Uludağ University Journal of The Faculty of Engineering, Vol. 28, No. 2, 2023 RESEARCH

DOI: 10.17482/uumfd.1286863

THE EXPERIMENTAL AND NUMERICAL INVESTIGATION OF THE THERMAL BEHAVIORS OF A JACKETED HEAT EXCHANGER WITH A STIRRER

Halil BAYRAM [*]
Gökhan SEVİLGEN [**]
Mert CESUR [***]

Received: 24.04.2023; revised: 09.06.2023; accepted: 21.06.2023

Abstract: In this study, the thermal performance of a jacketed heat exchanger with a stirrer was experimentally and numerically investigated under different jacket side water inlet temperature and flow rate conditions. In the numerical study, a three-dimensional computational fluid dynamics (CFD)model of the heat exchanger was generated and the analyzes were performed with the ANSYS-Fluent software package. In addition, a stirrer was added to both the experimental and the numerical study to obtain the effects of the stirrer on the temperature and velocity values of the water in the tank. It was seen that the results of the analyzes performed under similar conditions to the experimental study were in good agreement with the experimental study. It was concluded that the effect of the flow rate decreases with increasing inlet temperature. When the stirrer was activated and the flow rate was increased from 0.5 to 2.5 l/min in 30, 40, and 50°C inlet temperature conditions, the time to reach the target temperature inside the tank decreased from approximately 1600 to 900, from 2500 to 1350, and from 4900 to 1400, respectively. In general, it was observed that the effect of the stirrer increased with increasing inlet temperature in all flow rate conditions.

Keywords: Stirred Tank, Jacketed Heat Exchanger, CFD

Karıştırıcılı Ceket Tip Isı Değiştiricilerin Isıl Davranışlarının Deneysel ve Sayısal Olarak İncelenmesi

Öz: Bu çalışmada, karıştırıcılı ceket tip bir ısı değiştiricisinin ısıl performansı ceket tarafı su giriş sıcaklık ve debi koşulları altında deneysel ve sayısal olarak incelenmiştir. Sayısal çalışmada, ısı değiştiricisinin üç boyutlu hesaplamalı akışkanlar dinamiği (HAD) modeli oluşturulmuş ve analizler ANSYS-Fluent paket programı kullanılarak yapılmıştır. Ayrıca tank içerisindeki suyun sıcaklık ve hızına olan etkisini inceleyebilmek için hem deneysel hem de sayısal çalışmada tank içerisine bir karıştırıcı eklenmiştir. Deneysel çalışmayla benzer koşullar altında gerçekleştirilen analiz sonuçlarının deneysel çalışma sonuçlarıyla uyumlu olduğu görülmüştür. Artan giriş sıcaklığıyla debi etkisinin azaldığı sonucuna varılmıştır. Karıştırıcı devredeyken ve debi 0.5'ten 2.5 l/dk'ya çıkarıldığında, hedeflenen sıcaklığa ulaşma süreleri 30, 40 ve 50°C giriş sıcaklık koşullarında sırasıyla 1600'den 900'e, 2500'den 1350'ye ve 4900'den 1400'e azalmıştır. Genel olarak tüm debi koşullarında karıştırıcı etkisinin artan giriş sıcaklık değeriyle birlikte arttığı gözlenmiştir.

Anahtar Kelimeler: Karıştırıcılı Tank, Ceket Tip Isı Değiştirici, HAD

[*] Department of Mechanical Engineering, Engineering Faculty, Amasya University, 05100, Amasya/Türkiye
[**] Department of Automotive Engineering, Engineering Faculty, Bursa Uludağ University, 16059, Bursa/Türkiye
[***] TOFAŞ, Turkish Automotive Factory, R&D, Propulsion Systems, Bursa, Türkiye
Corresponding Author: Halil Bayram (halil.bayram@amasya.edu.tr)

1. INTRODUCTION

The heat transfer between two or more fluids is carried out by using the heat exchangers. The heat exchangers are examined under different groupings according to their different properties such as process type, flow arrangements, heat transfer mechanisms, etc. (Bhutta et al., 2012; Kakac et al., 2020; Bayram, 2022a). The jacketed heat exchangers examined in this study are basically obtained by adding a jacket to the outer surface of a mixing tank. In this added jacket part, fluids at different temperatures can circulate with desired flow rate values. This type of jacketed heat exchanger is widely used in heat rejection from internal combustion engines (ICEs) (Gavali et al., 2007; Kessler et al., 2007; Kruger et al., 2008; Karras et al., 2014). When impellers are added to these tanks, it is also known that they are used in many different areas such as food, medicine, cosmetics, petrochemistry, etc. (Lakghomi et al., 2008; Kakac et al., 2020). Basically, it is aimed that the fluids in these tanks reach the desired properties such as temperature, homogeneity, etc. in the desired time. In order to control the time to reach the desired condition, the impellers in the tank play an important role. (Rezend, 1996). The sizing parameters affecting the thermal performance and processing time of a jacketed heat exchanger with a stirrer can be listed as dimensions of the tank, the location of the stirrer(s) and the number of the stirrers, distance between the stirrer edge and the flat bottom of the tank, the geometrical type of baffles, and the thickness of the wall etc. There are also rating parameters affecting the thermal performance and processing time of a jacketed heat exchanger with a stirrer and can be listed as the mass flow rate, the temperature difference between hot and cold fluids separated by wall, the rotational velocity of the stirrer etc. (Rezend, 1996; Sharafani, 2015; Bayram and Sevilgen, 2017a). When the studies on these parameters are examined, it has been seen that the undesired vortexes in the tank can be prevented by the baffles added to the inner walls of the tank (Rezend, 1996; Missen et al., 1999). Cui et al. experimentally and numerically investigated the mixing process for laminar flow in pitched-blade turbine stirred tanks (Cui et al, 2018). They concluded that the impeller diameter was the parameter that most influenced the mixing time. After the impeller diameter, the parameters affecting this duration were respectively discharge angle and blade width. Tong et al. investigated the effects of the heat transfer coefficient, the temperature value of circulating heating water and the pressure on the dehydriding performance by using a jacketed and coiled-tube heat exchanger that consisted of phase change material (PCM) (Tong et al., 2021). Delgado et al. experimentally investigated of heat transfer coefficient and volumetric energy density in a coiled stirred tank containing a low cost PCM emulsion. They stated that the overall heat transfer coefficient was calculated as around 3.5–5.5 times higher when the stirring rate was 290-600 rpm (Delgado et al., 2017). In addition, they noted that even at the lowest stirring rate, PCM use caused an improvement of almost 100%. Lakghomi et al. numerically investigated the effects of stirred coil and jacket on the flow behaviors and heat transfer inside the tanks. They observed that the use of coils resulted in a more uniform temperature distribution and higher heat transfer coefficient. On the other hand, they also stated that the power required for impellers was higher than the jacketed case in coiled stirred tanks (Lakghomi et al., 2008). Major-Godlewska investigated the parameters affecting the heat transfer in the agitated vessel equipped with a jacket or coil. As a result, it was concluded that heat transfer depends on the impeller shape and the properties of the liquid (Major-Godlewska, 2014). Debab et al. conducted a study in a jacketed vessel equipped with turbine impellers to obtain optimum experimental conditions for a non-Newtonian fluid. According to the obtained optimization results, they stated that an agitated vessel equipped with a Flat Blade Disc Turbine (FBDT) of diameter ratio d/D = 0.6 and baffles are the most advantageous properties for heat transfer (Debab et al., 2011). When the numerical studies on heat exchangers are examined, a carefully generated CFD model can make accurate predictions about the thermal performance and flow characteristics of the heat exchanger (Bayram and Sevilgen, 2018; Sevilgen and Bayram, 2020). Daza et al. generated a CFD model of a jacketed stirred tank equipped with a six-blade Rushton turbine impeller to obtain a Nusselt number correlation due to dependency of the heat transfer coefficient on the impeller and the speed of rotation. The authors stated that the results of the Nussxelt number correlation were in a good agreement with the experimental results obtained from a reliable representation of the heat transfer in the tank (Daza et al., 2019). This type of heat exchangers were also used in Organic Rankine Cycle (ORC) systems. Zaniewski et al. examined a 10 kWe ORC micro-turbine generator to provide better heat transfer coefficient, reduce the mean temperature at the shell wall tangential to the electric generator stator and decrease in fin size and increase in fin density at the finned wall surface by optimization of the water jacket geometry of the generator (Zaniewski et al., 2023). In addition, it can be seen from the literature that the jacketed heat exchangers were also used in metal hydride (MH) reactor systems that was expected to performed good performance considering possess compact structure, enhanced heat transfer, good gas tightness etc. (Bai et al., 2020). In this study, the heat exchanger used in the experimental study was modeled in three dimensions and analyzed under similar conditions to the experimental study. The obtained results were also compared with the experimental study results.

