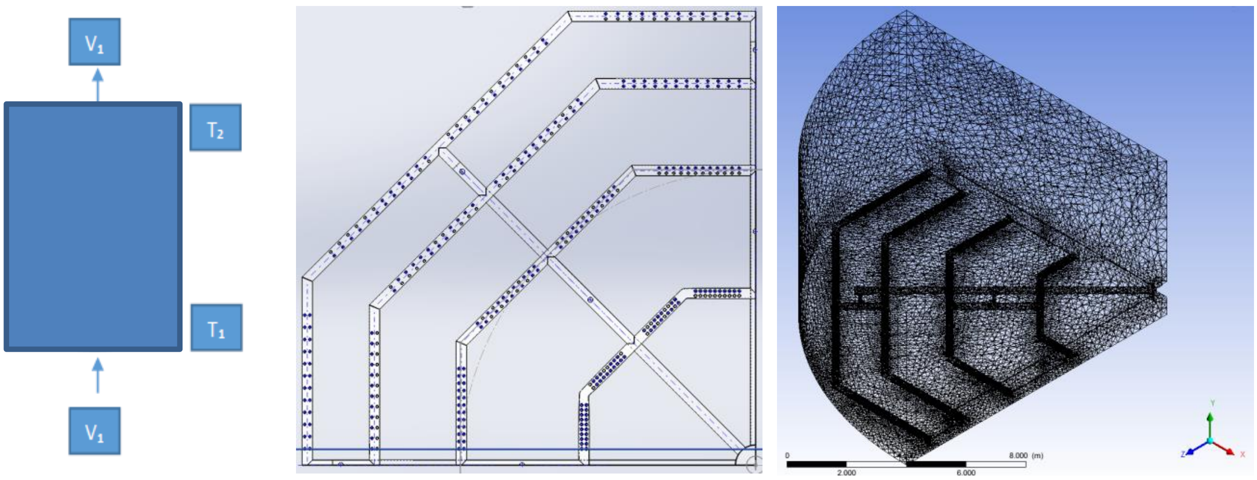
(b) 3D diffuser without tank (1/4) (Case B)

Nozzle diameter	0.032 m
Number of nozzles/sets	2 nozzles
Angle between both nozzles	120°
Ring 1, R1 (inner)	10 x 4 sets
R2	12 x 4 sets
R3	12 x 4 sets
R4	12 x 4 sets
Total number of nozzles	368
Inlet velocity (header pipe)	1.82 m/s

(c) 3D tank and diffuser (1/4 and 1/8 height) (Case C)

Nozzle velocity (as inlet into tank), V_n	0.2m/s

Schematics of each case are presented in Figure 2. For thermocline analysis, 2D simulation is sufficient to yield thermal stratification. Thus, 3D simulations of Case B and Case C are conducted to study flow behaviour with constant temperature. Flows in Case A and Case C are laminar. Meanwhile, k-epsilon turbulence model is used for Case B. Main inlet pipe (header pipe) is simplified on Case B for symmetrical purpose.



(a) (b) (c)

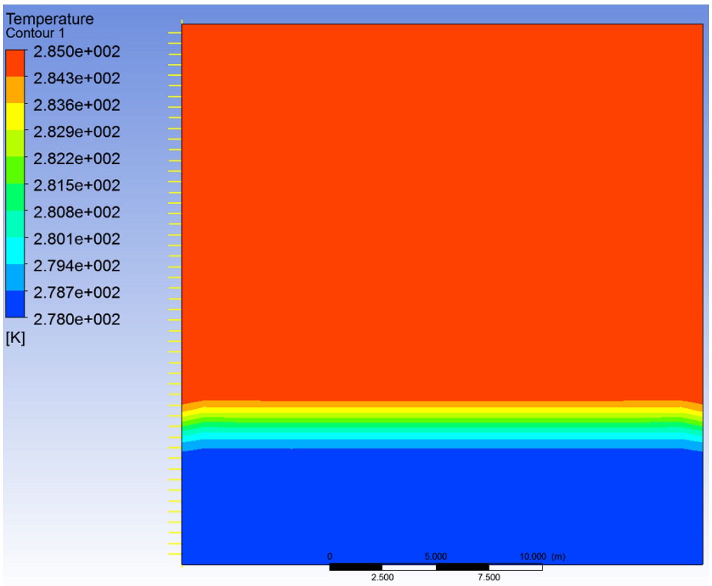
Fig. 2. (a) 2D tank without diffuser (b) 3D diffuser without tank (1/4) (c) 3D tank and diffuser (1/4 and 1/8 height)

