SolarPACES 2023, 29th International Conference on Concentrating Solar Power, Thermal, and Chemical Energy Systems

Thermal Energy Storage Materials, Media, and Systems

https://doi.org/10.52825/solarpaces.v2i.948

Published: 24 Jul. 2024

Thermal Energy Storage With Horizontal Thermocline Tank for Sludge Drying Application

Luca Imponenti[1], Ryan Shininger[1], Hank Price[1], and Kyle Kattke[1]

[1]Solar Dynamics LLC, USA

Abstract. Solar thermal technologies can be an attractive option for the decarbonization of process heat applications; in particular, thermal energy storage is a high value proposition for continuous processes that may operate at a range of thermal conditions. A thermal-fluid model is developed to investigate the buoyant flow in a horizontal thermocline storage tank for a sludge-drying application. The process conditions and tank geometry considered are unique compared to the domestic water heating application typically considered in the literature and well suited for parabolic trough collectors. It was found that good separation of hot and cold fluids can be achieved with the long horizontal tank geometry depending on the inlet configuration. Uniform flow distribution along the length of the tank is required to establish a thermocline. Assuming desirable inlet conditions can be achieved, the tank outlet temperature predicted by the thermal-fluid model shows good agreement with the system-level performance model.

Keywords: Process Heat, Thermal Energy Storage, Thermocline

1. Introduction

In recent years there has been increased interest from several industry sectors to decarbonize their processes. Solar thermal can play an important role in decarbonizing process heat applications since energy is collected and used as heat, thus avoiding inefficient conversion processes; this application is referred to as Solar Industrial Process Heat (SIPH). Thermal energy storage (TES) will play an important role in SIPH since these processes often run continuously. Sludge drying is a potential SIPH application with a significant market potential: natural gas powered air dryers operating around 150°C are typically used to dry sludge as a pre-cursor for fertilizer production.

A solar thermal system with direct TES using pressurized water as the working fluid and parabolic trough collectors is being designed to provide thermal energy for a sludge-drying application. Options for TES at the process temperatures are abundant, including: two tank storage with pressurized water, thermocline tank with pressurized water, and various solid media. This work investigates a horizontal thermocline tank for direct TES of pressurized water. While this geometry is not ideal for thermocline performance it provides many benefits for ease of construction and total cost of storage; thus, it is important to model the thermocline performance under various conditions to understand if this storage system is suitable from a technical point of view.

2. Model description

A CFD model was developed within the OpenFOAM library [1] to simulate the horizontal thermocline tank storage system. The model is based on the buoyantPimpleFoam solver, which is a transient solver capable of resolving buoyant and turbulent flow. The PIMPLE algorithm is used for pressure-velocity coupling. The thermophysical properties of water are taken from [2] and are assumed to vary only with temperature. Although the pressure-dependence of the thermophysical properties is negligible (<0.1% change in volume from atmospheric to design pressure), it is important to prevent any boiling for the model formulation to be accurate. One of the main objectives of the CFD analysis is to verify that the system-level performance model (PM) does a reasonable job of thermally characterizing the thermocline tank for plant design.

2.1 Process Conditions

The solar thermal system feeds a water-air heat exchanger which provides hot air to the dryer. The dryer can operate with air inlet temperatures between 120 and 160°C. During daytime operation a portion of the flow from the solar field is directed to the dryer, bypassing the storage system. The parabolic trough solar field can easily provide higher temperature heat than the required 160°C without significant efficiency penalties; thus, the storage system is designed for a hot temperature of 200°C to reduce the cost of storage and provide some allowance for mixing in the demo system. The flow from the solar field which bypasses the storage system is attemperated as necessary before flowing to the dryer system. The boiling point of water at the design pressure is 205°C. Compared to the well-studied conditions for domestic water heating application [3]–[5] this process operates at higher temperature and pressure. The density difference between the hot and cold fluids at the design point is significantly larger compared to domestic water heating, resulting in stronger buoyant forces; however, the temperature difference and volumetric thermal expansion coefficient (β) are also larger promoting heat transfer between the hot and cold regions. The differences in water properties in the two applications are highlighted in Table 1.