2. MATERIALS AND METHOD

2.1. Experimental Setup

In the experimental study, GUNT WL110 main unit and GUNT WL 110.04 model jacketed heat exchanger were used. Water was used as the working fluid in both the tank and the jacket sides. The outer surface of the water tank was isolated, and it has been accepted as adiabatic as in similar studies in the literature (Bayram and Sevilgen, 2017b; Bayram, 2022b). Therefore, it was assumed that the heat transfer only occurred between the two fluids through the wall of the water tank. On the jacket side, the hot water was circulated at the desired flow rates thanks to the pump located in the main unit. In addition, thanks to the resistance in the main unit, the temperature values of the hot water can also be adjusted to desired values. On the tank side, cold water was charged to the tank, and then at the end of the heating process, the water was discharged manually. This process was repeated for all cases in experimental studies. The heat exchanger used in the experiments can be seen in Figure 1.

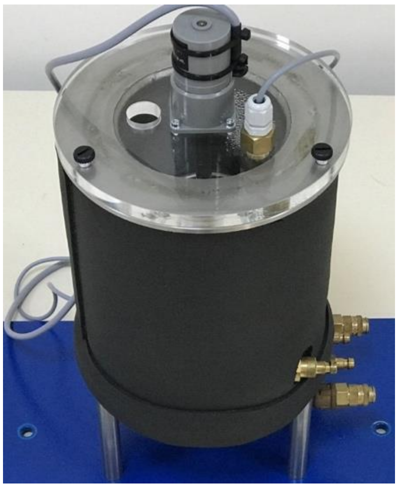
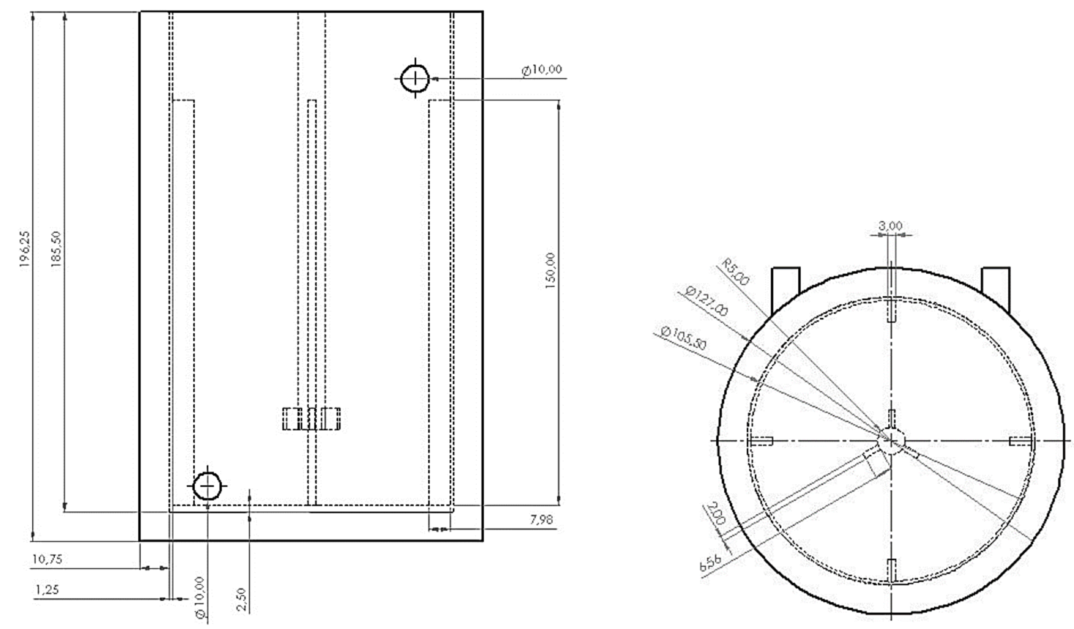


Figure 1:
The jacket heat exchanger used in the experiments

In the numerical study, the heat exchanger used in the experiments was modeled in three dimensions, and numerical calculations were performed with transient conditions by using the ANSYS-Fluent software package. The dimensions of the modeled heat exchanger are presented in Figure 2.

