VAASAN AMMATTIKORKEAKOULU
UNIVERSITY OF APPLIED SCIENCES

Osku Väyrynen

ENSURING THE AVAILABILITY OF TURBULATORS FOR END-OF-LIFE CYCLE OF THE NXW'S CH6X DEVICES

Danfoss Drives Vaasa

Tekniikka
2023

VAASAN AMMATTIKORKEAKOULU
UNIVERSITY OF APPLIED SCIENCES
Konetekniikka

ABSTRACT

Author	Osku Väyrynen
Title	Ensuring the Availability of Turbulators for End-of-Life Cycle of the NXW's CH6x Devices
Year	2023
Language	English
Pages	58
Name of Supervisor	Sami Elomaa

This thesis done in partnership with Danfoss Drives, and it addresses the critical issue of ensuring an uninterrupted supply of turbulators for the cooling systems in the NXW CH6x family of drives. Turbulators, essential for improving heat transfer rates by creating turbulence, are important cooling system components and the availability of turbulators is paramount to prevent costly production downtimes. The project aims to solve the problem that arises when these components face supply shortages, particularly when original models become obsolete, or suppliers discontinue them. The primary objective is to explore and evaluate alternative turbulator designs and identify new suppliers, ensuring the continuous production of NXW CH6x drives without supply-related issues.

The research employs Computational Fluid Dynamics (CFD) simulation, a key method in analyzing and solving complex fluid flow and heat transfer problems. This approach models a fluid flow using equations for mass, momentum, and energy conservation, solved numerically to provide visualizations and graphs. These simulations are crucial in assessing the performance of various turbulator designs and their impact on cooling efficiency.

The findings reveal that different turbulator designs significantly influence the efficiency of the cooling system. CFD simulations provide insights into fluid behavior, including velocity, pressure, and temperature distributions, crucial for optimizing cooling system designs. The study concludes that the strategic selection and optimization of turbulators is critical for maintaining uninterrupted production and enhancing the overall efficiency and longevity of the cooling systems in electronic devices.

Keywords	Heat sink, simulation, water cooling, turbulator, power electronics.

VAASAN AMMATTIKORKEAKOULU
UNIVERSITY OF APPLIED SCIENCES
Konetekniikka

TIIVISTELMÄ

Tekijä	Osku Väyrynen
Opinnäytetyön nimi	NXW CH6x -turbulaattoreiden saatavuuden varmistaminen NXW-laitteiden loppu elinkaarelle
Vuosi	2023
Kieli	englanti
Sivumäärä	58
Ohjaaja	Sami Elomaa

Tämä Danfoss Drivesin kanssa yhteistyössä tehty opinnäytetyö käsittelee kriittistä haastetta turbulaattorien keskeytymättömän toimituksen varmistamisesta NXW CH6x -taajuusmuuttajien jäähdytysjärjestelmiin. Turbulaattorit, jotka ovat kriittisiä lämmönsiirron parantamiselle luomalla turbulenssia, ovat kriittinen jäähdytysjärjestelmän osa, ja turbulaattorien saatavuus on ensiarvoisen tärkeää kalliiden tuotantoseisokkien estämiseksi. Projekti pyrkii ratkaisemaan ongelman, joka syntyy, kun komponenttien saatavuus heikkenee, etenkin kun alkuperäiset mallit vanhenevat tai toimittajat lopettavat niiden valmistuksen. Ensisijainen tavoite on tutkia ja arvioida vaihtoehtoisia turbulaattori malleja ja löytää uusi toimittaja, mikä varmistaa NXW CH6x -taajuusmuuttajien jatkuvan tuotannon ilman toimitusongelmia.

Tutkimuksessa käytetään Computational Fluid Dynamics (CFD) -simulaatiota, joka on keskeinen menetelmä monimutkaisten nestevirtaus- ja lämmönsiirto ongelmien analysoinnissa ja ratkaisemisessa. Tämä lähestymistapa mallintaa nestevirtausta käyttämällä massan, liikemäärän ja energian säilymisen yhtälöitä, jotka ratkaistaan numeerisesti visualisointien ja kuvaajien luomista varten. Nämä simulaatiot ovat ratkaisevia arvioitaessa erilaisten turbulaattori mallien suorituskykyä ja niiden vaikutusta jäähdytystehokkuuteen.

Tulokset paljastavat, että erilaiset turbulaattori mallit vaikuttavat merkittävästi jäähdytysjärjestelmän tehokkuuteen. CFD-simulaatiot tarjoavat tietoa nesteen käyttäytymisestä, mukaan lukien nopeus-, paine- ja lämpötilajakaumista, jotka ovat ratkaisevan tärkeitä jäähdytysjärjestelmän suunnittelun optimoinnissa. Tutkimuksessa todetaan, että turbulaattorien strateginen valinta ja optimointi on kriittistä jatkuvan tuotannon ylläpitämiseksi ja elektronisten laitteiden jäähdytysjärjestelmien kokonaistehokkuuden ja pitkäikäisyyden parantamiseksi.

Avainsanat	Jäähdytyselementti, simulointi, vesijäähdytys, turbulaattori, tehoelektroniikka.

TABLE OF CONTENTS

FIGURES

TABLES

TERMS AND ABBREVIATIONS

CFD	Computational fluid dynamics
FEM	Finite element method
EOL	End-of-life
IGBT	Insulated Gate Bipolar Transistors
MOSFET	Metal-oxide-semiconductor field-effect transistor
V	Voltage
VFD	Variable frequency drives
PWM	Pulse-width modulation

1 INTRODUCTION

This thesis is done in partnership with Danfoss Drives, and it focuses on the challenge of securing a continuous supply of turbulators for the cooling system in NXW CH6x family of drives. Turbulators are a key component in a cooling system; they create more turbulence and by doing so, improve the heat transfer rate. When these components are in short supply, it can lead to production line downtime and significant costs.

Given this, the aim of this thesis is to explore and evaluate different turbulator designs and to identify new suppliers. The goal is to avoid any interruption in availability, which is especially critical when the original turbulator model may become outdated or suppliers discontinues them. By investigating various alternatives, this thesis aims to ensure that the production lines of NXW CH6x drives continues to operate effectively without facing supply-related issues.

The thesis starts with the presentation of the existing turbulator designs and how well they perform. Then, using Computational Fluid Dynamics (CFD) simulations, it tests various alternative designs. The focus is on finding new turbulators that can perform as well as or better than the original models within the specific project constraints, such as the need for non-disruptive changes and backward compatibility during the end-of-life phase of the devices.

Ultimately, the findings from this thesis should provide practical options for maintaining the performance of NXW CH6x drives cooling system and the efficient running of the production lines producing NWX CH6x family of drives by securing a reliable supply of turbulators, thereby avoiding downtime, and keeping efficiency high.

2 HEAT SINKS, HEAT EXCHANGERS, COLD PLATES AND WATER BLOCKS

Heat sinks, heat exchangers, cold plates, and water blocks are key elements in cooling electronic devices. Heat sinks, often made of aluminum or copper, release heat into the air, while heat exchangers transfer heat between fluids. The archetypical function of any heat exchanger is getting energy from one fluid mass to another (Lienhard & Lienhard, 2019, p. 99). Water blocks, used in liquid cooling, draw heat from high-performance devices directly. All these components are crucial for preventing electronic overheating and ensuring device longevity. (Shah & Sekulić, 2003)

2.1 Heat Sinks to Transfer Heat Energy

Heat sinks are critical devices used to dissipate heat in electronic components and machinery, safeguarding against overheating and potential damage. These components operate by absorbing heat produced during the function of devices, such as computers, televisions, and other heat-generating appliances, and distributing it away from sensitive areas. Predominantly crafted from high thermal conductivity materials, such as aluminum, copper, or graphite, heat sinks are designed in various forms to meet specific thermal requirements. The efficiency of heat sinks is further enhanced by thermal interface materials, which ensure a better thermal bond between the component and the heat sink. (Lienhard & Lienhard, 2019)

Moreover, heat sinks may employ passive cooling techniques, such as natural convection, or active cooling, including forced air or liquid, to manage the dissipation process effectively. They must also be designed considering environmental variables such as ambient temperatures and airflow, which greatly influence performance. (Kordyban, 1998)

Within this realm, turbulators play a pivotal role. These elements are incorporated into heat sink designs to disrupt laminar flow, creating turbulence that facilitates increased thermal transfer efficiency. As such, heat sinks, augmented by turbulators, constitute an essential facet of modern thermal management strategies, ensuring devices operate within safe temperature ranges and maintain optimal performance. (Allen, 2020)

2.2 Forced Air or Water Cooling

Forced air cooling leverages fans or blowers to move air over electronic components and heat sinks, facilitating heat dissipation. This method is not only cost-effective but also simple to implement. However, its cooling capacity is somewhat limited, and it is susceptible to environmental factors such as ambient temperature and humidity. Moreover, forced air systems can contribute to acoustic noise, which may be a concern in noise-sensitive environments. (Kordyban, 1998)

On the other hand, water cooling systems, which pump liquid coolant through the electronic systems, are superior in heat absorption due to the higher thermal conductivity of liquids over air. This makes water cooling an optimal choice for high-power electronics, enabling it to maintain lower operational temperatures consistently. Nonetheless, this method introduces complexities in installation with the need for an elaborate network of plumbing, pumps, and valves. It also entails a higher cost, not just in initial setup but also in maintenance, requiring vigilant monitoring for potential leaks that can harm electronic components. (Shah & Sekulić, 2003)

In terms of scalability, forced air cooling can become impractical in high heat density scenarios due to the larger and more numerous fans required, whereas water cooling systems can handle increased heat loads with relatively small increases in system size. Additionally, the quieter operation of water cooling can be a deciding factor in environments where noise is a significant concern. (Kordyban, 1998; Shah & Sekulić, 2003)

The selection between air and water cooling ultimately hinges on the specific demands of the electronic system in question, considering factors such as cooling efficiency requirements, potential risks, environmental conditions, space availability, and noise tolerances. At Danfoss, most high-power drives are offered in both water- and air-cooled versions. (Kordyban, 1998; Shah & Sekulić, 2003)

2.3 Power Electronics Cooling in Variable Frequency Drives.

Managing heat is essential for the reliability and efficiency of power electronics. This is particularly true for devices such as Variable Frequency Drives (VFDs), which control the speed of electric motors. As these components operate, they convert power, resulting in the generation of heat. If this heat is not dissipated effectively, it can reduce the performance or even damage the electronics. (Kraus & Bar-Cohen, 1983; Thompson, 2007)

Traditionally, cooling has relied on passive methods such as heat sinks or active ones like air cooling. Heat sinks draw the heat away from electronic components, while fans move air to keep temperatures down. These methods, however, are becoming less effective as devices such as VFDs become more compact, with less room for air flow and larger heat generation in a smaller area. (Kraus & Bar-Cohen, 1983)

To address the increased heat density, engineers are devising advanced cooling solutions. Fluid-based cooling, for example, employs liquids that can be cycled through micro-channels to absorb heat directly from hot spots. Alternately, immersion cooling submerges electronics in a non-conductive liquid, providing uniform cooling without electrical hazards. Additionally, modern computational systems are used to regulate these cooling processes, adapting in real time to the thermal demands. By integrating these innovative cooling strategies, the latest generation of power electronics, including VFDs, can maintain optimal temperatures for peak operation, even under intense power loads. (Shah & Sekulić, 2003)

2.3.1 IGBT

Insulated Gate Bipolar Transistors, or IGBTs, are a type of semiconductor device that is widely utilized in power electronics. They combine the simple gate-drive characteristics of MOSFETs with the high-current and low-saturation-voltage capability of bipolar transistors, making them ideal for high-power applications. (Schulz, 2019)

