



Validated CFD modelling of an air-cooled motor controller heatsink in electric sailplane applications

L Kirchberger

 **orcid.org/ 0000-0002-8817-1714**

Dissertation accepted in fulfilment of the requirements for the degree *Master of Engineering in Mechanical Engineering* at the North-West University

Supervisor: Prof JJ Bosman

Graduation: July 2026

Acknowledgements

This journey would not have been possible without the love and support of the people who mean the most to me.

To my girlfriend, Chantel - thank you for being my anchor. Your patience, encouragement, and constant belief in me made even the most difficult days feel lighter. You've celebrated every small win with me and reminded me of what truly matters when I lost perspective. I am deeply grateful for your love and for the quiet strength you bring into my life.

To my family - your steady presence and unconditional support gave me the courage to keep moving forward. You've always been the people I could lean on, no matter the challenge, and I carry with me the values you've instilled. I would not be who I am without you.

To my friends - thank you for reminding me that life is bigger than deadlines and obstacles. The laughter, conversations, and moments we've shared brought balance and joy when I needed it most. Your friendship has been a gift throughout this journey and hopefully longer.

To my supervisor, Bossie - your guidance shaped not only this project but also the way I think and work. Our many conversations, both serious and lighthearted, kept me focused yet reminded me to approach challenges with perspective. I could not have asked for a better mentor.

And to my loyal dog - thank you for the unspoken comfort of companionship. The small moments of joy, energy, and peace you brought into my days mattered more than words can describe.

I am incredibly fortunate to be surrounded by such remarkable people. Their belief in me, in ways big and small, has made all the difference. This achievement is as much theirs as it is mine.

"I am not a product of my circumstances. I am a product of my decisions - and the guidance of those who believed in me."

— Adapted from Stephen R. Covey

Abstract

This thesis presents a validated three-dimensional computational fluid dynamics (CFD) investigation of the air-cooled motor-controller heat sink used in electric sailplane applications, with specific focus on the JS3 RES system. The objective was to establish a reliable, certification-relevant simulation framework capable of accurately predicting thermal behaviour and supporting future design improvements.

A conjugate heat transfer (CHT) approach was employed to capture the interactions between the heat sink, the integrated heat pipes, and the external fuselage airflow. Heat pipes were modelled as high-conductivity solid rods, providing a computationally efficient yet physically representative approximation of lateral heat spreading. Commercial CFD tools (STAR-CCM+) were used, with careful selection of material properties, boundary conditions, and solver settings to ensure model fidelity. Experimental validation was performed using instrumented ground tests, with air-density corrections applied to enable consistent comparison between measured and simulated temperatures.

The results demonstrate that the CFD model reproduces the thermal behaviour with high accuracy, deviating by less than 1% across the entire verification range of a benchmark test. This deviation is smaller than those observed in both experimental measurements and comparable simulations performed in SimScale. It therefore falls well within the acceptable limits typically expected for validated CFD simulations when compared against theoretical results. Key findings include the influence of mesh resolution on numerical accuracy, the critical role of heat pipes in lateral heat transport, and the sensitivity of validation outcomes to sensor placement and experimental precision.

Beyond the JS3 RES application, this work establishes a validated CFD baseline suitable for parametric and topological optimisation, targeted thermal design improvements, and the development of efficient cooling strategies for other small-scale electric aircraft systems. The methodology and results contribute to aerospace engineering by providing a reliable, simulation-driven approach for the design and evaluation of compact thermal management systems.

Keywords: CFD; Conjugate Heat Transfer; Air-Cooled Heat Sink; STAR-CCM+; Thermal Management; Electric Sailplanes; Aerospace Engineering

Ethical Use of Artificial Intelligence

In accordance with the *NWU Guidelines on the Responsible and Ethical Use of Artificial Intelligence* (as of 11 March 2024 North-West University (2024)), I declare that the use of artificial intelligence (AI) in this work complies with the principles of transparency, honesty, integrity, accountability, privacy, and proper referencing.

Throughout this thesis, generative AI tools, including large language models (LLMs) such as ChatGPT, were used in the following limited and responsible ways:

- **Drafting and structuring support:** Assistance was sought during the early structuring phases of certain sections to brainstorm appropriate headings and improve clarity and conciseness of argumentation.
- **Proofreading and language refinement:** AI tools were utilised exclusively for grammatical and stylistic feedback. Rewriting occurred only where needed and all suggested changes were critically reviewed and finalised by me.
- **Formatting and $\LaTeX$ assistance:** AI tools were consulted for guidance on $\LaTeX$ syntax and document structuring, without replacing any scientific content.
- **Literature verification and referencing support:** While AI-assisted tools provided recommendations, all references included in this thesis were manually verified using scholarly databases to ensure correctness and credibility.

All outputs from AI tools were critically reviewed, edited, and contextualised by me, the author, to ensure accuracy, academic rigor, and alignment with the learning outcomes of the programme. No AI-generated content has been submitted as my own without substantial human oversight, revision, and critical engagement.

My supervisor is fully aware of the extent to which AI tools were used during the research and writing of this thesis. The use of AI was discussed openly and received explicit approval in accordance with the ethical research practices outlined by the university.

I take full accountability for the integrity and accuracy of all content contained herein. All sources, including AI-assisted contributions, have been acknowledged appropriately. The use of AI in this research adheres to NWU's policy on academic integrity and responsible innovation.

Declaration: I, the undersigned, confirm that this work has been completed in accordance with the *NWU Guidelines on the Responsible and Ethical Use of Artificial Intelligence*, and I take full ownership of its contents.

Student Name

20/05/2026
Date

Contents

1	Introduction	1
1.1	Background	1
1.1.1	Electric Sailplane Context	1
1.1.2	Propulsion System and Thermal Challenge	2
1.1.3	Thermal Management System	3
1.2	Certification and Operating Requirements	4
1.3	Thermal Behaviour and System Limitations	5
1.4	Design Evolution and Motivation	6
1.5	CFD Modelling Context	6
1.6	Problem Description	6
1.7	Objectives	7
2	Literature review	8
2.1	Introduction	8
2.2	Principles of Heat Sink Design	8
2.3	Heat transfer	9
2.3.1	Conduction	10
2.3.2	Convection	10
2.3.3	Radiation	11
2.4	Material selection	11

2.4.1	Aluminium	11
2.4.2	Copper	12
2.4.3	Advanced Materials	13
2.5	Thermal Analysis and Simulation	13
2.5.1	Computational Fluid Dynamics (CFD)	14
2.5.2	Thermal Analysis Software (TAS)	16
2.5.3	Differences between CFD, CHT, and TAS	18
2.5.4	Siemens Star CCM+	19
2.6	Different Modelling Approaches	20
2.6.1	Full 3D Analysis	20
2.6.2	Thin-Walled Approach	22
2.6.3	Porous Media Model	23
2.6.4	Baffles Approach	25
2.6.5	Conjugate Heat Transfer (CHT) Model	27
2.6.6	Dimensionality Reduction Approach	28
2.6.7	Summary and Selection Criteria	29
2.7	Innovations in Heat Sink Design	29
2.7.1	3D Printing	30
2.7.2	Advanced Materials	30
2.7.3	Applications	30
2.8	Comparative Analysis of Cooling Systems	31
2.8.1	Air-Cooled vs. Water-Cooled Systems	31
2.8.2	Comparative Studies on Fin Geometries and Orientations	32
2.8.3	Applications	33
2.9	Heat Pipe and Vapour Chamber Technology and Numerical Modelling	33
2.9.1	Vapour Chambers	34

2.9.2	CFD / Numerical Modelling Approaches	35
2.9.3	Research Gaps and Motivation	35
2.10	Literature Review Summary	36
2.11	Key Insights	37
2.12	Future Directions	37
2.13	Chapter Conclusion	38
3	Methodology	40
3.1	Overview	40
3.2	Full 3D Approach	41
3.3	CFD Modelling	42
3.3.1	Numerical Implementation and Initial Setup	49
3.3.2	CFD Simulation Workflow	50
3.3.3	Mesh Independence Study (MIS) - Full 3D	50
3.3.4	Material Properties and Thermal Interactions	63
3.3.5	Environmental Conditions and Atmospheric Modelling	65
3.3.6	Summary	67
4	Simulation Setup, Implementation, and Comparison	68
4.1	Initial Material Property Selection and Conduction Analysis	69
4.1.1	Flow Domain	69
4.1.2	Heat Sink	70
4.1.3	Heat Pipes	70
4.1.4	Heat Source	71
4.2	Initial Thermal Interface Analysis (Thermal Resistance)	71
4.2.1	Heat Pipes to Heat Sink	71
4.2.2	Heat Source to Heat Sink	72

4.3	Verification and validation	72
4.4	Benchmark Verification	73
4.4.1	Selection of Appropriate Benchmark Test	74
4.4.2	CFD Simulation Results vs. Benchmark Data	74
4.4.3	Refinement of CFD Simulation Based on Benchmark	87
4.5	Validation with physical data	88
5	Results and Discussion	92
5.1	Interpretation of Results	92
5.1.1	Simulation with 40 kW and 25 kW Battery Power	92
5.1.2	Model Refinements to Address Oscillating Residuals	95
5.1.3	937.5 W Heat Influx Case	97
5.1.4	Temperature Distribution on the Heat Sink Fins	99
5.1.5	Reasons for Differences	102
5.2	Comparison with Tested Data	103
5.2.1	Practical Test Setup	103
5.2.2	Simulation Results	105
5.2.3	Conclusion of Practical Results vs Simulation	106
6	Conclusion	108
6.1	Summary of Findings	108
6.2	Contributions to Aerospace Engineering	111
6.3	Recommendations for Future Research and Projects	112
A	Further Simulation Data Not Shown in the Main Body	117
A.1	Preliminary Simulation – Additional Information	117
A.1.1	Mesh	118
A.1.2	937.5 W Simulation	121

A.2	Comparison with Experimental Data – Additional Information	123
A.3	Test Flight Data	131
A.3.1	General Remarks on Data Quality	131
A.3.2	Air-Cooled Motor Controller — Self-Launch Case	131
A.3.3	Water-Cooled Motor Controller — Sustainer Case	134
A.3.4	Water-Cooled Motor Controller — Self-Launch Case	134
B	Datasheets and Drawings	136
B.1	Heat Pipe Datasheet	136
B.2	Electric Motor Datasheet	139
B.3	Motor Controller Datasheet	143
B.4	Heat Sink Drawing	147
B.5	Motor Controller Box-Top Drawing	149
B.6	Motor Controller Box-Bottom drawing	151
B.7	Motor Controller Box-Divider Drawing	151
B.8	Motor Controller Box-Fan Assembly Drawing	153
B.9	Motor Controller Assembly (Air cooled) drawing	155
B.10	Motor Controller Assembly (Water-Cooled) Drawing	155
B.11	Cooling Fans Datasheet	160

List of Figures

1.1	Jonker Sailplanes JS3 RES.	2
1.2	Schematic layout of the JS3 RES highlighting major propulsion components.	3
1.3	Effect of pilot position on fuselage airflow.	3
1.4	Motor controller heat sink and internal thermal path.	4
1.5	Motor controller installed in wing spar compartment.	4
1.6	Heat sink and CFD-predicted temperature distribution under hot-day conditions.	5
2.1	Different types of heat transfer (Source: Duke University (2016)).	9
2.2	Heat sink with full 3-D CFD analysis (Source: Galliera, 2018).	21
2.3	Thin-Walled CFD Model	22
2.4	Porous media modelling approach for a microchannel heat sink (Source: Mahgoub and Ghani (2021)	24
2.5	Baffles approach for enhancing shell-and-tube heat exchanger performance (Source: CFD Land, 2022).	25
2.6	Baffles approach for enhancing shell-and-tube heat exchanger performance (Source: CFD Land, 2022).	26
2.7	Different fin geometries of heat sinks (adapted from Kumar et al. (2021))	33
2.8	Functional principle of heat pipes. Adapted from Dai Nippon Printing Co., Ltd., "The Basics of Heat Pipes – Their History, Principle, and Varieties Explained" Dai Nippon Printing Co. (2022).	34
3.1	Illustrative heat-pipe heat-spreading principle. https://myheatsinks.com	43
3.2	Original heat sink design from NX	44

3.3	Actual heat sink installed in the aircraft	45
3.4	Original heat source design from NX	46
3.5	Heat source integrated between heat sink and power electronics	46
3.6	CFD flow domain defined in Siemens NX (airflow from left to right)	47
3.7	Motor-controller packaging inside the fuselage (cockpit-side view)	47
3.8	Complete motor-controller assembly inside the fuselage (side view, airflow from left to right)	48
3.9	Complete original design from NX	48
3.10	Original starting mesh with base size 50 mm, showing coarse resolution and limited geometric detail.	51
3.11	Location of Pressure and Velocity measurements	53
3.12	Maximum Temperature measurement	54
3.13	Flow Domain MIS - 3D: Percentage differences in key parameters relative to previous mesh refinement iterations.	54
3.14	Flow Domain with final base mesh size of 3 mm, showing refined regions near heat sink fins.	55
3.15	Heat Sink MIS - 3D: Maximum temperature variations with mesh refinement.	57
3.16	Heat Sink with final mesh base size of 2 mm, showing refined fin and body details.	57
3.17	Heat Pipes MIS - 3D: Maximum temperature variations with mesh refinement.	59
3.18	Heat Pipes with final mesh base size of 5 mm.	59
3.19	Heat Source MIS - 3D: Maximum temperature variation with mesh refinement.	61
3.20	Heat Source with final mesh base size of 10 mm.	61
3.21	Final 3D MIS with all components.	63
4.1	Numerical simulation workflow implemented in STAR-CCM+.	69
4.2	SimScale Heat Sink	75
4.3	SimScale Heat Source	76

4.4	SimScale Flow Domain	76
4.5	SimScale setup showing the half-domain assembly with internal mesh and conjugate interfaces	77
4.6	External view of the SimScale flow domain with mesh and local refinement in the heat sink region	77
4.7	Mesh of the heat sink used for the verification study, highlighting the refined discretization on the fins.	79
4.8	Mesh of the heat source block used in the verification study.	80
4.9	External view of the flow domain mesh showing local refinement around the heat sink region.	80
4.10	Internal view of the flow domain mesh illustrating refined regions near the heat sink.	81
4.11	External view of the verification-case mesh with visible refinement zones.	82
4.12	Original mesh used by SimScale for the benchmark study.	83
4.13	Thermal Resistance Comparison	87
5.1	Temperature distribution for the 1500 W heat influx simulation before flow-domain extension. Airflow direction is from left to right.	93
5.2	Temperature distribution for the refined 1500 W heat influx simulation. Airflow direction is from left to right.	94
5.3	Residuals during the original 1500 W heat influx simulation before flow-domain extension	95
5.4	Residuals during the refined 1500 W heat influx simulation	95
5.5	Temperature distribution for the original 937.5 W heat influx simulation before flow-domain extension. Airflow direction is from left to right.	98
5.6	Temperature distribution for the refined 937.5 W heat influx simulation. Airflow direction is from left to right.	98
5.7	Residuals during the original 937.5 W heat influx simulation before flow-domain extension	99
5.8	Residuals during the refined 937.5 W heat influx simulation	99
5.9	Temperature distribution on the heat sink fins (bottom surface) at 1500 W heat influx before flow domain extension	100

