## **PROJECT REVIEWS**: **CFD Analysis of Pillow Plate Heat Exchanger**

# Sujay Pakhare1, Nikhil Mankar2, Shravan Pathak3, Bhushan Jadhav4

1,2,3,4Department of Mechanical Engineering, Shri Sant Gajanan Maharaj College of Engineering, Shegaon. India

**ABSTRACT**

This study looks at how pillow plate heat exchangers work by using computer simulations to understand both their heat transfer performance and structural strength. These exchangers are widely used in industries because they’re compact, durable, and good at transferring heat. We used CFD (Computational Fluid Dynamics) to study fluid flow and heat movement, and FEA (Finite Element Analysis) to check the unit's strength under pressure. To make sure our results are reliable, we compared them with existing experimental or theoretical data. We also explored how design changes—like plate thickness, channel spacing, and flow direction—affect performance. The aim is to find ways to boost heat transfer without weakening the structure, helping engineers design more efficient and long-lasting systems for real-world industrial use.

**Keywords**- Pillow Plate Heat Exchanger,Computational Fluid Dynamics (CFD) Heat Transfer Enhancement,

Pressure Drop Analysis, Thermohydraulic Performance, Turbulence Modeling

# **INTRODUCTION**

In thermal and industrial energy systems, the efficient operation of pillow plate heat exchangers is essential for achieving better performance and energy savings. These systems are widely used in industries where optimizing heat transfer and maintaining pressure stability can lead to significant improvements in overall efficiency. Even small enhancements in the design can make a big difference. The pillow-like shape of the plates isn’t just for looks—it plays a key role in how fluids move through the system. This unique geometry increases surface area, promotes turbulence, and improves heat exchange by allowing more interaction between the fluid and the plate surface.

The performance of pillow plate heat exchangers largely depends on the shape of their internal channels and how fluids flow through them. That’s why it’s important to study these details closely. In this research, we use pillow plate heat exchangers as a model for advanced CFD (Computational Fluid Dynamics) analysis. Our goal is to understand how different design features impact heat transfer and flow behavior. By improving these aspects, we aim to support the development of more efficient, reliable heat exchangers that can handle the demands of real-world industrial applications.

**LITERATURE REVIEW**

In 2011, Mitrovic and Maletic made a key contribution to the understanding of pillow plate heat exchangers (PPHEs) by exploring how fluids flow and transfer heat inside their uniquely inflated, wavy structures. Unlike traditional flat or tubular designs, PPHEs create complex flow patterns that can boost heat transfer but also increase pressure drop. Using Computational Fluid Dynamics (CFD), they visualized how fluid separates, swirls, and reattaches as it moves over the inflated surfaces and weld spots—insights that simple models couldn’t provide. Their work showed that design features like weld spacing and inflation height significantly impact both heat transfer and flow resistance. They also found that standard turbulence models often missed key flow behaviors, recommending more advanced approaches like Reynolds Stress Models for better accuracy. Their research became a foundation for future studies, influencing how engineers design and optimize PPHEs by highlighting the importance of geometry, local flow behavior, and advanced simulation techniques[1].

As industries pushed for more compact and efficient heat exchangers, pillow-plate heat exchangers (PPHEs) gained attention for their unique design and performance benefits. The inflated, pillow-like plates create a complex, three-dimensional flow that enhances heat transfer through increased turbulence. However, understanding and accurately predicting fluid and heat behavior in such intricate geometries was a major challenge. In 2014, Piper, Zibart, Tran, and Kenig made a significant leap forward by using advanced numerical methods to closely study turbulent single-phase flow and heat transfer within PPHEs, offering much deeper insights than earlier research had achieved[5].

*Fundamentals of Heat and Mass Transfer* by Incropera, DeWitt, Bergman, and Lavine (2007) is a cornerstone in thermal engineering education. Widely respected for its clarity and depth, this textbook has guided generations of students and professionals alike. It starts with the basics—conduction, convection, and radiation—building gradually from simple to more advanced concepts. What sets it apart is how it blends theory with real-world relevance, making complex ideas approachable. Its strong focus on convection, including boundary layers and key dimensionless numbers, is especially helpful for those working with heat exchangers or HVAC systems. The book also offers thorough coverage of radiation, often overlooked in other texts, making it a truly comprehensive resource[7].

this research paper, a comprehensive survey of computer vision and artificial intelligence applications in precision agriculture is presented, focusing on the integration of vision-based intelligent systems to enhance farming practices. It explores the complete digital life cycle of crops; from image acquisition, metrics, including vegetation indices, are discussed as essential tools for quantifying crop health and properties. The paper highlights advancements in imaging techniques, machine learning algorithms, and AI-driven decision-making for tasks such as pest detection, growth monitoring, and yield optimisation. It also addressed the challenges of implementing generalised, real-time computer vision models in agricultural environments, emphasising the potential for these technologies to improve productivity, sustainability, and food security. By offering a holistic view of the digital agriculture pipeline, this survey aims to bridge gaps in existing literature and inspire further research in the field[10].