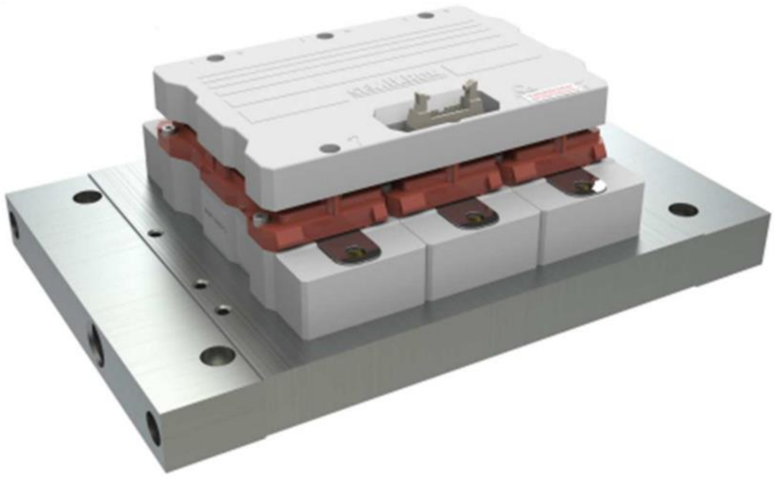


Figure 1. Semikron Danfoss SKiiP 3 package. (Semikron-Danfoss, 2017)

An IGBT has three terminals – the collector, the emitter, and the gate, similar to a standard MOSFET. They are particularly valued for their ability to efficiently switch high voltages and currents, which is crucial in applications such as motor control, power supplies, and inverters. With their low on-state voltage drop and the capacity to handle high power, IGBTs are engineered for performance. Additionally, their high input impedance means they can be controlled with minimal power – a feature that simplifies the design of control circuits. (Schulz, 2019)

IGBTs play a critical role in Variable Frequency Drives (VFDs) which are employed for controlling the speed of motors. Their high efficiency, fast switching speeds, and capability to manage substantial currents make them far more suitable for this purpose than traditional MOSFETs, especially in high-temperature environments. (Schulz, 2019)

Within a VFD, IGBTs are integral to the inverter module, which is responsible for transforming DC power back into AC at varying frequencies and voltages to control motor speed. (Semikron-Danfoss, 2024)

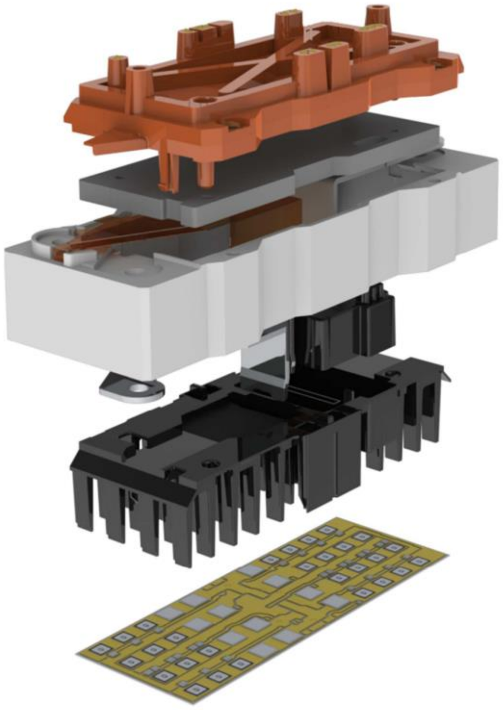


Figure 2. Semikron Danfoss SKiiP 3 half bridge exploded view. (Semikron-Danfoss, 2017)

The IGBTs switch on and off quickly, generating a pulse-width modulated (PWM) waveform. This PWM control is what allows for the precise adjustment of both frequency and voltage supplied to the motor, leading to precise motor speed regulation. (Schulz, 2019)

Moreover, since IGBTs must cope with the high currents and voltages typical in AC outputs (commonly 230V, 480V or even 690V), and dissipate significant heat during operation, robust thermal management is essential. Proper heat sinking and other cooling methods are crucial to maintain their performance and ensure reliability. (Schulz, 2019)

In summary, IGBTs are a cornerstone of modern VFDs, enabling the precise and efficient control of motor speed while optimizing energy consumption and ensuring the safety and reliability of the system.

2.3.2 Resistors and Capacitors

In Variable Frequency Drives (VFDs), resistors and capacitors play critical roles in managing power and protecting the system. Resistors in VFDs, such as braking resistors, are crucial for dissipating excess energy generated during motor deceleration, preventing overvoltage conditions. Bleeder resistors safely discharge stored energy in capacitors to avoid electric shocks when the drive is powered off. Additionally, resistors are used for current sensing to monitor electrical flow and in gate circuits to control the switching of semiconductor devices like IGBTs, ensuring proper operation and longevity of the system. (*What Do Capacitors Do in VFDs?*, n.d.)

Capacitors are equally important in VFDs, with DC bus capacitors smoothing out the DC power supply, critical for the stable operation of the inverter stage. Snubber capacitors protect against voltage spikes that can harm the power switching devices. Decoupling capacitors work to stabilize voltage levels close to these power devices during fast switching operations. Lastly, filter capacitors reduce harmonic distortion in the power output of VFDs, ensuring clean operation and minimal interference with other equipment. (*What Do Capacitors Do in VFDs?*, n.d.)

These components are meticulously selected to match the specifications of the drive, ensuring efficiency, regulatory compliance, and safety in the VFD’s operation. Their performance is fundamental to the ability of the VFD to provide precise motor control and to its overall reliability and efficiency. (*What Do Capacitors Do in VFDs?*, n.d.)

3 LAMINAR FLOW AND TURBULENT FLOW

Laminar flow is smooth with fluid particles gliding in parallel lines, typically at lower velocities, leading to less efficient heat transfer due to minimal mixing. Turbulent flow, on the other hand, is characterized by high velocities causing chaotic eddies and swirls that enhance heat transfer by promoting greater fluid mixing. While laminar flow is energy-efficient, turbulent flow increases thermal efficiency but requires more energy to maintain, influencing the design and operation of systems like heat exchangers and heat sinks. (Cadence, 2023)

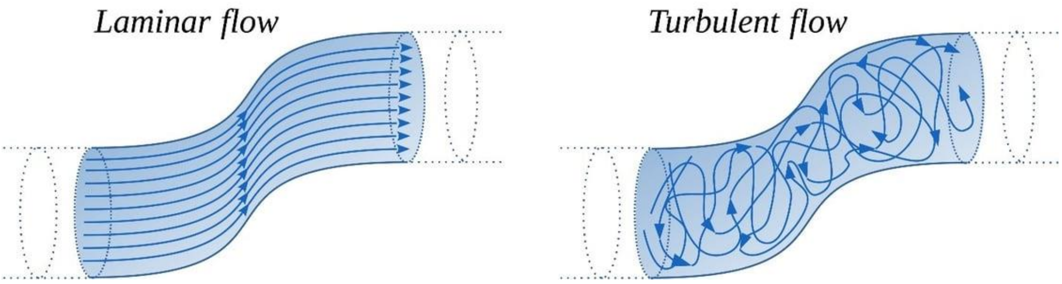


Figure 3. Comparison of laminar and turbulent flow. (Cadence, 2023)

3.1 Laminar Flow

Laminar flow, characterized by its smooth, layered movement of fluid, facilitates a serene and controlled environment where the fluid travels in parallel layers with minimal crossflow. In this regime, the fluid's motion is highly predictable, which allows for precise control in applications where such stability is paramount. This is particularly beneficial in scenarios like microfluidic devices used in biomedical engineering, where the consistency of flow rates is crucial for the accurate analysis and handling of minute fluid volumes. (Chang, 1970; Schobeiri, 2014)

Moreover, the laminar flows gentle nature translates into reduced frictional forces, which in turn minimizes energy losses and prevents erosion within the flow channels. This can result in quieter operation and less wear on system components, prolonging their life and ensuring reliability. However, due to the orderly flow pattern, there is a limitation in the convective heat transfer as the layers of fluid do not mix, which leads to a predominantly conductive heat transfer mechanism. As such, systems relying on laminar flow may incorporate enhanced surface designs like micro-fins or textured surfaces to increase the effective surface area, thereby improving the rate of heat exchange. Additionally, utilizing materials with high thermal conductivity can help to overcome the inherent limitations of laminar flow, ensuring efficient energy transfer in applications ranging from electronics cooling to environmental control systems. (Chang, 1970; Schobeiri, 2014)

3.2 Turbulent Flow

Turbulent flow is a complex type of fluid flow characterized by chaotic and stochastic fluctuations in velocity and pressure, leading to extensive mixing of fluid particles. This regime is governed by a high Reynolds number, indicating the dominance of inertial forces over viscous forces in the fluid's motion. As fluid moves rapidly or encounters obstructions, the smooth laminar flow breaks down into a cascade of swirling eddies and vortices, resulting in enhanced thermal exchange and mixing capabilities, which are beneficial in many industrial processes, such as in heat exchangers or combustion systems. (Chang, 1970)

Despite these advantages, the inherent unpredictability of turbulence poses significant challenges in controlling flow and predicting system behavior. The small-scale vortices within turbulent flow contribute to energy dissipation, converting organized flow kinetic energy into random thermal energy, an effect that can increase drag and reduce system efficiency. Modeling such a flow requires sophisticated computational fluid dynamics techniques, ranging from highly detailed DNS to more statistically averaged approaches such as RANS, making the accurate prediction of turbulent flow an intensive but crucial aspect of fluid dynamics studies in environmental, aeronautical, and process engineering fields. (Chang, 1970)

4 PRESSURE LOSS IN HEAT SINKS

Pressure loss in water-cooled heat sinks is an influential factor that can impact the efficacy of a thermal management system. This loss of pressure occurs due to a combination of factors during the circulation of the coolant through the heat sink. (Shah & Sekulić, 2003; *What Is Pressure Drop?*, n.d.)

Pressure losses are caused by frictional losses, flow direction changes and flow rate.

As water flows through the channels within the heat sink, it encounters resistance, primarily due to surface friction. This friction is dependent on both the viscosity of the water and the surface roughness of the channel, leading to a loss of pressure that increases with the length of the flow path. (Shah & Sekulić, 2003; *What Is Pressure Drop?*, n.d.)

The design of the heat sink often necessitates changes in flow direction, which can introduce additional pressure drops. Each turn, bend, or sudden expansion in the geometry of the channel can contribute to a loss of pressure due to the disturbance in the flow pattern. (Shah & Sekulić, 2003; *What Is Pressure Drop?*, n.d.)

The velocity of the water has a direct relationship with pressure loss; higher flow rates can cause greater frictional losses. However, this is a double-edged sword as a higher flow rate also typically means improved heat transfer capabilities. (Shah & Sekulić, 2003; *What Is Pressure Drop?*, n.d.)

5 HEAT TRANSFER FROM SOLID TO LIQUID

Heat transfer from a solid to a liquid is a fundamental process that occurs when the two states of matter are in thermal contact. This transfer of thermal energy can happen through different mechanisms: conduction, convection, and radiation, each playing a distinct role depending on the conditions. Thermal energy and heat are taken as synonyms in engineering context (Lienhard & Lienhard, 2019, p. 3).

