

Tek Uçlu Eğimli Boru-Tank Sisteminin HAD Modellemesi

Burhan BAYHAN ^{1*}  Gökhan ARSLAN ² 

¹ Selçuk University, Cihanbeyli Vocational School, Department of Mechanical and Metal Technology, Konya, Türkiye

² Mersin University, Faculty of Engineering, Department of Mechanical Engineering, Mersin, Türkiye

Makale Bilgisi	ÖZET
Geliş Tarihi: 19.10.2024 Kabul Tarihi: 02.04.2025 Yayın Tarihi: 31.12.2025	Bu çalışmada, tek uçlu eğimli boru-tank sisteminin laboratuvarında yapılan doğal dolaşım deneylerinin sayısal analizinde Hesaplamalı Akışkanlar Dinamiği (HAD) yöntemi kullanılmıştır. Sayısal analiz sonucunda hız vektörleri ve sıcaklık konturları elde edilmiştir. Bu sonuçlara göre, doğal dolaşımın var olduğu ve eğimli borunun tank girişine yakın yerde daha etkili olduğu ve borunun kapalı dip kısmına doğru gidildikçe bu etkinliğin azaldığı gibi çıkarımlar yapılmıştır. Ayrıca 6 saatlik deney sonucunda elde edilen tank ortalama sıcaklığı ile sayısal analiz sonucunda elde edilen ortalama tank sıcaklığı kıyaslandığında, eğimli boruya uygulanan 600 ve 800 W yüksek güçteki deneylerde ısı rejiminin sağlandığı ve 200 ve 400 W daha düşük deneylerde ise ısı rejiminin sağlanması için daha çok zaman gerektiği sonucuna varılmıştır.
Anahtar Kelimeler: Doğal akış, Tek uçlu eğimli boru, HAD.	

CFD Modeling of the One-Ended Inclined Pipe-Tank System

Article Info	ABSTRACT
Received: 19.10.2024 Accepted: 02.04.2025 Published: 31.12.2025	In this study, Computational Fluid Dynamics (CFD) method was used in the numerical analysis of the natural circulation experiments of the one-ended inclined pipe-tank system conducted in the laboratory. As a result of the numerical analysis, velocity vectors and temperature contours were obtained. According to these results, it was concluded that natural circulation exists and that it is more effective near the tank inlet of the inclined pipe and that this effectiveness decreases as one moves towards the closed bottom of the pipe. In addition, when the tank average temperature obtained as a result of the 6-hour experiment is compared with the average tank temperature obtained as a result of the numerical analysis, it is concluded that the thermal regime is provided in the experiments with high power of 600 and 800 W applied to the inclined pipe and that more time is required to provide the thermal regime in the experiments with lower power of 200 and 400 W.
Keywords: Natural flow, One-ended inclined pipe, CFD.	

To cite this article:

Bayhan, B. & Arslan, G. (2025). CFD modeling of the one-ended inclined pipe-tank system. *Necmettin Erbakan University Journal of Science and Engineering*, 7(3), 361- 371. <https://doi.org/10.47112/neufmbd.2025.98>

*Corresponding Author: Burhan Bayhan, bayhanburhan@gmail.com



This article is licensed under a Creative Commons Attribution-NonCommercial 4.0 International License (CC BY-NC 4.0)

INTRODUCTION

The unsustainability and unequal distribution of fossil fuels are increasingly directing countries to renewable energy sources [1]. Increasing energy demands and technological advances have accelerated this transition [2]. This issue is now an incredibly attractive one for Turkey in terms of reducing its energy dependency on energy [3]. Therefore, energy is of significant importance in ensuring the welfare and economic development of countries [4]. Renewable energy offers a sustainable and environmentally friendly alternative in various fields by using resources that are continuously or repeatedly renewed by natural processes [5-7].

The efficient use of energy, which is an indispensable part of human life, is one of the most studied current issues today [8]. Natural circulation systems can be given as an example. These systems work according to natural laws such as gravity and buoyancy and automatically start the flow in the system without the need for any machinery, with the density difference caused by heating and cooling. Therefore, the maintenance and operating costs of these systems are quite low. For example, the use of natural ventilation systems in the air conditioning of buildings is of significant importance to reduce energy consumption and maintain the ecosystem balance [9]. Another example is the water heating systems frequently used today. In these systems, the flow occurs in single-ended inclined pipes.

Natural circulation solar water heating systems; These systems, which are widely used in our country and are becoming increasingly popular in other countries, consist of collectors, storage tanks and connection pipes and include energy collection and use systems [10, 11].