Brief descriptions of diffuser are as follow; inlet water flows from header pipe (0.45m dia.) at tank center into 8 allocating pipes. For each allocating pipe, 2 allocating-distributing connector pipes link R1/R2 and R3/R4 diffusing pipes. Lastly, 4 octagonal diffusing pipes (R1, R2, R3 and R4) contains nozzle configuration as shown in Figure 2 (b).

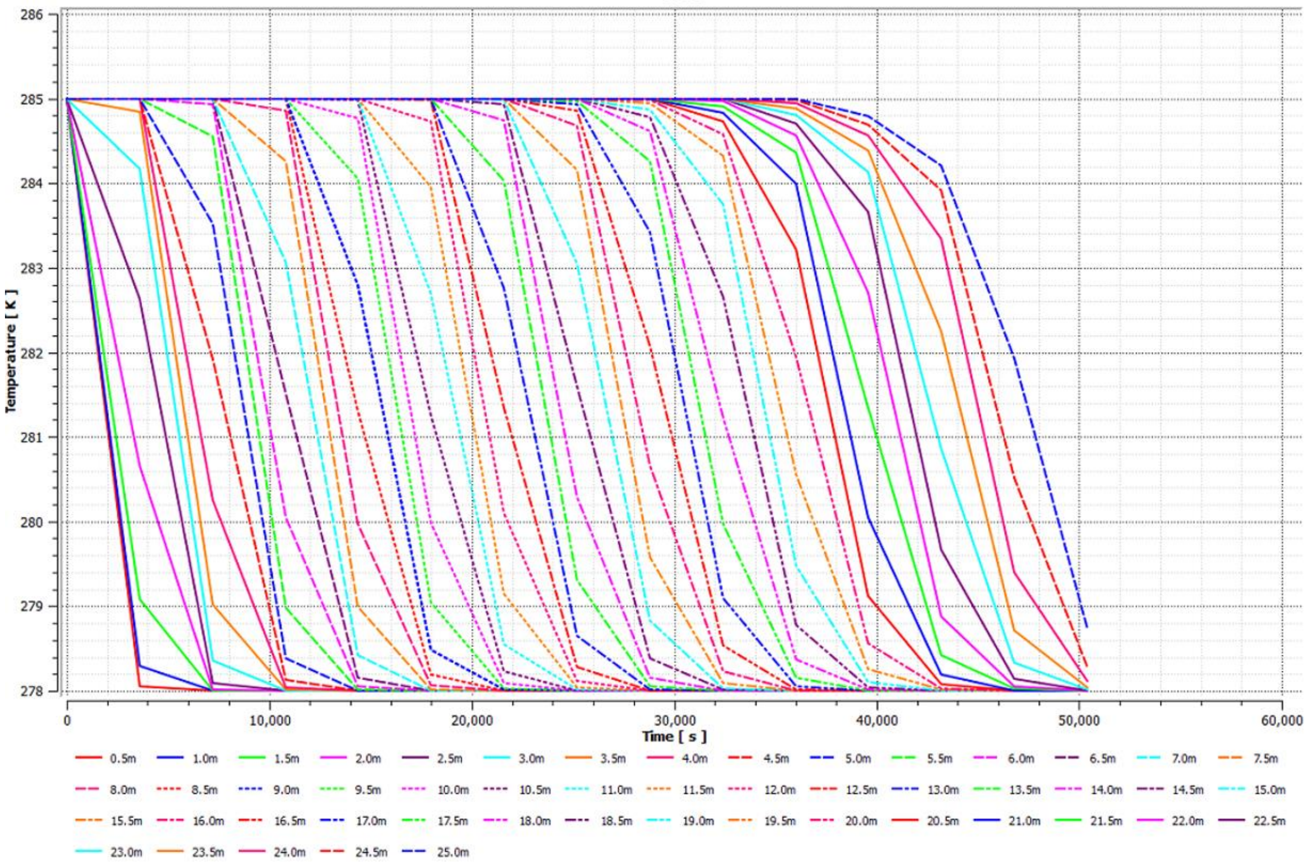
3. Results

3.1 Case A - 2D Tank Without Diffuser

During charging, chilled water from bottom diffuser slowly pushes the warm water upwards through top diffuser out of the tank. Here, thermocline thickness is constantly moving upwards and the temperature for thermocline calculation is taken from 280K to 285K. From Figure 3 (a), the thermocline thickness is approximately 1.3m compared to 1.1m obtained by