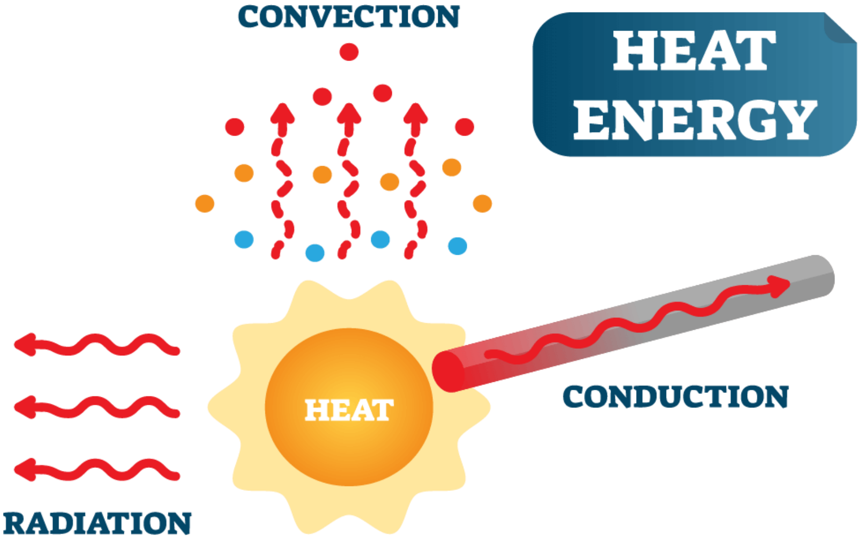


Figure 4. Three modes of heat transfer. (*Concept Group LLC*, n.d.)

Conduction is the direct transfer of heat through matter, resulting from the physical contact of the molecules. In the interface between a solid and a liquid, thermal energy is passed from the more energetic particles of the solid to the less energetic particles of the liquid. This mechanism relies on the vibrational energy of atoms and molecules and is particularly significant in metals and other highly conductive materials. (Lienhard & Lienhard, 2019)

Convection is the process where heat is carried away by the movement of the liquid itself. As the liquid touches the heated solid surface, it absorbs heat, becomes less dense, and rises. Cooler, denser liquid then replaces it, setting up a convective current that helps to distribute the heat more evenly throughout the liquid volume. (Lienhard & Lienhard, 2019)

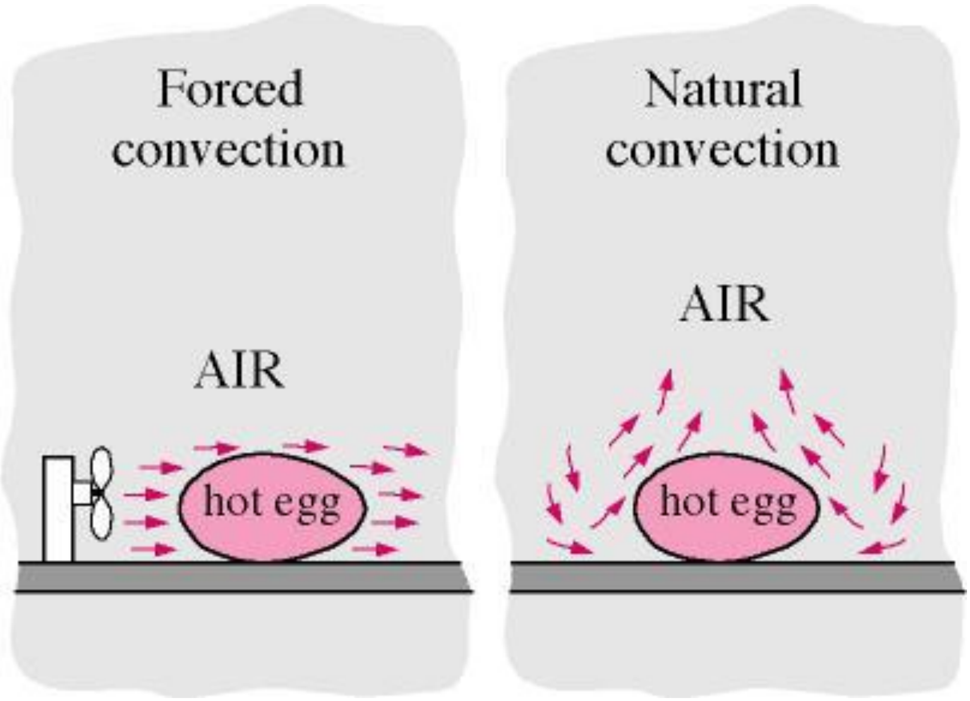


Figure 5. Picture showing the difference between forced and natural convection. (McGraw-Hill Higher Education, 1998)

Radiative heat transfer occurs without direct contact, as energy is transferred in the form of electromagnetic waves. Any solid at a temperature above absolute zero emits radiation, which can be absorbed by the surrounding liquid, thus heating it up. This mode of heat transfer is less significant in solid-liquid interfaces unless the solid is at a very high temperature, or the liquid is transparent to the radiative energy spectrum involved. (Lienhard & Lienhard, 2019)

In summary, while conduction is typically the dominant mechanism in most solid-liquid heat transfer scenarios, convection and radiation also contribute to the overall process. The efficiency and rate of heat transfer depend on the properties of the materials involved, the surface area in contact, and the temperature difference between the solid and the liquid.

5.1 Heat Transfer Enhancements

Implementing features such as turbulators, can increase the heat transfer coefficient but may also contribute to a higher pressure drop due to the induced turbulence and additional flow obstructions. (Shah & Sekulić, 2003)

Understanding and minimizing pressure loss is critical in heat sink design as it directly affects the pumping power required to maintain the desired coolant flow rate. A well-optimized system must carefully balance the heat removal needs with the acceptable pressure drop to ensure high performance without undue energy expenditure. (Shah & Sekulić, 2003)

Optimizing a water-cooled heat sink involves a full-fledged approach to design, where the flow rate, flow path geometry, and heat transfer rates are meticulously tailored to mitigate pressure losses while maximizing thermal performance. This balance ensures operational efficiency, maintains component temperatures within safe limits, and conserves energy. (Shah & Sekulić, 2003)

6 WAYS TO ACHIEVE TURBULENT FLOW

Turbulent flow is a key concept in thermal management systems, especially when seeking to improve heat transfer efficiency. By transitioning from laminar to turbulent flow within a cooling medium, the thermal exchange between hot surfaces and the fluid is significantly enhanced. Achieving this turbulent state can be done through various methods, including the strategic use of turbulators such as twisted tapes, wire coils, or shaped inserts that disrupt smooth flow patterns. Adjusting the geometry of cooling channels, such as using dimpled or roughened surfaces, also promotes turbulence. These techniques are critical in industries where cooling high-power electronic systems, such as in variable frequency drives (VFDs) and other advanced electronic devices, is crucial for performance and longevity. (Lienhard & Lienhard, 2019; Shah & Sekulić, 2003)

6.1 Integrated Elements in Heat Sinks

Integrated elements within water-cooled heat sinks are intrinsic design features that amplify their heat dissipation capabilities. These enhancements, tailored to the specific needs of the application, often include an array of fins, pins, dimples, or various geometric structures that serve to expand the surface area for heat exchange and foster turbulence within the fluid flow. (Shah & Sekulić, 2003)

Incorporating such elements directly into the heat sink's design obviates the need for additional parts, contributing to a leaner and more effective cooling solution. This is particularly beneficial in scenarios where space is at a premium or where superior thermal management is crucial. The strategic integration of these features enables a significant uptick in thermal performance, ensuring that the heat sink operates at optimal efficiency and effectiveness. (Shah & Sekulić, 2003)

6.1.1 Cold Plate

Cold plates are essential for cooling high-power electronics, utilizing direct contact with components to transfer heat to a circulating liquid coolant. Made typically from high thermal conductivity materials like aluminum or copper, they feature internal channels that guide the coolant and maximize heat transfer. This method allows for a more compact and efficient cooling solution compared to traditional air-cooled methods, making cold plates crucial for managing the thermal load in densely packed electronic systems. Generally, these plate-type heat exchanges cannot accommodate very high pressures, temperatures or pressure and temperature differences (Shah & Sekulić, 2003, pp. 22–23).

6.1.2 Micro Channels

Microchannels are a sophisticated feature engineered into water-cooled heat sinks to boost their thermal efficiency. Essentially narrow, often only millimeters in cross-section, these channels significantly increase the contact surface area between the heat sink and the coolant. This design encourages turbulent flow, which is far superior for heat transfer compared to a laminar one because it facilitates more effective heat exchange. (Zhao et al., 2014)

These micro-sized passages help in creating a more compact heat sink while ensuring a more even distribution of temperature across its surface, enhancing both performance and reliability. Additionally, microchannels allow for a greater heat flux density; in other words, they enable the heat sink to handle more heat per unit area, which is crucial in high-power applications. (Zhao et al., 2014)

Nevertheless, the adoption of microchannels isn't without its trade-offs. The increased surface area can lead to a higher pressure drop, requiring more powerful pumps to maintain optimal coolant flow. There's also an increased susceptibility to clogging, making the purity of the coolant a top priority. Furthermore, producing heat sinks with microchannels can be a complex process, potentially escalating manufacturing costs. Despite these challenges, the use of microchannels is often justified by their substantial contribution to the efficiency of water-cooled systems, especially in scenarios where space-saving and high thermal performance are of paramount importance. (Zhao et al., 2014)

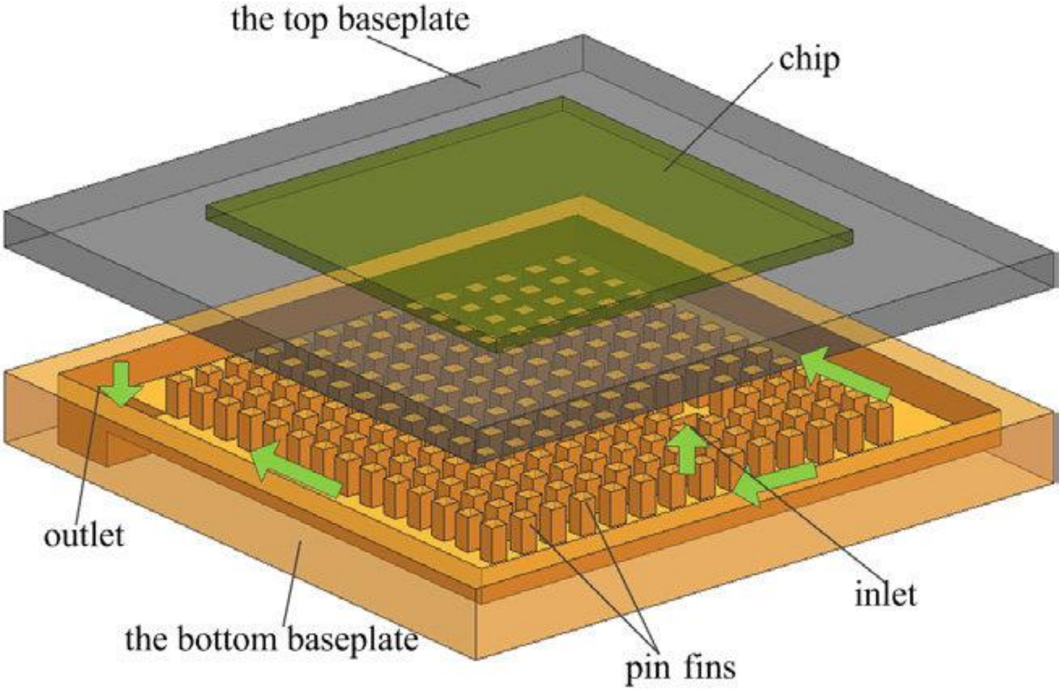


Figure 6. Microchannel heat sink with a pin-fin array (Zhao et al., 2014)

6.1.3 Jet Impingement

Jet impingement cooling is a sophisticated technique that employs direct fluid jets to manage the temperature of high-power electronic components effectively. This method works by forcing a coolant at high velocity through nozzles, which then strikes the surface needing cooling. The kinetic energy of the liquid disrupts the boundary layer of thermal resistance directly above the heated surface, thereby enhancing the heat transfer rate. This direct approach allows for significant removal of heat over small surface areas, making it ideal for densely packed or high-heat-flux applications. (*What Is Jet Impingement Cooling and How Is It Applied for Thermal Management of Electronics (Part 1 of 2) | Advanced Thermal Solutions*, n.d.)