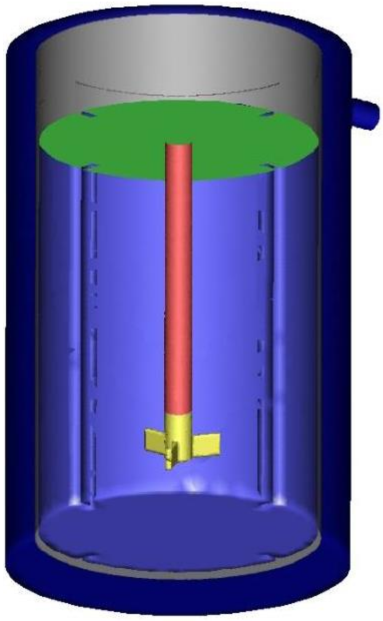


a. *b.*

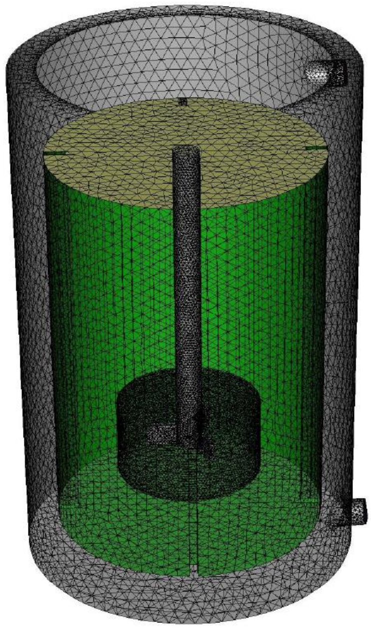
Figure 2:
*The **a.** side and **b.** top views of the heat exchanger*

2.2. Numerical Setup

Considering the dimensions of the heat exchanger used in the experimental study, the CAD model of the jacketed heat exchanger with a three-bladed stirrer was given in Figure 3 (a). The mesh structure of the model consists of approximately 3.8 million tetrahedron elements and can be seen in Figure 3 (b). In the generated CFD model, the movement of the stirrer was carried out using the Single Rotating Reference Frame method and the created reference frame around the stirrer can also be seen in Figure 3 (b). The purpose of this method is to stabilize a stationary problem relative to rotating equipment. This method is used to model fluids that start to move with a constant speed rotating equipment like a stirrer and accelerating in the radial direction (Cerisola, 2012; ANSYS, 2013).



a.



b.

Figure 3:
*The **a.** CAD model **b.** mesh structure views of the heat exchanger*

When the rotational speed is constant, it can be transformed the equations of fluid motion to the rotating frame. And it can be possible by adding two extra acceleration terms in the momentum equation named Coriolis and centripetal. And also, there must be a relation between the absolute and relative velocity. The stationary frame velocity of the fluid can be transformed to the rotating frame one by using the following equation:

$$\overrightarrow{v_r} = \vec{v} - (\vec{\omega} - \vec{r}) \quad (1)$$

There are two different methods to express the equations of motion. These are the relative velocity and the absolute velocity formulations (Equations 2-8). The conservation of mass is calculated by using Equation 2 for both methods.

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \overrightarrow{v_r} = 0 \quad (2)$$

$$\frac{\delta}{\delta t}(\rho \overrightarrow{v_r}) + \nabla . (\rho \overrightarrow{v_r} \overrightarrow{v_r}) + \rho(2\vec{\omega} \text{ x } \overrightarrow{v_r} + \vec{\omega} \text{ x } \vec{\omega} \text{ x } \vec{r}) = -\nabla p + \nabla . \overline{\overline{\tau}}_r + \vec{F} \quad (3)$$

$$\frac{\delta}{\delta t}(\rho E_r) + \nabla . (\rho \overrightarrow{v_r} H_r) = \nabla . (k \nabla T + \overline{\overline{\tau}}_r . \overrightarrow{v_r}) + S_h \quad (4)$$

$$E_r = h - \frac{p}{\rho} + \frac{1}{2}(v_r^2 - u_r^2) \quad (5)$$

$$H_r = E_r + \frac{p}{\rho} \quad (6)$$

$$\frac{\delta}{\delta t}(\rho\vec{v}) + \nabla.(\rho\vec{v_r}\vec{v}) + \rho(\vec{\omega} \times \vec{v}) = -\nabla p + \nabla.\bar{\bar{\tau}}_r + \vec{F} \tag{7}$$

$$\frac{\delta}{\delta t}(\rho E) + \nabla.(\rho\vec{v_r}H + \rho\vec{u_r}) = \nabla.(k\nabla T + \bar{\bar{\tau}}_r.\vec{v}) + S_h \tag{8}$$

The effects of adding the stirrer to the tank side, the temperature, and the flow rate of the water circulating in the jacket side on the thermal behaviors of the heat exchanger have been experimentally and numerically investigated. When the stirrer was used in the experiments, its speed was kept constant at 330 rpm. The inlet temperature value of the hot water on the jacket side was determined as 30, 40, and 50°C. In addition, experiments were carried out at constant volumetric flow rates of 0.5, 1.5, and 2.5 l/min at each temperature value. In addition, the initial temperature value of the water in the tank was kept constant at approximately 11°C in all experiments. The case list in detail was presented in Table 1.

Table 1. The boundary conditions used in the experimental study

Case No	Stirrer [rpm]	$\dot{V}$inlet,jacket [l/min]	Tinlet, jacket [°C]
1	-	0.5	30
2	-	1.5	
3	-	2.5	
4	-	0.5	40
5	-	1.5	
6	-	2.5	
7	-	0.5	50
8	-	1.5	
9	-	2.5	
10	330	0.5	30
11	330	1.5	
12	330	2.5	
13	330	0.5	40
14	330	1.5	
15	330	2.5	
16	330	0.5	50
17	330	1.5	
18	330	2.5	

A validation study was also carried out to assure the accuracy of the results of the numerical study. In this validation study, the results of the experimental study performed at 50°C and 2.7 l/min were used. In the generated CFD model, a point named P0 was assigned to a position equivalent to the temperature sensor in the tank (Figure 4). And the values obtained at this point were used in the validation study.