analytical data. This is caused by diffuser simplifications, initial temperature domain patch, and slight variation in upwards velocity, V1, as well as effective tank width and height. Figure 3 (b) shows temperature history of tank water at tank wall (thermocline). Temperature values are stored on hourly basis which result in uneven thermocline lines.



(a)



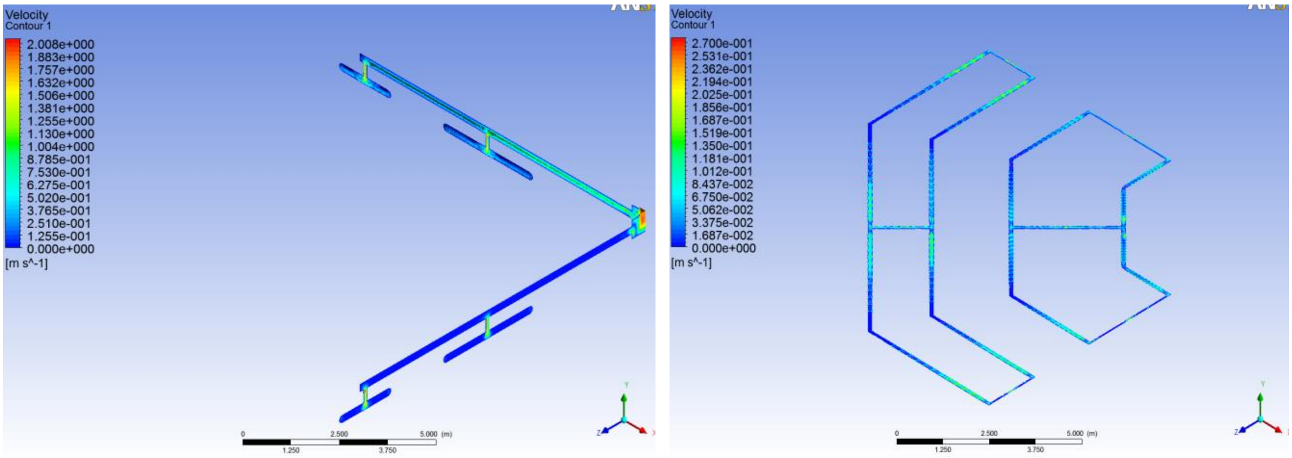
(b)

Fig. 3. (a) Temperature distribution 3 hours during charging (b) temperature vs time for each 0.5m tank height (thermocline)

Based on the figure, time taken for temperature drop is slightly longer as charging process approaches completion compared to earlier charging process. This is due to diffused heat in time that causes slightly thicker thermoclines as the warm water leaves the tank.