# **METHODOLOGY**

**1. Understanding the Problem**

We started by identifying our goal — to study how heat and fluid flow inside a **pillow plate heat exchanger**. These exchangers have a special pillow-shaped design that allows efficient heat transfer, and we wanted to simulate this behavior using CFD tools.

**2. Creating the Geometry**

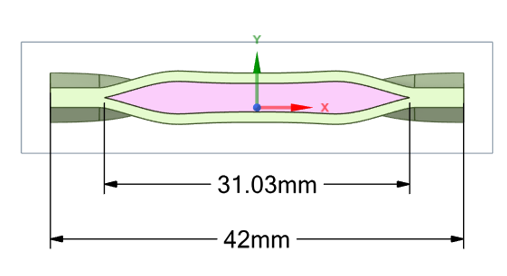
We created a 3D model of the pillow plate using ANSYS SpaceClaim. The model included:

* Two thin metal plates joined together with spot welds.
* Inflated pillow-like shapes formed between the plates.
* An internal flow path for water.

We made sure the geometry was simple but realistic, just enough to capture the fluid dynamics and heat transfer without making the simulation too heavy.

Diagram of a diagram of a metal structure

Description automatically generated

Fig; Pillow Plate Concept

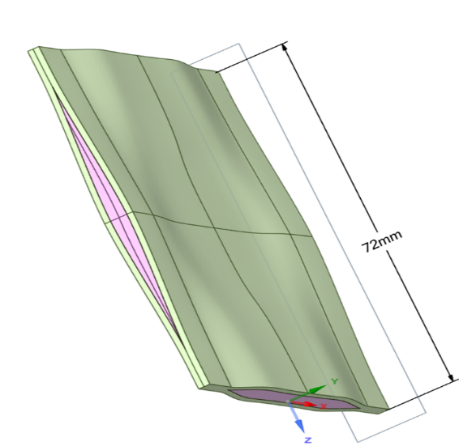


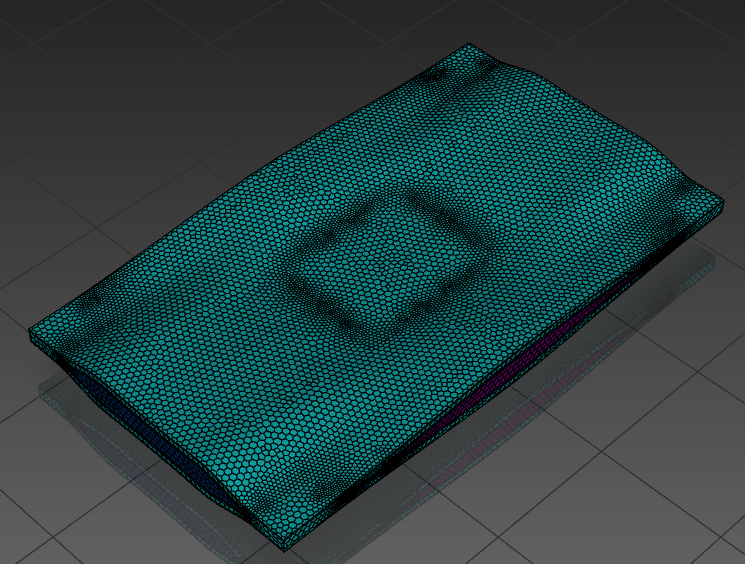
Fig: CAD model of Pillow Plate

**3. Meshing the Model**

Next, we moved to the meshing stage. This is where we break the 3D model into small cells that the software can calculate.

* We used a **finer mesh** near the wall surfaces to accurately capture boundary layer effects.
* **Inflation layers** were added at the walls to get better resolution of the thermal and velocity gradients.

