CFD-Driven Development of a Laboratory-Scale Thermocline Thermal Energy Storage System

Summary

This work presents the CFD-based design of a compact high-temperature thermal energy storage system for small and medium-sized enterprises. The system is based on a packed-bed concept using ceramic honeycomb structures and aims to enable the decarbonization of industrial process heat. Due to scaling effects, maintaining thermocline stability and uniform flow distribution represents a key challenge. A two-stage CFD approach is applied, including geometry selection of storage structures and system-level design optimization of the laboratory demonstrator. The results highlight the strong influence of channel geometry and inlet design on thermo-fluid performance. The developed laboratory system provides the basis for future experimental validation and scaling of the miniTESS concept.

Keywords: Thermal Energy Storage, CFD, Thermocline, Power-to-Heat, Process Heat, Decarbonization, High-Temperature Storage

Field of application for the findings: Thermal Energy Storage

Addressed audience: Researchers and engineers working on TES and energy system optimization

1. Introduction

Industrial heat demand accounts for a substantial share of global final energy consumption and CO_2 emissions. Although considerable progress has been made in decarbonizing electricity generation, the provision of industrial process heat remains strongly dependent on fossil fuels. This challenge is particularly pronounced for small and medium-sized enterprises (SMEs), which often require reliable heat at elevated temperatures but typically lack access to economically viable decarbonization technologies. In sectors such as bakery production, process heat is commonly needed at temperature levels of around 300 °C, often during night-time or early morning hours when on-site photovoltaic generation is unavailable.

Power-to-Heat (PtH) systems combined with thermal energy storage (TES) offer a promising approach to decouple electricity supply and heat demand. However, the current state of the art in high-temperature TES is dominated by large-scale, application-specific systems. Existing approaches include solid storage media such as ceramic structures, granulates, bricks, steel, sand, and concrete-like materials, with air or liquids as heat transfer media. These systems typically address storage capacities well above 1 MWh and are tailored to large industrial users, limiting their suitability for SMEs due to cost, complexity, and lack of standardization.

The miniTESS concept addresses this gap by developing, constructing, and commissioning a compact, standardized high-temperature thermal energy storage system based on ceramic honeycomb structures with air as a heat transfer medium. Building on previous large-scale developments, the concept is transferred to a smaller capacity range suitable for SME applications, with the main objective of enabling the cost-effective decarbonization of industrial process heat.

A key technical challenge in this context is the preservation of thermocline behavior in a reduced system size. The thermocline, defined as the transition zone between hot and cold regions within the storage, is critical for maintaining a stable outlet temperature. Its degradation due to scaling effects, flow maldistribution, or thermal dispersion directly impacts system performance.

2. System Concept and Design Challenges

The miniTESS system is designed as a compact, modular high-temperature thermal energy storage unit for SME applications. It consists of a packed-bed storage filled with ceramic honeycomb structures, an electrical heating unit for charging, and a heat exchange system for discharging heat to the process. The system will be implemented in a containerized, standardized configuration to enable simple integration.

Compared to large-scale systems, the reduced dimensions introduce several design challenges. A key issue is the stability of the thermocline, which is essential for maintaining a constant outlet temperature. The shorter flow path in a small system increases the risk of thermocline spreading and reduced usable storage capacity.

In addition, uniform flow distribution across the storage is critical. Maldistribution at the inlet can lead to distorted thermoclines and reduced performance, while the limited space within the container constrains the design of flow-conditioning elements. Buoyancy effects further influence the flow and temperature field and may either stabilize or degrade thermal stratification.

To address these challenges, a laboratory-scale model is developed to evaluate and compare different storage stone geometries. A simulation-driven design approach is applied for the optimization of the lab-scale model. Computational Fluid Dynamics (CFD) is used to analyze flow distribution, heat transfer, and thermocline behavior, and to support the design of the laboratory-scale demonstrator. A schematic diagram of this demonstrator is shown in Fig. 1.

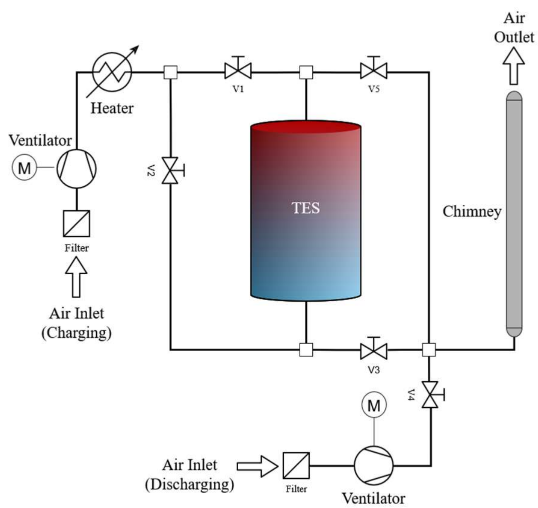


Fig. 1: Schematic diagram of the miniTESS test platform concept

3. CFD-Based Design of the Laboratory System

The design of the miniTESS laboratory demonstrator is primarily guided by Computational Fluid Dynamics (CFD) simulations, which serves as the basis for subsequent scaling and experimental validation. The CFD investigations are carried out in two consecutive steps. First, a comparative analysis of different ceramic storage geometries is performed to identify a suitable honeycomb structure for the application. Second, the overall storage system and inlet configuration are optimized with respect to flow distribution and thermal performance.