3.2 Case B - 3D Diffuser Without Tank

Based on Figure 4 (a), the velocity magnitude distribution is high near the centre of diffuser (header pipe). Header pipe velocity is approximately 0.8m/s compared to 0.6m/s from analytical data. This is due to inlet simplification that forces faster flow downwards inside the header pipe. However, result shows higher velocity at the allocating-diffusing connector pipe compared to allocating and diffusing pipe due to smaller diameter. Allocating pipe velocity is approximately 1.25m/s compared to 1.1m/s from analytical data. In Figure 4 (b), the velocity is higher near the allocating-diffusing connector pipe and velocity reduces further away from it in the diffusing pipe. Besides, the velocity at outer diffusing pipe is also slightly higher than inner diffusing pipe. Diffusing pipe velocity is approximately 0.25m/s compared to 1.4m/s from analytical data. Water momentum distributed further in allocating pipe as water reaches pipe end wall compared to momentum at the middle of allocating pipe. Asymmetrical meshes are the reason for less symmetrical velocity contours obtained here.



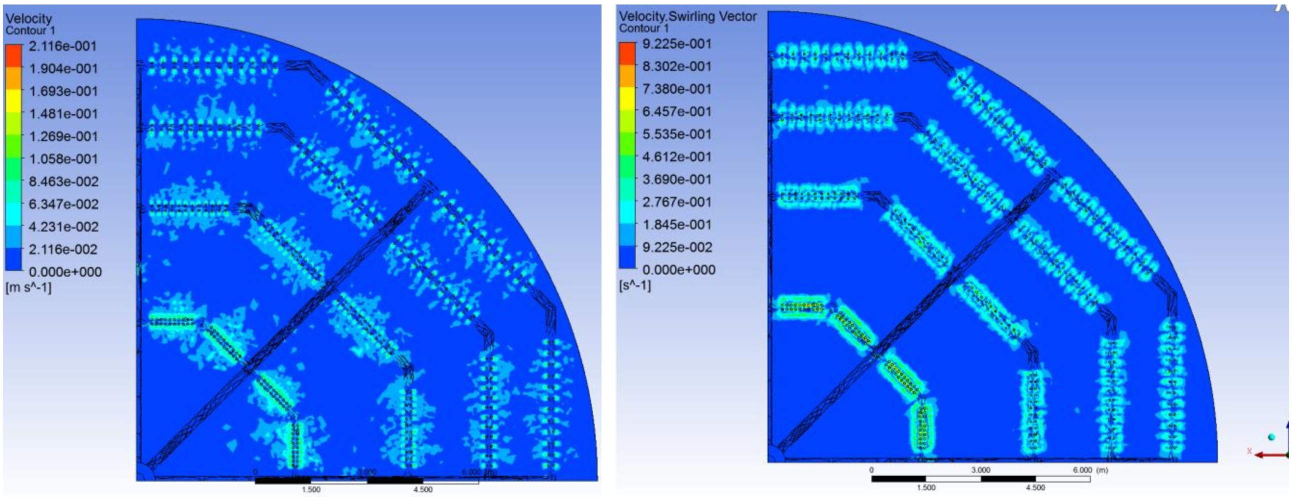
(a) (b)

Fig. 4. (a) Velocity magnitude contour of symmetry plane (b) plane with nozzles

Conclusively, the maximum nozzle velocity obtained is approximately 0.22 m/s located at nozzles near allocating-diffusing connector pipe compared to 0.26m/s from analytical data. Further away, the nozzle velocity magnitudes are around 0.01m/s.