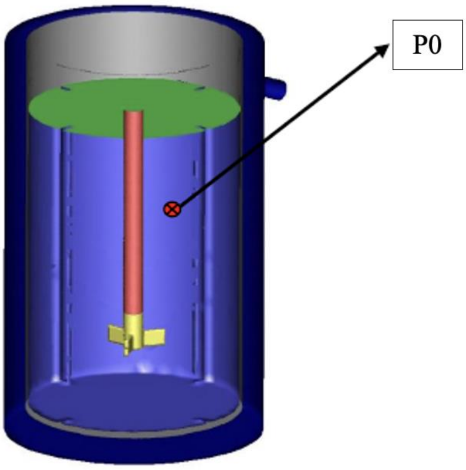


Figure 4:
The generated point to determine the temperature values inside the tank

Solver settings preferred in the numerical study were presented in Table 2. As indicated in the table, the k-omega SST turbulence model was used in the analysis. This turbulence model was preferred due to the precision and stability of the numerical results in the available literature (Hellsten and Laine, 1997; Sadino-Riquelme et al., 2002; Dhakal and Walters, 2009; Fernandez and Neuhez, 2022). In order to examine the temperature and velocity distributions obtained from these analyses in detail, from the tangent of the lower and upper sections of the volume created for SRRF (Y0 and Y4), from the tangent of the lower and upper sections of the stirrer (Y1 and Y3), and from the middle section of the stirrer (Y2) imaginary surfaces were assigned (Figure 5).

Table 2. Solver settings used in the numerical simulations

Solver type	Pressure-based
Time	Transient
Equations	Combined simulation of flow and energy
Flow type	k-omega SST turbulence model
Solution method	Coupled

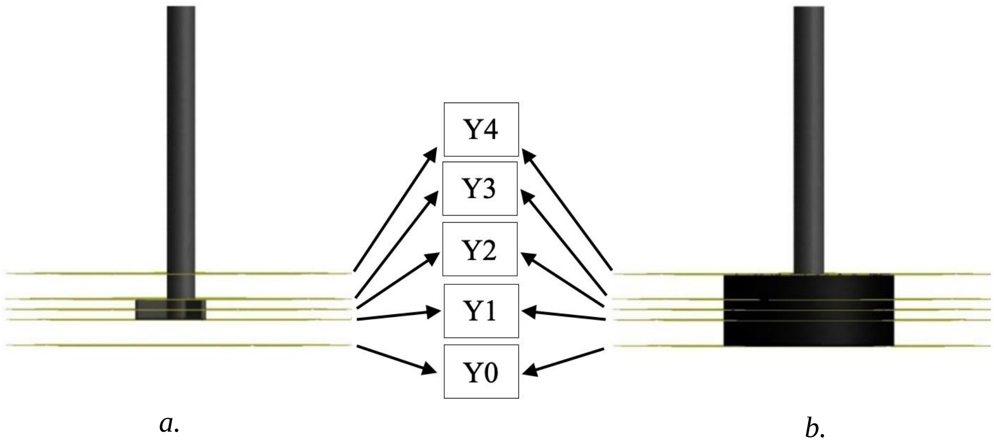


Figure 5:
The generated surfaces around the stirrer to determine the temperature and velocity distributions inside the tank

2.3. Numerical Setup

In the validation process, the results obtained from the analyzes with the results of the experimental study carried out under similar conditions were compared and can be seen in Figure 6. According to the results of the experimental study, which were carried out at 50°C inlet temperature, 2.7 l/min flow rate and the stirrer was active boundary conditions, the reaching time to the target temperature was approximately 1300 seconds. This time value was approximately 1000 seconds in the numerical study. In addition, although temperature differences were observed between the experimental and numerical study at the beginning of the comparison, it was observed that the temperature values were in quite agreement after approximately 400 seconds. These differences in temperature values were thought to be since the outside of the tank was adiabatic in the numerical study, but the insulation material outside the tank in the experimental study. Although this insulation material reduced heat losses, it could not completely prevent it. In addition, there was no heat loss in the numerical study and all heat transfer occurred between the jacket and tank sides. Consequently, it can be said that the results of the analyses were in good agreement with the results of the experimental study.

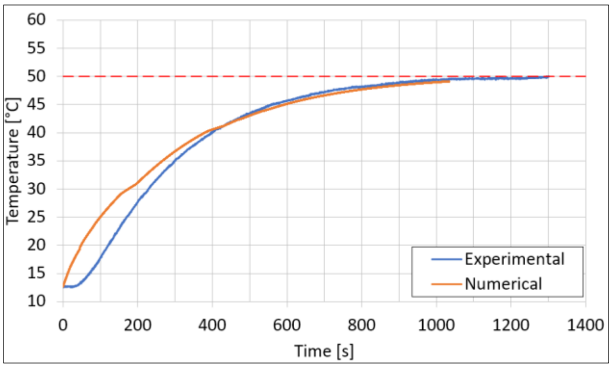


Figure 6:
The comparison of the results of the experimental and numerical studies

3. RESULTS AND DISCUSSIONS

In Figure 7, the effects of jacket side flow rate are presented in detail. When the results obtained from the experiments are examined, it was observed that the cases with the lowest flow rate values reached the target temperature in the longest time in all experiments. In cases where the stirrer was not active, the reaching time value varied between approximately 2150 and 5250 seconds at 0.5 l/min flow rate condition. This value was determined as approximately 2900 seconds at 40°C jacket side inlet temperature condition. Although the target temperature value was reached at relatively close time values under the inlet temperature condition of 30 and 40°C, this time value was considerably longer when the inlet temperature value was 50°C. The highest and lowest time values for 30, 40, and 50°C inlet temperature conditions were determined as 1700-1600, 2150-2000, and 3000-2600 seconds, respectively. In cases where the stirrer was active, the reaching time values varied between about 1600 and 4900 seconds at 0.5 l/min flow rate condition. This value was determined as approximately 2500 seconds at 40°C inlet temperature condition. Under the 30°C inlet temperature condition, the reaching time value increased from approximately 1600 to 900 seconds with an increase in flow rate from 0.5 to 2.5 l/min. This value decreased from approximately 2500 to 1350 and from 4900 to 1400 seconds at 40 and 50°C inlet temperature conditions, respectively. Although the differences in the reaching time values of 1.5 and 2.5 l/min conditions were close to when the stirrer was not active, the difference values increased with the activation of the stirrer. It was also found that these differences decreased with increasing inlet temperature conditions.