Table 1. *Temperature range and water properties relevant for buoyant flow and natural convection heat transfer.*

Application	Temp. Range [°C]	Density Change [kg m^{-3}]	Avg. β [K^{-1}]
Domestic Water Heating	10 – 90	34.5 (3.45%)	4.37e – 4
Sludge-drying	112 – 200	85.3 (8.98%)	1.07e – 3

The dryer system can operate over a range of inlet conditions so the temperature delivered from the storage system also has some allowance for mixing; the discharging mass flow rate of the tank is therefore a function of the outlet temperature, and the storage system can still deliver usable thermal energy when the outlet temperature begins to decay. This feature of the storage system results in some flexibility in the thermocline performance compared to power generation applications. A system level PM was developed in TRNSYS and uses the built-in thermocline tank module for a reduced order representation of the storage system [6]. The PM is not discussed in detail in this manuscript, but it is important to resolve the variable process conditions described above.

2.1 Horizontal Tank Geometry

There are several theoretical studies considering horizontal thermocline tanks for domestic water heating [3], [5], but there are few experimental studies or test data available. In comparison, there is significant data available for vertical tank thermocline systems [7]. This is likely due to the fact that the vertical geometry has a few geometrical features that promote thermocline formation, i.e., larger distance between hot and cold inlets are possible, the area for heat conduction is constant and is smaller for a similar storage volume. However, the horizontal tank geometry has the potential for cheaper cost of storage and ultimately a lower levelized cost of heat (LCOH) so there is motivation to evaluate if the technical performance is sufficient. In this application, to achieve 8 hours of storage a 90,000 gallon horizontal tank with 11 ft diameter and an elliptical head was selected (340.69 m^3, 3.35 m diameter, approximate length 39 m). A previous study with a validated model achieved favorable thermocline performance in horizontal tanks with different inlet configurations and a similar diameter for a domestic water heating application at lower temperature and pressure [4]. Although the tank length and process conditions differ significantly, the referenced study was used to verify the model developed in this study due to the similarities in diameter and the lack of other CFD studies considering horizontal tank thermoclines in the literature.

2.1.1 Reduced Geometry Model

Due to the large volume of fluid in the storage system and relatively small flow features to be resolved at the tank inlets the model is computationally intensive. To simulate different geometries in a reasonable timeframe an additional model is developed with a significantly reduced volume; this is referred to as the tank slice model as it is effectively a small segment of the tank volume. A small 3D section of the tank is selected instead of a 2D model to better preserve the flow features at the inlets (which were found to have a significant impact on the thermocline performance). Each tank slice model is named similarly to the matching full-size tank geometry, i.e., tank slice 2280 orifices is a small slice of a full size geometry with 2280 orifices, thus the tank slice model will have significantly fewer orifices.

2.1.2 Flow Distribution

The horizontal tank geometry being investigated is not ideal for thermocline performance due to the relatively short distance in the vertical direction between the inlets, and the length in the axial direction. For ideal thermocline performance, any fluid motion perpendicular to the direction of gravity is undesirable [8]. For the geometry considered, axial flow distribution prior to introducing water into the main tank volume can greatly improve the thermocline performance. The tank geometry and one of the more promising axial distribution systems is illustrated in Figure 1. The inlet distribution system in Figure 1, also referred to as the sparger box, consists of an inner pipe designed to spread flow axially, and a surrounding box designed to reduce the inlet velocity before introduction to the main tank volume. The sparger box concept is inspired by similar inlet configurations for vertical thermocline tanks [5].