5.10 Temperature distribution on the heat sink fins (bottom surface) at 1500 W heat influx	101
5.11 Temperature distribution on the heat sink fins (bottom surface) at 937.5 W heat influx before flow domain extension	101
5.12 Temperature distribution on the heat sink fins (bottom surface) at 937.5 W heat influx	101
A.1 Preliminary simulation – general geometry	118
A.2 Preliminary simulation – inlet mesh of flow domain	118
A.3 Preliminary simulation – outlet mesh of flow domain	119
A.4 Preliminary simulation – general mesh of flow domain	119
A.5 Preliminary simulation – heat pipe mesh	120
A.6 Preliminary simulation – heat sink mesh	120
A.7 Preliminary simulation – heat source mesh	121
A.8 Preliminary simulation – complete mesh	121
A.9 Residuals of the preliminary simulation at 937.5 W heat input, showing poor convergence and oscillatory behavior	122
A.10 Temperature distribution predicted by the preliminary 937.5 W simulation	122
A.11 Maximum temperature predicted by the preliminary 937.5 W simulation	123
A.12 Maximum temperature predicted by the experimental-data comparison simulation (uncorrected for altitude)	124
A.13 Temperature distribution within the flow domain (uncorrected for altitude)	124
A.14 Solver residuals of the experimental-data comparison simulation (uncorrected for altitude)	125
A.15 Maximum temperature predicted by the experimental-data comparison simulation (corrected for altitude)	125
A.16 Temperature distribution within the flow domain (corrected for altitude)	125
A.17 Solver residuals of the experimental-data comparison simulation (corrected for altitude)	126

A.18 Comparison of measured and simulated temperatures and pressure drop (corrected for altitude)	126
A.19 General Experimental Setup	129
A.20 Water heating of experimental Setup	130
A.21 Temperature probe locations of experimental setup	130
B.1 RES system bench test conducted by SOLO Aircraft Engines.	143

List of Tables

3.1	Flow Domain Mesh Independence Study (MIS) - 3D	53
3.2	Heat Sink Mesh Independence Study (MIS) - 3D	56
3.3	Heat Pipes Mesh Independence Study (MIS) - 3D	58
3.4	Heat Source Mesh Independence Study (MIS) - 3D	60
3.5	Final base sizes and corresponding cell counts for the 3D MIS.	62
3.6	Material properties of JS3 RES components used in the CFD model.	63
4.1	SimScale Heat Sink Dimensions	75
4.2	SimScale Heat Source Dimensions	76
4.3	SimScale Flow Domain Dimensions	76
4.4	Number of cells for each component in the benchmark verification.	78
4.5	Thermophysical properties of air used in the verification study	84
4.6	Thermophysical properties of the heat sink material (aluminium) used in the verification study	84
4.7	Thermophysical properties of the heat source used in the verification study	84
4.8	Inlet velocities, volumetric flow rates, and corresponding mass flow rates for the verification cases.	85
4.9	Mass flow rate, temperatures, power, and thermal resistances for different inlet conditions.	86
4.10	Percentage errors for different inlet conditions, comparing simulation, analytical, and experimental results.	86
4.11	Summary of average junction-to-air thermal resistances and deviations from theoretical predictions.	88

4.12 Physical values assigned to the flow domain.	89
4.13 Physical values assigned to the heat sink.	89
4.14 Physical values assigned to the heat pipes.	89
4.15 Physical values assigned to the heat source.	89
4.16 Thermal interface resistances between the primary conduction paths. . . .	90
5.1 Temperature and performance data obtained from practical testing ?. . . .	104
5.2 Comparison between experimental and simulated temperatures, pressure drop, and heat input.	105
A.1 Experimental vs. simulated values — percentage deviation.	128
A.2 Test flight data — Air-cooled motor controller — self-launch (2025)	133
A.3 Test flight data — Air-cooled motor controller — self-launch (2022)	133
A.4 Test flight data — Water-cooled motor controller — Sustainer case	134
A.5 Test flight data — Water-cooled motor controller — Self-launch case	135

Table of abbreviations

A table containing a list of abbreviations that will be used throughout the text.

AC	Alternating Current
AI	Artificial Intelligence
AM	Additive Manufacturing
ATS	Advanced Thermal Solutions
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
CNTs	Carbon Nanotubes
CPU	Central Processing Unit
CS 22	Certification Specification 22 (for Sailplanes and Powered Sailplanes)
DC	Direct Current
EASA	European Union Aviation Safety Agency
FVM	Finite Volume Method
GNTs	Graphene Nanotubes
IGBTs	Insulated-Gate Bipolar Transistors
ISA	International Standard Atmosphere
JS	Jonker Sailplanes
MCP	Maximum Continuous Power
MIS	Mesh Independence Study
MP	Maximum Power
OAT	Outside Air Temperature
PCB	Printed Circuit Board
RAM	Random-Access Memory
RANS	Reynolds-Averaged Navier–Stokes
RES	Retractable Electrical Propulsion System
RPM	Revolutions Per Minute
STAR-CCM+	Simulation of Turbulent flow in Arbitrary Regions - Computational Continuum Mechanics + (extended multiphysics capability)
TAS	Thermal Analysis Software
TIM	Thermal Interface Materials
V&V	Verification and Validation

Chapter 1

Introduction

1.1 Background

1.1.1 Electric Sailplane Context

Electric self-launching sailplanes represent a specialised class of lightweight aircraft that combine the aerodynamic efficiency of conventional gliders with the operational autonomy of self-powered take-off and climb. Unlike powered aircraft, sailplanes are optimised for minimum mass, low aerodynamic drag, and high glide performance across a wide speed envelope. These design priorities leave limited margin for additional systems, particularly those associated with propulsion and thermal management. As a result, the integration of electric propulsion into sailplane airframes presents tightly coupled engineering challenges, including mass control, spatial integration, reliability, and heat dissipation.

The Jonker Sailplanes JS3 RES (Retractable Electrical Propulsion System) is a modern electric self-launching sailplane designed to enable independent take-off and climb without compromising soaring performance. To achieve this, the aircraft integrates a retractable electric propulsion unit, high-voltage lithium-ion battery packs, and associated power electronics into an aerodynamically optimised fuselage. While this preserves aerodynamic efficiency, it introduces concentrated heat sources into a structure not originally designed for significant thermal loads.



Figure 1.1: Jonker Sailplanes JS3 RES.

1.1.2 Propulsion System and Thermal Challenge

Within the electric propulsion architecture, the motor controller represents a primary thermal source. It converts high-voltage DC from the battery system into three-phase AC for the propulsion motor. During high-power operation, significant heat is generated due to switching losses, conduction losses, and resistive heating.

If not managed effectively, this heat leads to power derating, accelerated ageing, or protective shutdown.

The physical integration of the propulsion system further exacerbates the thermal challenge. As illustrated in Figure 1.2, the propulsion motor is mounted on a retractable pylon, while the motor controller is installed in a confined fuselage region between the wing roots. Battery packs are distributed along the fuselage sides for mass balance.

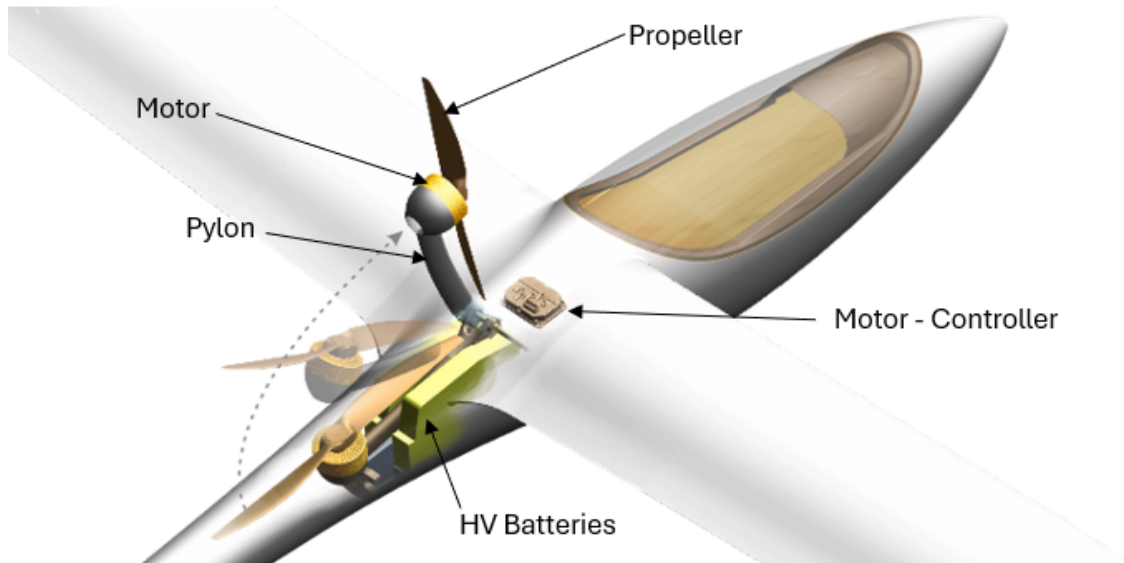


Figure 1.2: Schematic layout of the JS3 RES highlighting major propulsion components.

This configuration severely restricts available airflow and cooling surface area. Additionally, pilot positioning can influence inlet airflow, as shown in Figure 1.3, introducing variability in cooling performance under real operating conditions.



(a) Restricted airflow due to pilot position.



(b) Unobstructed airflow reference case.

Figure 1.3: Effect of pilot position on fuselage airflow.

1.1.3 Thermal Management System

To address these constraints, the original JS3 RES cooling concept employs an air-cooled heat sink incorporating embedded heat pipes and forced convection via electrically driven fans. The aluminium heat sink provides extended fin surface area, while heat pipes distribute heat laterally.

Figure 1.4 shows the disassembled motor controller, highlighting the heat sink and full heat transfer path from electronics to airflow.



Figure 1.4: Motor controller heat sink and internal thermal path.

Figure 1.5 shows the installed configuration, illustrating the constrained fuselage integration.

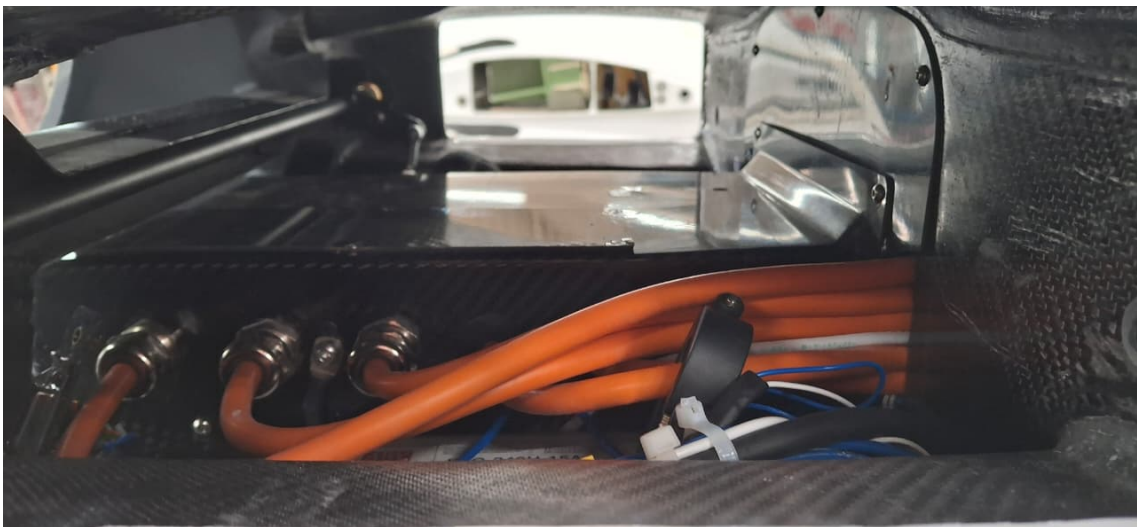


Figure 1.5: Motor controller installed in wing spar compartment.

Heat pipes are passive thermal devices that transport heat via phase-change mechanisms, enabling high effective thermal conductivity. However, their performance depends on orientation, manufacturing quality, and thermal interface conditions.

1.2 Certification and Operating Requirements

From a certification perspective, thermal performance is governed by EASA CS-22, with specific test conditions defined in CS 22.1047 (Cooling test procedure for

engine-powered sailplanes) and associated special conditions for electric propulsion installations (SC-22.2014-01).

For the JS3 RES, compliance requires demonstration of adequate cooling under a representative flight envelope consisting of take-off or maximum power operation, followed by a sustained climb at maximum continuous power under hot-day ambient conditions of 38 °C at sea level.

Under these conditions, the motor-controller temperature must remain below 85 °C (corrected to reference ambient conditions), ensuring compliance with certification requirements.

1.3 Thermal Behaviour and System Limitations

Heat transfer in the system is governed by:

- conduction within solids,
- thermal contact resistance at interfaces,
- convection to surrounding airflow.

Radiation is negligible at operating temperatures.

Early analytical and CFD models indicated compliance under nominal assumptions. However, flight test data at elevated ambient conditions showed that thermal limits were occasionally approached or exceeded.

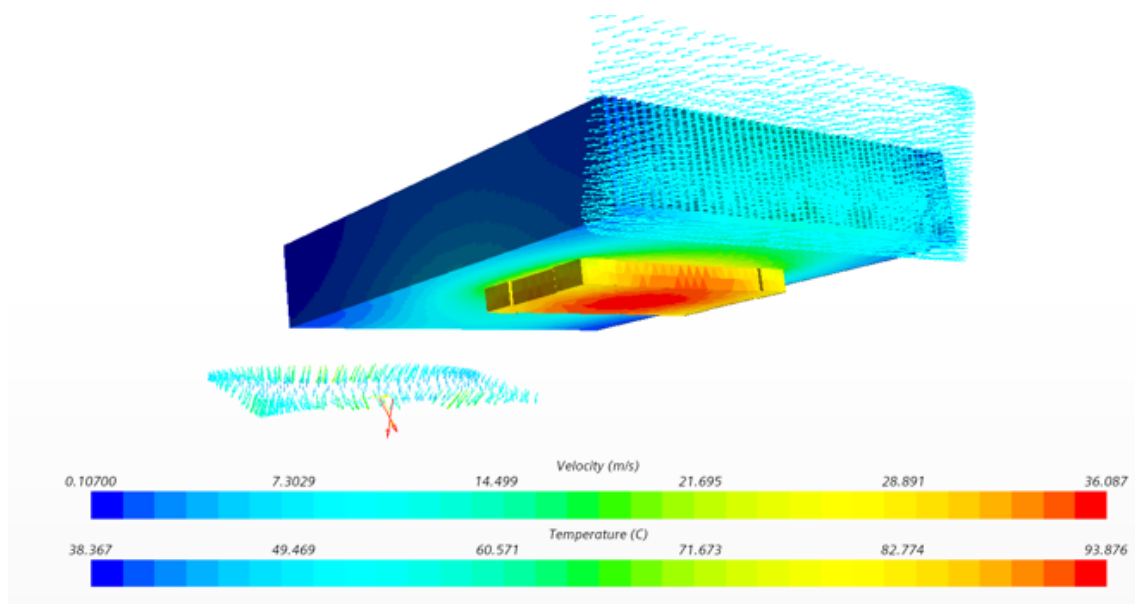


Figure 1.6: Heat sink and CFD-predicted temperature distribution under hot-day conditions.

Key uncertainties include:

- internal airflow distribution,
- fan performance in installed configuration,
- geometric effects on convection,
- thermal contact resistances,
- simplified heat pipe modelling.