The effects of the stirrer on the results of experiments performed under the same conditions are presented in detail in Figure 8. By activating the stirrer at 30 and 40°C inlet temperature conditions at the lowest flow rate, the reaching time value to the target temperature decreased by approximately 500 seconds. The decreasing in this value was determined as approximately 700 seconds at 50°C inlet temperature condition. When the flow rate value was 1.5 l/min, a decrease of approximately 400 seconds was determined in the reaching time at the lowest temperature condition. A decrease of approximately 700 seconds was observed in other temperature conditions. At the highest flow rate, activating the stirrer at 30 and 40°C inlet temperature conditions resulted in a relatively low reduction in the reaching time to the desired temperature in the tank. However, although the target temperature was reached for a very long time when the stirrer was not active at 50°C inlet temperature condition, the reaching time was reduced by about half when the stirrer was active. In general, the effect of the stirrer increased with increasing inlet temperature in all flow rate conditions. An improvement in the reaching time was observed under 30 and 40°C inlet temperature conditions. However, when the inlet temperature value was 50°C, this decrease in the reaching time was observed to be quite high.

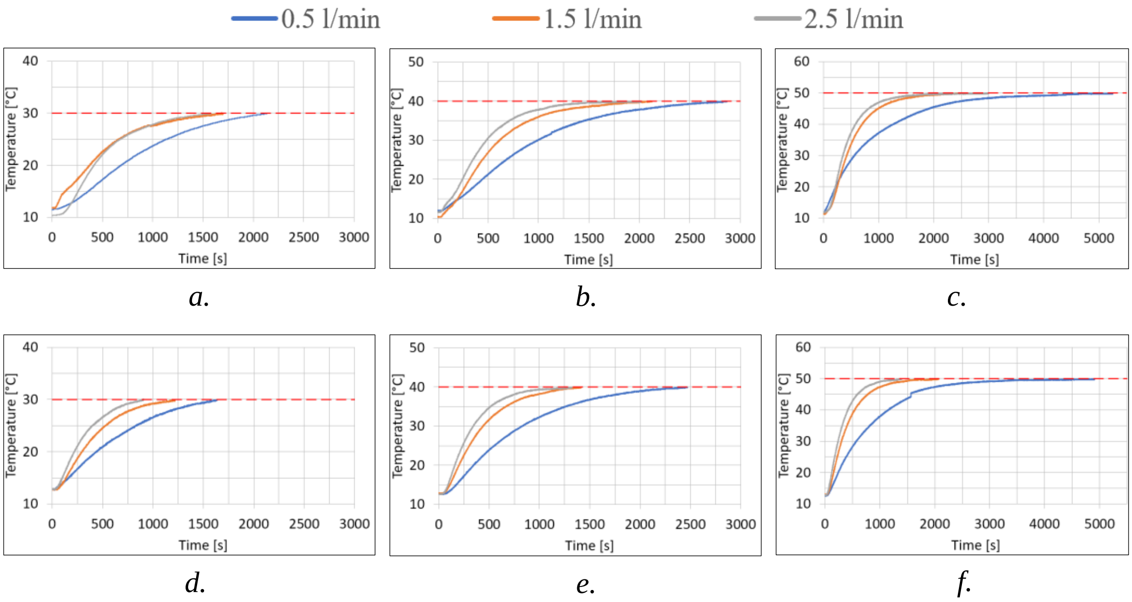


Figure 7:
The comparison of the reaching time values under the same temperature and stirrer conditions ***a.*** *0 rpm/30°C* ***b.*** *0 rpm/40°C* ***c.*** *0 rpm/50°C* ***d.*** *330 rpm/30°C* ***e.*** *330 rpm/40°C* ***f.*** *330 rpm/50°C*

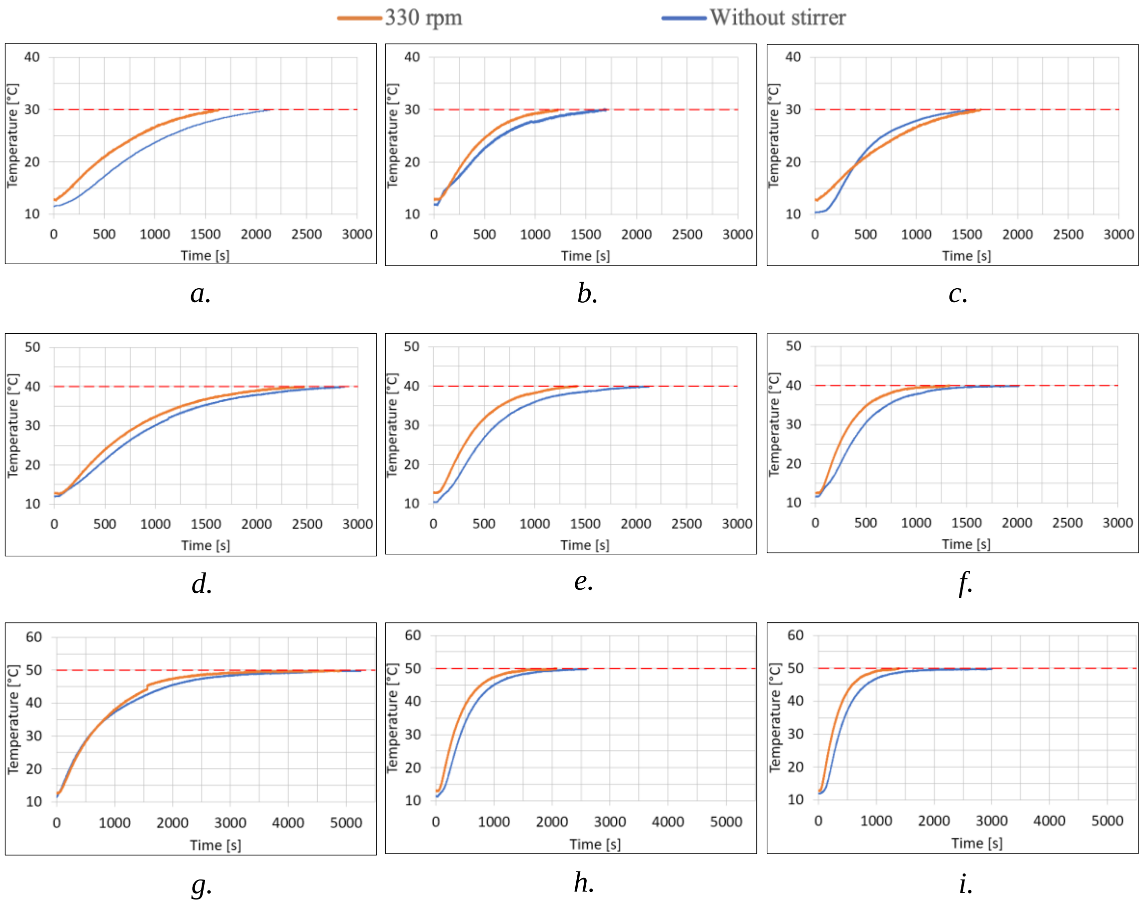


Figure 8:
The comparison of the reaching time values under the same temperature and stirrer conditions ***a.*** *0.5 l/min /30°C* ***b.*** *0.5 l/min /40°C* ***c.*** *0.5 l/min /50°C* ***d.*** *1.5 l/min /30°C* ***e.*** *1.5 l/min /40°C* ***f.*** *1.5 l/min /50°C* ***g.*** *2.5 l/min /30°C* ***h.*** *2.5 l/min /40°C* ***i.*** *2.5 l/min /50°C*