There are many studies in the literature on natural flow in pipes: For example, Bejjam and Kiran [12] performed a 3-D simulation of a single-phase natural circulation loop with Ansys 14.5 (Fluent) program. In this loop, they performed steady-state analysis of water and water-based Al_2O_3 (Al_2O_3 -water) nanofluid at 1%, 3%, 5% and 6% particle volume concentrations. They adopted the mixing model to simulate the nanofluid-based natural circulation loop depending on the Stokes number. Li et al. [13] performed a numerical and theoretical study on a single-ended, horizontal, and vacuum tube. As a result, they found that the secondary flow in natural circulation has a significant effect on the temperature distribution and flow rate in the tube. Pleshanov et al. [14] designed two types of natural circulation loops in a fluidized bed steam boiler. The first type was designed as a complex loop, while the second type was divided into independent sections. As a result, it was observed that the specific metal amount in both designed circulation circuits was almost the same. Riahi and Taherian [15] tested a closed thermosyphon flat plate solar water heater with natural circulation and found that, contrary to widespread belief, such systems could provide sufficient energy to meet the hot water demand. Misale et al. [16] conducted an experimental study on natural circulation in interconnected vertical rectangular loops. With the results obtained from the study, this study was accepted as the first attempt to optimize several interconnected natural circulation loops. Bocanegra et al. [17] investigated different heater-cooler configurations in the natural circulation loop. Bayhan and Arslan [18] created a model for thermal calculations in a single-ended inclined pipe, calculation of natural circulation flow rate, energy balance in the tank to which the pipe is connected and estimation of pipe outlet temperature. They also compared this model with models in literature. As a result, the model created was easier and more successful. Bayhan and Arslan [19] aimed to experimentally investigate the natural circulation solar energy system containing parabolic trough solar collector under real field conditions in the outdoor environment. For this purpose, an experimental setup system was established that included a parabolic trough solar collector, followed the sun in one dimension from east to west on the N-S axis and operated with natural circulation. Experiments were conducted on different dates when solar radiation values varied. With the data obtained from these experiments, it was found that the radiation values coming to the opening of the movable collector and 56% of this radiation could reach the vacuum tube glass pipe located at the

focus of the collector. In all experiments, information about the existence of natural circulation was obtained by finding the Rayleigh numbers in the tank-vacuum tube glass pipe connection section. Additionally, studies on CFD were utilized during this study. For example, Jaber et al. [20] aimed to increase the thermal stratification in a cylindrical heat storage tank. For this purpose, they placed a conical passive flow guide and a heat exchanger coil in the tank. It was investigated both experimentally and numerically by writing codes. As a result, a significant increase in efficiency was achieved. Canlı et al. [21] numerically investigated the velocity and turbulence distributions in the hydrodynamic inlet length of the pipes depending on the axial and radial positions. Ceviz et al. [22] investigated the steady laminar natural convection heat transfer due to gravity acceleration through a vertical pipe with zero wall thickness using CFD. Canlı et al. [23] aimed to calculate the heat transfer in pipes for flows changing from laminar to transitional and turbulent. For this purpose, three types of computational fluid dynamics thermal boundary conditions were used. These are constant surface temperature (type I), heat flux (type II) and convection heat transfer coefficient (type III). Canlı et al. [24] aimed to determine the aerodynamic resistances of two agricultural tractors. For this purpose, computational fluid dynamics modeling was performed at nine different speeds.

In this study, the aim is to investigate the natural circulation inclined pipe-tank system numerically. For this purpose, the numerical analysis of the experiments conducted in the laboratory with Ansys 19 (Fluent) was performed and the velocity vectors and temperature contours were obtained. According to these results, the existence of natural circulation in the system, its continuity, and the locations where it is more effective in the system were commented on. Considering the difficulty of experimental study in terms of application and cost and the importance of developing more practical methods for such complex problems, we believe that this study will contribute to literature. In addition, in this study, the numerical study modeling steps revealed are clearly stated. These details are not included in the studies on similar subjects in the literature on the modeling of natural circulation. Therefore, we think that this study will be extremely useful for researchers who will work on the modeling of natural circulation.

MATERIALS AND METHODS

Within the scope of this study, numerical modeling studies of the experiments conducted at 100, 200, 400, 600 and 800 W thermal power were conducted in two and three dimensions using the CFD method.

Experimental Setup in a Laboratory Environment

The experimental setup set up in the laboratory environment is given in Figure 1. In this setup, the Mobilterm-605 heat transfer oil is heated by giving the desired electrical power to the rod resistances on the surface of the inclined galvanized pipe with a boxed variac. The heat transfer oil heated in the inclined pipe is directed to the tank consisting of 2 mm galvanized sheet metal at a lower temperature connected to the inclined pipe by natural circulation. In order to minimize heat losses during the experiment, the tank is insulated with 10 cm thick glass wool, and the inclined pipe is insulated with 20 cm thick glass wool. If desired, the heat transfer oil heated in the tank can be cooled by activating the cooling system and circulating cooling water through the copper serpentine placed inside the tank with a centrifugal pump. However, only the heating system was operated in this study [25].

The measurement system of the experimental setup; It consists of a Keithley 2700 data logger with $\pm 0.15\%$ uncertainty, a computer and a 5 kVA single-phase boxed variac with a digital sensitive multimeter display where current and voltage values can be read. Glass fiber insulated K-type thermocouple wire was preferred as a thermocouple. Kline and McClintock methods were used to

determine the uncertainty of the thermal power measured in the experiments [26]. As a result, it was found that the uncertainty was below 3%.