In addition to its high cooling efficiency, jet impingement allows for precise temperature control, as the flow rate and jet parameters can be finely adjusted. This adaptability makes it a go-to method for thermal management in unevenly heated or spatially constrained environments. By configuring the jet velocity and impact area, engineers can tackle hot spots aggressively, ensuring the longevity and performance of sensitive electronic components. (*What Is Jet Impingement Cooling and How Is It Applied for Thermal Management of Electronics (Part 1 of 2) | Advanced Thermal Solutions*, n.d.)

Moreover, the versatility of jet impingement enables its integration into various cooling systems, ranging from microchannel heat sinks to large-scale industrial equipment. The technology can be designed to circulate either single-phase coolants for standard applications or phase-change fluids that absorb additional heat through vaporization for more demanding scenarios. Despite potential challenges such as noise and increased power demand for the pumps, the precise cooling and high efficiency of jet impingement solidify its position as a critical asset in modern thermal management strategies. (*What Is Jet Impingement Cooling and How Is It Applied for Thermal Management of Electronics (Part 1 of 2) | Advanced Thermal Solutions*, n.d.)

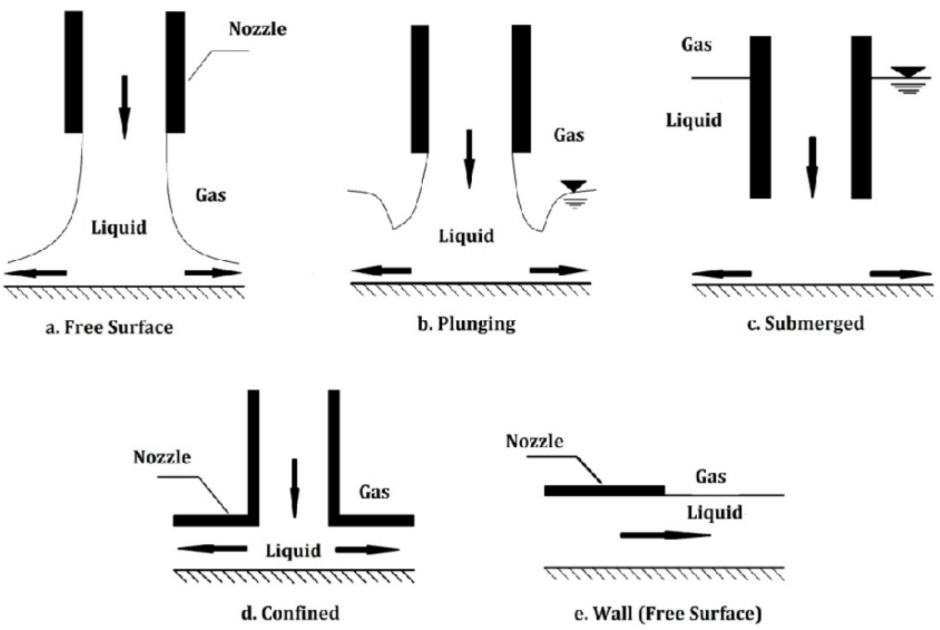


Figure 7. Different types of impingement jets (Molana & Banooni, 2013)

6.2 Turbulators

Turbulators are devices or structures that are added to water-cooled heat sinks to increase their heat transfer efficiency. They are designed to disrupt the flow of water through the heat sink and promote turbulence, which increases the contact between the water and the heat sink surface, and therefore enhances the heat transfer. (Allen, 2020)

There are various types of turbulators that can be used in water-cooled heat sinks, including matrix turbulators, twisted tape turbulators, ball turbulators and spring turbulators. These turbulators are typically placed inside the channels or passages of the heat sink, and they are designed to generate turbulence by creating eddies and vortices in the flow of water. (Allen, 2020)

Turbulators can be effective at improving the heat transfer efficiency of water-cooled heat sinks, particularly in situations where the flow of water is laminar or slow-moving. By increasing the turbulence of the water flow, turbulators can help to break up the boundary layer of water that forms near the surface of the heat sink, which can improve heat transfer by exposing more water molecules to the heat sink surface. (Allen, 2020)

However, it is important to note that the use of turbulators can also increase the pressure drop across the heat sink, which can reduce the flow rate of cooling water and require additional pumping power to maintain the desired flow rate. Additionally, the design and placement of turbulators can be complex, and it may require careful consideration of the flow conditions and heat sink geometry to achieve optimal performance. (Allen, 2020)

6.2.1 Matrix Turbulator

In a water-cooled heat sink, a matrix turbulator significantly enhances thermal transfer efficiency by inducing turbulence in the water flow. The design of these turbulators, involving a matrix of small, coiled wires, typically crafted from conductive materials like metal, maximizes contact with the water, thereby optimizing heat absorption. (Allen, 2020; Calgavin, 2022)

As water navigates the complex geometry of the matrix turbulator, it undergoes a deliberate disturbance in flow. This not only facilitates a more dynamic mix but also intensifies the thermal exchange between the water and the heat sink surfaces. The resulting turbulence ensures a more uniform temperature distribution and accelerated heat dissipation. (Allen, 2020; Calgavin, 2022)

While matrix turbulators offer superior heat transfer capabilities, especially in high heat flux scenarios, they also introduce a higher pressure drop in the system. This necessitates a careful balance between thermal performance and hydraulic resistance, as the increased pressure drop might require more potent pumps, impacting the system's energy efficiency. Moreover, the open structure of matrix turbulators mitigates the risk of clogging, a common concern in fluid dynamics, ensuring long-term, reliable operation in various cooling applications. (Allen, 2020; Calgavin, 2022)

Figure 8. CALGAVIN's hiTRAN® Thermal Systems matrix turbulator (Calgavin, 2022)

6.2.2 Twisted Tape Turbulator

A twisted tape turbulator is a clever device designed to boost the effectiveness of a water-cooled heat sink. It's a strip of metal, coiled into a spiral shape, which is placed in the coolant's path. This spiral shape forces the water to swirl as it flows, enhancing the heat transfer from the heat sink to the water. (Allen, 2020)

The tape's design, including its width and the tightness of the twist, is customized to match the heat sink's needs. This design ensures that the water doesn't just flow smoothly over the heat sink but moves in a more chaotic manner that increases contact and improves heat absorption. (Allen, 2020)

While a twisted tape turbulator can improve heat transfer efficiency, it also creates more resistance against the water flow, meaning the pump must work harder. So, it's important to consider the balance between the better heat transfer and the extra power needed for the pump. Despite the potential for increased pumping costs, the benefits of using a twisted tape are often significant, especially in systems that need to manage a lot of heat. (Allen, 2020)

Figure 9. Twisted tape turbulator (Allen, 2020)

6.2.3 Ball Turbulator

A ball turbulator is a dynamic device used to increase the efficiency of heat transfer within a water-cooled heat sink. It comprises numerous small, spherical balls, typically fashioned from conductive materials like metal or durable alternatives like plastic or ceramic. These turbulators are placed directly in the path of the cooling water, and as they move freely, they induce turbulence in the water flow. (Allen, 2020)

The motion of the balls disrupts the flow, creating pressure variations and turbulent eddies that enhance the mixing of the water, thus facilitating a more effective heat exchange between the water and the heat sink's surface. By promoting more intense and even heat absorption, ball turbulators can significantly bolster the performance of a water-cooled heat sink, particularly in systems dealing with high thermal loads or those requiring fast and efficient heat dissipation. (Allen, 2020)

However, it is critical to consider the trade-off introduced by ball turbulators, as they can lead to an increased pressure drop in the system. This may necessitate more powerful pumps or increased energy expenditure to sustain the desired flow rate. The selection of ball size and material, as well as the overall design of the turbulator system, should be optimized for the specific heat sink and application to ensure enhanced cooling without disproportionate increases in operational costs. (Allen, 2020)

Figure 10. Ball turbulator (Allen, 2020)

6.2.4 Spring Turbulator

Spring turbulators serve as an innovative means to elevate the efficiency of heat transfer in water-cooled heat sinks. These devices, typically small, coiled springs made of metal or plastic, are strategically placed within the flow path of the cooling water. Their presence induces a deliberate turbulence as the water maneuvers around them. (Allen, 2020)

This turbulent action incites a series of pressure changes and the formation of eddies, which in turn foster a more vigorous mix of the cooling water. The result is a more efficient thermal exchange from the heat source to the water, thereby optimizing the heat sink’s cooling performance. (Allen, 2020)

Spring turbulators are especially advantageous in scenarios where a moderate level of turbulence is desired. They stand out among other turbulators due to their lower likelihood of clogging from debris or scaling, promoting long-term efficiency and ease of maintenance. Nonetheless, the enhanced turbulence comes with an associated pressure drop, potentially necessitating more powerful pumps to uphold the cooling system's flow requirements. The design and implementation of spring turbulators must, therefore, consider the balance between improved heat transfer and the cooling system's operational demands. (Allen, 2020)

Figure 11. Ball turbulator (Allen, 2020)

7 CURRENT TURBULATOR MODEL

The currently used turbulators, featuring a matrix design as show in **Figure 8**, are a key component in the cooling system of the CH6x family of drives. Manufactured locally in Finland, these turbulators offer significant benefits, including reduced logistics costs and enhanced supply chain efficiencies, thereby bolstering the Finnish economy. Their competitive pricing has established these matrix turbulators as the obvious choice for use. However, the high performance of matrix turbulators comes with challenges in price, complexity, and difficulties in 3D modeling and efficient simulation. Despite the success and longstanding adoption of the current model, the search for alternative designs and manufacturers that can match or exceed the efficiency and cost-effectiveness of the current matrix turbulator is a critical task, especially in the face of supply chain challenges. The focus is on evaluating new materials and designs to potentially replace or supplement the existing model, thereby balancing innovation with economic practicality.

7.1 Positives of The Matrix Turbulator Against Other Designs

In exploring the advantages of the matrix turbulator over other turbulator designs, a key aspect to highlight is its high cooling performance. The matrix turbulator excels in achieving a balance between efficient cooling and acceptable flow and pressure loss. This balance is crucial for applications that demand high cooling efficiency without significantly affecting the system's overall flow dynamics. The design's ability to maintain this equilibrium makes it particularly valuable in cooling high power electronics.

Another significant advantage of the matrix turbulator is its status as a tried and tested model with CH6x family of drives. This long-standing reliability means that new productions of the matrix turbulator require less real-world testing compared to newer, less established designs. The reduced need for extensive testing not only saves time but also minimizes the resources required for validating performance, making it a more practical choice in terms of speed and efficiency of implementation.

The matrix turbulator also offers adjustable flow characteristics, a feature that adds to its versatility. The ability to modify the number of loops and the angle of loops within the turbulator allows for customization of flow properties to suit specific requirements. This adjustability is particularly beneficial in applications where fine-tuning of flow dynamics is essential. While this flexibility is an advantage, it's important to note that it could also introduce complexities, as will be discussed in the subsequent section on negatives.

Each of these advantages underscores why the matrix turbulator remains a preferred choice during this project, striking an effective balance between performance, reliability, and practicality.

7.2 Negative of The Matrix Turbulator Against Other Designs

One of the primary challenges with the matrix turbulator is its complexity in design and manufacturing. This complexity requires a higher degree of precision during production, potentially elevating production costs. The matrix turbulators are made using specialized, one-off machines that are expensive to fabricate and require specific skills to operate. This situation can limit the flexibility to adapt quickly to changes in design specifications or to meet custom requirements. While this complexity contributes to its effectiveness, it can pose significant challenges, especially in scenarios requiring rapid production or modifications.