Simulation of the full-size tank and sparger box system is a very computationally intensive task. To help with computer resources separate models were developed for each component: (i) the sparger box model resolves the turbulent flow in the main inlet pipe that dissipates as it moves through the sparger box, (ii) the tank model solves the buoyant-driven flow within the main tank volume with a velocity boundary condition at the sparger box inlet/outlet. This document will focus on the tank model and thermocline performance, with an assumed velocity profile at the tank inlet (sparger box outlet). Although analysis of the sparger box is currently underway, results to date suggest it is reasonable to assume a uniform tank inlet velocity distribution.

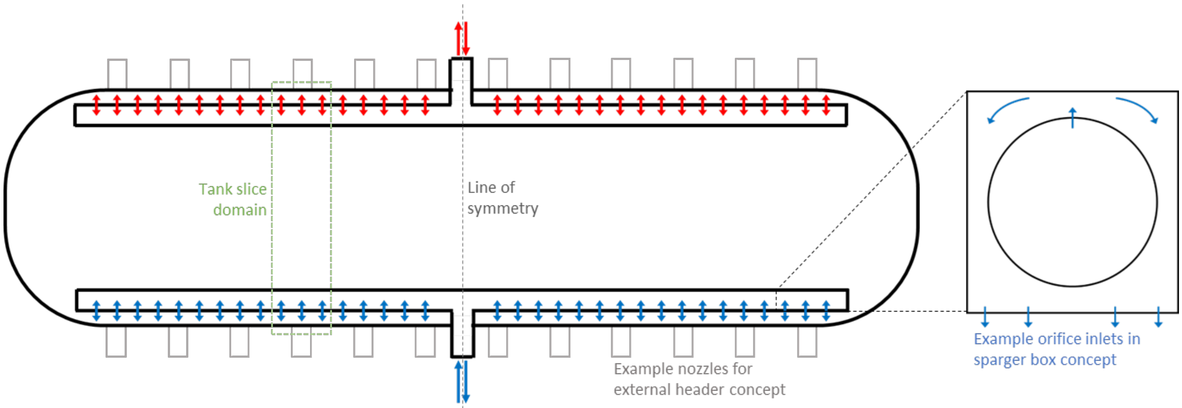

Figure 1. *Modelled tank geometry showing the sparger box concept for inlet flow distribution, the external header concept where the nozzles flow directly into the tank is also illustrated.*

2.2 Boundary Conditions

Two different types of simulations are considered: ideal simulations with constant charge/discharge conditions based on the design point, and PM simulations which use variable process conditions from the system-level model. The ideal simulations are used to compare the thermocline performance of different inlet geometries, while the PM simulations consider multiple days of real operational conditions to verify the reduced-order model of the TES system within the PM. All ideal simulations are initialized with constant temperature and no-flow conditions, these simulations consider low heat loss conditions such that while the PM simulations are initialized by solving two clear-sky days and the successive discharging process (see 2nd day of operation in Figure 5). The boundary conditions during operation for the two types of simulations are described in Table 2. At the inlet, a uniform temperature and a user-defined velocity profile are set (and the relevant turbulence parameters, if applicable); while at the outlet a uniform pressure is specified. The model is set up to be able to switch the pressure outlet and velocity inlet depending on the process; in addition, the inlets and outlets can be changed to walls to simulate periods where the tank is in stand-by. In this system there is a large range of charge and discharge conditions the TES system can be bypassed during day-time operation, and the flow from the storage tank is attemperated in some cases.

Table 2. *Boundary conditions for different types of simulations.*

Simulation Type	**Discharge Inlet Temp. [°C]**	**Charge Inlet Temp. [kg m^{-3}]**	**Discharge Mass Flow [kg s^{-1}]**	**Charge Mass Flow [kg s^{-1}]**	**External HTC [W m^{-2} K^{-1}]**
Ideal	112.0	200.0	21.6	12.4	0.1
Performance Model	96.2 – 112.1	166.6 – 200.2	3.8 – 21.6	0.1 – 12.4	5.0