We kept an eye on mesh quality to avoid errors and to ensure smooth simulation



**4. Setting Up the Physics (Boundary Conditions)**

In ANSYS Fluent, we defined how the system behaves:

* **Inlet**: Water enters at 0.5 m/s and 350 K temperature.
* **Outlet**: Pressure outlet at 0 Pa (atmospheric pressure).
* **Walls**: Set as *adiabatic* (no heat loss) and no-slip (fluid sticks to the wall).
* **Fluid**: Water was selected as the working fluid with real physical properties.

**5. Choosing the Solver Settings**

To make the simulation work properly, we picked the right models:

* **Solver Type**: Pressure-based, steady-state (because we’re assuming flow and temperature stay constant over time).
* **Turbulence Model**: Standard *k-ε* model (works well for general cases).
* **Energy Equation**: Enabled, since heat transfer is important here.

**6. Initializing and Running the Simulation**

We initialized the flow from the inlet and started the calculation. We monitored key parameters like:

* Temperature change from inlet to outlet
* Pressure drop
* Flow pattern (to see how water flows inside the pillow plate)

The simulation was run until the solution converged (residuals dropped below 10⁻⁵), which means the results became stable and reliable.

1. **RESULT**

**A. Velocity Distribution – How the Water Flows** Inside the pillow plate channels, the water speeds up where the space gets tight and slows down where the channel widens. This creates a wavy, swirling flow pattern that’s great for mixing and heat transfer.

# Maximum velocity: ~0.7 m/s at narrow spots

# Average flow speed: ~0.5 m/s

# Observation: Tiny recirculation zones form around the "bulges" of the pillow plate, enhancing local mixing

# **B. Temperature Distribution – How Heat is Transferred**

# As the water flows through the exchanger, it cools down — a clear sign that the system is transferring heat effectively.

# Inlet temperature: 350 K

# Outlet temperature: ~335 K

# Plate surface temperature: ranges between 330 K and 345 K

# Observation: The wavy design causes better contact with the surface, improving heat absorption.

# **C. Pressure Drop – Resistance to Flow**

# The water loses some pressure as it pushes through the narrow, curving channels.

# Inlet pressure: 15,000 Pa

# Outlet pressure: 0 Pa (gauge)

# Total pressure drop: 15,000 Pa

# Observation: Most of the drop happens near tight gaps and sharp bends. This could be reduced by optimizing the channel design.

# **D. Heat Transfer Rate – How Much Heat Was Moved?**

# Using the simulation data and water’s specific heat, we estimated the total heat absorbed:

# Mass flow rate: 0.0005 kg/s

# Temperature drop: 15 K

# Heat transfer rate (Q): ~31.36 W

# Observation: The pillow plate design performs well, even at relatively low flow rates.

# **E. Wall Heat Flux – Where Heat Transfers Best**

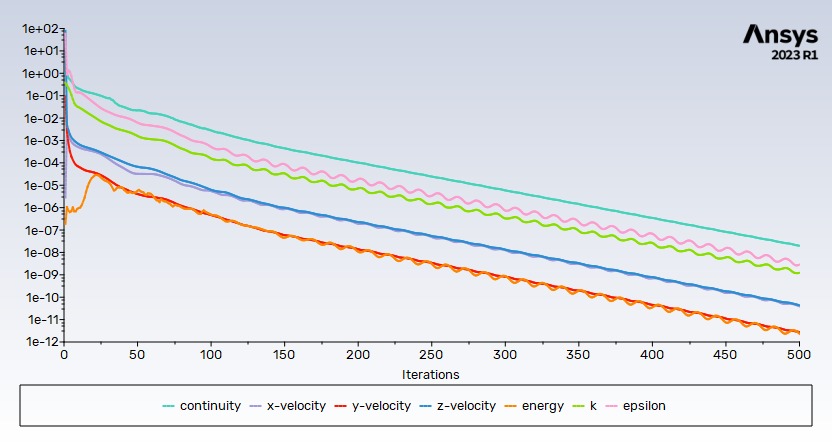
# Heat flux tells us how efficiently heat is moving across the plate surface.