Compared to alternatives like twisted tape and spring turbulators, the matrix turbulator presents greater challenges in accurate simulation and modeling due to its intricate design. These simpler designs are far easier to model and simulate in 3D, streamlining their development and testing processes. Consequently, the matrix turbulator demands more resource-intensive simulations, extending the time and cost involved in the development phase, potentially slowing down the innovation cycle.

The complex design and manufacturing process of the matrix turbulator might also result in higher upfront costs compared to simpler designs like twisted tape and spring turbulators. This factor can be a major consideration for projects with constrained budgets or those that require a large number of turbulators. Although the long-term benefits of the matrix design might offset these initial costs, the immediate financial impact is an important consideration in cost-sensitive projects.

Additionally, the specific design of the matrix turbulator might limit its versatility in certain applications. Despite its adjustability, there could be scenarios where simpler turbulators provide better performance or easier integration. This limitation is particularly relevant in specialized applications where different designs, like the straightforward twisted tape or spring turbulators, might be more advantageous.

Lastly, concerns regarding maintenance and longevity are notable with the matrix turbulator. Its intricate structure may pose challenges in regular maintenance and servicing, particularly in environments that demand frequent cleaning. The complexity of the design might also affect its longevity, as more intricate components are susceptible to wear and tear over time.

Understanding these negatives is crucial for a comprehensive assessment of the matrix turbulator’s overall suitability compared to other designs. This section aims to provide insights into areas where the matrix turbulator might fall short, guiding future improvements or the development of alternative technologies.

8 SIMULATION

8.1 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) simulation is a technique used to analyze and solve complex fluid flow problems using numerical methods and algorithms. It is a powerful tool used in engineering and science to simulate and analyze fluid flow, heat transfer, and other related phenomena. (Anderson, 1995; Ansys, n.d.-b)

In CFD simulation, the fluid flow is modeled using mathematical equations that describe the conservation of mass, momentum, and energy. These equations are solved using numerical methods on a computer, and the results are presented in the form of visualizations and graphs. (Anderson, 1995)

CFD simulation can be used to model a wide range of fluid flow problems, including turbulent flow, laminar flow, compressible flow, and multiphase flow. It can be used to analyze and optimize the design of a wide range of devices and systems, such as aircraft wings, heat exchangers, pumps, and combustion chambers. (Anderson, 1995)

CFD simulation can provide a wealth of information about fluid flow, including velocity profiles, pressure distributions, and temperature distributions. It can also provide insight into complex phenomena, such as vortex shedding, boundary layer separation, and turbulence. (Anderson, 1995)

Overall, CFD simulation is a powerful tool for analyzing and solving complex fluid flow problems in a wide range of engineering and scientific applications. It can provide valuable insights into fluid flow behavior and help engineers optimize the design of devices and systems to improve their performance and efficiency.

Figure 12. CDF simulation of straight channel.

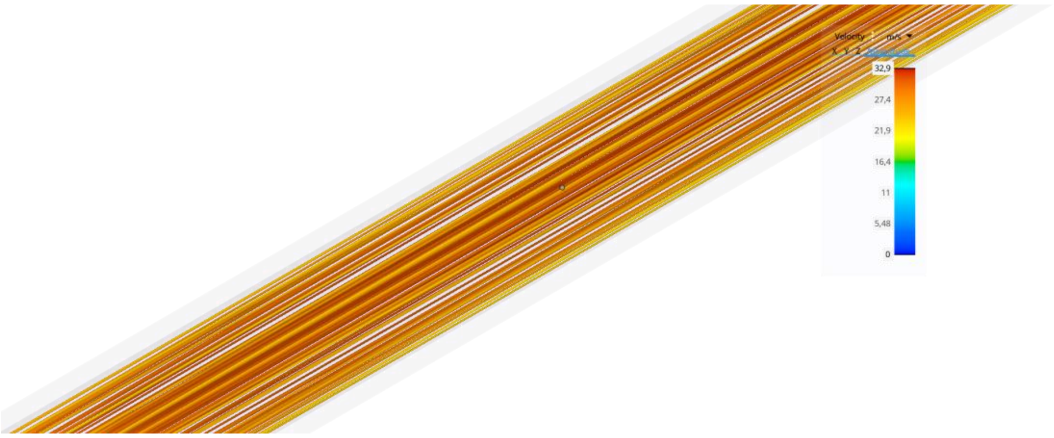


Figure 13. Close up of the streamlines displaying flow.

8.2 Finite Element Method

The Finite Element Method (FEM) is a numerical technique used to solve a wide range of engineering and scientific problems, including structural analysis, heat transfer, fluid mechanics, and electromagnetics. It involves breaking down a complex system or structure into smaller, more manageable components called finite elements, which are then analyzed using mathematical equations and numerical methods. (Ansys, n.d.-a; Ottosen & Petersson, 1992)

In FEM simulation, the structure or system being analyzed is modeled using a mesh of interconnected finite elements. Each element is defined by a set of mathematical equations that describe its behavior, and these equations are combined to form a system of equations that describe the behavior of the entire system. (Ottosen & Petersson, 1992)

The equations are solved numerically using a computer, and the results are presented in the form of visualizations and graphs. The FEM simulation can provide information about stress and strain distributions, deformation, temperature distributions, and other related phenomena. (Ottosen & Petersson, 1992)

FEM simulation is a powerful tool that can be used to optimize the design of structures and systems, predict their behavior under different conditions, and identify potential problems and failure modes. It can be used to analyze a wide range of structures and systems, including bridges, buildings, aircraft wings, heat exchangers, and electronic components. (Ottosen & Petersson, 1992)

Overall, FEM simulation is a valuable tool for engineers and scientists, providing a way to analyze and optimize complex systems and structures, and to gain insight into their behavior under different conditions. It is widely used in industry, research, and academia, and it has revolutionized the way that engineers and scientists approach problem-solving and design optimization.

8.3 Flow Distribution in The Heat Sink.

The distribution of flow in the heat sink gives valuable information about the behavior of the flow of water in the heat sink and its effects on the cooling efficiency of the cooling system. Key parameters are uniformity of the flow distribution and its effect on pressure drop across the heat sink. The flow in distribution in manifolds is a commonly encountered process in heat sinks, fuel cells and irrigation. The used heat sink is basically a U type manifold that is commonly encountered manifold type. (McNown, 1954)

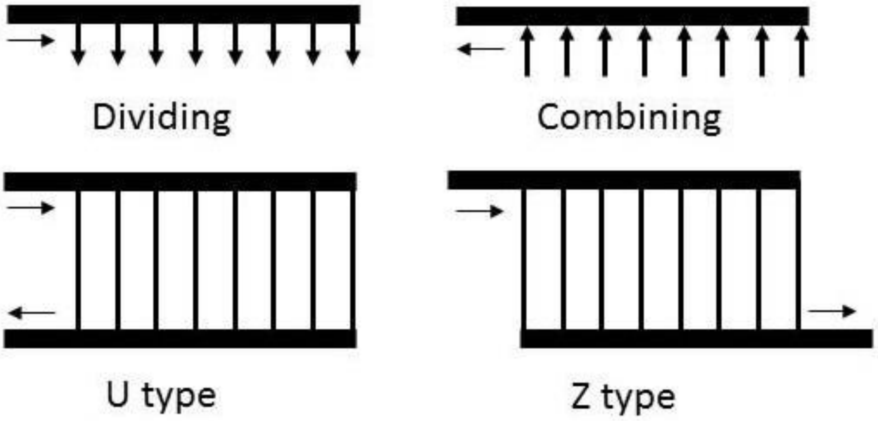
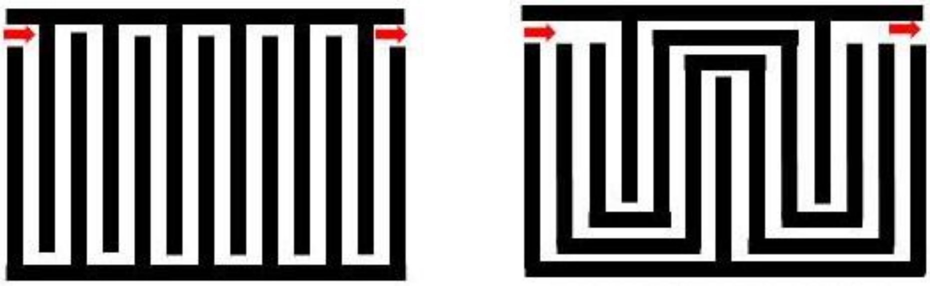
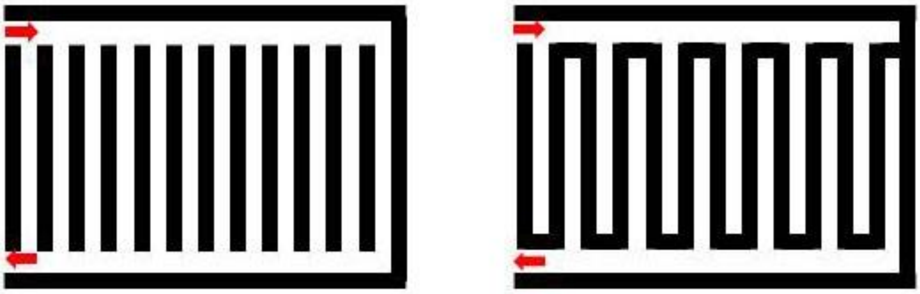


Figure 14. Picture explaining dividing, combining, U type and Z type manifolds. (Wang, 2013b)



a) Single serpentine

b) Multiple serpentine with three channels



c) Straight parallel

d) Interdigitated

Figure 15. Pictures of few common manifold configurations. (Wang, 2013a)

8.3.1 Simulation of The Heat Sink Body for Flow Distribution.

The flow rate of every cooling channel was measured at the end of it. This setup enables precise simulation of flow, velocity, and pressure for each cooling channel, based on a known inlet flow condition.

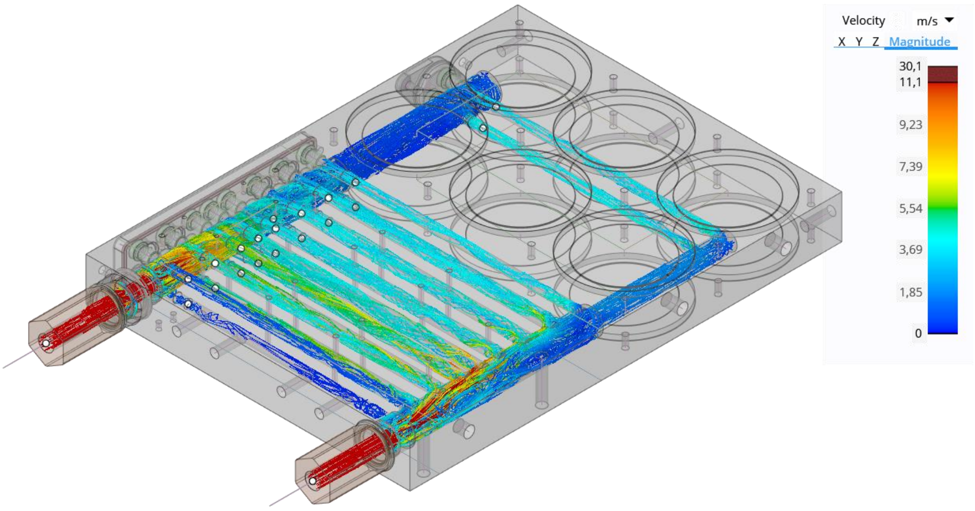


Figure 16. Simulated flow distribution in the heat sink body, with white dots marking the points of flow measurement within the channels.

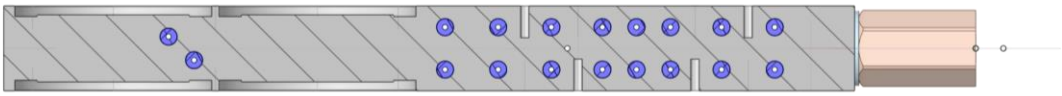


Figure 17. Eighteen flow channels where the turbulators are installed.