3. Results & Discussion

3.1 Verification

The CFD model developed in OpenFOAM is compared to previous literature results for verification [4]. The comparison, shown in Figure 2, is considered a verification as opposed to a proper validation due to some key differences in the model, namely the model in [4] uses the Boussinesq approximation to resolve the buoyant flow within the tank, and all other thermophysical properties are assumed constant [4]. Since the goal of the model developed in this study is to look at conditions with a larger ΔT the variation in properties will be more significant and the Boussinesq approximation will not be applicable; thus, the water within the tank is modelled as an incompressible fluid where the density, specific heat, thermal conductivity, and viscosity vary as a function of temperature. The temperature dependent properties and removal of the Boussinesq approximation results in more mixing in the OpenFOAM model compared to the model in [4]. This effect can be seen in Figure 2 where the width of the thermocline is approximately 20 cm wider for this study; the thermocline is effectively a mixed zone of hot and cold fluids; thus a larger thermocline after the same volumetric input indicates more mixing in the tank. Although the error between the two simulations is significant (maximum error of nearly 48%), the progression of the thermocline is similar and indicates the model is functioning as intended. A stricter validation of the model is planned when more applicable data is available.

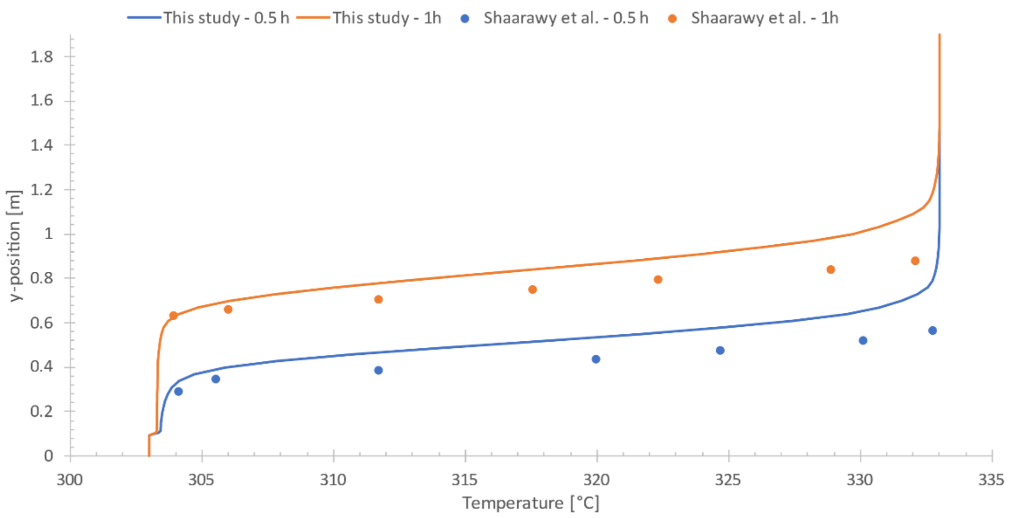

Figure 2. *Comparison of temperature profiles from CFD model developed in this work to validated literature model with similar geometry [5].*

3.1 Thermocline Performance

3.1.1 Ideal Conditions

An ideal discharge process starting from a fully charged tank is simulated as a benchmark case to compare different geometries. The outlet temperature throughout the discharge process is plotted for several different tank slice models (and the full-size model for the best performing geometry) in Figure 3; the time for a full-tank volume exchange is indicated by the light vertical line. All results in Figure 3 represent the same 90,000 gallon tank with different inlet configurations, the tank slice model is a more computationally efficient representation of the same geometry as discussed in Section 2.1.1. At the given ideal conditions, the inlet velocity is > 1 m/s causing significant turbulence and mixing in the storage tank. To reduce the inlet velocity results, two different concepts are simulated: the sparger box illustrated in Figure 1, and an external header concept that flows into the tank through a series of nozzles (effectively splitting the tank in the axial direction). The best performing geometry was the sparger box configuration with 2280 1” orifices at the interface to the main tank volume and round-trip efficiency of 95.8%; the tank slice and full-size models are compared for this geometry. The number of orifices was selected to have sufficient flow area for a Reynolds number below 2000.