# Range: 200–500 W/m²

# Hotspots: Near flow-contact zones — especially where the velocity is highest.

# Observation: The pillow shape creates alternating zones of strong and weak heat transfer, which is normal and helps mix

# the flow.



**DISCUSSIONS**

# The imulation results for pillow plate heat exchangers (PPHEs) reveal distinct fluid dynamics in their inner and outer channels, shaped by the exchanger's unique geometry. In the inner channels, the flow meanders and forms recirculation zones around welding spots, enhancing mixing and overall heat transfer, despite localized reductions in efficiency. Compared to traditional pipe systems, PPHEs demonstrate superior thermal performance at low Reynolds numbers due to this enhanced mixing. However, as Reynolds numbers increase, the associated pressure losses diminish this advantage. In the outer channels, periodic changes in cross-sectional area lead to boundary layer separations and vortex formations near welding spots, significantly impacting pressure loss and heat transfer. Utilizing the elliptic blending k-ε turbulence model in simulations provided more accurate predictions of these complex behaviors, aligning closely with experimental data. Design correlations for Darcy friction factor and Nusselt number were derived, aiding in performance estimation. When comparing PPHEs to pipes under equal pumping power, PPHEs generally outperform due to better mixing. However, under equal Reynolds numbers, pipes often exhibit lower pressure losses, highlighting the importance of evaluation criteria in performance assessment

# **CONCLUSION**

In This project took a deep dive into understanding how a pillow plate heat exchanger performs by using Computational Fluid Dynamics (CFD) simulations. We used ANSYS Fluent to model the system, considering water as the working fluid, with an inlet temperature of 350 K and an inlet velocity of 0.5 m/s. These conditions helped create a realistic scenario for evaluating how the heat exchanger operates. The results from the CFD analysis gave us some valuable insights. For example, we could see the velocity distribution within the heat exchanger, highlighting areas of high turbulence. While turbulence is great for enhancing heat transfer, it also creates pressure losses that need to be carefully managed. This kind of detailed flow information is essential for improving how efficiently the heat exchanger works. We also looked at the pressure drop, which showed us the resistance the fluid faces as it moves through the system. This is important because it tells us how much energy is needed to keep the flow steady. The visualizations of streamlines and path-lines helped us spot any areas where the flow was inefficient or where there might be recirculation, helping us identify potential improvements in the design. In the end, this project really highlighted how valuable CFD simulations can be when it comes to optimizing heat exchanger designs. The insights we gathered could help engineers make more efficient, cost-effective, and environmentally friendly systems, ultimately improving the performance of industrial heat exchangers.

# REFERENCES

1. M. R S. Mitrovic and D. Maletic, "Numerical Investigation of Flow and Heat Transfer in Pillow Plate Heat Exchangers," *International Journal of Heat and Mass Transfer*, vol. 54, no. 15–16, pp. 3433–3441, July 2011. DOI: 10.1016/j.ijheatmasstransfer.2011.03.034
2. B. Piper, S. Mitrovic, and D. Maletic, "CFD Analysis of Flow and Heat Transfer in Pillow Plate Channels Using Advanced Turbulence Models," Applied Thermal Engineering, vol. 73, no. 1, pp. 145–157, Dec. 2014. DOI: 10.1016/j.applthermaleng.2014.07.039
3. J. M. Tran, M. Piper, and E. Y. Kenig, "Experimental Investigation of Convective Heat Transfer and Pressure Drop in Single-Phase Flow Through Pillow Plates," Chemie Ingenieur *Technik*, vol. 87, pp. 226–234, 2015.
4. X F. Billard and D. Laurence, "A Robust k–ε–v²/k Elliptic Blending Turbulence Model Applied to Near-Wall, Separated and Buoyant Flows," International Journal of Heat and Fluid Flow, vol. 33, pp. 45–58, 2012.
5. M. Piper, A. Zibart, J. M. Tran, and E. Y. Kenig, "A Numerical Study on Turbulent Single-Phase Flow and Heat Transfer in Pillow Plates," in *15th International Heat Transfer Conference*, Kyoto, 2014.
6. A. Olenberg, M. Piper J. M. Tran, and E. Y. Kenig, "Determination of the geometric design parameters of pillow plate heat exchangers," Applied Thermal Engineering, vol. 91, pp. 1168–1175, 2015
7. F. P. Incropera, D. P. DeWitt, T. L. Bergman, and A. S. Lavine, Fundamentals of Heat and Mass Transfer, 6th ed. Hoboken, NJ: John Wiley & Sons, 2007
8. ANSYS, Fluent Theory Guide 19.1, Chapter 18: Solidification and Melting, ANSYS Inc., 2018.literature review in this point
9. Y. C. Shih and H. Chou, "Numerical study of solidification around staggered cylinders in a fixed space," Numerical Heat Transfer, Part A: Applications, vol. 48, no. 3, pp. 239–260, 2005
10. E. Y. Kenig, "Numerical investigation of turbulent forced convection heat transfer in pillow plates," Int. J. Heat Mass Transfer, vol. 94, pp. 516-527, 2016