Table 1. Flow distribution across the eighteen cooling channels in the heat sink.

4.8%	5.0%	5.3%	5.2%	5.2%	6.3%	7.3%	7.2%	2.1%
5.2%	5.2%	5.5%	5.7%	5.9%	6.6%	7.3%	6.7%	3.4%

Figure 16 and **Figure 17** illustrates the simulation of flow distribution within the heat sink body, showing the eighteen flow channels where turbulators are installed. The flow percentages across these channels are outlined in **Table 1** in same positions as channels are seen in **Figure 17**.

The simulation data reveals a non-uniform flow distribution. Channels nearest to the inlet exhibit lower flow rates and velocities compared to their downstream counterparts. The central channels display the highest flow rates. Although this variation exists, it is less significant than initially predicted and is unlikely to critically impact the overall cooling performance of the heat sink. Therefore, this minor flow imbalance does not justify altering the current turbulator design to account for it.

While optimizing thermal management is paramount, and strategies like flow equalization or adjusting turbulator placement could significantly improve uniformity and heat transfer efficiency, such optimizations are not feasible within the scope of this project due to time and resource constraints. Nonetheless, understanding their potential advantages, we will continue with the current turbulator design specifications. Future projects, especially those involving new designs of cooling systems and with more extensive resources and time allowances, might provide opportunities to investigate and implement these advanced thermal management strategies.

8.4 Turbulators

The investigation examines the influence of varied turbulator styles on the flow and pressure in a channel. By assessing common configurations such as matrix, twisted tape, ball, and spring turbulators, the study aims to quantify the pressure drop they induce and the corresponding changes in flow velocity. The findings are instrumental for enhancing heat sink and turbulator designs, where controlled flow velocity and pressure are critical for optimal cooling performance.

8.4.1 Matrix Turbulator

The simulation results for the Matrix turbulator reveal significant findings regarding pressure loss and flow velocity. Across the 200 mm length of the turbulator, a consistent pressure loss of 24% was observed, regardless of the inlet pressure, which ranged from 2 bar to 8 bar. This indicates a stable pressure reduction performance of the Matrix turbulator within the tested pressure range. Additionally, the study noted a 12% increase in flow velocity across the same pressure range. This increase remained constant from 2 bar to 8 bar, suggesting that the turbulator effectively accelerates flow without dependence on the varying inlet pressures. These results are crucial for understanding the Matrix turbulator's efficiency and effectiveness in modulating flow and pressure in a controlled environment.

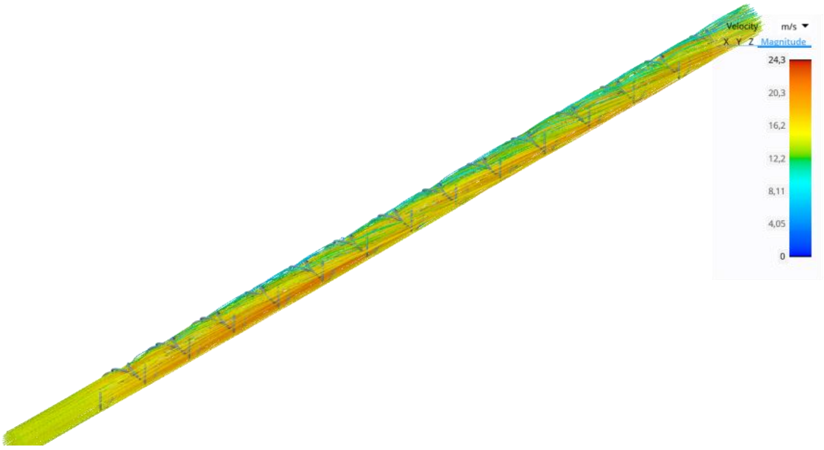


Figure 18. CDF simulation of matrix turbulator in straight channel.

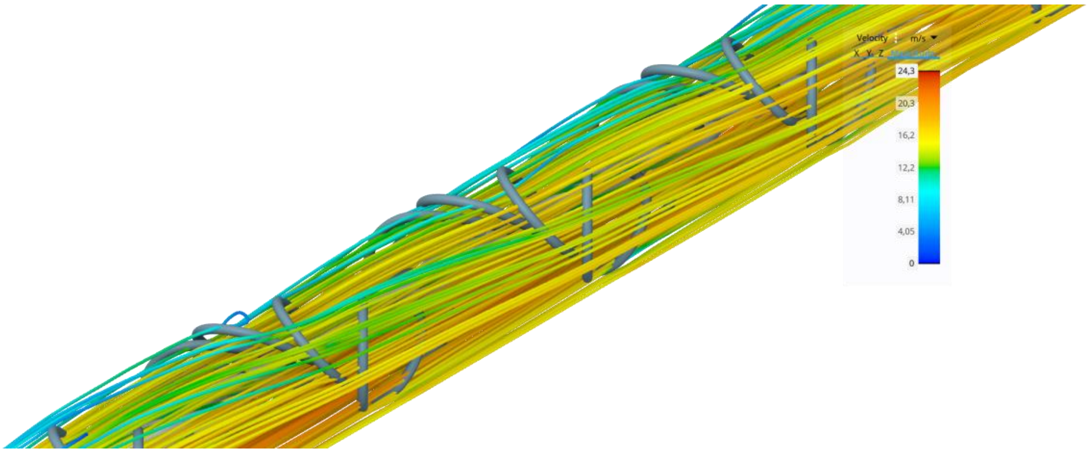


Figure 19. Close up of the streamlines displaying flow.

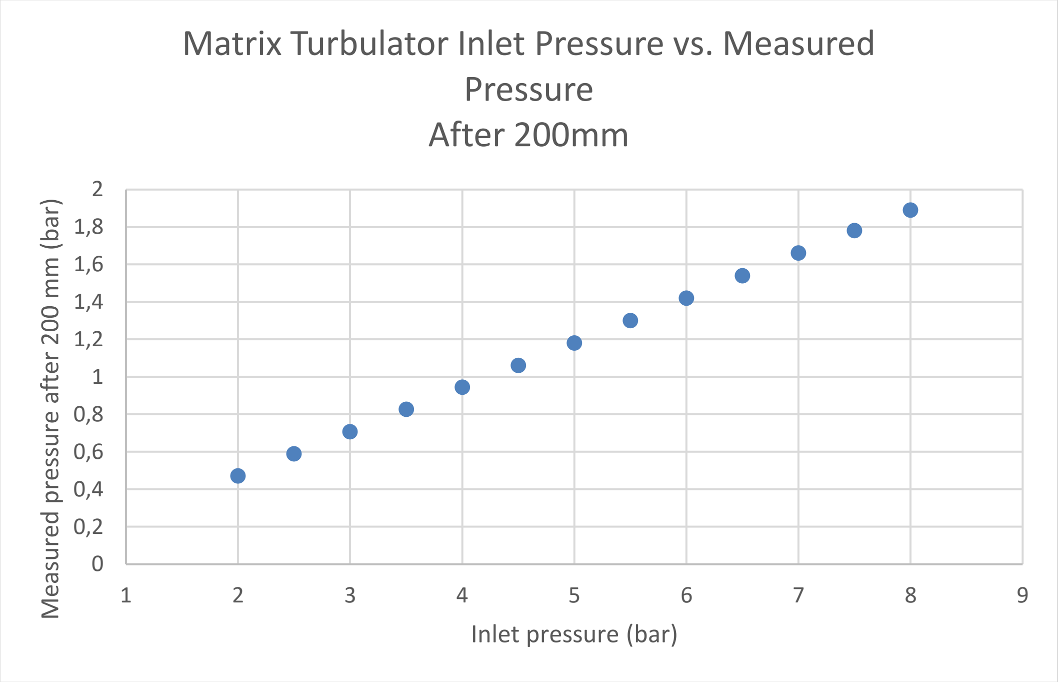


Figure 20. Matrix Turbulator Inlet Pressure vs. Measured Pressure

8.4.2 Twisted Tape Turbulator

The simulation of the Twisted Tape turbulator, designed with a single twist per 100 mm length, yielded significant findings in terms of pressure loss and flow velocity. The pressure loss across a 200 mm section of the turbulator ranged from 53% to 56%, showing remarkable consistency across a wide range of inlet pressures from 2 bar to 8 bar. This indicates a substantial and uniform reduction in pressure, highlighting the efficiency of the Twisted Tape design in creating resistance within the flow.

In terms of flow velocity, there was a moderate increase observed, ranging from 10% to 11%, which remained constant regardless of the varying inlet pressures. This consistent increase in velocity demonstrates the Twisted Tape turbulator's capacity to enhance flow dynamics within the specified parameters.

It is important to note that this simulation focused on a plain Twisted Tape design without any additional modifications. While other studies have explored the potential for increased efficiency with the incorporation of holes or cuts on the edges of the Twisted Tape, these variations were not included in the current simulation. This leaves scope for future research to examine the effects of such modifications on the turbulator's performance.

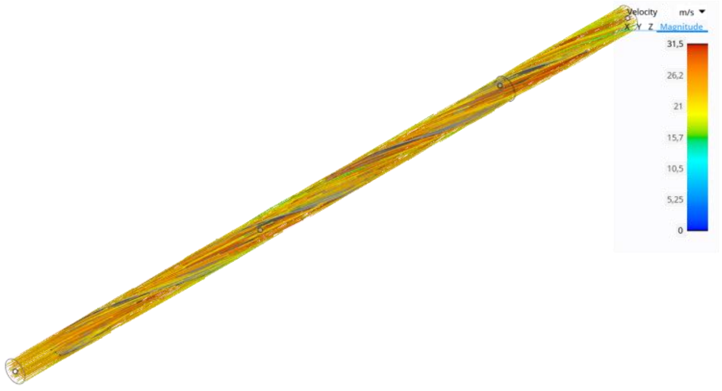


Figure 21. CDF simulation of twisted tape turbulator in straight channel.

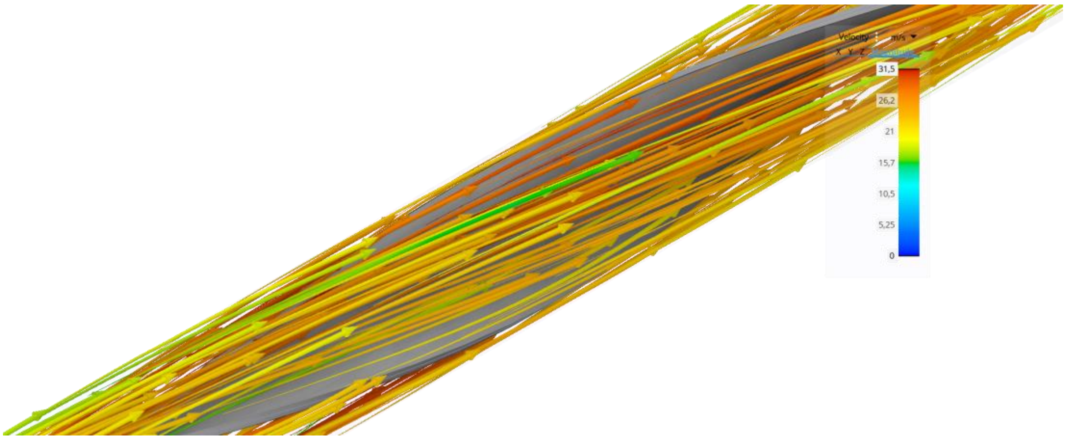


Figure 22. Close up of the streamlines displaying flow.