The tank slice model captures the initial decay in the outlet temperature as the thermocline begins to move through the outlet nozzle. There is additional mixing in the full-size tank which causes a widening of the thermocline compared to the tank slice model (up to 16% error). As mentioned previously if all other conditions are held constant, the thermocline which is an indication of the level of mixing between hot and cold fluids. The additional mixing is attributed to geometry differences at the tank head and near the centre of the tank near the tee; however, this results in only a small reduction in the round-trip efficiency compared to the tank slice model, 95.1% compared to 95.8%. Based on this comparison, the tank slice model is considered an acceptably accurate representation for the comparative analysis in Figure 3(a) and will be used for further analysis of the thermocline behavior over multiple days of operation.

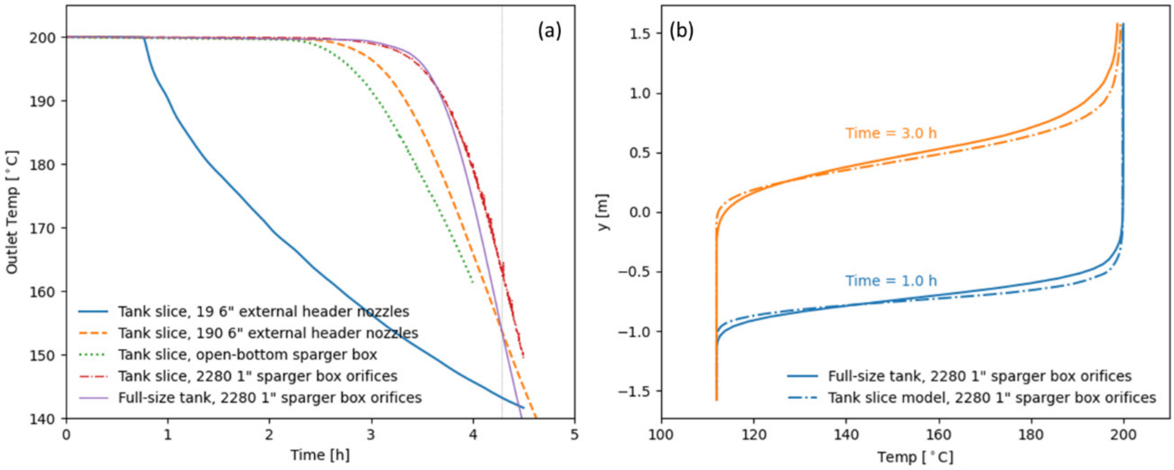

Figure 3. *Comparison of CFD model results for different tank and inlet geometries: (left) outlet temperature, and (right) vertical temperature profile near the center of geometry.*

After downselecting the sparger box geometry with 2280 1 inch orifices the simulation was continued for several charge and discharge cycles at ideal condition, results are plotted in Figure 4. There is a 30-minute standby period with no flow in or out of the tank in between each charge and discharge operation. The inlet conditions are switched while the mixed zone of fluid is still present in the tank to evaluate the stability of the thermocline through successive cycles. After the first cycle, (initialized at constant pressure and velocity conditions) the temperature profile in successive cycles is very similar indicating the thermocline is stable through the changes in inlet conditions similar to what could occur when responding to solar resource availability. This is considered an important result due to the variable cross-sectional area in the horizontal tank configuration, which results in the thermocline changing width as it moves through the tank; in Figure 4(b) it is evident that these variations in the thermocline properties as the mixed zone of fluid moves through the tank results in minimal additional mixing.