3.1. Geometry Selection of Ceramic Storage Structures

In an initial step, three commercially available channel geometries (square, hexagonal, and circular) as well as different sizes of the square-shaped channels are evaluated using transient 3D CFD simulations. To isolate the effect of geometry, all configurations are analyzed under identical boundary conditions, including constant hydraulic diameter, minimum wall thickness, and identical operating conditions.

The simulations assess both aerodynamic and thermal performance metrics, including pressure loss, stored thermal energy, and energy density. The results indicate distinct differences in thermo-fluid behavior. The square geometry exhibits the lowest pressure loss due to its larger open frontal area, while the hexagonal structure achieves the highest specific energy density and a more homogeneous temperature distribution. In contrast, the circular geometry shows increased thermal lag in peripheral regions, reducing the effectively usable storage volume.

Based on a combined evaluation of hydraulic and thermal performance, the optimized channel geometry is selected as the reference design for the demonstrator. Subsequent work focused on the parametric optimization of this geometry, including channel spacing, wall thickness, and open frontal area.

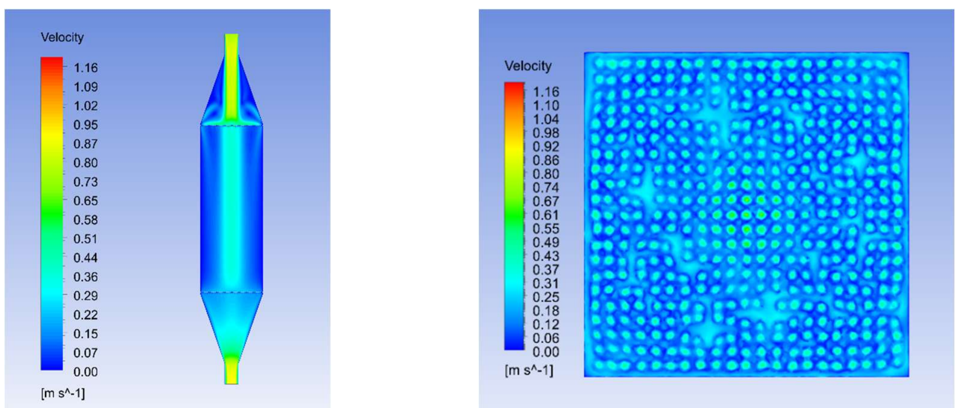
3.2. System-Level CFD Analysis and Design Optimization

In a second step, CFD simulations are extended to the full laboratory storage system in order to optimize the flow direction and overall design. Particular attention is given to achieving a uniform flow distribution across the storage cross-section, which is essential for stable thermocline formation.

A set of sensitivity analyses is conducted to evaluate the influence of key design parameters, including:

- Air mass flow rate
- Geometry of the inlet diffuser
- Design of the perforated plate upstream of the storage
- Entrance channel diameter

The simulation model of the storage system, implemented in ANSYS, is shown in Fig. 2 (a), while the perforated plate is presented in Fig. 2 (b). The figure illustrates the velocity contours of the airflow within the reference model. The results show that even small geometric variations in the inlet region can significantly affect flow uniformity and, consequently, the thermal performance of the storage. Non-uniform flow leads to localized temperature gradients and a distorted thermocline, reducing the effective storage capacity. The CFD analysis is therefore used to iteratively refine the inlet design and define a configuration that ensures a homogeneous velocity profile at the storage entrance.



(a) (b)

Fig. 2: Velocity contours of (a) the storage system and (b) the perforated plate using ANSYS.

The final design of the laboratory demonstrator is derived from these simulation results and provides a well-defined basis for subsequent experimental investigations.

4. Outlook and Next Steps

At the current stage, the laboratory demonstrator is under construction. The CFD-based design approach presented in this work provides the foundation for the experimental phase, in which the thermo-fluidic behavior of the storage system will be investigated under controlled conditions.

Future work will focus on the commissioning of the test rig and the execution of systematic measurement campaigns to characterize thermocline formation, pressure losses, and dynamic charging and discharging behavior. The experimental data will be used to validate and further refine the numerical models.

In subsequent project phases, the validated models will be extended to the design, optimization, construction and commissioning of the full miniTESS system, including its integration into an industrial application. This will enable the development of a standardized, scalable storage solution tailored to the requirements of small and medium-sized enterprises.