1.4 Design Evolution and Motivation

Supply chain disruptions led to a transition toward a liquid-cooled system. While thermally effective, this introduced:

- increased mass (4–6 kg),
- higher system complexity,
- additional failure modes,
- increased maintenance requirements,
- centre-of-gravity shift.

These drawbacks motivated renewed investigation of the air-cooled concept.

1.5 CFD Modelling Context

Conjugate heat transfer (CHT) CFD is widely used for coupled solid–fluid thermal analysis. However, it is computationally expensive and sensitive to boundary conditions and thermal interface modelling. Heat pipe modelling remains particularly challenging and is typically simplified.

Without experimental validation, CFD predictions for compact thermal systems remain limited in reliability.

1.6 Problem Description

Accurate prediction of thermal performance in compact air-cooled motor-controller systems remains challenging due to:

- coupled conduction–convection physics,
- uncertain interface resistances,
- airflow variability,
- simplified heat pipe representation.

A validated CFD methodology is therefore required for reliable prediction under certification-relevant conditions.

1.7 Objectives

The objective of this research is to develop and validate a CFD-based modelling methodology for the JS3 RES motor-controller cooling system.

1. Develop and Validate a CFD-Based Thermal Model.
2. Establish a CFD-Based Modelling Approach for Heat Pipes.
3. Investigate Deviations Between Simulated and Measured Behaviour.
4. Recommend Design and Methodological Improvements.

Chapter 2

Literature review

2.1 Introduction

Thermal management is critical in electronic systems to prevent overheating and ensure component reliability and longevity. As electronic devices continue to increase in power density, effective heat dissipation becomes essential for maintaining optimal operating temperatures and preventing thermal failure. Heat sinks play a pivotal role in this process by enhancing the transfer of heat away from electronic components into the surrounding environment.

A comprehensive review of heat transfer in electronic systems is provided by Kadum et al. (2022), which discusses the fundamentals of heat transfer, various cooling strategies, and the impact of material properties and geometrical configurations on thermal performance. The paper highlights the importance of efficient heat dissipation for maintaining the performance and lifespan of electronic components, making it highly relevant to air-cooled heat sink design in compact, high-power systems such as the JS3 RES motor-controller.

With advancements in computational methods and material science, heat sink design has evolved significantly, offering innovative solutions to meet the thermal management challenges posed by modern electronic devices. This literature review provides a comprehensive overview of these advancements, highlighting key studies and innovations in heat sink design and thermal management.

2.2 Principles of Heat Sink Design

Heat sinks are critical components designed to maximize the dissipation of heat from electronic devices, thereby preventing overheating and ensuring reliable operation. Effective heat sink performance relies on the three primary mechanisms of heat transfer:

conduction, convection, and radiation.

Designing an efficient heat sink requires careful consideration of multiple factors, including geometry, fin design, surface area, and airflow dynamics. Optimizing these factors ensures that heat is effectively transferred from the heat-generating components to the surrounding environment. In compact, high-power systems such as the JS3 RES motor-controller, these design considerations are particularly important due to tight spatial constraints and variable airflow conditions.

Advanced numerical modelling, including computational fluid dynamics (CFD), is often employed to predict and enhance heat sink performance. Such models allow engineers to evaluate the impact of design choices on thermal performance, assess uncertainties in material properties and boundary conditions, and guide iterative design improvements even when only limited experimental data are available.

By understanding and optimizing these principles, engineers can design heat sinks that efficiently manage thermal loads, extend the lifespan of electronic devices, and ensure stable operation under diverse environmental and operational conditions.

2.3 Heat transfer

As shown in Figure 2.1 Duke University (2016), heat transfer occurs through conduction, convection, and radiation.

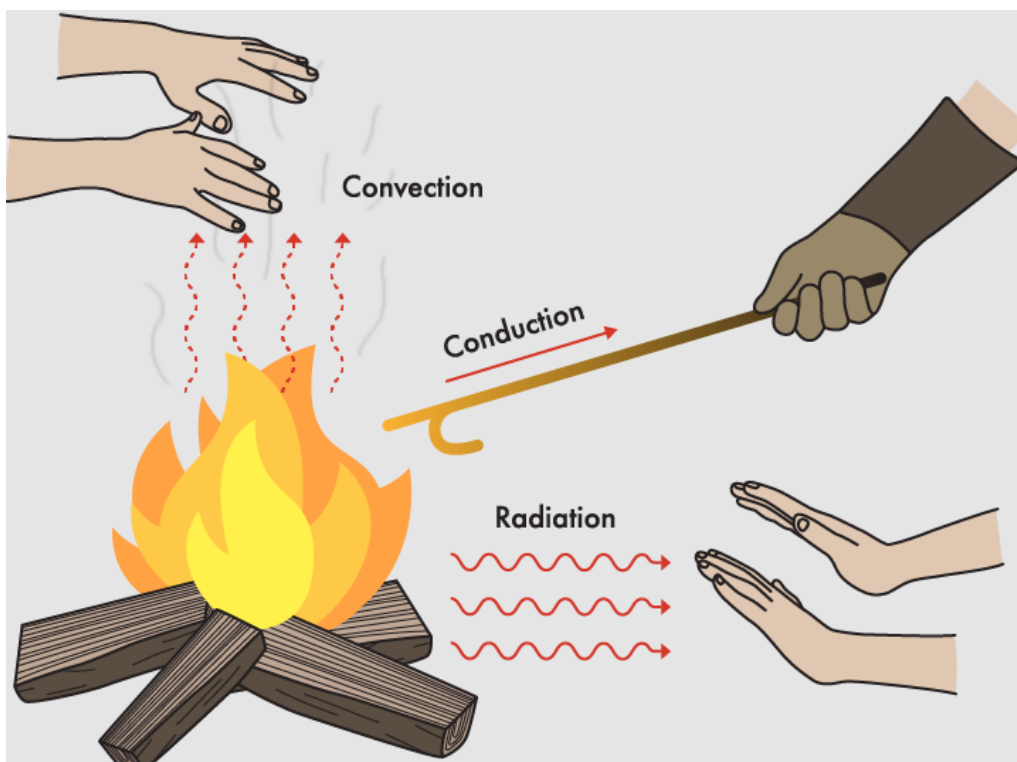


Figure 2.1: Different types of heat transfer (Source: Duke University (2016)).

2.3.1 Conduction

Conduction is the primary mechanism of heat transfer within a heat sink, allowing thermal energy to flow from the electronic component to the surrounding material. The efficiency of this process is largely determined by the thermal conductivity of the material used in the heat sink. Materials such as copper and aluminium are commonly selected due to their high thermal conductivities, which enable effective heat transfer from the component to the heat sink fins Ekpu et al. (2011).

Copper-based heat sinks typically offer superior thermal conductivity compared to aluminium, resulting in enhanced heat dissipation capabilities, particularly in air-cooled systems. However, aluminium remains widely used due to its lower weight and cost, making it a practical choice for applications where these factors are critical.

The performance of the heat sink also depends on the quality of the thermal interface between the electronic component and the heat sink surface. Thermal interface materials (TIMs), including thermal greases, phase change materials, and thermal pads, reduce thermal resistance across the interface and improve the overall efficiency of heat conduction Narumanchi et al. (2010). Proper selection and application of TIMs are therefore essential to ensure that the heat sink can effectively manage the thermal load and maintain the reliable operation of electronic devices.

2.3.2 Convection

Convection refers to the transfer of heat through the motion of a fluid, which can occur naturally due to temperature gradients (natural convection) or be actively driven by fans or blowers (forced convection). In heat sink applications, convection plays a critical role in removing heat from the fins into the surrounding air.

The geometry and arrangement of heat sink fins significantly influence convective heat transfer. Fins increase the effective surface area available for heat exchange and guide airflow to optimize cooling performance. Advanced heat sink designs, such as pin-fin arrays and microchannel structures, further enhance convective heat transfer by increasing turbulence, promoting uniform airflow, and maximizing the surface area in contact with the moving fluid Ekpu et al. (2011).

Effective convective cooling requires careful consideration of airflow patterns, fin spacing, and fluid velocity. In air-cooled systems like the JS3 RES motor-controller, forced convection from embedded fans is essential to maintain acceptable operating temperatures under high-power conditions, especially given the limited space and weight constraints of the aircraft.

2.3.3 Radiation

Radiation is the emission of heat in the form of electromagnetic waves. While generally less significant than conduction and convection in heat sinks, radiative heat transfer becomes more relevant at higher temperatures. Materials with high emissivities, such as anodized aluminium, are utilized to enhance radiative heat transfer from heat sinks. Anodizing aluminium increases its surface emissivity from approximately 0.04 to 0.86, significantly improving its ability to radiate heat QATS (2010).

Anodizing is an electrochemical process that thickens and toughens the naturally occurring protective oxide layer on the surface of aluminium parts. This process not only enhances the material's corrosion resistance but also increases its emissivity, thereby improving thermal radiation efficiency. The increased emissivity allows heat sinks to more effectively dissipate heat through radiation, which is particularly beneficial in applications where convective cooling is limited or in vacuum environments where convection is absent QATS (2010).

In summary, while radiation may not be the primary mode of heat transfer in all heat sink applications, anodizing aluminium significantly enhances its radiative properties, contributing to more efficient thermal management in electronic systems.

In this study, radiation is not expected to play a critical role, as the operating temperature is anticipated to remain below 75 °C. At these temperatures, the contribution of radiative heat transfer is minimal compared to conduction and convection, as discussed by Holman in his comprehensive work on heat transfer Holman (2010).

2.4 Material selection

The choice of materials significantly influences the thermal performance, weight, and cost-effectiveness of heat sinks used in electronic devices. The two primary materials commonly employed are aluminium and copper, each offering distinct thermo-physical properties that directly affect heat transfer performance and system mass.

2.4.1 Aluminium

Aluminium is widely used in heat sink applications due to its favourable combination of thermal conductivity, low density, and cost-effectiveness. Its density is approximately 2700 kg/m³, which is roughly one-third that of copper. This makes it particularly suitable for aerospace and portable electronic applications where mass reduction is critical. Aluminium also offers a thermal conductivity in the range of 200–235 W/m·K (depending on alloy), which is sufficient for many air-cooled heat sink designs.

Additionally, aluminium is relatively inexpensive, typically costing significantly less per kilogram than copper, and is easily manufactured using extrusion, machining, and casting processes. This allows for complex fin geometries that enhance surface area and improve convective heat transfer.

However, its lower thermal conductivity compared to copper can limit performance in high heat flux applications where rapid heat spreading is required.

The primary advantages of aluminium for heat sink applications include:

- **Thermal Conductivity:** Approximately 200–235 W/m·K, which is adequate for many forced and natural convection applications but lower than copper.
- **Density:** Approximately 2700 kg/m³, resulting in significantly reduced component mass compared to copper (8960 kg/m³).
- **Cost:** Typically 3–5 times cheaper per kilogram than copper, depending on market conditions.
- **Machinability:** High machinability and suitability for extrusion enables complex and high surface-area fin designs.
- **Corrosion Resistance:** Natural oxide layer formation provides good resistance to environmental degradation.

2.4.2 Copper

Copper is widely recognised for its superior thermal conductivity, making it highly effective in applications requiring rapid heat spreading and low thermal resistance. Its thermal conductivity is approximately 385–400 W/m·K, nearly twice that of aluminium, which enables more efficient heat transfer away from concentrated heat sources.

However, copper has a significantly higher density of approximately 8960 kg/m³, making it more than three times heavier than aluminium. This limits its suitability in weight-sensitive applications such as aerospace systems. Copper is also considerably more expensive, often costing several times more per unit mass than aluminium.

Manufacturing complexity is higher as well, since copper is harder to machine and deform, increasing production effort and cost.

Compared to aluminium, copper offers the following characteristics:

- **Thermal Conductivity:** Approximately 385–400 W/m·K, enabling superior heat spreading performance.

- **Density:** Approximately 8960 kg/m^3 , resulting in significantly increased system mass.
- **Cost:** Typically 3–6 times more expensive than aluminium, depending on global commodity prices.
- **Machinability:** Lower machinability than aluminium, increasing manufacturing effort and tool wear.
- **Corrosion Behaviour:** Susceptible to oxidation and corrosion in certain environments, often requiring protective coatings.

2.4.3 Advanced Materials

Emerging materials such as graphene and carbon nanotubes (GNTs and CNTs) have attracted considerable attention for next-generation heat sink designs due to their exceptional thermal properties. Graphene exhibits reported in-plane thermal conductivities in the range of $2000\text{--}5300 \text{ W/m}\cdot\text{K}$, significantly exceeding that of copper, making it a highly attractive candidate for advanced thermal management applications Balandin et al. (2008).

Similarly, CNTs can exhibit thermal conductivities ranging from $1000\text{--}3000 \text{ W/m}\cdot\text{K}$ depending on structure and alignment, positioning them as promising reinforcements in composite thermal materials Yan et al. (2015).

Recent research has explored hierarchical three-dimensional structures combining CNTs, graphene nanosheets, and related nanostructures. These hybrid materials have demonstrated improved effective thermal conductivity and reduced thermal resistance compared to conventional metals, although large-scale manufacturing remains challenging Yan et al. (2015).

In addition to nanomaterials, hybrid metallic designs are widely used in practical applications. A common approach combines a copper base (thermal conductivity approximately $385 \text{ W/m}\cdot\text{K}$) with aluminium fins (approximately $205 \text{ W/m}\cdot\text{K}$), leveraging copper for efficient heat spreading and aluminium for reduced mass and improved convection performance. Studies have shown that such hybrid heat sinks can achieve thermal performance close to full copper designs while reducing weight by up to 40–60%, depending on geometry and configuration Cermak et al. (2020).

2.5 Thermal Analysis and Simulation

Advances in computational fluid dynamics (CFD) and thermal analysis software have significantly enhanced the design and optimization of heat sinks for electronic devices.

These tools allow engineers to model heat transfer processes, airflow behavior, and temperature distributions before physical prototyping, reducing development time and cost. By integrating CFD with thermal analysis, engineers can predict thermal performance in complex electronic systems, identify potential hotspots, optimize heat sink geometry, and ensure reliable thermal management across a range of operating conditions.

2.5.1 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) simulations provide detailed insights into heat sink performance by resolving heat transfer mechanisms and airflow patterns within the system. Through CFD, engineers can assess the effects of various operating conditions and design configurations, enabling optimization of heat sink geometry, fin layout, and material selection. This approach has been shown to be highly effective in guiding thermal design for advanced electronic systems Yan et al. (2015). By leveraging CFD, designers can predict critical performance indicators such as maximum temperature, temperature distribution, and airflow efficiency, facilitating a simulation-driven workflow for thermal management.

Heat Transfer Mechanisms: CFD simulations enable engineers to investigate the detailed mechanisms of heat transfer within a heat sink, including conduction, convection, and radiation. As highlighted by Yan et al. (2015), CFD can accurately model the thermal conductivity and heat capacity of the heat sink materials, providing valuable insight into how heat is conducted from the electronic component through the heat sink and subsequently dissipated into the surrounding environment.

Airflow Patterns: CFD studies have shown that modifying fin configurations can substantially influence airflow distribution, leading to improved overall cooling performance. For example, Abdullah and Jubear (2022) used CFD simulations to investigate the thermal behavior of copper foam fin heat sinks under forced convection. Their results demonstrated that specific fin designs could significantly enhance airflow patterns and increase thermal efficiency, highlighting the critical role of fin geometry in optimizing heat sink performance.