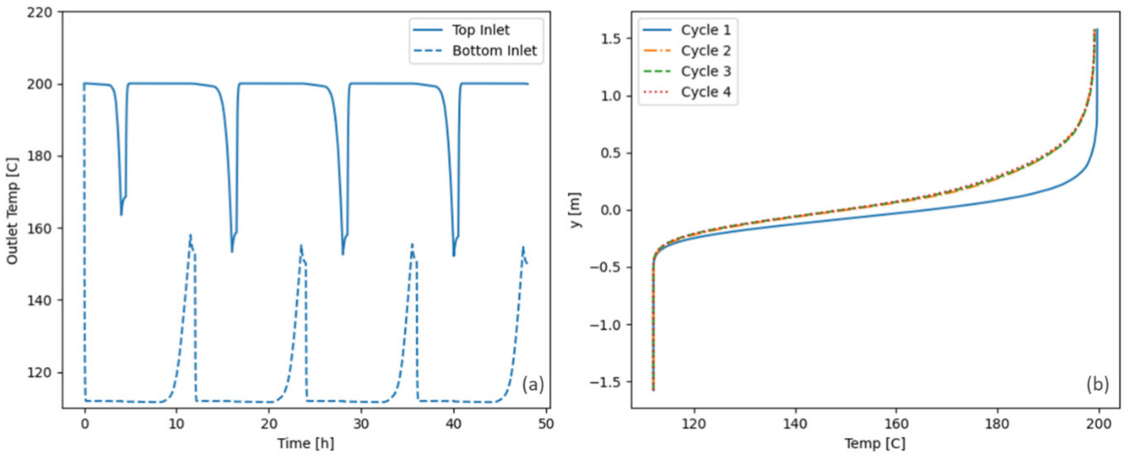

Figure 4. *CFD model results during four successive discharge – charge cycles, (a) inlet and outlet flow temperature, and (b) thermocline profile 2 hours into discharging for each cycle.*

3.1.1 Performance model comparison

Simulations with PM data are used to evaluate the thermocline stability under variable conditions, and to verify the 1D sub-model does an adequate job of representing the thermocline performance. During periods of favorable DNI conditions operating near the design point, the CFD model predicts less mixing than the 1D sub-model; however, during weather transients the temperature decay is also more significant in the CFD model. Overall, the PM captures the correct trends for the thermocline tank, assuming uniform axial distribution is achieved. The 1D sub-model contains no information regarding the specific inlet geometry, but the number of nodes and a mixing rate parameter can be adjusted to improve the agreement with the CFD model;at this point the number of nodes is the only parameter that was varied, while the impact of the mixing parameter is still to be investigated. Without the axial flow distribution provided by the sparger box, the CFD model performance will be significantly worse, and the storage tank will not be well represented in the system-level model; thus, it will be important to validate the storage system performance after construction to ensure the PM model is an accurate tool once operation commences.

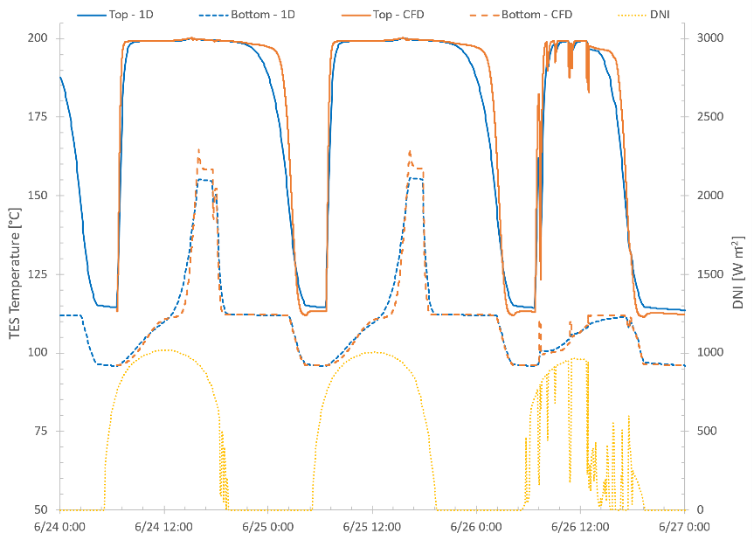

Figure 5. *Comparison of thermocline hot and cold inlet temperatures predicted with CFD model and 1D sub-model used in system-level performance model over 3 days of operation.*