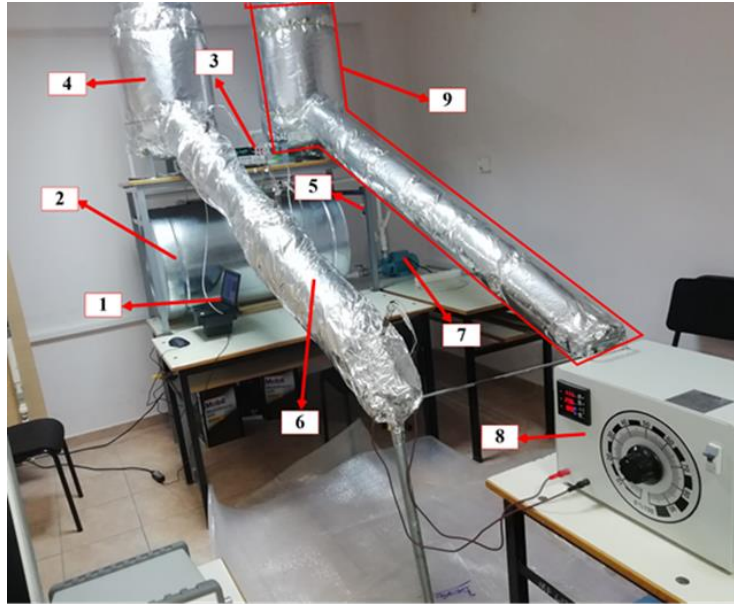


Figure 1

The experimental setup established in the laboratory environment is (1) PC, (2) Cooling Water Tank, (3) Data Logger, (4) Oil Tank, (5) Flowmeter, (6) Inclined Pipe, (7) Centrifugal Pump, (8) Boxed Variac and (9) Unused Part [18, 20].

Methodology of CFD

CFD refers to an iterative process that is expensive in terms of memory and computation for the computer, which develops in the form of numerical solutions of the equilibrium equations (continuity, Navier-Stokes equations, etc.) in a flow field. In order to perform a numerical analysis, the problem is first determined, expressed mathematically, and then this mathematical expression is concretized and calculated with the CFD software. The results obtained by the expert who analyzes the problem are visualized and interpreted with colored shape themes [27].

Two-Dimensional Numerical Study

For the experiments conducted at 100, 200, 400, 600 and 800 W thermal power, numerical modeling studies were conducted separately in two dimensions. By converting the thermal power values to heat flux and subtracting the lost energy, net heat fluxes of 287, 573, 1147, 1720 and 2294 W/m² were defined on the inclined pipe surface, respectively. The heat flux on the tank surface was entered as zero. The properties of Mobiltherm605 heat transfer oil were entered. A time-dependent (transient) solution was made. Gravitational acceleration in the Y axis was entered as -9.81 m/s². The flow was assumed to be laminar. The speed was accepted as 0.001 m/s as a reference. Coupled algorithm was selected as the solution method and the spatial discretization part was selected as completely second order. As the numerical modeling equivalent of the experiments that lasted a total of 6 hours in the laboratory and temperature measurements were made at 5-minute periods; The number of time steps was entered as 21600 and the time step as one second. In numerical modeling, All Triangles Method was used to create a solution mesh-1 consisting of 11786 nodes and 22444 elements (Figure 2.a). In order to make the analysis result independent of the solution mesh, the value in the Refinement command was increased from 1 to 2 and the solution mesh was made denser (solution mesh-2 (Figure 2.b); number of nodes: 25879, number of elements: 50067) and the results were compared. Since there was no significant difference in the results, the velocity vectors obtained as a result of the numerical analysis performed

with the solution mesh-2 were obtained.

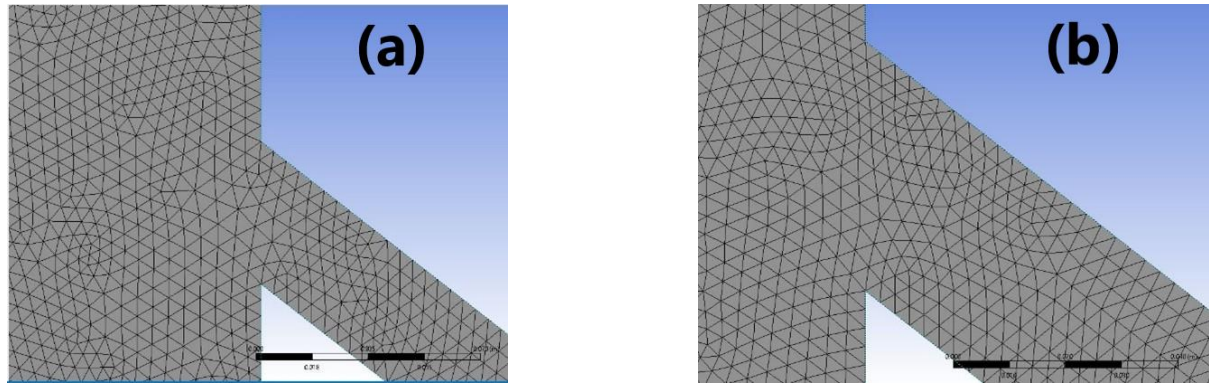


Figure 2
Two-Dimensional Numerical Study Mesh; (a) Mesh-1, (b) Mesh-2.