The velocity distributions on defined surfaces in the tank were presented in Figure 9. It was seen that the velocity distributions on the Y1 and Y3 surfaces located just below and above the stirrer were quite similar. The highest velocity distribution was observed at Y2 surface which located at the center of the stirrer. Velocities of about 0.8 m/s were observed in the rear regions of the stirrer blades. It has also been observed that there was a velocity region of about 0.5 m/s in a very wide area in the outer parts of the blades. In the Y0 plane, which was defined at the bottom, a very low velocity distribution was observed compared to the stirrer region. On the Y4 surface, which was the top surface, it has been observed that the stirrer had almost no effect on the velocity distribution. It has also been observed that the velocity values decreased at a high rate as they moved away from the stirrers on all created surfaces.

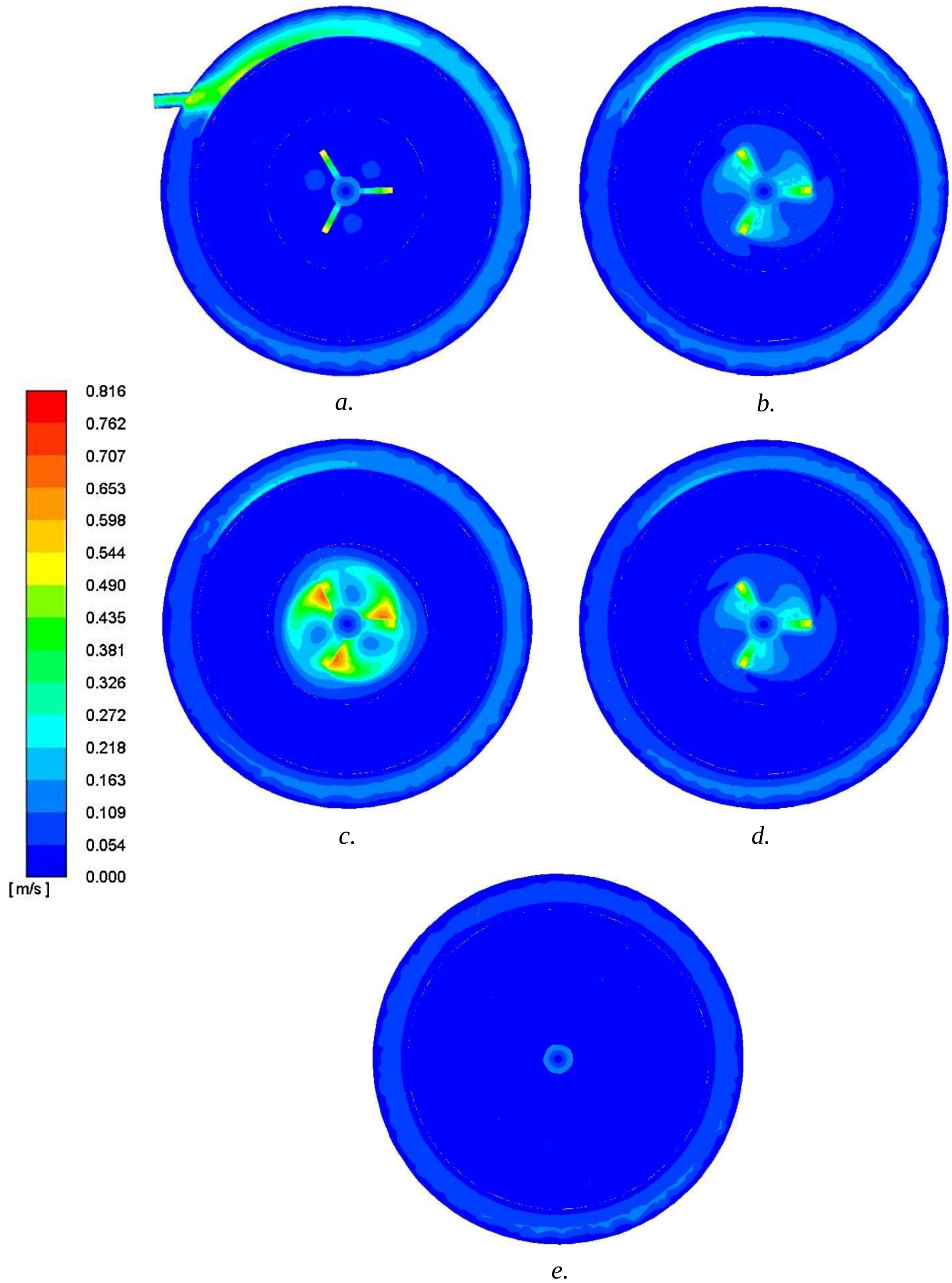


Figure 9:
The velocity distributions at the generated surfaces inside the tank

Considering the temperature distributions obtained in the Y2 plane, which was the plane in the center of the stirrer (Figure 10), it was seen that there was a relatively high temperature increase in the outer region of the stirrer. In addition, it has been observed that there was a relatively higher heat transfer rates in the regions where the baffles were located, thanks to the movement of the fluid created by the stirrer. When the temperature distribution in the section taken in the middle section of the tank was examined, the effect of the stirrer on the temperature distribution can be seen more clearly (Figure 11). It is seen that the fluid movement caused by the rotation of the stirrer considerably increased the convection heat transfer. It has been observed that there was a high rate of heat transfer especially by the jacket side in the lower region of the stirrer. In addition, it has been determined that the temperature values of the water in the area where the baffles interact with the rotation of the stirrer increase considerably compared to the other regions.