Optimization of Heat Sink Geometry: CFD simulations allow engineers to evaluate and refine heat sink geometries without the need for physical prototypes, offering a cost-effective and time-efficient approach. By testing multiple configurations virtually, designers can identify the optimal geometry that maximizes heat dissipation while balancing constraints such as weight, material cost, and manufacturability. Studies have shown that specific geometric modifications, such as increasing the surface area

exposed to airflow or optimizing fin spacing, can result in substantial improvements in thermal performance and overall cooling efficiency.

Fin Designs: The configuration of heat sink fins plays a critical role in determining thermal performance. CFD simulations allow engineers to study how variations in fin thickness, height, spacing, and shape influence both heat transfer and airflow. By optimizing these parameters, designers can maximize the surface area available for heat dissipation while maintaining efficient airflow through the heat sink.

Experimental and numerical studies have demonstrated that specific fin arrangements, such as tapered or staggered designs, can enhance convective heat transfer. For example, Patil and Saini (2020) investigated a cylindrical, staggered pin-fin heat sink and reported substantial improvements in heat transfer efficiency, highlighting the importance of careful fin design in thermal management applications.

Material Properties: The selection of heat sink materials is one of the most critical factors influencing thermal performance, weight, and overall system cost. Materials directly impact thermal conductivity, specific heat capacity, and compatibility with other components, all of which determine how effectively heat can be removed from electronic devices. CFD simulations are particularly valuable in quantifying the influence of material properties under various operating conditions and geometries, allowing engineers to optimize heat sink performance before physical prototypes are manufactured.

A study by Abdullah and Jubear (2024) employed CFD simulations to systematically compare aluminium and copper heat sinks with multiple fin configurations and flow rates. Their analysis revealed that copper heat sinks consistently provided higher heat dissipation due to their superior thermal conductivity, making them more suitable for high-power applications. However, aluminium heat sinks offered substantial advantages in terms of weight and cost, which can be critical in applications where mass reduction is a priority, such as aerospace or portable electronics. These trade-offs emphasize the importance of considering the full operational context when selecting heat sink materials.

Similarly, Advanced Thermal Solutions (2017) analyzed heat sinks with embedded heat pipes, comparing copper and aluminium designs. Their findings indicated that aluminium heat sinks could achieve nearly the same thermal performance as copper variants when optimized properly, particularly when heat pipe integration and fin geometry were carefully considered. This demonstrates that material choice does not exist in isolation; rather, it interacts with heat sink design parameters such as fin density, surface area, and thermal interface materials (TIMs).

Overall, these studies underline the critical role of material properties in the design and optimization of heat sinks. Using CFD simulations to model different materials allows engineers to evaluate the balance between thermal efficiency, weight, cost,

and manufacturability. By integrating material selection with geometric and airflow optimization, designers can achieve heat sink solutions that meet stringent performance requirements while maintaining practical considerations for production and system integration.

2.5.2 Thermal Analysis Software (TAS)

Thermal Analysis Software (TAS) generally refers to computational tools focused on solving heat transfer problems, primarily emphasising conduction and radiation within solid domains. These tools typically do not explicitly resolve the surrounding fluid flow field. Instead, convective effects are often represented using empirical correlations or prescribed heat transfer coefficients.

In contrast, computational fluid dynamics (CFD)-based solvers resolve the full set of governing conservation equations for mass, momentum, and energy. This allows for the direct simulation of fluid flow and its coupling with heat transfer mechanisms. As a result, CFD inherently performs thermal analysis by simultaneously capturing conduction, convection, and, where applicable, radiation within a fully coupled framework. Modern CFD solvers can therefore predict detailed temperature distributions in both solid and fluid regions, enabling high-fidelity modelling of complex thermo-fluid interactions Tang (2024).

Examples of widely used CFD platforms include STAR-CCM+ and ANSYS Fluent, which are capable of performing conjugate heat transfer (CHT) simulations where both solid conduction and fluid convection are resolved within a single computational domain.

TAS tools typically employ reduced-order modelling approaches or simplified boundary conditions to represent convective heat transfer. This results in significantly lower computational cost and faster setup times compared to CFD simulations. Consequently, TAS is well suited for early-stage design, rapid thermal screening, or conduction-dominated problems such as multilayer electronic assemblies or preliminary printed circuit board (PCB) thermal assessments.

In contrast, CFD is required when fluid flow strongly influences thermal behaviour, such as in forced convection heat sinks where local velocity fields, turbulence effects, and flow separation significantly affect heat transfer performance.

For air-cooled heat exchanger applications, where heat transfer is strongly coupled to local airflow characteristics and geometric features such as fin spacing and flow separation, CFD-based conjugate heat transfer simulations are therefore required to accurately capture the governing thermal behaviour.

Thermal Interface Materials (TIMs): Thermal Interface Materials (TIMs) are critical in improving the thermal connection between electronic components and heat sinks. Microscopic surface irregularities and air gaps between the component and the heat sink can significantly impede heat transfer, leading to elevated component temperatures. TIMs, such as thermal greases, phase change materials, and thermal pads, fill these gaps and enhance the effective thermal conductivity across the interface.

Thermal analysis software allows engineers to model the behavior of TIMs under varying operational conditions, including changes in temperature, pressure, and surface contact quality. This enables the evaluation of different materials, thicknesses, and application methods to determine the optimal configuration for efficient heat dissipation. Accurately accounting for TIM properties is essential for reliable simulation results, as interface resistance often represents a significant portion of the total thermal resistance in electronic systems. By integrating TIM characteristics into both CFD and TAS simulations, engineers can better predict maximum component temperatures, optimize heat sink performance, and ensure the reliability and longevity of the electronic device under diverse operating conditions.

Coatings: Surface coatings can significantly influence the thermal performance and durability of heat sinks. Coatings with high emissivity, such as anodized or black oxide layers, enhance radiative heat transfer, particularly in applications where convection is limited. Additionally, protective coatings reduce the risk of corrosion and oxidation, which can degrade thermal conductivity over time and affect long-term reliability.

Thermal analysis software allows engineers to simulate the effects of various coating materials, thicknesses, and surface treatments, quantifying their impact on overall heat dissipation and thermal resistance. By incorporating coating characteristics into CFD and TAS models, designers can make informed decisions on material selection and surface treatment strategies to optimize heat sink performance while ensuring durability and longevity under operational conditions.

Other Factors Influencing Heat Transfer Efficiency: Thermal analysis software also accounts for a range of operational and environmental factors that can significantly influence heat transfer efficiency. These include ambient temperature, airflow distribution, orientation of the heat sink, and the presence of nearby components that may obstruct flow or create hotspots. By accurately modelling these variables, engineers can predict real-world thermal performance and optimize heat sink designs to maintain effective cooling under diverse operating conditions.

The integration of thermal analysis software with CFD provides a comprehensive framework for evaluating both solid and fluid heat transfer mechanisms. This combined approach enables a holistic understanding of thermal behavior, allowing engineers to make data-driven decisions that enhance overall system performance, reliability, and

safety.

In summary, thermal analysis software is an essential tool in heat sink design and optimization. By simulating the effects of TIMs, surface coatings, geometric configurations, and operational conditions, engineers can develop robust designs that ensure electronic components operate within safe temperature limits, thereby extending their service life and maintaining consistent performance across all operating scenarios.

2.5.3 Differences between CFD, CHT, and TAS

Accurate thermal management is critical in high-performance systems, and engineers have several computational tools at their disposal. Among the most commonly used are Computational Fluid Dynamics (CFD), Conjugate Heat Transfer (CHT), and Thermal Analysis Software (TAS). Understanding their scope, capabilities, and limitations is essential for selecting the appropriate approach for a given application.

Computational Fluid Dynamics (CFD): CFD is a numerical method for analyzing fluid flow and the associated convective heat transfer. It is particularly effective when detailed knowledge of fluid behavior is required, such as airflow around heat sink fins, through channels, or inside enclosures. CFD solves the governing equations for fluid momentum, energy, and turbulence, enabling the prediction of velocity fields, pressure distributions, and convective heat fluxes. While CFD can incorporate heat transfer from solid surfaces to the fluid via boundary conditions, in standard CFD simulations, the solid domain is often treated simplistically, using prescribed temperatures or heat fluxes rather than solving the conduction within the solid explicitly. This makes CFD ideal for studying the fluid-dominated aspects of heat dissipation but less accurate when solid conduction plays a significant role.

Conjugate Heat Transfer (CHT): CHT is essentially an extension of CFD that simultaneously solves heat transfer in both solid and fluid domains. In a CHT simulation, conduction within solids and convection in fluids are coupled at the solid–fluid interface, ensuring continuous temperature and heat flux predictions across boundaries. This approach is particularly important for systems where the thermal behavior of the solid significantly influences the fluid temperature distribution, such as in complex heat sinks, electronic components, or systems with temperature-dependent material properties. In other words, while standard CFD may approximate the solid as a thermal boundary, CHT explicitly models the solid and fluid together, providing a high-fidelity representation of the entire thermal system Leyli et al. (2019).

Thermal Analysis Software (TAS): TAS typically focuses on conduction, radiation, and overall thermal response of components, either in steady-state or transient scenarios. Unlike CFD, TAS does not solve detailed fluid flow; instead, it often uses simplified models or empirical correlations for convective heat transfer. TAS is ideal for evaluating temperature distributions within solids, predicting the effects of thermal interface materials (TIMs), multi-layer structures, and surface coatings, and for performing transient thermal analysis under varying operational loads. TAS is computationally less demanding than CFD or CHT but cannot capture detailed fluid-mediated effects such as localized hotspots due to stagnation or turbulence.

Key Differences and Complementarity:

- **Focus:** CFD emphasizes fluid behavior and convective heat transfer, TAS emphasizes solid conduction and transient response, while CHT captures both simultaneously.
- **Fidelity:** CHT provides the highest fidelity for coupled solid-fluid thermal systems, CFD provides detailed fluid insights but approximates the solid, and TAS provides reliable solid-domain predictions but uses simplified fluid models.
- **Computational Requirements:** TAS is generally fastest and least resource-intensive, CFD requires more computational effort depending on flow complexity, and CHT is the most demanding due to the coupled solution of solid and fluid domains.
- **Applications:** CFD is used for airflow optimization and convective analysis, TAS for material evaluation and transient temperature prediction, and CHT for scenarios where both solid conduction and fluid convection strongly interact, such as in heat sink optimization.

2.5.4 Siemens Star CCM+

Siemens Star CCM+ is a simulation platform that integrates both CFD and TAS capabilities, allowing engineers to perform high-fidelity thermal simulations, including CHT, within a single environment. This integration facilitates a holistic understanding of thermal behavior in complex systems.

CFD Capabilities: Within Star CCM+, CFD can model detailed fluid flow patterns, pressure distribution, turbulence, and convective heat transfer around components. Engineers can evaluate airflow over heat sink fins, detect stagnation zones, and optimize fin design, fan placement, and enclosure ventilation to enhance convective cooling efficiency.

TAS Capabilities: Star CCM+ also provides TAS functionality, allowing detailed analysis of heat conduction in solids, radiation effects, and transient thermal response. This is essential for predicting temperature evolution in heat sinks, thermal interface materials, coatings, and multi-layer structures under variable power loads.

CHT Integration: Using Star CCM+, engineers can perform CHT simulations where conduction in solids and convection in fluids are solved simultaneously. This allows precise evaluation of thermal interactions between the heat sink and the surrounding air, capturing localized temperature gradients, heat flux continuity at interfaces, and the influence of material properties on overall thermal performance. By combining CFD, TAS, and CHT in a single platform, Star CCM+ supports a comprehensive, simulation-driven design workflow.

Summary: In summary, TAS, CFD, and CHT serve complementary purposes in thermal management: TAS focuses on solid-domain analysis, CFD on fluid-domain analysis, and CHT couples both for high-fidelity simulations. Star CCM+ provides a unified environment to leverage all three approaches, enabling optimized heat sink design and reliable thermal management in complex engineering systems.

2.6 Different Modelling Approaches

2.6.1 Full 3D Analysis

The full 3D approach involves detailed modelling of the heat sink's geometry, capturing all its features, including the base, fins, and any intricate structural details. This method is often considered the gold standard for Computational Fluid Dynamics (CFD) simulations, as it provides the most accurate representation of both fluid flow and heat transfer phenomena Turner and Smith (2020); Pollini et al. (2019).

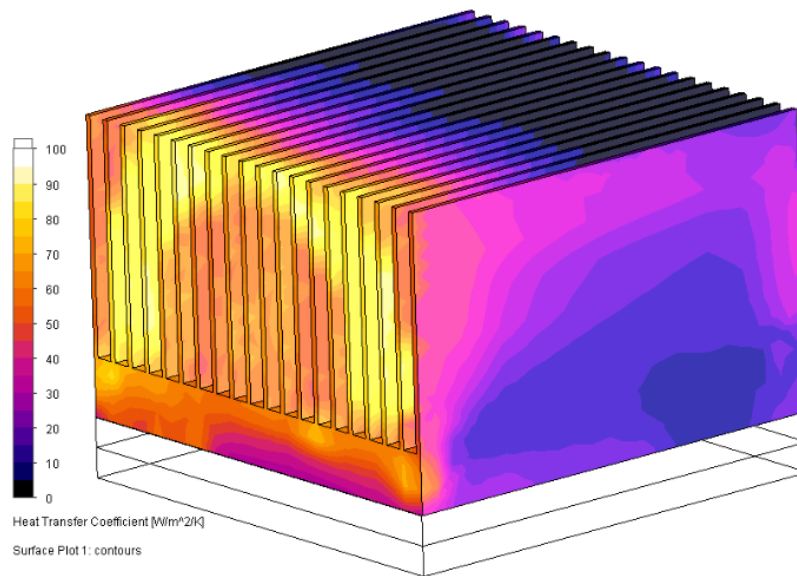


Figure 2.2: Heat sink with full 3-D CFD analysis (Source: Galliera, 2018).

As shown in Figure 2.2, a full 3-D conjugate heat transfer simulation illustrates the complex temperature gradients in a heat sink assembly Galliera (2018)

- **Advantages**

High Accuracy: Full 3D models provide precise results by accounting for all relevant thermal and fluid flow characteristics. The detailed geometric representation allows for the capture of complex interactions between solid and fluid domains, which is critical for accurate prediction of conduction, convection, and conjugate heat transfer processes.

Comprehensive Analysis: This approach is particularly suitable for complex geometries and tightly coupled physical phenomena. It enables the simulation of laminar and turbulent flows, localized hotspots, and intricate heat transfer interactions within fins, base plates, and heat pipes. Additionally, full 3D simulations can incorporate transient effects and multi-physics interactions if necessary, providing an in-depth understanding of thermal performance under realistic operating conditions.

- **Disadvantages**

Computationally Intensive: Full 3D simulations demand significant computational resources, including memory and processing power. High-resolution meshes and fine temporal discretization can result in long simulation times, making rapid iterative design studies challenging.

High Complexity: Setting up a full 3D simulation is complex and time-consuming. Accurate representation requires careful geometry creation, mesh refinement, and detailed specification of boundary conditions. Expertise is needed to ensure numerical stability, convergence, and meaningful results, which can be a barrier for rapid design optimization.

2.6.2 Thin-Walled Approach

The thin-walled approach simplifies the heat sink by modelling the fins as thin surfaces or walls, thereby reducing the overall geometric complexity of the system. This method is particularly advantageous when the fin thickness is much smaller than other dimensions of the heat sink, such as the base or fin height, and is most useful in scenarios where computational efficiency is a priority.