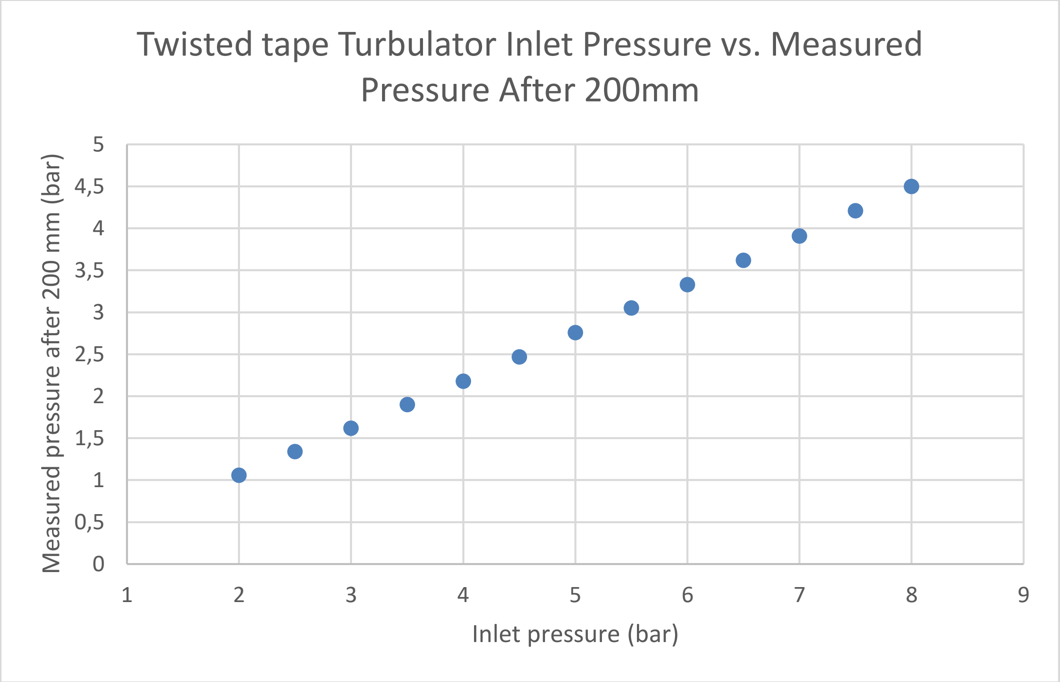


Figure 23. Twisted tape Turbulator Inlet Pressure vs. Measured Pressure

8.4.3 Ball Turbulator

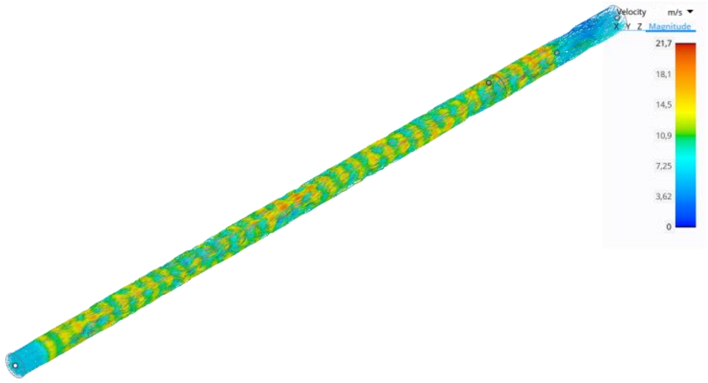


Figure 24. CDF simulation of ball turbulator in straight.

The simulation of the ball turbulator, featuring a 6 mm balls within an 8 mm flow channel, offered key insights into its impact on the coolant flow. The pressure loss was consistently observed to be between 14% to 17% across a 200 mm segment, irrespective of the inlet pressure ranging of 2 bar to 8 bar. This indicates a stable effect on the pressure by the Ball turbulator under varying conditions.

A notable increase in flow velocity, up to 100%, was consistent across the inlet pressures. This substantial rise is mainly due to the measurement point's proximity to the 6 mm ball obstructing the 8 mm channel. When the flow velocity was measured away from the ball's immediate influence, it showed a significant yet lower increase of about 70%, accompanied by a pronounced wavy flow velocity pattern, as clearly depicted in the simulation picture from **Figure 24** to **Figure 27**

An important aspect highlighted by the simulation is the potential for customizing the turbulator system by varying the size of the ball. Using balls of different sizes could provide a means to fine-tune the system, adjusting both pressure loss and flow velocity according to specific requirements. However, the simulation did not account for additional structures necessary to stabilize the ball's position in the channel, indicating the need for further design considerations to ensure the practical viability of this turbulator configuration.

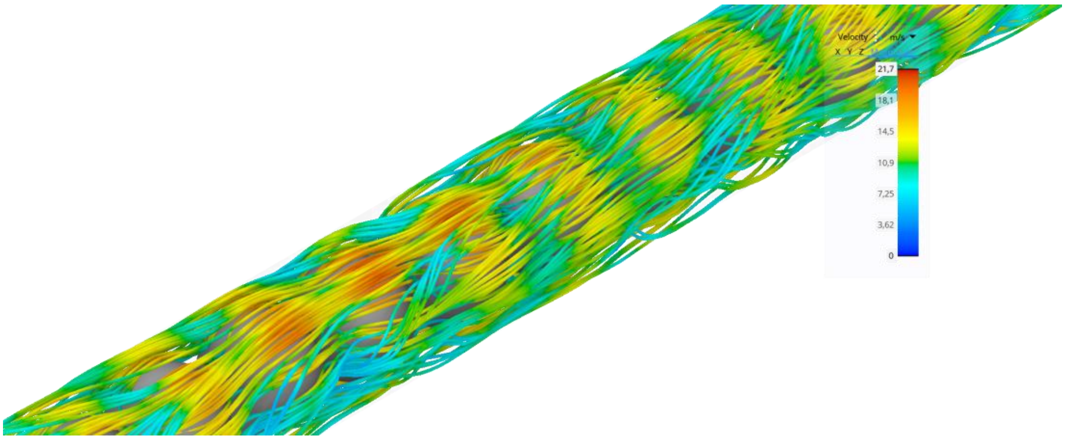


Figure 25. Close up of the streamlines displaying flow.

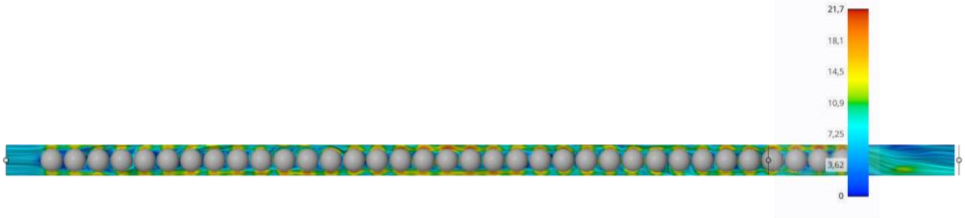


Figure 26. Direction field display of ball turbulator in straight channel.

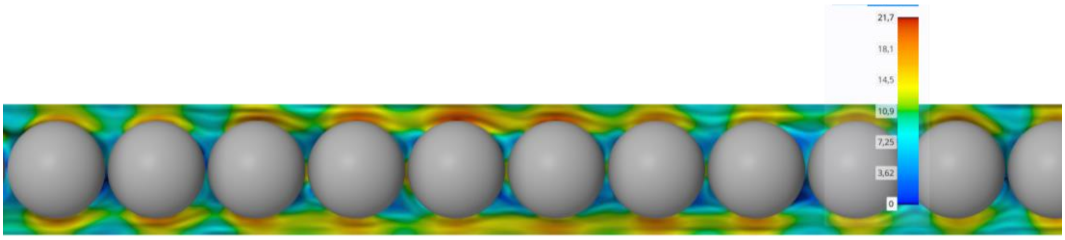


Figure 27. Close up of the direction field display of the flow.

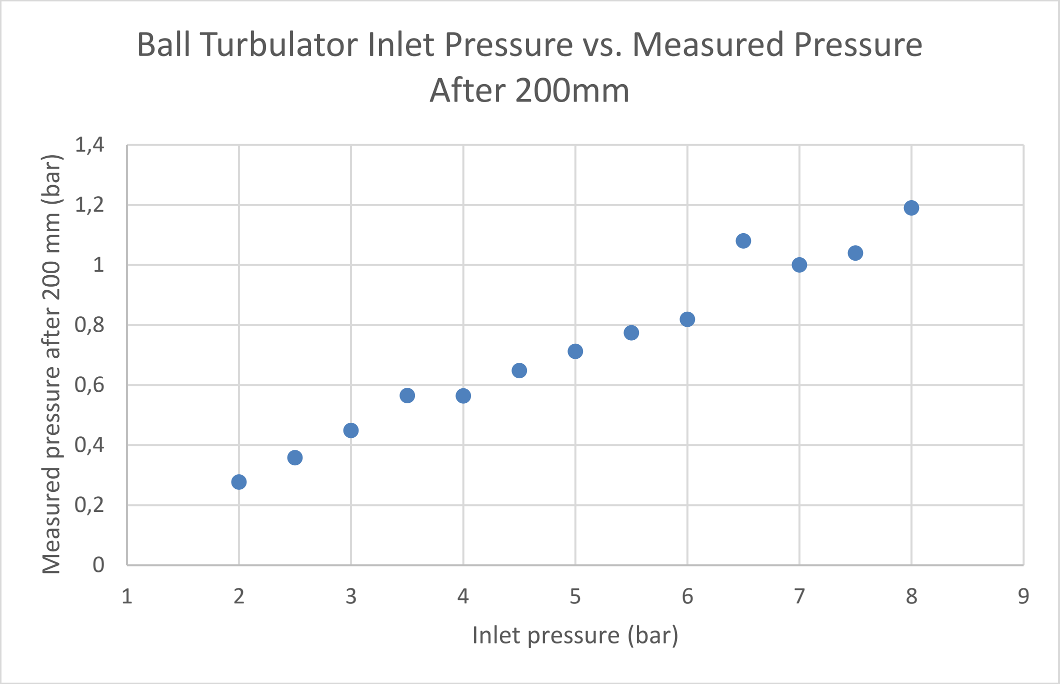


Figure 28. Ball Turbulator Inlet Pressure vs. Measured Pressure

8.4.4 Spring Turbulator

The simulation of the Spring turbulator yielded noteworthy results in terms of pressure loss and flow velocity changes. Across the 200 mm section of the turbulator, a consistent pressure loss of 13% was recorded, which remained unchanged for inlet pressures ranging from 2 bar to 8 bar. This consistent performance across a wide range of pressures highlights the Spring turbulator's stable pressure reducing capabilities. In terms of flow velocity, there was an observed increase of 7 to 8%, which also remained constant across the tested inlet pressure range from 2 bar to 8 bar. These findings demonstrate the Spring turbulator's effectiveness in moderately enhancing flow velocity without being significantly influenced by varying inlet pressures.

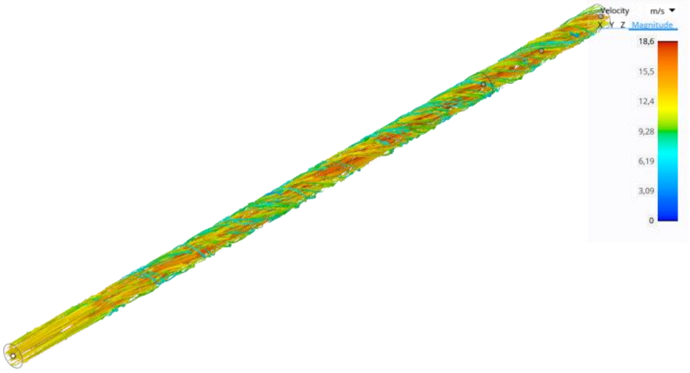


Figure 29. CDF simulation of spring turbulator in straight channel.