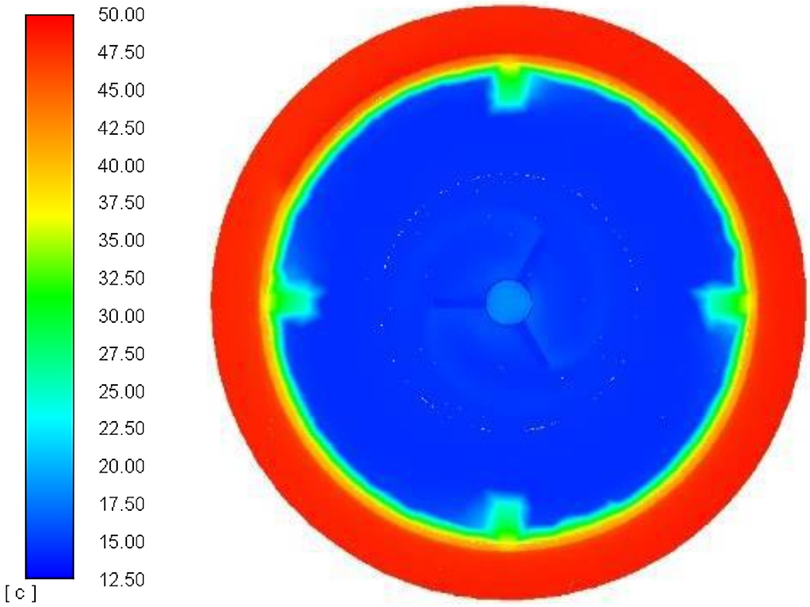


Figure 10:
The temperature distribution at the Y2 surface inside the tank

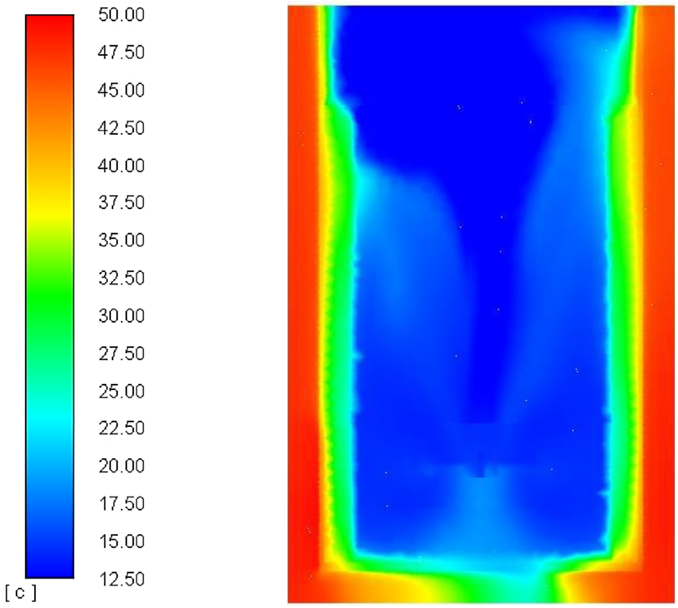


Figure 11:
The temperature distribution at the middle section of the tank

4. CONCLUSIONS

In this study, the thermal behaviors of a jacketed heat exchanger that used in many working areas such as automotive, food, medicine, cosmetics, petrochemistry etc. were experimentally investigated for getting the effects of jacket side inlet temperature and flow rate and adding a stirrer to the tank. We also numerically investigated by using the generated three-dimensional CFD model of the heat exchanger to obtain the velocity and temperature distributions inside the tank. According to the performed experiments and numerical analyses, the main results of this study are described below:

- In all experiments, the cases with the lowest flow rate value had the longest reaching time to the target temperature.
- In cases where the stirrer was not active, the reaching time value varied between approximately 2150 and 5250 seconds at 0.5 l/min flow rate condition.
- In cases where the stirrer was active, when the flow rate value increased from 0.5 to 2.5 l/min in 30, 40, and 50°C inlet temperature conditions, the reaching time values decreased from approximately 1600 to 900, from 2500 to 1350, and from 4900 to 1400 seconds, respectively.
- By activating the stirrer at 30 and 40°C inlet temperature conditions at the lowest flow rate, the reaching time value to the target temperature decreased by approximately 500 seconds. The decreasing in this value was determined as approximately 700 seconds at 50°C inlet temperature condition.
- At the highest flow rate, although activating the stirrer at 30 and 40°C inlet temperature conditions resulted in a relatively low reduction in the reaching time to the desired temperature in the tank, the reaching time was reduced by about half when the inlet temperature value was set to 50°C.
- According to the results of the analyses, the highest velocity values were observed as 0.8 m/s in the rear region of the blades on the Y2 surface defined in the middle of the stirrer.
- It is seen that the fluid movement caused by the rotation of the stirrer considerably increased the convection heat transfer. It has been observed that there was a high rate of heat transfer especially by the jacket side in the lower region of the stirrer.

CONFLICT OF INTEREST

The authors declare no conflicts of interest.

AUTHOR CONTRIBUTION

Halil BAYRAM, Gökhan SEVİLGEN, and Mert CESUR performed the experiments and analyses, analyzed the research data to contribute to the design configurations, and also wrote this paper.

REFERENCES

1. ANSYS Fluent User's Guide, Release 15.0, (2013).

2. Bai, X. S., Yang, W. W., Zhang, W. Y., Yang, F. S., and Tang, X. Y. (2020) Hydrogen absorption performance of a novel cylindrical MH reactor with combined loop-type finned tube and cooling jacket heat exchanger, *International Journal of Hydrogen Energy*, 45(52), 28100-28115. doi: 10.1016/j.ijhydene.2020.04.209

3. Bayram, H. and Sevilgen, G. (2017a) The effects of using a stirrer on the thermal performance of a water jacketed heat exchanger, *International Conference on Engineering Technologies (ICENTE'17)*, 937-939.