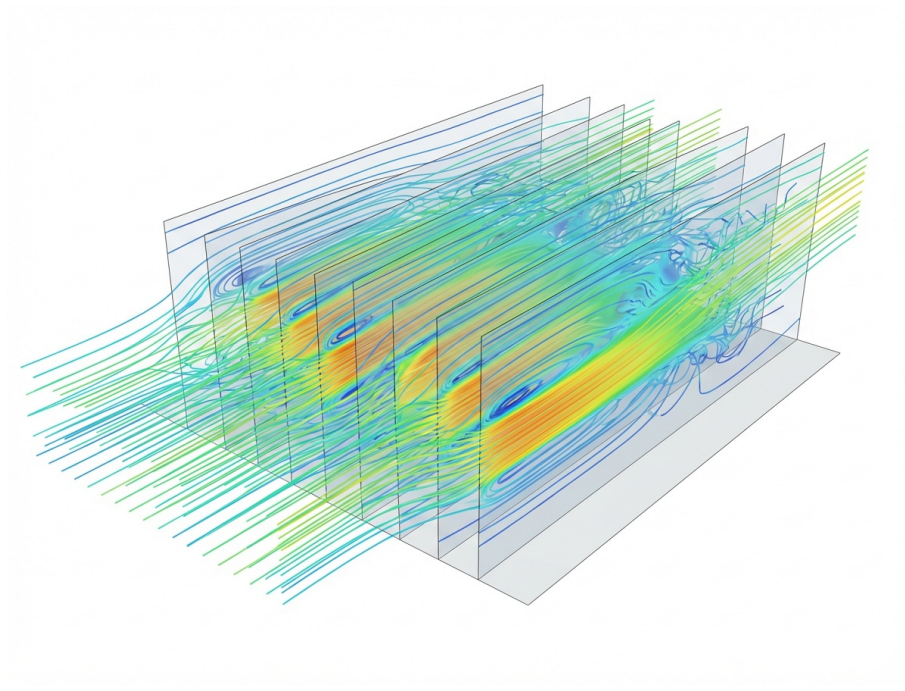


Figure 2.3: Thin-Walled CFD Model

The primary assumption in this approach is that heat transfer through the fin thickness is negligible relative to other dimensions. Consequently, simplified equations can be used to approximate heat conduction within the fins, significantly reducing the computational resources required. Fluid-structure interactions are also simplified, focusing mainly on convective heat transfer, which is typically the dominant mechanism for thin fins.

- **Advantages:**

Reduced Computational Cost: The thin-walled approach drastically lowers computational requirements compared to full 3D simulations. Fewer geometric details result in smaller meshes, reduced memory usage, and shorter simulation times, making this approach well-suited for preliminary design studies or parametric sweeps (Rahman et al. (2024)).

Effective for Thin Fins: When fin thickness is small relative to other dimensions, the approximation has minimal impact on accuracy. Studies have shown that for such geometries, modelling fins as thin walls provides a reasonable representation of overall heat transfer while significantly reducing computational effort (Leontiou and Fyrillas (2015)).

- **Disadvantages:**

Lower Accuracy for Thicker Fins: If fins are relatively thick, heat conduction through the fin itself can no longer be neglected. In these cases, the thin-walled approximation may underestimate thermal resistance and lead to inaccurate temperature predictions (Silva et al. (2020)).

Limited Capture of Flow Interactions: The approach may fail to capture complex flow phenomena near the fin edges, such as vortex shedding, boundary layer development, and localized turbulence. These effects can influence convective heat transfer, particularly at higher flow velocities, potentially leading to oversimplified predictions of heat sink performance.

2.6.3 Porous Media Model

The porous media model approximates the heat sink as a homogenized medium with effective properties that account for the combined influence of fins and air gaps, as described by Nield and Bejan (2006). Instead of representing the detailed geometry of individual fins, the model treats the heat sink as a continuum with averaged transport properties, capturing the bulk effects on pressure drop and heat transfer.

As illustrated in Figure 2.4, the porous media approach can be used to approximate the heat pipe or microchannel heat sink as a simplified computational domain Mahgoub and Ghani (2021).

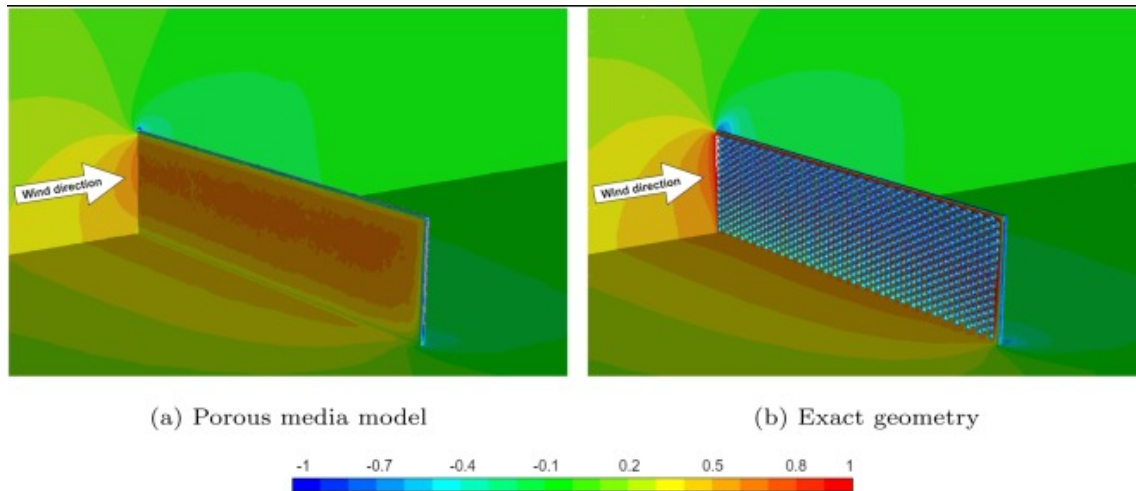


Figure 2.4: Porous media modelling approach for a microchannel heat sink (Source: Mahgoub and Ghani (2021))

- **Advantages**

Computational Efficiency: By simplifying the detailed fin geometry, the porous media approach significantly reduces the number of mesh elements, lowering computational cost and simulation time. This makes it particularly advantageous for iterative design studies and parametric optimization.

Balanced Accuracy: The method provides a reasonable trade-off between accuracy and efficiency for predicting overall heat transfer and pressure drop. It is suitable for preliminary evaluations where detailed local thermal or flow features are less critical.

- **Disadvantages**

Empirical Dependence: The model relies on empirically derived effective properties, such as permeability and thermal conductivity. The accuracy of the results is therefore dependent on the quality and applicability of these correlations.

Limited Local Detail: While effective for global performance assessment, the porous media model cannot resolve fine-scale flow structures or localized temperature gradients within the heat sink. This limits its usefulness for detailed thermal management studies where precise local conditions are important.

2.6.4 Baffles Approach

The baffles approach simplifies the heat sink fins as thin obstacles within the flow domain, guiding and obstructing the airflow without modelling the full 3D geometry, as explained by Holman (2010). These baffles are treated as solid surfaces that influence the flow path, promoting fluid mixing and enhancing heat transfer by increasing convective interactions with the heat sink surfaces.

The baffles approach can improve heat transfer by inducing turbulence and guiding flow through desired paths, as shown in Figure 2.5 CFD Land (2022).

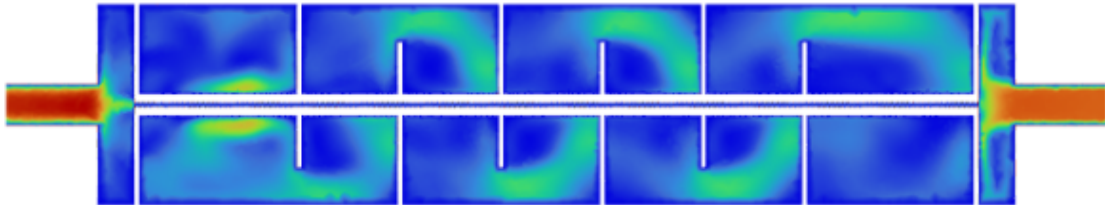


Figure 2.5: Baffles approach for enhancing shell-and-tube heat exchanger performance (Source: CFD Land, 2022).

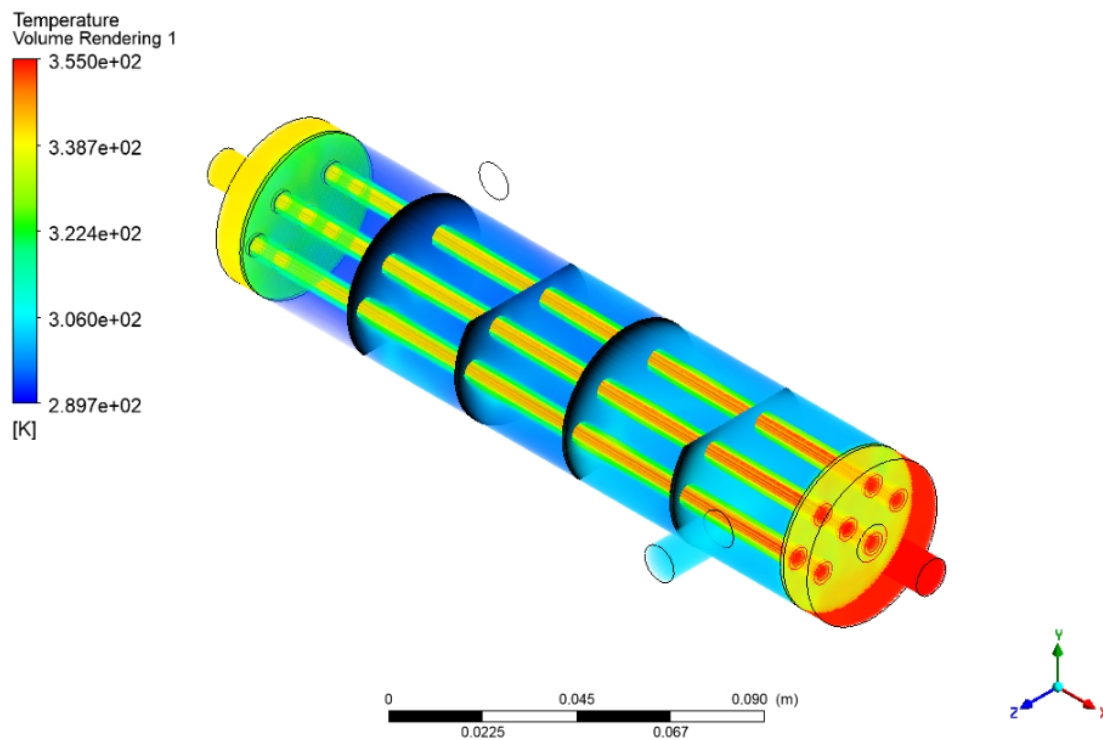


Figure 2.6: Baffles approach for enhancing shell-and-tube heat exchanger performance (Source: CFD Land, 2022).

- **Advantages**

Lower Complexity: modelling fins as baffles significantly reduces geometric and computational complexity. This lowers the number of mesh elements and computational requirements, making the method suitable for rapid prototyping and preliminary design evaluations.

Preliminary Analysis: The approach enables quick assessment of multiple design configurations and their effects on heat transfer and pressure drop. It is particularly useful for identifying general trends and guiding initial design decisions before committing to more detailed simulations.

- **Disadvantages**

Lower Accuracy: Compared to full 3D modelling, the baffles method provides less accurate predictions. The simplification may lead to errors in estimating detailed flow patterns, local temperature gradients, and precise heat transfer rates Holman (2010).

Simplified Flow Interactions: This approach cannot fully capture detailed thermal and fluid dynamic interactions, such as boundary layer development, flow separation, or local turbulence effects, which may impact the accuracy of thermal performance predictions, especially under high flow velocities.

2.6.5 Conjugate Heat Transfer (CHT) Model

The Conjugate Heat Transfer (CHT) approach simultaneously models heat transfer within the solid heat sink and the surrounding fluid, solving the governing equations for both domains concurrently Bousseth et al. (2020). By coupling conduction within the solid and convection within the fluid, this method captures the thermal interactions at the solid–fluid interface, ensuring accurate heat flux continuity and temperature prediction across boundaries. Unlike simplified approaches that treat the two domains separately, CHT simulations provide a more realistic representation of the system's thermal behavior, particularly in applications involving complex geometries or variable material properties. In the context of this study, the CHT approach enables a detailed analysis of the heat dissipation characteristics of an air-cooled rectangular fin heat sink used in the motor-controller system of the JS3 RES. This coupling between the solid and fluid domains allows for a precise evaluation of how design and flow parameters influence overall thermal management performance.

- **Advantages**

High Fidelity: CHT provides highly accurate predictions by incorporating both conduction within the solid and convection in the fluid. This comprehensive modelling ensures that thermal resistances, heat transfer coefficients, and localized temperature gradients are represented accurately, which is critical for high-performance thermal management.

Detailed Thermal Insights: The approach captures complex interactions between the fluid and solid domains, including temperature-dependent material properties, thermal contact resistances, and conjugate effects. This makes CHT particularly suitable for detailed performance evaluations and design optimization of heat sinks in confined or complex environments.

- **Disadvantages**

Computationally Intensive: CHT simulations require substantial computational resources and extended simulation times. The coupling of fluid and solid domains

increases the complexity of the numerical solution and often necessitates access to high-performance computing facilities.

Complex Setup: Setting up CHT simulations can be challenging due to the need for detailed boundary conditions, accurate material properties, and proper interface definitions between fluid and solid regions. Careful mesh generation and solver configuration are essential to achieve reliable results.

2.6.6 Dimensionality Reduction Approach

Dimensionality reduction models simplify the problem by approximating the 3D geometry using 2D planes or "slice" representations, where cross-sections are used to represent the full structure (Tian (2019)). This approach is often applied in the early stages of design and optimization, where rapid evaluation of multiple configurations is required. By reducing the model's dimensionality, computational complexity is significantly decreased, enabling faster simulations and more efficient exploration of design variations.

- **Advantages**

Quick Simulations: Reduced computational cost and shorter simulation times. Simplifying the geometry to 2D planes drastically lowers the number of mesh elements and numerical operations required.

Preliminary Design: Particularly useful for initial design studies. Engineers can quickly assess the effects of various configurations on thermal performance, enabling rapid iteration and design exploration.

- **Disadvantages**

Reduced Accuracy: Less precise for complex 3D geometries and interactions. Important three-dimensional phenomena, such as secondary flows, vortices, and localized heat transfer enhancements, may be overlooked.

Limited Insight: May fail to capture detailed thermal and flow characteristics. Local variations in temperature or flow that are critical for high-fidelity thermal management studies might be missed.

2.6.7 Summary and Selection Criteria

The selection of an appropriate CFD modelling approach for heat sink analysis requires careful consideration of multiple factors, including the required accuracy, available computational resources, complexity of the geometry, and the specific objectives of the study. Each approach presents trade-offs between computational cost, simulation time, and fidelity of results.

For applications where detailed thermal and flow predictions are critical, the full 3D approach, particularly the Conjugate Heat Transfer (CHT) model, is preferred. CHT simulations account for both conduction in the solid heat sink and convection in the fluid domain, providing high-fidelity predictions of temperature distribution, heat fluxes, and potential hotspots. This approach is essential for final design verification, certification purposes, or when evaluating designs operating under extreme thermal conditions.