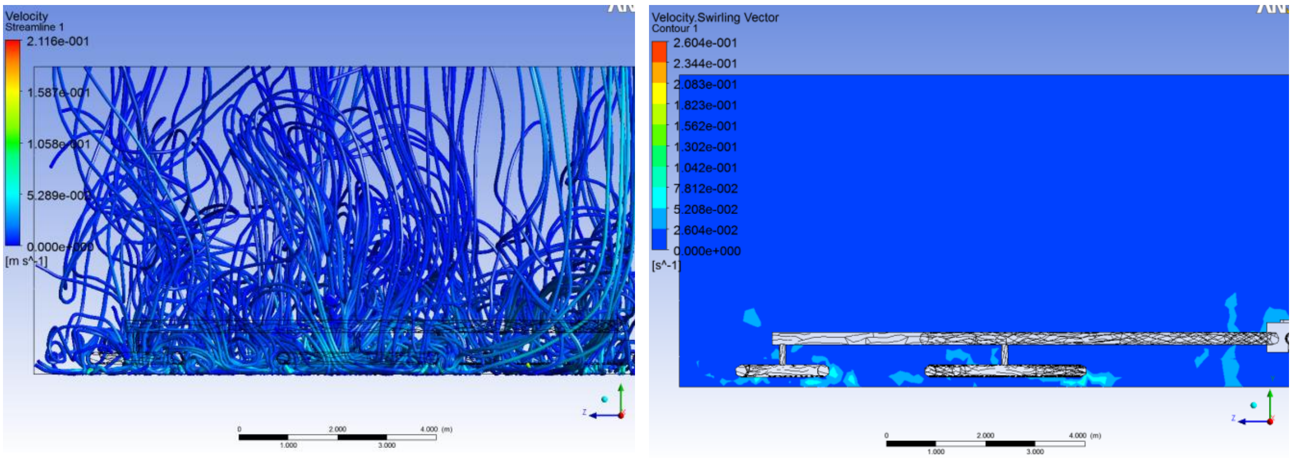
3.3 Case C - 3D Tank and Diffuser

In previous section, non-uniform velocities are obtained for each nozzle. In this section, uniform nozzle outlet velocity is assumed (Vn=0.2m/s) to eliminate nozzle-tank interface setup. The purpose of this simulation case is to identify flow circulation that potentially contribute to mixing of thermoclines. Swirling parameter is used to measure mixing tendancy as in Figure 5 and Figure 6.



(a) (b)

Fig. 5. Velocity magnitude contour (a) and swirling contour (b) at tank bottom surface



(a) (b)

Fig. 6. Streamlines of particles released from tank bottom surface (a) and swirling contour of symmetry plane (b)

Figure 5 (a) shows higher velocity magnitudes at inner distributing pipe, R1. It is important to note that, as previously discussed in Figure 4 (b), velocities in R1 should be slightly slower due to water momentum variations across allocating pipe, instead, constant and uniform nozzle velocities is used in this simulation case. Figure 5 (b) also shows higher swirling values near R1 pipe region. However, that critical region does not generate intense mixing or circulation as shown in Figure 6 (a). Streamlines of particles show smooth (almost straight) fluid movements upward in upper region of bottom diffuser. In addition, swirling contour on symmetry plane also shows low mixing tendency as in Figure 6 (b).

4. Conclusions

Present results show good agreement with analytical data. Besides, this study provides a more comprehensive understanding on flow and thermal behaviours of a TES diffuser system. In conclusion, present results can be summarized as below:

a) Thermocline thickness is approximately 1.3m compared to 1.1m obtained by analytical data.
b) Maximum nozzle velocity obtained is approximately 0.22 m/s located at nozzles near allocating-diffusing connector pipe compared to 0.26m/s from analytical data.
c) The TES system design have good thermal stratification.

Present CFD results provide thorough analysis that complement limited calculated values in analytical data. This study provides validation basis for future TES diffuser system optimization study.

Acknowledgement
This study is funded by Universiti Teknologi Malaysia under Research University Grant (GUP) entitled "Optimisation of Stratification Diffuser Chilled Water Thermal Storage Tank for District Cooling Appication" with registered Vot No. Q.J130000.2545.17H54 It is a university-private sector collaboration with AHAR Consultants PLT. It is also funded by Research University Grant (GUP) entitled “Improvement on Nozzle Design and Spray Optimisation for Fogging System” with registered Vot. No. Q.J130000.2545.11H55.