4. Conclusion

A horizontal tank thermocline TES system with pressurized water was evaluated for a sludge-drying application. The process conditions for this application are considered favorable for thermocline storage with pressurized water due to the large density change of water over the temperature limits and flexibility of the dryer system which can operate across a range of inlet temperatures. The horizontal tank geometry was selected for its lower cost and ease of construction, despite some evident disadvantages for the thermocline, i.e. short distance in the direction of gravity and a large, variable heat conduction area.

CFD model simulations at ideal discharge conditions indicate that significant mixing can occur in the long horizontal tank. To maintain good separation of hot and cold fluids the flow distribution before entering the main tank volume is critical. A sparger box concept has been designed and is currently under evaluation to distribute flow along the length of the tank and reduce momentum at the inlets. If desirable inlet conditions can be maintained, the CFD model shows reasonable agreement with the system-level model.

Future work will be focused on the design of the sparger box for axial distribution of flow along the length of the tank. Once the inlet piping design is finalized additional verification of the full tank model compared to the tank slice model will be completed, including the effects of varying inlet velocity along the length. This work on the sparger box and achieving favorable inlet conditions is considered critical to the success of this concept in a commercial setting. In general, future work will be focused on verifying the thermocline stability through different weather and operational conditions before construction.

Data availability statement

Data from the CFD model developed in this study is not publicly available at this time.

Author contributions

Luca Imponenti – Conceptualization, investigation, software, writing (original draft); Ryan Shininger – Conceptualization, project administration, writing (review & editing); Hank Price – Conceptualization, software, writing (review & editing); Kyle Kattke – Conceptualization, software

Competing interests

The authors declare that they have no competing interests.

Funding

This material is based upon work supported by the U.S. Department of Energy, Office of Energy Efficiency and Renewable Energy, Research under Award Number DE-EE0009640.

References

[1] H. G. Weller et al., "A tensorial approach to computational continuum mechanics using object-oriented techniques," *Comput. Phys.*, vol. 12, no. 6, pp. 620–631, Nov. 1998, doi: 10.1063/1.168744.

[2] W. C. Reynolds, *Thermodynamic properties in SI: graphs, tables, and computational equations for forty substances*. Dept. of Mechanical Engineering, Stanford University, 1979.

[3] D. L. Savicki et al., "Three-dimensional analysis and investigation of the thermal and hydrodynamic behaviors of cylindrical storage tanks," *Renew. Energy*, vol. 36, no. 5, pp. 1364–1373, May 2011, doi: 10.1016/j.renene.2010.10.011.

[4] M. Shaarawy and M. Lightstone, "Numerical Analysis of Thermal Stratification in Large Horizontal Thermal Energy Storage Tanks," *J. Sol. Energy Eng. Trans. ASME*, vol. 138, no. 2, Apr. 2016, doi: 10.1115/1.4032451.

[5] P. Trinuruk et. al., "Comparative Study of Inlet Structure and Obstacle Plate Designs Affecting the Temperature Stratification Characteristics," *Energies*, vol. 15, no. 6, p. 2032, Mar. 2022, doi: 10.3390/en15062032.

[6] University of Wisconsin--Madison. Solar Energy Laboratory. (1975). TRNSYS, a transient simulation program. Madison, Wis. :The Laboratory.

[7] M. Pilotelli et al., "Performance analysis of a large TES system connected to a district heating network in Northern Italy," *Energy Rep.*, vol. 8, pp. 1092–1106, Nov. 2022, doi: 10.1016/j.egyr.2022.07.094.

[8] J. Rendall et al., "Nondimensional convection numbers modeling thermally stratified storage tanks: Richardson's number and hot-water tanks," *Renew. Sustain. Energy Rev.*, vol. 150, p. 111471, Oct. 2021, doi: 10.1016/j.rser.2021.111471.