In contrast, reduced-order approaches such as the thin-walled, porous media, baffles, or dimensionality reduction models are advantageous for early-stage design exploration and parametric studies. These methods simplify the geometry or homogenize the domain, substantially lowering computational requirements while still providing reasonable estimates of overall thermal performance. They are particularly useful for screening multiple design alternatives, performing optimization studies, or conducting sensitivity analyses when rapid results are necessary.

The choice of modelling approach should therefore be guided by a combination of project goals and constraints. High-fidelity models are best suited for final validation and critical applications, whereas simplified models provide efficient tools for iterative design and preliminary evaluations. In practice, a hierarchical approach is often employed: initial designs are screened using simplified methods, and the most promising configurations are then analyzed with detailed CHT simulations to ensure accurate performance predictions under operational conditions.

2.7 Innovations in Heat Sink Design

The evolution of heat sink design has been largely driven by the increasing miniaturization of electronic devices and the higher power densities they generate, both of which impose stringent requirements on thermal management. To provide a complete overview of current research and trends in the field, this section briefly summarizes notable innovations in heat sink design reported in the literature. It should be noted that this study does not propose or implement new heat sink designs; the discussion here is intended solely for context and completeness.

2.7.1 3D Printing

Additive manufacturing, commonly referred to as 3D printing, allows engineers to fabricate complex geometries and intricate internal structures that are difficult or impossible to achieve with conventional manufacturing methods. This capability enables the design of customized heat sinks with optimized fin shapes, microchannels, and enhanced airflow paths, which improve thermal performance while minimizing material usage and overall weight Günel and Yılmaz (2018). Additionally, 3D printing supports rapid prototyping and iterative design processes, facilitating faster development cycles for advanced thermal management solutions.

2.7.2 Advanced Materials

The adoption of advanced materials has further expanded the potential of heat sink designs. Phase-change materials (PCMs) can absorb and release large amounts of heat during phase transitions, effectively managing transient thermal loads in devices with fluctuating power levels. Composite materials, on the other hand, combine high thermal conductivity with lightweight and mechanically robust substrates, providing an optimal balance between thermal performance, weight, and durability Zeng et al. (2012). These materials also enable hybrid designs, where multiple materials are integrated to leverage their respective strengths, resulting in heat sinks that are both highly efficient and practical for real-world applications.

2.7.3 Applications

The rapid advancement of electronic devices, driven by increasing miniaturization and higher power densities, presents significant thermal challenges. Traditional cooling solutions are often insufficient for managing the heat generated by compact, high-performance components. In response, innovative approaches such as 3D printing and the use of advanced materials have become essential for enhancing heat sink designs.

Additive manufacturing, or 3D printing, enables the fabrication of highly complex geometries that are difficult or impossible to achieve with conventional manufacturing techniques. This capability allows engineers to design heat sinks with optimized fin structures and airflow channels, improving heat dissipation efficiency and meeting the specific thermal demands of modern electronics. Furthermore, 3D printing facilitates customization for individual applications, offering flexibility to optimize thermal management solutions according to device-specific constraints, including size, weight, and geometry.

In parallel, the adoption of advanced materials, such as phase-change materials (PCMs)

and high-performance composites, is revolutionizing heat sink performance. PCMs are particularly advantageous because they absorb and release substantial amounts of heat during phase transitions, providing effective thermal buffering in systems with fluctuating power levels. This capacity to stabilize temperature variations makes them ideal for mitigating thermal peaks in electronic devices. Composite materials combine the high thermal conductivity of metals with the lightweight, durable properties of non-metallic substrates, offering an excellent balance between performance, weight, and structural practicality.

Together, 3D printing and advanced materials enable the development of heat sinks that not only meet the growing thermal demands of modern electronics but also optimize space and weight constraints, resulting in more efficient, compact, and reliable thermal management solutions suitable for applications ranging from portable electronics to aerospace systems.

2.8 Comparative Analysis of Cooling Systems

Comparative studies of heat sink models and cooling systems are crucial for understanding the performance, benefits, and limitations of different approaches in electronic applications. These analyses evaluate a range of design parameters, including geometry, material properties, thermal conductivity, and overall system configuration, to identify the most efficient solutions for specific thermal management challenges.

By systematically comparing different designs, engineers can assess key factors such as thermal performance, pressure drop, material selection, manufacturability, and cost-effectiveness. Such studies also help highlight trade-offs between cooling efficiency and other constraints, including system size, weight, and energy consumption.

Ultimately, comparative analysis provides valuable insights for optimizing thermal management strategies and selecting the most appropriate heat dissipation solutions for applications such as high-performance computing, power electronics, and compact mobile devices. Through rigorous evaluation of various cooling technologies, engineers can determine the most effective design approach for each application, ensuring reliable operation, enhanced component longevity, and consistent performance under diverse operating conditions.

2.8.1 Air-Cooled vs. Water-Cooled Systems

Comparative studies of air-cooled and water-cooled heat sinks provide valuable insights into the advantages and limitations of each approach for various electronic applications. Research (Zhang et al. (2018)) has demonstrated that water-cooled systems generally offer superior thermal performance, particularly in high-power applications where heat

dissipation demands are substantial. Water-cooling leverages the high specific heat capacity of water, enabling more efficient heat absorption and transfer than air-cooled systems. This makes water-cooled solutions particularly suitable for applications such as data centers, high-performance computing, and power electronics, where maintaining low operating temperatures is critical for performance and reliability.

Despite their enhanced cooling efficiency, water-cooled systems involve greater complexity, cost, and maintenance requirements. They often necessitate additional components such as pumps, heat exchangers, and fluid channels, which increase both initial installation costs and long-term operational overhead. Integration into existing systems can also be challenging, requiring careful management of fluid flow and leak prevention.

In contrast, air-cooled systems are simpler, more cost-effective, and easier to maintain. They rely on natural convection or forced airflow from fans to dissipate heat, making them practical for lower-power or compact electronic devices where thermal management requirements are less stringent. Air-cooled solutions are widely used in consumer electronics due to their ease of integration and lower overall cost.

Ultimately, the choice between air-cooled and water-cooled systems depends on the specific thermal requirements, spatial constraints, and budgetary considerations of the application. While water cooling provides superior performance for high-power devices, air cooling remains a robust and practical solution for many lower-power and space-limited scenarios.

2.8.2 Comparative Studies on Fin Geometries and Orientations

Recent research into the optimization of fin geometries for CPU and electronic cooling has highlighted the critical influence of geometric configurations on thermal performance. For example, Bangalee et al. (2021) demonstrated that thermal performance improves when fin channels are oriented perpendicular to the surface parallel to the outlet. Configurations with perpendicular fin channels resulted in lower average system temperatures, indicating more efficient heat dissipation compared to alternative orientations.

As shown in Figure 2.7, heat sinks can adopt a variety of fin geometries, including straight, pin, and flared designs. Each geometry affects the surface area, airflow distribution, and overall thermal performance of the heat sink. Straight fins are simple and efficient for uniform airflow, pin fins enhance turbulence and local heat transfer, and flared or tapered fins can improve cooling in constrained spaces. These variations highlight the importance of selecting an appropriate fin design based on system requirements and available volume Kumar et al. (2021).

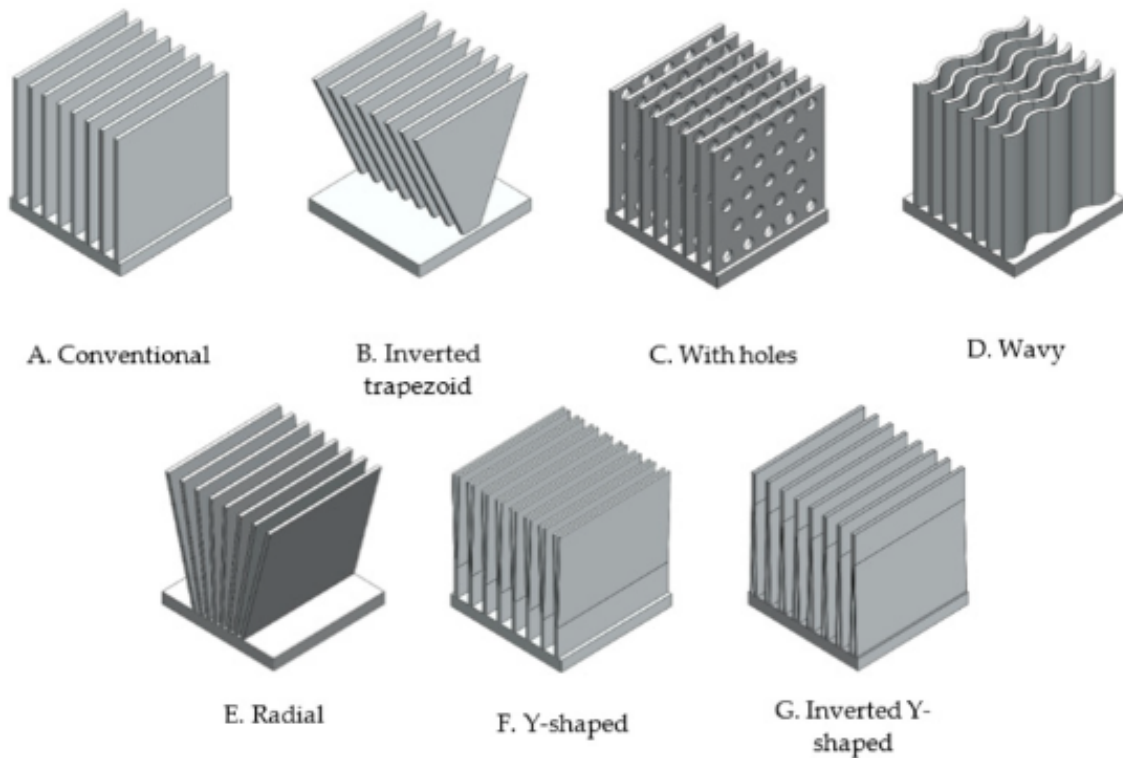


Figure 2.7: Different fin geometries of heat sinks (adapted from Kumar et al. (2021))

2.8.3 Applications

Comparative analyses of different cooling solutions are vital for providing engineers and designers with a comprehensive understanding of their performance characteristics. By evaluating parameters such as power dissipation, spatial constraints, and cost-effectiveness, engineers can make informed decisions on the most appropriate thermal management strategy for a given application. Thorough comparisons of various cooling system designs allow for the selection of solutions that balance thermal efficiency, reliability, and operational practicality. This approach ensures that electronic devices operate within safe temperature ranges, enhancing longevity, performance, and overall system stability across diverse operating conditions.

2.9 Heat Pipe and Vapour Chamber Technology and Numerical Modelling

Heat pipes are passive thermal devices that transfer heat efficiently by leveraging the phase change of a working fluid. Internally, heat is absorbed at the evaporator section, causing the liquid to vaporize; the vapour then travels along a sealed channel to the condenser section, where it releases heat and condenses back to liquid. The liquid is returned to the evaporator by capillary action through a wick structure or by

gravity, enabling continuous thermal transport with minimal temperature gradient. This mechanism allows heat pipes to achieve effective thermal conductivities several orders of magnitude higher than typical solid metals, making them highly attractive for compact, weight-constrained systems such as the JS3 RES motor-controller cooling system.

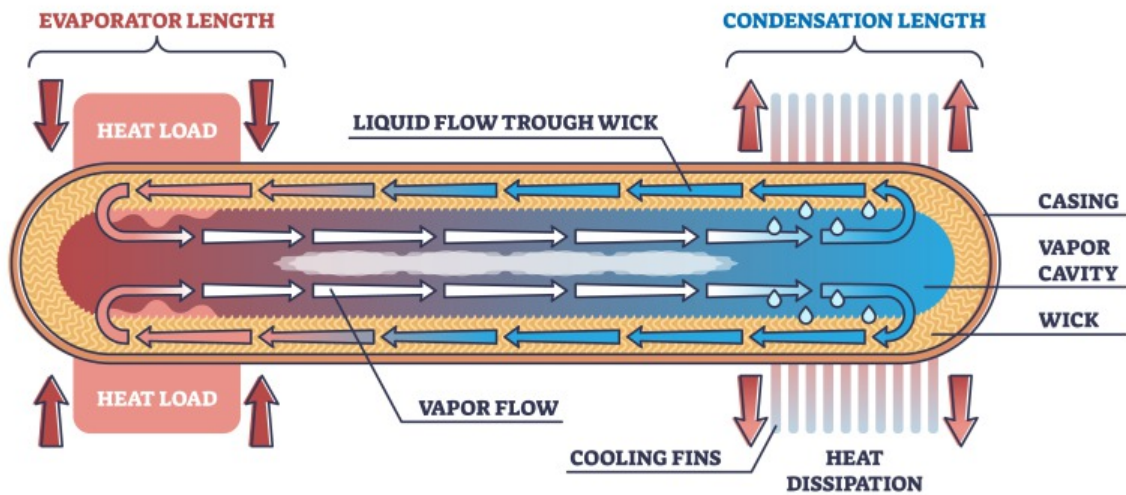


Figure 2.8: Functional principle of heat pipes. Adapted from Dai Nippon Printing Co., Ltd., “The Basics of Heat Pipes – Their History, Principle, and Varieties Explained” Dai Nippon Printing Co. (2022).

In addition to standard cylindrical heat-pipes, planar variants called vapour chambers employ the same physical principle but spread heat more uniformly in two dimensions rather than simply transmitting it along one axis. Due to their thin profile and high in-plane thermal conductivity, these are increasingly used in electronic cooling applications and are highly relevant to our motor-controller cooling system modelling.

2.9.1 Vapour Chambers

Vapour chambers are essentially flat or plate-type heat pipes (sometimes called thermal ground planes) where the ratio of width to thickness is large, enabling rapid heat spreading across a plane before dissipation. They essentially perform two functions: (1) distribute heat laterally (spread) from a concentrated source, and (2) transfer that heat to a fin or external surface for convection. Because of their geometry, vapour chambers are often used in electronics cooling (e.g., power modules, CPUs) and increasingly in aerospace/compact systems where space and thickness are constrained. In the context of the JS3 RES cooling system, a vapour-chamber style concept (or flat heat pipe) could be beneficial to spread heat before fins or airflow regions, thereby reducing local temperature hot spots and improving uniformity.

2.9.2 CFD / Numerical Modelling Approaches

Despite wide usage of heat pipes and vapour chambers, accurate numerical modelling of them within CFD frameworks remains challenging due to several coupled phenomena: multiphase flow, phase-change (evaporation/condensation), capillary flow in wick structures, thin film dynamics, orientation and gravity effects, contact resistances, and the interaction with ambient convection. Several approaches have been reported:

- **Effective Thermal Conductivity (ETC) Models:** Simplify the heat pipe or vapour chamber as a solid domain with an artificially high or effective thermal conductivity, capturing the steady-state heat transfer but neglecting transient, orientation- or wick-specific effects.
- **Equivalent Thermal Resistance Networks:** Represent the evaporator, adiabatic, and condenser sections (or spreading and wick sections in a vapour chamber) as lumped resistances. Useful for quick estimates but offers limited spatial resolution for complex geometries.
- **Porous Media or Source-Term Models:** Approximate the wick and vapour flow regions via porous media modelling or introduce source terms for phase change (evaporation/condensation) and capillary return. This provides more detail but at higher computational cost and calibration effort.
- **Full Multiphase CFD Models:** Resolve the vapour-liquid flow, phase change, wick fluid flow, and interface phenomena directly. This is the most physically accurate but typically computationally prohibitive for large/complex systems or iterative design studies.