4. Bayram, H., and Sevilgen, G. (2017b) Numerical investigation of the effect of variable baffle spacing on the thermal performance of a shell and tube heat exchanger, Energies, 10(8), 1156. doi: 10.3390/en10081156

5. Bayram, H. and Sevilgen, G. (2018) Numerical investigation of the effects of different baffle types on the thermal performance of a shell and tube heat exchanger, *Academic Platform-Journal of Engineering and Science*, 6(3), 58-66. doi:10.21541/apjes.397414

6. Bayram, H. (2022a) Analysis of numerical and grey relation method of the effect of helical tape insert density on the hydrothermal performance, *Case Studies in Thermal Engineering*, 39, 102406. doi:10.1016/j.csite.2022.102406

7. Bayram, H. (2022b) Numerical investigation of heat transfer improvement of a double pipe heat exchanger with koch snowflake fractal and longitudinal fin designs, International Journal of Fluid Mechanics Research, 49(1). doi: 10.1615/InterJFluidMechRes.2022042499

8. Bhutta, M. M. A., Hayat, N., Bashir, M. H., Khan, A. R., Ahmad, K. N., and Khan, S. (2012) CFD applications in various heat exchangers design: A review, *Applied Thermal Engineering*, 32, 1-12. doi: 10.1016/j.applthermaleng.2011.09.001

9. Cerisola, A. (2012). Numerical analysis of tidal turbines using virtual blade model and single rotating reference frame, Department of Mechanical Engineering-University of Washington, Seattle (WA)-USA: IRENAV, ARTS ET METIERS ParisTech.

10. Cui, Y., Zhang, H., Li, X., Yang, M., and Guan, Z. (2018) Computational and experimental investigation of laminar flow mixing system in a pitched-blade turbine stirred tank, *International Journal of Agricultural and Biological Engineering*, 11(4), 111-117. doi: 10.25165/j.ijabe.20181104.2729

11. Daza, S. A., Prada, R. J., Nunhez, J. R., and Castilho, G. J. (2019) Nusselt number correlation for a jacketed stirred tank using computational fluid dynamics, *The Canadian Journal of Chemical Engineering*, 97(2), 586-593. doi: 10.1002/cjce.23385

12. Debab, A., Chergui, N., Bekrentchir, K., and Bertrand, J. (2011) An investigation of heat transfer in a mechanically agitated vessel, Journal of Applied Fluid Mechanics, 4(2), 2-50. doi: 10.36884/jafm.4.02.11915

13. Delgado, M., Lázaro, A., Mazo, J., Peñalosa, C., Marín, J. M., and Zalba, B. (2017) Experimental analysis of a coiled stirred tank containing a low cost PCM emulsion as a thermal energy storage system. *Energy*, 138, 590-601. doi:10.1016/j.energy.2017.07.044

14. Dhakal, T. P., and Walters, D. K. (2009) Curvature and rotation sensitive variants of the K-Omega SST turbulence model, In Fluids Engineering Division Summer Meeting, Vol. 43727, pp. 2221-2229.

15. Fernandes, L. B., and Nunhez, J. R. (2022) CFD Simulation of Mixing Tank with Different Rushton Agitator Diameters and Constant Power Consumption, Proceedings of the 7th World Congress on Momentum, Heat and Mass Transfer (MHMT'22), Lisbon, Portugal. doi: 10.11159/icmfht22.135

16. Gavali, M. G., Subbarao, A. V., and Marathe, N. V. (2007) Optimization of water jacket using CFD for effective cooling of water-cooled diesel engines, *SAE Technical Paper*, 2007-26-049. doi: 10.4271/2007-26-049

17. Hellsten, A., Laine, S., Hellsten, A., and Laine, S. (1997) Extension of the k-omega-SST turbulence model for flows over rough surfaces. In 22nd atmospheric flight mechanics conference, 3577, 252-260.

18. Kakac, S., Liu, H., & Pramuanjaroenkij, A. (2020) Heat exchangers: selection, rating, and thermal design. CRC press.

19. Karras, N., Kuthada, T., and Wiedemann, J. (2014) An approach for water jacket flow simulations, *SAE Technical Paper*, 2014-01-0659. doi: 10.4271/2014-01-0659

20. Kessler, M. P., Kruger, M., Ataídes, R., de La Rosa Siqueira, C., Argachoy, C., and Mendes, A. S. (2007) Numerical analysis of flow at water jacket of an internal combustion engine, *SAE Technical Paper*, 2007-01-2711. doi:10.4271/2007-01-2711

21. Kruger, M., Kessler, M. P., Ataídes, R., de La Rosa Siqueira, C., dos Reis, M. V. F., Mendes, A. S., and Argachoy, C. (2008) Numerical analysis of flow at water jacket of an internal combustion engine, *SAE Technical Paper*, 2008-01-0393. doi: 10.4271/2008-01-0393

22. Lakghomi, B., Kolahchian, E., Jalali, A., and Farhadi, F. (2008) Coil and jacket's effects on internal flow behavior and heat transfer in stirred tanks, *International Journal of Chemical and Molecular Engineering*, 2(12), 383-387. doi:10.5281/zenodo.1086197

23. Major-Godlewska, M. (2014) An effect of different factors on heat transfer process in an agitated vessel, *Czasopismo Techniczne*, 2, 85-94.

24. Missen, R. W., Missen, R. W., Mims, C. A., and Saville, B. A. (1999) Introduction to chemical reaction engineering and kinetics. John Wiley & Sons Incorporated.

25. Rezend, C. (1996). CFD analysis of flow field in a mixing tank with and without baffles, *Master of Science*, Department of Mechanical Engineering College of Engineering Rochester Institute of Technology Rochester, New York.

26. Sadino-Riquelme, M. C., Rivas, J., Jeison, D., Donoso-Bravo, A., and Hayes, R. E. (2022) Computational modelling of mixing tanks for bioprocesses: Developing a comprehensive workflow, The Canadian Journal of Chemical Engineering, 100(11), 3210-3226. doi: 10.1002/cjce.24220

27. Sevilgen, G., and Bayram, H. (2020) Numerical analysis of heat transfer of a brazed plate heat exchanger, *Academic Platform-Journal of Engineering and Science*, 8(3), 491-499. doi: 10.21541/apjes.683151

28. Sharafani, A. T. (2015). Simulation of The Agitated Batch, *Master of Science*, Czech Tecnical University in Prague, Faculty of Mechanical Engineering, Department of Process Engineering.

29. Tong, L., Yuan, Y., Yang, T., Bénard, P., Yuan, C., and Xiao, J. (2021) Hydrogen release from a metal hydride tank with phase change material jacket and coiled-tube heat exchanger, *International Journal of Hydrogen Energy*, 46(63), 32135-32148. doi: 10.1016/j.ijhydene.2021.06.230

30. Zaniewski, D., Klimaszewski, P., Klonowicz, P., Witanowski, Ł., Lampart, P., Jędrzejewski, Ł., and Suchocki, T. (2023) Organic Rankine Cycle turbogenerator cooling–optimization of the generator water jacket heat exchange surface, *Applied Thermal Engineering*, 120041. doi: 10.1016/j.applthermaleng.2023.120041