Three-Dimensional Numerical Study

In two-dimensional studies, three-dimensional studies were needed due to divergences in energy calculations. Similar to two-dimensional numerical studies; numerical modeling studies for experiments performed at 100, 200, 400, 600 and 800 W thermal power were performed separately in three dimensions. By converting thermal power values to heat flux and subtracting the lost energy, net heat fluxes of 287, 573, 1147, 1720 and 2294 W/m² were defined on the inclined pipe surface, respectively. Heat flux on the tank surface was entered as zero. Properties of Mobiltherm605 heat transfer oil were entered. Gravitational acceleration on the Y axis was entered as -9.81 m/s². Flow was assumed to be laminar. Velocity was accepted as 0.001 m/s as reference. The coupled algorithm was selected as the solution method and spatial discretization part was selected as completely second-order. The convergence criterion was entered as 0.001. Unlike the two-dimensional numerical study, a stable solution was made. Because the transient solution requires a lot of time and powerful computer hardware. Energy calculation was activated. The time step number was entered as 10000 and the time step was entered as one second. In addition to the All-Triangles Method in numerical modeling, the inflation command was used to make the solution mesh tighter near the surface. The solution mesh consisting of 148235 nodes and 318260 elements was created as shown in Figure 3.

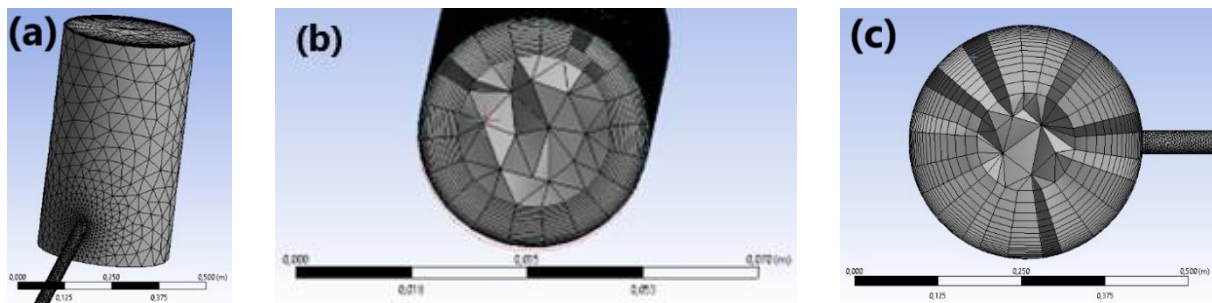


Figure 3
Three-Dimensional Numerical Study Mesh; (a) Tank-Pipe, (b) Pipe, (c) Tank.

RESULTS AND DISCUSSION

Two-Dimensional Numerical Modeling Study Results

When Figure 4 is examined, it is seen that the velocity vectors are directed upwards at the tank inlet and show a movement tendency similar to Benard cells. The highest flow rates are 1.61; 2.79; 4.04;

5.17 and 5.64 mm/s for 100, 200, 400, 600 and 800 W, respectively.

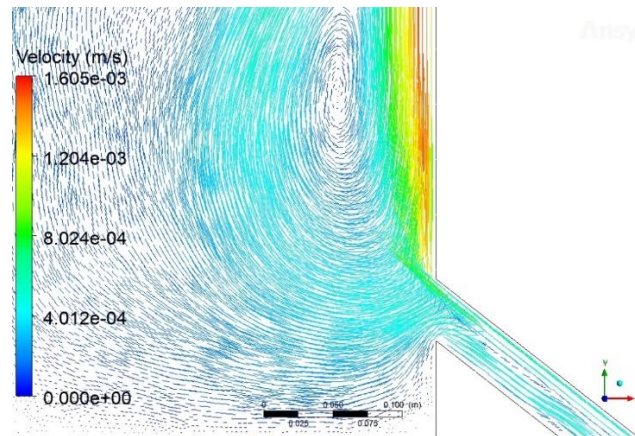


Figure 4
Velocity Vectors for 100 W thermal power.

Three-Dimensional Numerical Modeling Study Results

When Figure 5 is examined, it is seen that the velocity vectors are directed upwards at the tank inlet and show a movement tendency similar to Benard cells. The highest flow velocities are 0.186; 0.235; 0.241; 0.264 and 0.212 m/s for 100, 200, 400, 600 and 800 W, respectively.

When Figure 6 is examined; the streamlines of the numerical analysis performed at 800 W thermal power are seen. It is understood that these streamlines are continuous in the pipe-tank system. In addition, while the natural flow follows a laminar profile in the lower and upper sections of the pipe, it follows a spiral profile in the middle section.