In the context of vapour chambers, recent CFD studies have examined spreading thermal resistance and effective conductivity for flat geometries Kim et al. (2022). Similarly, detailed models of heat pipes in aerospace cooling systems have been published Scigliano et al. (2024), which show effective conductivities and simulation frameworks.

2.9.3 Research Gaps and Motivation

Although many analytical models and simplified numerical approaches exist, a widely validated CFD methodology suitable for embedded heat pipes and vapour chambers in compact, thermally constrained aerospace systems remains elusive. Key gaps include:

- Modelling of the combined effect of embedded heat pipes/vapour chambers, fin geometry, airflow constraints, and contact resistances within compact fuselage compartments.

- Transient behaviour and the impact of manufacturing / assembly variation (e.g., clamping force, wick porosity, fill fraction) on performance.
- Computational efficiency: balancing sufficient physical fidelity for certification / engineering use with acceptable simulation time and resource use.
- Specific modelling of vapour chambers connected to fin/airflow systems in constrained 3D geometries under aerospace loads.

This research, therefore, targets those gaps by developing and validating a CFD - compatible modelling method that explicitly incorporates the embedded heat pipe and vapour chamber concept within the motor-controller cooling system of the JS3 RES. The outcome is intended not just for this specific case, but as a methodology for future lightweight aerospace thermal systems.

2.10 Literature Review Summary

Effective thermal management is essential for ensuring the reliability, performance, and longevity of electronic systems, particularly as power densities increase and device miniaturisation continues. Optimised heat sink design, leveraging the fundamental principles of conduction, convection, and radiation, is central to this process. Advanced computational methods, including numerical simulations and modelling techniques such as CFD and TAS, allow engineers to evaluate and refine heat sink designs, providing detailed insights into thermal performance and enabling the prediction of potential bottlenecks.

In addition to conventional heat sinks, passive thermal devices such as heat pipes and vapour chambers have emerged as critical components in compact, weight-constrained systems. Heat pipes transfer heat efficiently via phase change and capillary-driven fluid return, achieving effective thermal conductivities orders of magnitude higher than solid metals. Vapour chambers, flat or plate-type heat pipes, enable rapid lateral heat spreading, reducing hotspots and improving temperature uniformity. Both technologies are widely applied in aerospace, electronics, and high-power devices, particularly where forced airflow is limited or weight and reliability constraints preclude active liquid cooling.

Despite their practical advantages, accurate numerical modelling of heat pipes and vapour chambers remains challenging due to multiphase flow, phase-change dynamics, capillary action, and orientation sensitivity. Simplified approaches, such as effective thermal conductivity models, resistance networks, and porous-media representations, are commonly used in CFD studies. Full multiphase models offer high accuracy but are computationally prohibitive, particularly in iterative design studies or compact geometries. This highlights a gap in validated CFD methodologies capable of reliably predicting embedded heat pipe and vapour chamber performance, motivating research

into efficient and accurate modelling approaches suitable for aerospace and electronics applications.

2.11 Key Insights

Several key insights have emerged from this literature review, contributing to a comprehensive understanding of heat sink and passive thermal device design:

- **Principles of Heat Transfer:** Understanding conduction, convection, and radiation remains fundamental for designing efficient heat sinks and integrating heat pipes or vapour chambers, ensuring reliable operation across varying thermal loads.
- **Material Selection:** Choice of materials such as aluminium, copper, and emerging high-conductivity composites (e.g., graphene, carbon nanotubes) directly affects thermal performance, weight, and manufacturability. For heat pipes and vapour chambers, working fluid properties, wick structures, and container materials additionally influence overall effectiveness.
- **Advanced Design and Manufacturing:** Additive manufacturing enables complex geometries for heat sinks and integrated heat pipes or vapour chambers, enhancing surface area, fin configurations, and heat spreading capabilities that are difficult with traditional machining.
- **Thermal Analysis and Simulation:** Computational tools such as CFD and TAS are indispensable for predicting thermal performance, including the interaction of heat sinks with embedded heat pipes and vapour chambers. Verification and validation against experimental or flight data remain crucial for model reliability.
- **Comparative Analysis:** Systematic comparisons between conventional air-cooled heat sinks, liquid-cooled systems, and passive devices like heat pipes or vapour chambers inform design trade-offs between thermal performance, complexity, weight, and cost.

2.12 Future Directions

Future research in thermal management for electronics and compact aerospace systems is expected to focus on several key areas:

- **Integration of Heat Pipes and vapour Chambers:** Developing validated, CFD-compatible models for embedded heat pipes and vapour chambers is essential. Research should explore computationally efficient methods that capture

phase-change dynamics, capillary return, transient behaviour, and spatial heat spreading in constrained geometries.

- **Advanced Materials:** Continued exploration of high-thermal-conductivity materials, including composites, phase-change materials, and advanced alloys, will support heat dissipation and enhance the performance of both conventional heat sinks and passive thermal devices.
- **Miniaturization and Integration:** With increasingly compact devices, integrated cooling solutions that combine heat sinks, heat pipes, and vapour chambers will be necessary to manage high thermal loads in limited form factors.
- **Sustainability:** Eco-friendly materials, energy-efficient manufacturing, and optimized design to minimize material use will become increasingly important in future heat sink and passive device development.
- **Automation and Artificial Intelligence:** AI-assisted design tools can optimize geometries, material selection, and integration of heat sinks, heat pipes, and vapour chambers. Machine learning can accelerate the design process, predict performance, and reduce reliance on trial-and-error physical testing.

In summary, future thermal management research will focus on combining advanced materials, passive heat transfer devices, compact integration strategies, and intelligent optimization methods to achieve highly efficient, reliable, and sustainable solutions for modern electronics and aerospace applications.

2.13 Chapter Conclusion

This literature review provided a comprehensive overview of heat sink design, passive thermal devices, and thermal management strategies in electronic and aerospace systems, with particular emphasis on their relevance to electric sailplane applications. The review highlighted the fundamental principles of heat transfer - conduction, convection, and radiation - and their critical role in guiding effective thermal management solutions.

Material selection, from conventional options such as aluminium and copper to advanced materials like graphene, carbon nanotubes, and phase-change composites, was shown to directly influence thermal performance, weight, and manufacturability. Similarly, advanced manufacturing techniques, particularly additive manufacturing, enable complex geometries that enhance heat dissipation while minimizing material use. Passive thermal devices, including heat pipes and vapour chambers, were identified as essential components in compact, weight-constrained systems due to their efficient, passive heat transport. However, accurate modelling of these devices remains challenging because of multiphase flow, phase-change phenomena, capillary effects, and orientation sensitivity.

Computational tools, particularly Computational Fluid Dynamics (CFD), were highlighted for their ability to predict, analyze, and optimize heat sink performance under varying operating conditions. While simplified models such as effective thermal conductivity or resistance networks are often employed for passive devices, gaps exist in validated CFD approaches that can reliably capture transient behavior, localized temperature gradients, and orientation effects in compact geometries.

The literature review indicates that designing and accurately simulating heat sinks for aerospace applications requires a careful combination of heat transfer fundamentals, material selection, manufacturing considerations, and computational modelling. These insights provide a clear foundation for the methodology employed in this thesis, which focuses specifically on the validated CFD modelling of an air-cooled motor-controller heat sink for the JS3 RES electric sailplane. The subsequent chapters build on this knowledge to develop a reliable, application-specific CFD model, ensuring that the thermal performance predictions align with experimental validation.

In summary, this review establishes the necessary background for applying and validating CFD techniques to a practical heat sink design, rather than developing a general framework, and highlights the critical considerations for high-fidelity thermal analysis in electric sailplane applications.

Chapter 3

Methodology

3.1 Overview

The air-cooled motor-controller system for the JS3 RES aircraft dissipates approximately 1.5 kW of energy, with incoming cooling air temperatures reaching up to 38 °C, within the confined fuselage volume of the airframe where both mass and available airflow are limited. Accurate thermal management is therefore critical to ensure safe and reliable operation.

Initially, several CFD modelling approaches were considered, including fully resolved three-dimensional (3D) geometries, thin-wall approximations, porous media representations, baffle models, and conjugate heat transfer (CHT) formulations. A comprehensive comparison of these approaches was conducted in Chapter 2, Section 2.6, where their respective advantages and limitations were evaluated with respect to accuracy, computational cost, and suitability for compact heat sink applications.

The analysis in Section 2.6 indicates that simplified methods such as porous media and thin-wall models reduce computational expense but rely on homogenisation assumptions that may limit their ability to capture localised flow structures and hotspot formation within compact fin geometries. Given the objective of this study to resolve detailed flow–thermal interactions within the motor-controller heat sink and to validate results against experimental data, a fully resolved 3D conjugate heat transfer approach was selected. This method provides the highest fidelity for the specific objectives of this study, as it directly resolves the three-dimensional flow field and solid–fluid heat transfer without introducing empirical homogenisation parameters.

Moreover, with the rapid growth of computational resources and high-performance computing capabilities, the limitations that previously necessitated lower-accuracy models are diminishing. This trend further supports the choice of full 3D CFD as the preferred method, as it enables highly detailed simulations with increasingly manageable computational costs.

Consequently, this study focuses exclusively on the fully resolved 3D CHT modelling approach. This method allows for a comprehensive representation of the geometry, including intricate fin structures, thermal interface materials, and detailed airflow paths, ensuring precise predictions of temperature distribution, heat flux, and overall thermal performance. By concentrating on a single, high-fidelity method, the study maximizes the accuracy of the analysis and provides a robust basis for potential future evaluation or optimization of the heat sink design, without performing a redesign within the scope of this thesis. All detailed information, including CAD drawings, fan specifications, and material datasheets, is provided in Appendix B.

3.2 Full 3D Approach

The full 3D CFD approach entails constructing a detailed, three-dimensional model of the heat sink, airflow, and all associated components within the system. This method simulates fluid flow and heat transfer in all three spatial dimensions, capturing complex interactions between the solid and fluid domains with the highest possible fidelity.

The full 3D approach is particularly advantageous for resolving intricate flow phenomena, including turbulence, secondary flows, recirculation zones, and detailed heat transfer between surfaces. These phenomena are critical for accurate predictions of temperature distribution, pressure drop, and overall thermal performance, especially in compact and high-power systems such as the JS3 RES motor-controller.

The primary limitation of this approach is its computational cost. High-fidelity 3D models require fine meshes and high spatial resolution, resulting in longer simulation times and significant demands on memory and processing power. However, with the continuous advancement of computational resources and high-performance computing, these limitations are becoming less restrictive. Consequently, full 3D simulations are increasingly feasible, offering accuracy and detailed insights that lower-fidelity models cannot achieve.

For the JS3 RES system, where precise thermal predictions are essential and the geometry is highly constrained, the full 3D CFD approach provides a robust and reliable method for evaluating and optimizing the heat sink design. While simpler methods may still be useful for rapid preliminary analyses, the high-resolution 3D simulations ensure that design decisions are based on the most accurate thermal data available, maximizing system reliability and performance.

3.3 CFD Modelling

For this study, Siemens Star CCM+ was employed using the fully resolved 3D approach in conjunction with conjugate heat transfer (CHT). Star CCM+ was chosen over alternative CFD packages, such as ANSYS Fluent, due to its robust handling of complex geometries, automated polyhedral meshing capabilities, and seamless integration with CAD models, which simplifies the import of NX files representing the motor-controller assembly. Additionally, the solver's coupled heat transfer algorithms and strong convergence monitoring are well suited to capture the tightly coupled solid–fluid interactions present in the heat sink and surrounding airflow domains.

The 3D CFD model includes all relevant components of the motor-controller system, such as the heat sink, air inlets, and surrounding airflow domains. This comprehensive representation allows the simulation to accurately capture the coupled heat transfer between solid and fluid regions, accounting for conduction within the heat sink, convection to the surrounding air, and thermal interactions with nearby components. This setup ensures that the simulated temperature distributions, pressure drops, and flow patterns closely reflect the actual operating conditions, providing a reliable basis for detailed thermal analysis of the JS3 RES system.

The characteristic airflow velocities and short fin-channel lengths resulted in forced convection dominating the thermal-fluid behaviour of the system. Consequently, the influence of gravity-induced buoyancy effects on the overall thermal performance was considered negligible and was not included in the simulations.

- **Heat Pipes:** Sealed and evacuated copper heat pipes were integrated into the design to enhance lateral heat distribution within the heat sink. Extending nearly the full width of the heat sink, the heat pipes significantly improve thermal performance by conducting heat away from the hot plate and redistributing it into the fin array. Full contact with both sides of the heat sink is ensured through a precisely machined press-fit interface, eliminating the need for additional thermal paste. Each heat pipe measures approximately 200 mm in length and 8 mm in diameter.

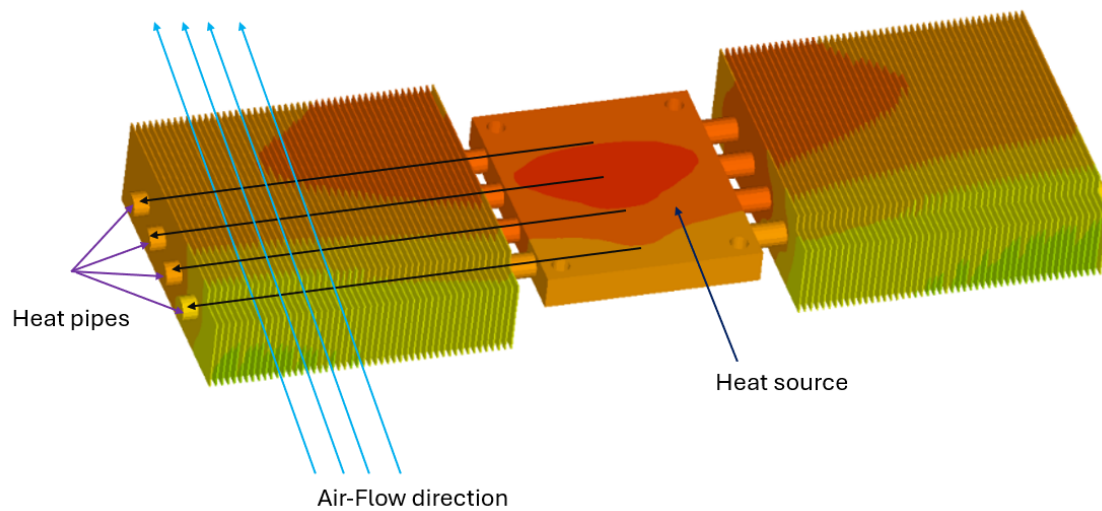


Figure 3.1: Illustrative heat-pipe heat-spreading principle. <https://myheatsinks.com>

As shown in Figure 3.1, the schematic illustrates how heat from the motor-controller is conducted laterally through the heat pipes from the hot plate to the fin array. The heat pipe axes are perpendicular to the airflow direction, enabling efficient redistribution of heat along the fins before it is convected away by the incoming air. While the figure is conceptual and does not represent the exact geometry of the JS3 RES heat sink, it effectively demonstrates the intended heat-spreading function of the heat pipes. Without the heat pipes, heat would not spread laterally fast enough, leading to higher temperatures near the center of the heat sink, while the outer fins contribute relatively little to heat transfer.