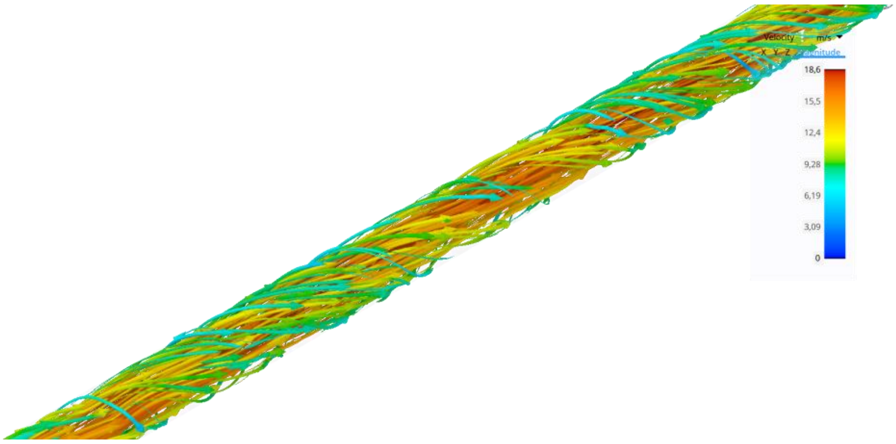


Figure 30. Close up of the streamlines displaying flow.

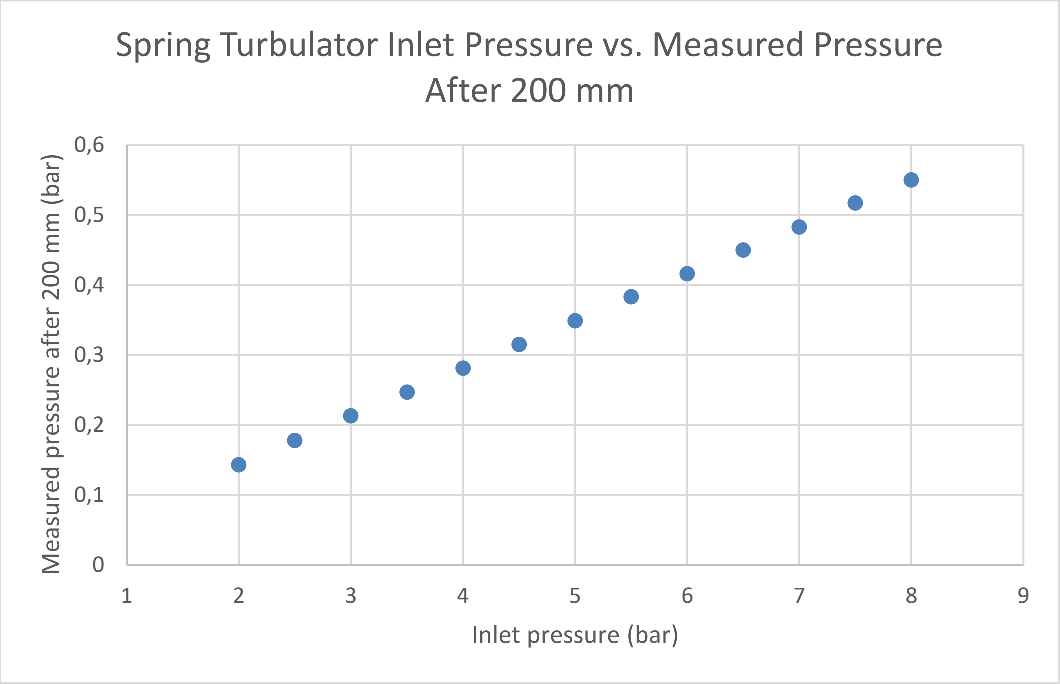


Figure 31. Spring Turbulator Inlet Pressure vs. Measured Pressure

9 REAL WORLD TEST DRIVES FOR TURBULATORS

While real-world testing of turbulators would have been a valuable addition to the thesis, practical considerations ultimately led to its exclusion from the scope. During the planning phase, real-world testing was considered a 'nice-to-have' feature rather than a core requirement, with the understanding that it would be pursued if time and resources permitted. However, as the research progressed, it became evident that the challenges associated with parts availability were more daunting than anticipated. The task of obtaining the alternative turbulators necessary for meaningful testing was not only complex but also threatened to introduce significant delays.

Moreover, the specific testing capabilities and infrastructure necessary to simulate real-world conditions were not within reach. Access to such specialized equipment and facilities often involves considerable expense and logistical planning, which were constraints that could not be overlooked. In addition to these practical problems, the chosen turbulator was manufactured with same drawings to same specification to one already used with a proven history in the field further diminished the justification for real-world testing. Given that the performance of this turbulator design had already been validated through extensive prior testing, additional real-world tests would have been interesting but not necessary.

The decision to forgo real-world testing was driven by a strategic choice to allocate resources efficiently and adhere to the project's timeline. It was a move to focus on leveraging existing data and research, rather than investing in an optional testing phase that would not fundamentally alter the thesis's conclusions. This decision ensured that the research stayed within the defined practical boundaries and timelines, emphasizing the most critical aspects of the study while acknowledging the limitations imposed by the circumstances.

10 CONCLUSIONS

The primary goal of this thesis was to identify an alternative turbulator design and manufacturer to replace the existing model, which may face future unavailability. Additionally, the study aimed to deepen the understanding of Computational Fluid Dynamics (CFD) simulations in the context of electronics cooling.

A significant achievement of this project was the discovery of a manufacturer capable of producing a turbulator design that matches the specifications of the older model without notably decreasing cooling capacity, increasing cost, and with good availability. This outcome is especially relevant in ensuring the availability of a key cooling system component amidst a potential future scarcity of the current model. The matching specifications also suggest a reduced need for extensive testing at this stage in the life cycle of the CH6x family of drives.

Furthermore, this research provided valuable insights into the application of CFD simulations in evaluating and predicting the performance of turbulator designs, along with enhancing the use of simulation software for the common design engineer. While the new turbulator did not demonstrate a marked improvement in cooling efficiency, the process of using CFD simulations enhanced the understanding of fluid dynamics in cooling systems, crucial for future innovations in this field.

The limitations of the study, particularly in long-term performance assessment and tests under varied conditions, suggest areas for future research. Future studies could focus on exploring innovative turbulator designs that push beyond current specifications and utilize advanced CFD simulations for comprehensive performance assessment. It is important to note that the turbulators studied represent older technology, with newer designs incorporating integrated elements for cooling.

During this project, a thorough search for manufacturers was conducted across several countries, including Finland, England, Italy, India, and China. It is important to recognize that turbulators, particularly of the matrix design, are highly specialized and complex components, with only a handful of companies worldwide that specialize in their production. This rarity adds a layer of complexity to the task of finding suitable manufacturers. This global exploration led to a promising potential partnership with a manufacturer in Italy, capable of producing turbulators to the same specifications as the old model. The process of validating this new manufacturer and the turbulator design is still ongoing, but early indications are promising. This international effort, in the context of a niche and complex component like the turbulator, underscores the importance of global collaboration and innovation in meeting the evolving needs of the electronics cooling industry.

In conclusion, this thesis not only identified a viable alternative to existing turbulator designs but also contributed to the broader knowledge of CFD simulations in electronics cooling. These findings are instrumental in preparing for future challenges in cooling system component availability and advancing the field's understanding of simulation-based design evaluation.

REFERENCES

Allen, K. (2020, July 4). *Turbulators in Heat Exchangers: Types and Purposes*. https://www.superradiatorcoils.com/blog/heat-exchanger-turbulators-types-and-purposes

Anderson, J. D. (1995). *Computational Fluid Dynamics: The Basics with Applications*. McGraw-Hill.

Ansys. (n.d.-a). *Ansys Mechanical | Structural FEA Analysis Software*. Retrieved 15 December 2023, from https://www.ansys.com/products/structures/ansys-mechanical

Ansys. (n.d.-b). *CFD Software: Fluid Dynamics Simulation Software*. Retrieved 15 December 2023, from https://www.ansys.com/products/fluids

Cadence. (2023). *The Differences Between Laminar vs. Turbulent Flow | Cadence*. https://resources.system-analysis.cadence.com/blog/msa2022-the-differences-between-laminar-vs-turbulent-flow

Calgavin. (2022). *hiTRAN® Thermal Systems*. https://www.calgavin.com/products-and-software/hitran-thermal-systems

Chang, P. K. (1970). *Separation of Flow*. Pergamon. https://doi.org/10.1016/B978-0-08-013441-3.50005-8

Concept Group LLC. (n.d.). Retrieved 16 December 2023, from https://conceptgroupllc.com/

Kordyban, T. (1998). *Hot Air Rises and Heat Sinks: Everything You Know about Cooling Electronics Is Wrong*. ASME Press. https://doi.org/10.1115/1.800741

Kraus, A. D., & Bar-Cohen, A. (1983). *Thermal Analysis and Control of Electronic Equipment*.

Lienhard, J. H., IV, & Lienhard, J. H., V. (2019). *A Heat Transfer Textbook* (5th ed.). Dover Publications. http://ahtt.mit.edu

McGraw-Hill Higher Education. (1998). *Convection Heat Transfer*. https://www.mhhe.com/engcs/mech/cengel/notes/ConvectionHeat-Transfer.html

McNown, J. S. (1954). Mechanics of Manifold Flow. *Transactions of the American Society of Civil Engineers*, *119*, 1103–1118.

Molana, M., & Banooni, S. (2013). Investigation of Heat Transfer Processes Involved Liquid Impingement Jets: A Review. *Brazilian Journal of Chemical Engineering*, *30*, 413–435. https://doi.org/10.1590/S0104-66322013000300001

Ottosen, N. S., & Petersson, H. (1992). *Introduction to the Finite Element Method*. Prentice Hall.

Schobeiri, M. T. (2014). Laminar-Turbulent Transition. In *Applied Fluid Mechanics for Engineers* (First edition.). McGraw-Hill Education. https://www.accessengineeringlibrary.com/content/book/9780071800044/chapter/chapter8

Schulz, M. (2019). *IGBT basic know how*.

Semikron-Danfoss. (2017). *Technical Explanation SKiiP®3 V3*. Semikron Danfoss.

Semikron-Danfoss. (2024). *IGBT Modules | Semikron Danfoss*. https://www.semikron-danfoss.com/products/product-classes/igbt-modules.html

Shah, R. K., & Sekulić, D. P. (2003). *Fundamentals of heat exchanger design*. John Wiley & Sons Hoboken, NJ.

Thompson, M. T. (2007). Notes 01 Introduction to Power Electronics. *Power Electronics*.

Wang, J. (2013a). *Different configuration*. https://commons.wikimedia.org/wiki/File:Manifold_Configuration01.jpg

Wang, J. (2013b). *Manifold arrangement for flow distribution*. https://commons.wikimedia.org/wiki/File:Manifold_arrangement.jpg

What do Capacitors do in VFDs? (n.d.). Retrieved 15 December 2023, from https://www.dosupply.com/tech/2022/05/12/what-do-capacitors-do-in-vfds/

What is Jet Impingement Cooling and How is it applied for Thermal Management of Electronics (Part 1 of 2) | Advanced Thermal Solutions. (n.d.). Retrieved 15 December 2023, from https://www.qats.com/cms/2010/09/13/what-is-jet-impingement-cooling-and-how-is-it-applied-for-thermal-management-of-electronics-part-1-of-2/

What is Pressure Drop? (n.d.). Retrieved 15 December 2023, from https://www.csidesigns.com/blog/articles/what-is-pressure-drop-and-how-does-it-affect-your-processing-system

Zhao, J., Wang, Y., Ding, G., Sun, C., & Wang, G. (2014). Design, fabrication and measurement of a microchannel heat sink with a pin-fin array and optimal inlet position for alleviating the hot spot effect. *Journal of Micromechanics and Microengineering*, *24*. https://doi.org/10.1088/0960-1317/24/11/115013