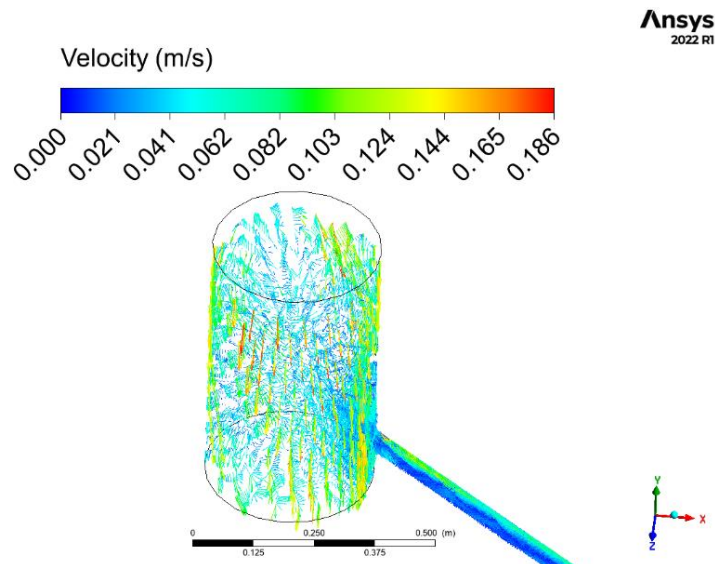


Figure 5
Velocity Vectors for 100 W thermal power.

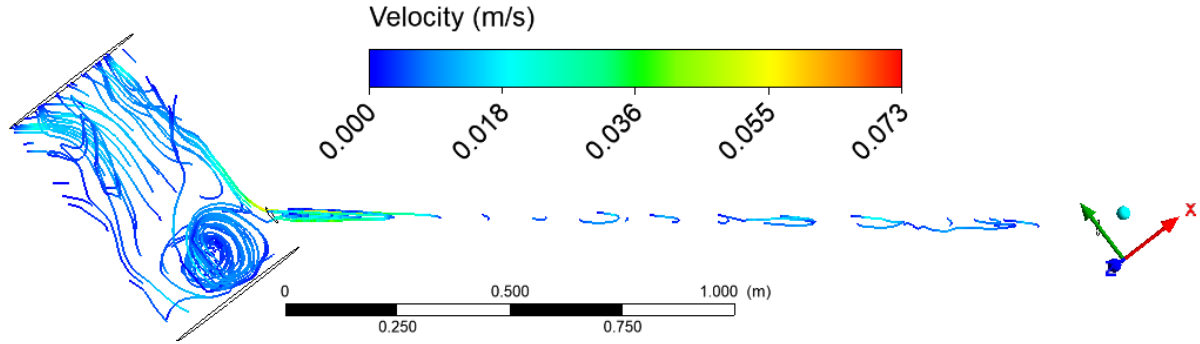


Figure 6
Streamlines for 800W thermal power.

The temperature distributions at the inlet, top, middle, and bottom sections of the inclined pipe tank are given in the range of Figures 7-10 for 200, 400, 600 and 800 W, respectively. Since there was a temperature divergence in the numerical analysis at 100 W, it was not included in the results. The temperature distribution in the tank appeared completely blue in the numerical analyses performed at all thermal powers and was 377, 407, 396 and 425 K for 200, 400, 600 and 800 W, respectively. At the end of the 6-hour experimental study, temperatures of 338, 372, 400 and 429 K were obtained in the tank, respectively. This shows that at the end of the experiment, the thermal regime was provided in the 600 and 800 W experiments performed at high thermal power, while more time was needed for the thermal regime in the 200 and 400 W experiments performed at low thermal power.

As we move from the tank inlet to the bottom of the inclined pipe towards the closed end, the color of the temperature contour turns almost into a single color. In this case, it can be said that natural circulation is not remarkably effective in this section and the temperature difference in the pipe section is small.

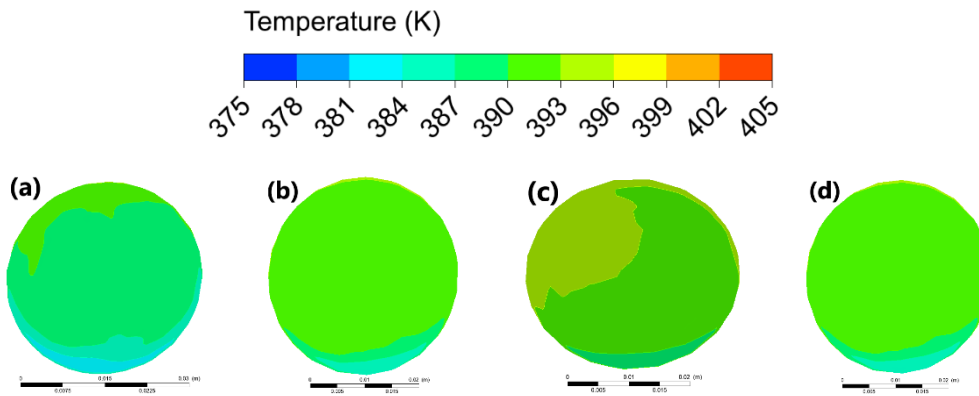


Figure 7
Temperature distribution in inclined pipe section (200 W) (a) Tank Inlet, (b) Top, (c) Middle and (d) Bottom.