References
[1] Alva, Guruprasad, Yaxue Lin, and Guiyin Fang. "An overview of thermal energy storage systems." *Energy* 144 (2018): 341-378.
[2] Nelson, J. E. B., A. R. Balakrishnan, and S. Srinivasa Murthy. "Parametric studies on thermally stratified chilled water storage systems." *Applied Thermal Engineering* 19, no. 1 (1999): 89-115.
[3] Mahmoud, Mohammed Hamed, and Ghanim Kadhem Abdulsada. "Enhancing Thermal Stratification in Liquid Storage Tanks During Relaxation Periods." *Journal of Engineering and Sustainable Development* 12, no. 1 (2008): 161-177.
[4] Zhang, Xinhe, Zhiyuan Liu, Bo Qu, Zishuo Huang, Xiaodong Chen, Wei Huang, and Yujie Kang. "The CFD simulation and analysis of energy station thermal storage’s influence to water storage tank’s sliding coil." *Energy Procedia* 142 (2017): 3617-3624.
[5] Romanchenko, Dmytro, Johan Kensby, Mikael Odenberger, and Filip Johnsson. "Thermal energy storage in district heating: Centralised storage vs. storage in thermal inertia of buildings." *Energy conversion and management* 162 (2018): 26-38.
[6] Fang, Yuhang, Jianlei Niu, and Shiming Deng. "Numerical analysis for maximizing effective energy storage capacity of thermal energy storage systems by enhancing heat transfer in PCM." *Energy and Buildings* 160 (2018): 10-18.
[7] Welsch, Bastian, Laura Göllner-Völker, Daniel O. Schulte, Kristian Bär, Ingo Sass, and Liselotte Schebek. "Environmental and economic assessment of borehole thermal energy storage in district heating systems." *Applied energy* 216 (2018): 73-90.
[8] Nelson, J. E. B., A. R. Balakrishnan, and S. Srinivasa Murthy."Experiments on stratified chilled-water tanks." *International Journal of Refrigeration.* 22 (1999): 216-234.
[9] Rezaie, Behnaz, Bale V. Reddy, and Marc A. Rosen. "Assessment of the thermal energy storage in friedrichshafen district energy systems." *Energy Procedia* 116 (2017): 91-105.
[10] Nandi, Bhaskar Rahul, Santanu Bandyopadhyay, and Rangan Banerjee. "Numerical modeling and analysis of dual medium thermocline thermal energy storage." *Journal of Energy Storage* 16 (2018): 218-230.
[11] Sivapalan, B., Neelesh Chandran, M., Manikandan, S., Saranprabhu, M. K., Pavithra, S., and Rajan, K. S. "Paraffin wax–water nanoemulsion: A superior thermal energy storage medium providing higher rate of thermal energy storage per unit heat exchanger volume than water and paraffin wax." *Energy Conversion and Management*, 162 (2018): 109–117.
[12] Bahnfleth, William P., Jing Song, and John M. Cimbala. "Measured and modeled charging of a stratified chilled water thermal storage tank with slotted pipe diffusers." *HVAC&R Research* 9, no. 4 (2003): 467-491.
[13] Hamad A., Syed M. A. A. and Kamarul A. A., "Reducing flow separation in T-junction pipe using vortex generator: CFD study." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 44, no. 1 (2018): 36-46.
[14] Kean, Tung Hao, Nor Azwadi Che Sidik, Yutaka Asako, Tan Lit Ken, and Siti Rahmah Aid. "Numerical study on heat transfer performance enhancement of phase change material by nanoparticles: a review." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 45, no. 1 (2018): 55-63.

[15] Ainaa M. M. M., Fazila M. N. and Jamaludin M. S., "Flow pattern and particle contamination on fluid flow through mitre band." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 45, no. 1 (2018): 128-140.