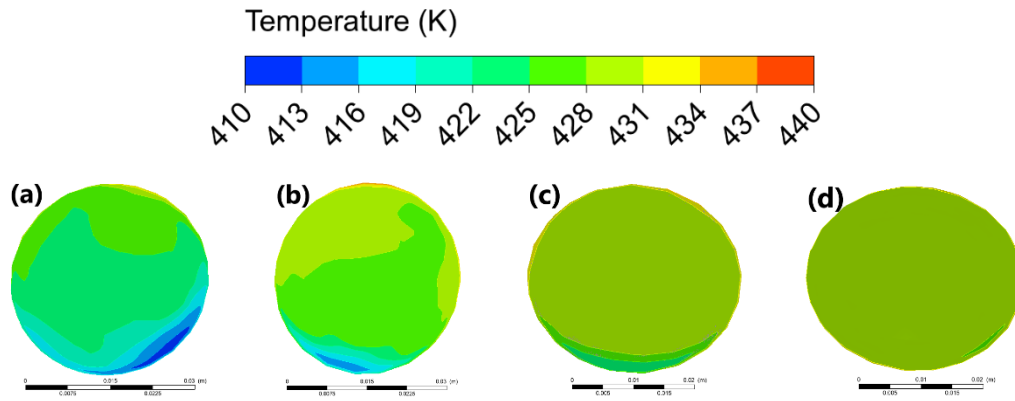


Figure 8
Te Temperature distribution in inclined pipe section (400 W) (a) Tank Inlet, (b) Top, (c) Middle and (d) Bottom.

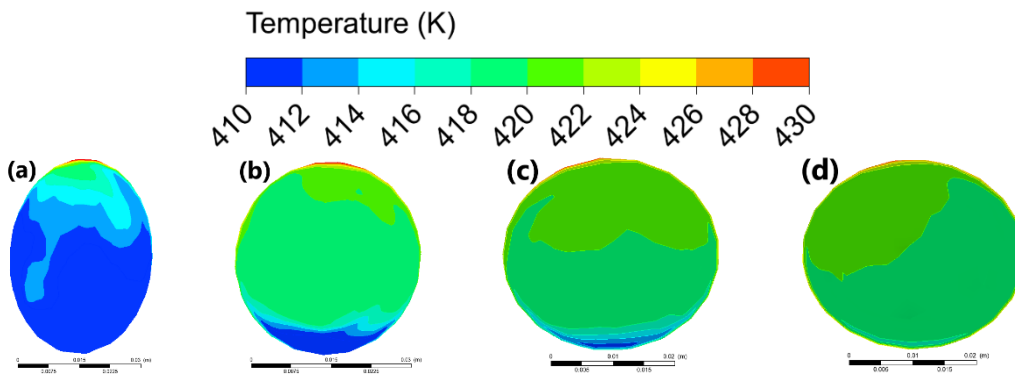


Figure 9
Temperature distribution in inclined pipe section (600 W) (a) Tank Inlet, (b) Top, (c) Middle and (d) Bottom.

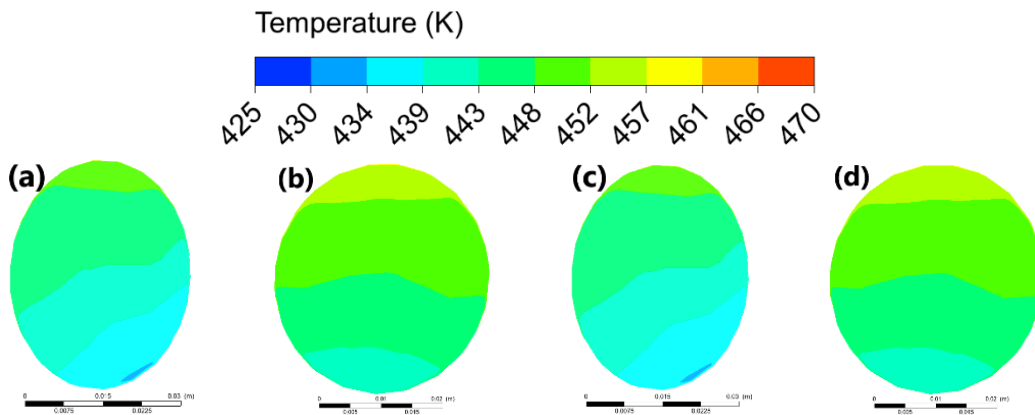


Figure 10
Temperature distribution in inclined pipe section (800 W) (a) Tank Inlet, (b) Top, (c) Middle and (d) Bottom.

CONCLUSIONS

When the velocity vectors obtained as a result of all numerical analysis are examined, it is seen that the fluid heated and reaching high temperature in the single-ended inclined pipe rises to the upper part of the pipe and moves towards the tank. Similarly, the fluid with lower temperature moves from the tank to the lower part of the inclined pipe. It is seen that the fluid moves like Benard cells in the tank. All these impressions are formed by the natural movement of the fluid due to the temperature difference in the system. It can be understood from the integrity of the streamlines obtained as a result of the numerical analysis that this movement of the fluid is also continuous in the single-ended inclined pipe-tank system.

When the temperature contours obtained as a result of the numerical analysis are examined, the color of the temperature contour changes almost to a single color as we move from the tank inlet to the bottom of the inclined pipe towards the closed end. In this case, it can be said that natural circulation is not remarkably effective in this section and the temperature difference in the pipe section is small.

It would be appropriate to increase scientific studies on natural flow in single-ended inclined pipes, which is the most common example of a water heating system. Especially when considered in terms of cost, it becomes clear that experimental work is very costly and instead it is important to turn to theoretical models and numerical analyses. There are not enough studies in the literature on modeling natural flow in single-ended inclined pipe-tank systems and similar systems. In this study, the parameters taken during modeling and the operations performed are given in detail. Thus, we believe that this study will be an example for future studies on similar subjects and will make a significant contribution to literature.

Ethical Statement

This study was produced from the doctoral thesis titled “Experimental and Numerical Investigation of Natural Circulation Solar Energy System Including Parabolic Trough Solar Collector” submitted on 12/30/2022 under the supervision of Assoc. Prof. Gökhan ARSLAN.

Author Contributions

Research Design (CRediT 1) : B.B. (%50) – G.A. (%50)

Data Collection (CRediT 2) : B.B. (%100)

Research - Data Analysis – Validation (CRediT 3-4-6-11) : B.B. (%70) – G.A. (%30)

Writing the Article (CRediT 12-13) : B.B. (%100)

Revision and Improvement of the Text (CRediT 14) : B.B. (%70) – G.A. (%30)

Financing

This study was supported by Mersin University Scientific Research Unit with project number 2018-2-TP3-2936.

Conflict of Interest

The authors have no conflicts of interest to disclose for this study.

Sustainable Development Goals (SDG)

Sustainable Development Goals: 7 Affordable and Clean Energy, 8 Decent work and economic growth, and 11 Sustainable Cities and Communities.

REFERENCES

- [1] G. Arslan, B. Bayhan, K. Yaman, Mersin/Türkiye için ölçülen global güneş ışınımının yapay sinir ağları ile tahmin edilmesi ve yaygın ışınlam modelleri ile karşılaştırılması, *Gazi University Journal of Science Part C: Design and Technology*. 7(1) (2019), 80-96. doi: 10.29109/gujsc.419473
- [2] B. Bayhan, G. Arslan, Applicability of solar and wind energy technologies for a non-residential Building, *Turkish Journal of Engineering*. 2 (1) (2018), 27-34. doi: 10.31127/tuje.341462
- [3] G. Arslan, B. Bayhan, Solar energy potential in Mersin and a simple model to predict daily solar radiation, *Mugla Journal of Science and Technology*. Special Issue (2016), 1-4. <https://dergipark.org.tr/en/pub/muglajsci/issue/25591/269956>
- [4] M. Hacıbeyoglu, M. Çelik, Ö. E. Çiçek, K en yakın komşu algoritması ile binalarda enerji verimliliği tahmini, *Necmettin Erbakan University Journal of Science and Engineering*. 5(2) (2023), 65-74. doi: 10.47112/neufmbd.2023.10
- [5] B. Akgayev, S. Akbayrak, M. Yılmaz, M. S. Büker, V. Unsur, Assessing the feasibility of photovoltaic systems in Türkiye: Technical and economic analysis of on-grid, off-grid, and utility-scale pv installations, *Necmettin Erbakan University Journal of Science and Engineering*. 6(1) (2024), 69-92. doi: 10.47112/neufmbd.2024.33
- [6] A. O. Özkan, H. B. Demir, Fotovoltaik panellerde sıcaklık ve zenit açısının panel güç üretimine etkisi, *Necmettin Erbakan University Journal of Science and Engineering*. 1(1) (2019), 1-9. <https://dergipark.org.tr/en/download/article-file/698145>
- [7] M. İ. Özgün, A. B. Batibay, B. Ünal, Y. R. Eker, A. Terlemez, Investigation of the use of TiO₂ obtained from endodontic NiTi files in dye-sensitized solar cells, *Necmettin Erbakan University Journal of Science and Engineering*. 5(1) (2023) 1-8. doi: 10.47112/neufmbd.2023.4
- [8] S. Ata, M. E. Boyacıoğlu, R. Şahin, A. Kahraman, ORÇ ile düşük sıcaklıklı ısı kaynaklarından elektrik üretilmesinde ıslak ve yeni nesil akışkanların çevresel ve termodinamik performanslarının karşılaştırılması, *Necmettin Erbakan University Journal of Science and Engineering*. 2(1) (2021), 1-13. doi: 10.47112/neufmbd.2021.6
- [9] H. D. Arslan, S. M. A. Bilgili, S. Doğan, Farklı ilkim bölgelerinde TOKİ tip konutlarının doğal havalandırma analizi, *Necmettin Erbakan University Journal of Science and Engineering*. 6(1) (2024), 21-39. doi: 10.47112/neufmbd.2024.30
- [10] P. M. Malkin, Design of Thermosyphon Solar Domestic Hot Water Systems, M. S. Thesis, *University of Wisconsin*, Madison, Wis., USA, 1985.
- [11] W. P. Akanmu, P. A. Bajere, Investigation of temperature and flow distribution in a serially connected thermosyphon solar water heating collector system, *Journal of Energy Technologies and Policy*. 5(2) (2015), 56-68. <https://core.ac.uk/download/pdf/234668006.pdf>
- [12] R. B. Bejjam, K. K. Kiran, Numerical study on heat transfer characteristics of nanofluid based natural circulation loop, *Thermal Science*. 22(2) (2018), 885-897. doi: 10.2298/TSCI160826087B
- [13] J. Li, X. Li, Y. Wang, J. Tu, A theoretical model of natural circulation flow and heat transfer within horizontal evacuated tube considering the secondary flow, *Renewable Energy*. 147(1) (2020), 630-638. doi: 10.1016/j.renene.2019.08.135
- [14] K. A. Pleshanov, E. G. Khlyst, M. N. Zaichenko, K. V. Sterkhov, Design of a natural circulation circuit for 85 MW steam boiler, *Thermal Science*. 21(3) (2017), 1503-1513. doi: 10.2298/TSCI161005320P
- [15] A. Riahi, H. Taherian, Experimental investigation on the performance of thermosyphon solar water heater in the south caspian sea, *Thermal Science*. 15(2) (2011), 447-456. doi: 10.2298/TSCI1102447R
- [16] M. Misale, J. A. Bocanegra, A. Marchitto, Thermo-hydraulic performance of connected single-phase natural circulation loops characterized by two different inner diameters, *International*

- Communications in Heat and Mass Transfer*. 125 (2021), 105309. doi: 10.1016/j.icheatmasstransfer.2021.105309
- [17] J. A. Bocanegra, A. Marchitto, M. Misale, Thermal performance investigation of a mini natural circulation loop for solar pv panel or electronic cooling simulated by lattice boltzmann method, *International Journal of Energy Production and Management*. 7(1) (2022), 1-12. <https://www.witpress.com/Secure/ejournals/papers/EQ070101f.pdf>
- [18] B. Bayhan, G. Arslan, Theoretical model of natural circulation flow and heat transfer within one-ended inclined pipe, *Thermal Science*. 26(6B) (2022), 5187-5198. doi: 10.2298/TSCI220402100B
- [19] B. Bayhan, G. Arslan, Experimental investigation of natural circulating solar energy system including a parabolic trough solar collector, *Journal of Solar Energy Engineering*. 147(2) (2025). doi: 10.1115/1.4066301
- [20] M. W. K. Jaber, et al., Transient evolution of thermal stratification and passive flow guidance inside a heat exchanger immersed thermal energy storage tank, *Journal of Energy Storage*. 88 (2024), 111472. doi: 10.1016/j.est.2024.111472
- [21] E. Canlı, A. Ates, Ş. Bilir, Developing turbulent flow in pipes and analysis of entrance region., *Academic Platform-Journal of Engineering and Science*. 9(2) (2021), 332-353. doi: 10.21541/apjes.818717
- [22] E. Canli, A. Ates, S. Bilir, Numerical Scheme for Dimensionless Natural Convection Analysis of Vertical Pipe, içinde: *Proceeding Book, 23rd Congress on Thermal Science and Technology with International Participation*. 2021, 1283-1293.
- [23] E. Canli, A. H. Altun, A. Ates, Hydrodynamic and Thermal Simultaneous Development in Pipes for All the Three Thermal Boundary Condition Types Using CFD, içinde: *Proceeding Book, 4th International Conference on Life and Engineering Sciences*, İstanbul, Turkey, 2021, 185-205.
- [24] E. Canli, H. Kucuksariyildiz, K. Carman. Impact assessment of new generation high-speed agricultural tractor aerodynamics on transportation fuel consumption and related phenomena, *Environmental Science and Pollution Research*. 30(3) (2023), 6658-6680. doi: 10.1007/s11356-022-22642-4
- [25] B. Bayhan, Parabolik Oluklu Güneş Kolektörü İçeren Doğal Dolaşımli Güneş Enerji Sisteminin Deneysel ve Sayısal İncelemesi, Doktora Tezi, *Mersin Üniversitesi Fen Bilimleri Enstitüsü, Makine Mühendisliği Anabilim Dalı*, Mersin, 2022.
- [26] S. J. Kline, F. A. McClintock, Describing uncertainties in single sample experiments, *Mechanical Engineering*. 75(1) (1953), 3-8.
- [27] N. Tokgoz, Ö. Süfer, Hesaplamalı akışkanlar dinamiğine genel bir bakış, *Osmaniye Korkut Ata Üniversitesi Fen Bilimleri Enstitüsü Dergisi*. 6(3) (2023), 2392-2408. doi: 10.47495/okufbed.1191498