Heat Pipe Working Principle: Heat pipes act as highly efficient thermal conductors by utilising a phase-change mechanism. When heat is applied at the evaporator section, the working fluid—de-ionised water in this case—evaporates and the resulting vapour travels towards the cooler condenser section. There, it releases heat and condenses back into liquid. A sintered wick lining the inner wall provides a capillary pathway that returns the liquid to the evaporator, enabling a continuous evaporation–condensation cycle. This process allows a heat pipe to transport large amounts of heat with minimal temperature gradients along its length, making it highly effective for distributing heat across the heat sink.

Modelling Approach in This Thesis: Accurately modelling the internal operation of a heat pipe—including the internal hollow geometry, phase-change behaviour, vapour transport, liquid return, and transient two-phase flow—is exceptionally challenging and remains an active research topic with limited high-fidelity CFD implementations. Achieving such a level of detail would require advanced multiphase modelling, specialised numerical methods, and extensive validation, making it more suitable for future dedicated research. Therefore, in this thesis, the heat pipes are modelled as solid copper rods with a high effective thermal conductivity. The exact thermal conductivity value used is based on the manufacturer’s datasheet provided

in Appendix B and is discussed in detail at a later stage. This simplification provides a computationally efficient yet physically reasonable approximation of the heat-spreading effect of the actual heat pipes.

- **Heat Sink:** The heat sink is an aluminium assembly designed to accommodate the embedded heat pipes. While some units were manufactured as two-part assemblies, others are produced as a single piece with the heat pipes pressed directly into the body. The single-piece version is primarily used in the JS3 RES system and offers superior thermal performance due to the intimate press-fit contact between the heat pipes and the aluminium body, improving lateral heat spreading compared to the two-part design.

The heat sink considered in this study measures $179.6\text{ mm} \times 213.36\text{ mm} \times 35.56\text{ mm}$ (L $\times$ W $\times$ H), featuring 47 fins that are 1 mm wide and 24 mm high, separated by 3 mm air gaps. CNC machining ensures precise fin geometry and proper integration of the heat pipes.

Figures 3.2 and 3.3 show, respectively, the original CAD model and a photograph of the heat sink installed in the aircraft. The CAD model provides a clear view of the intended geometry and fin layout, while the actual photo illustrates the real-world integration of the heat pipes and the final assembly. Including both images provides context for the geometry used in CFD simulations and the actual hardware constraints.

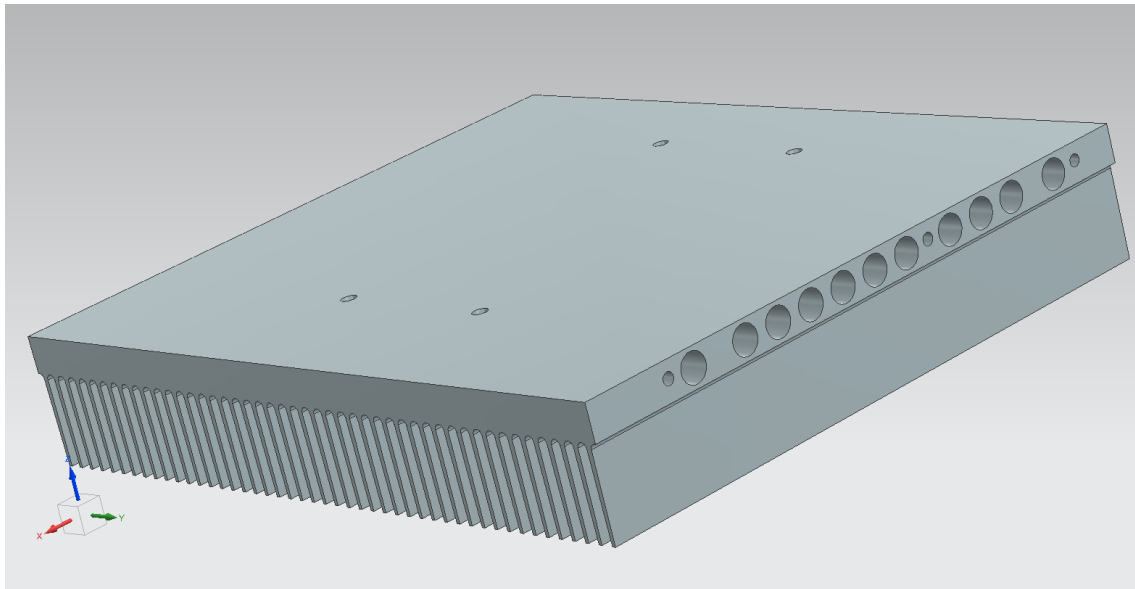


Figure 3.2: Original heat sink design from NX

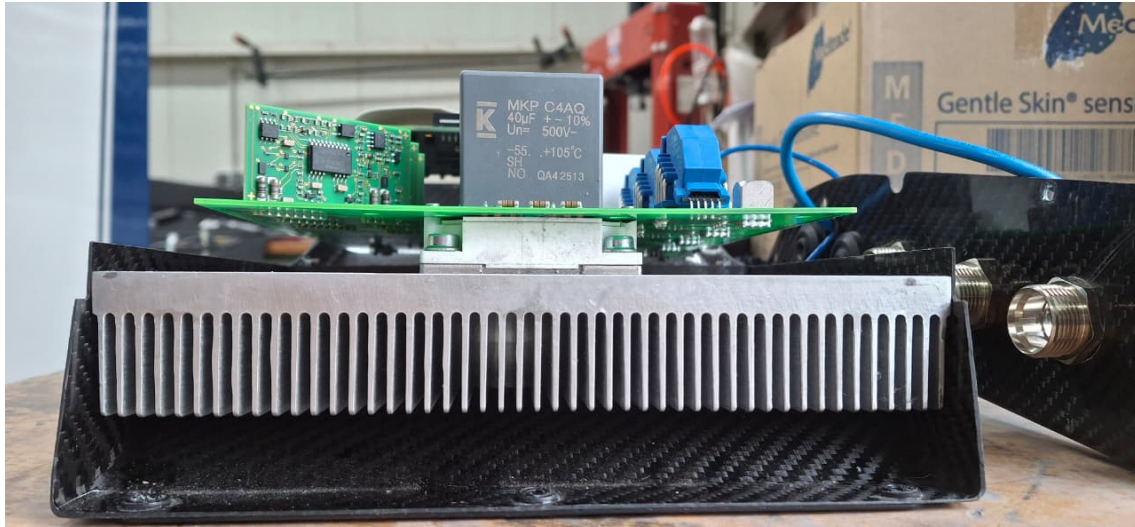


Figure 3.3: Actual heat sink installed in the aircraft

Note: For the purposes of CFD simulations in this study, the heat sink is modelled as the single-piece version with press-fit heat pipes, as this configuration is most commonly used and provides better heat transfer characteristics than the two-part design. This simplification also facilitates meshing and computational efficiency while still accurately capturing the essential thermal behavior.

- **Heat Source:** The copper heat source measures $122 \text{ mm} \times 62 \text{ mm} \times 15.6 \text{ mm}$ ($L \times W \times H$) and dissipates approximately 1.5 kW under maximum operating conditions. The design requirement specifies that its temperature must remain below 75°C after 90 s of maximum power operation at a 40 kW motor-controller load. The heat source is mechanically fastened to the heat sink to ensure adequate thermal contact. No internal airflow modelling is performed within the heat source itself.

Considering screw holes and slightly rounded edges, the effective contact area is approximately 6998 mm^2 , resulting in an average heat flux of about 0.214 W/mm^2 . This value is significantly lower than the heat flux typically encountered in high-performance CPU applications, indicating a moderate but spatially concentrated thermal load.

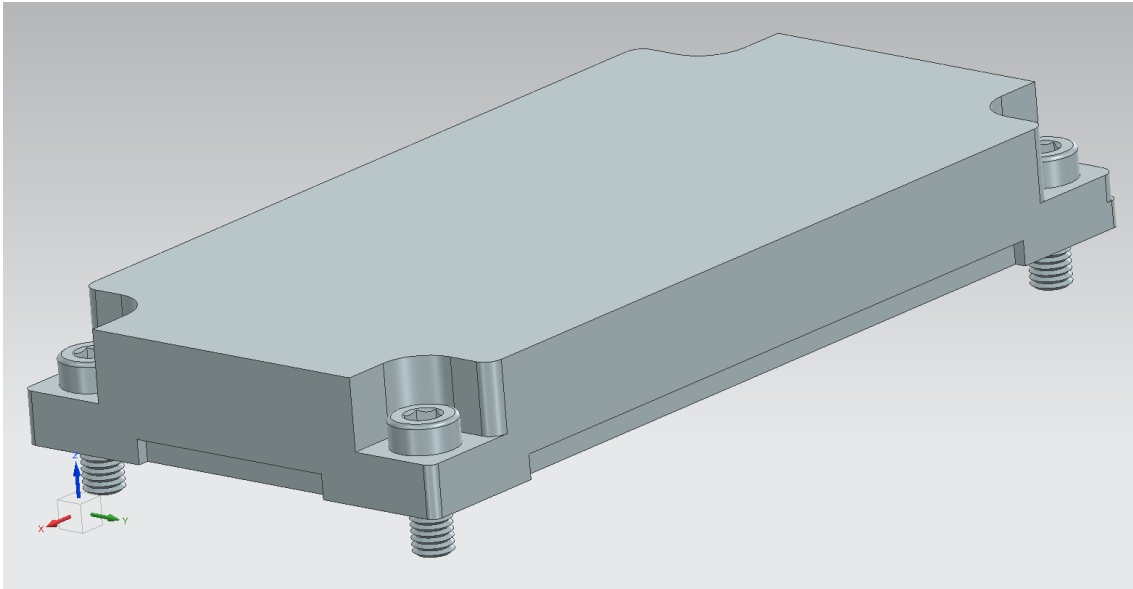


Figure 3.4: Original heat source design from NX

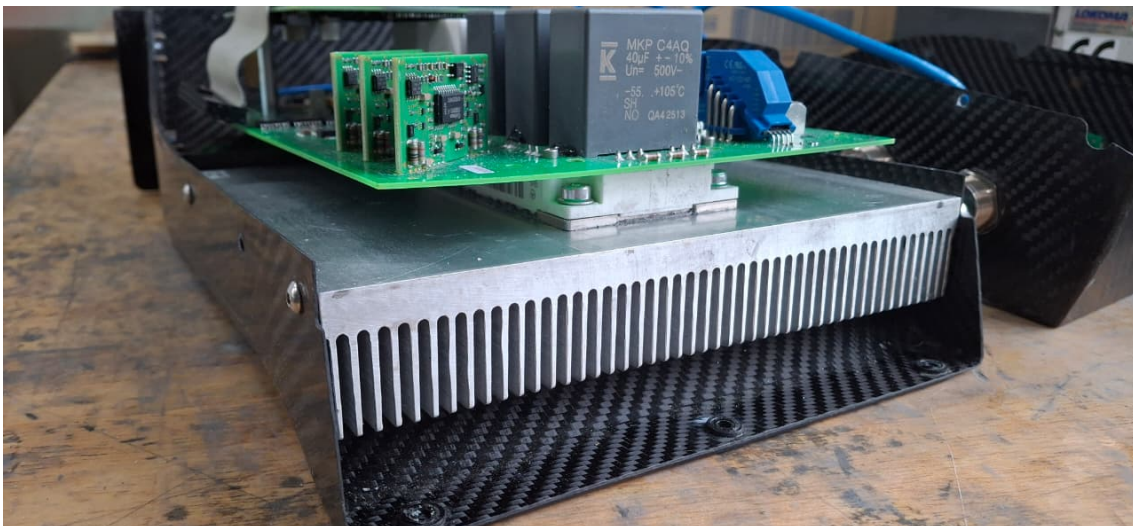


Figure 3.5: Heat source integrated between heat sink and power electronics

Figures 3.4 and 3.5 illustrate the heat source geometry and its integration within the motor-controller assembly. The CAD model defines the geometry used for CFD modelling, while the photograph shows the actual installation within the aircraft. In the assembled configuration, the heat source is positioned between the aluminium heat sink and the printed circuit board carrying the IGBTs. This stacked arrangement illustrates the confined spatial constraints inside the airframe and clarifies the direct thermal path from the power electronic components to the heat sink.

- **Flow Domain:** The flow domain includes the internal air volume surrounding the motor controller, covering both the upper and lower regions as well as the inlet and outlet sections. Two high-performance axial fans operate in a pull configuration and

are positioned downstream of the heat sink. This configuration was selected due to spatial constraints as well as cockpit noise and safety considerations. The total enclosed flow volume is approximately $2,523,385 \text{ mm}^3$ and is filled with ambient air.

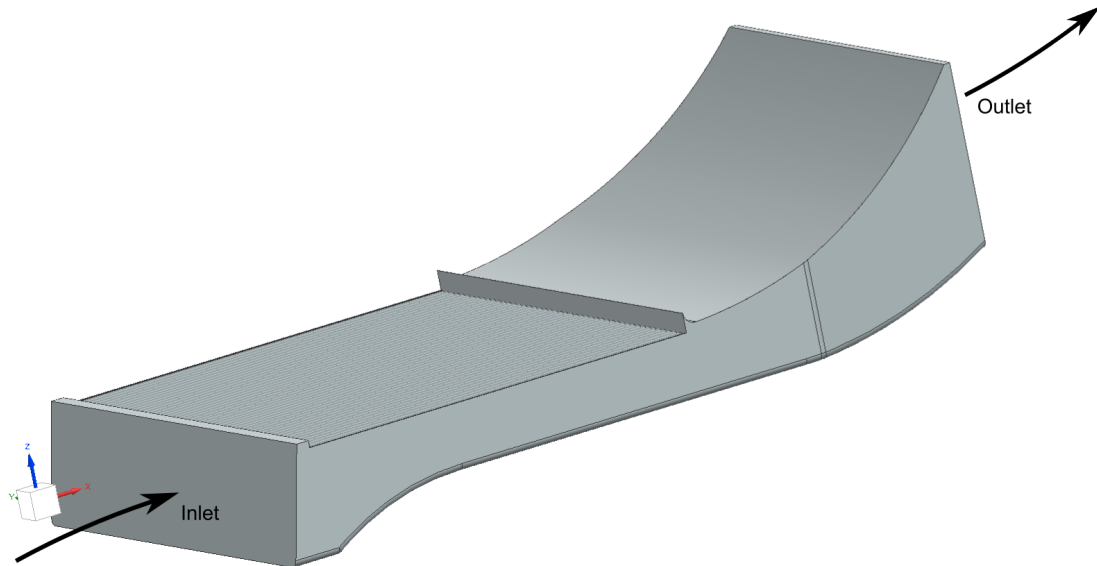


Figure 3.6: CFD flow domain defined in Siemens NX (airflow from left to right)

Figure 3.6 illustrates the computational air volume used for the CFD simulations. Air enters from the left, flows through the heat sink fin array, and is extracted on the right-hand side by the downstream fans.



Figure 3.7: Motor-controller packaging inside the fuselage (cockpit-side view)



Figure 3.8: Complete motor-controller assembly inside the fuselage (side view, airflow from left to right)

Figures 3.7 and 3.8 show the installed configuration within the aircraft fuselage. The images emphasize the confined installation space and illustrate the physical airflow path through the controller. Cooling air enters the fuselage section from the left, passes through the heat sink and embedded heat pipes, and is subsequently drawn through the fans before exiting downstream. The resulting thermal flow path is therefore: IGBTs → heat source → heat sink → airflow → fans → outlet.

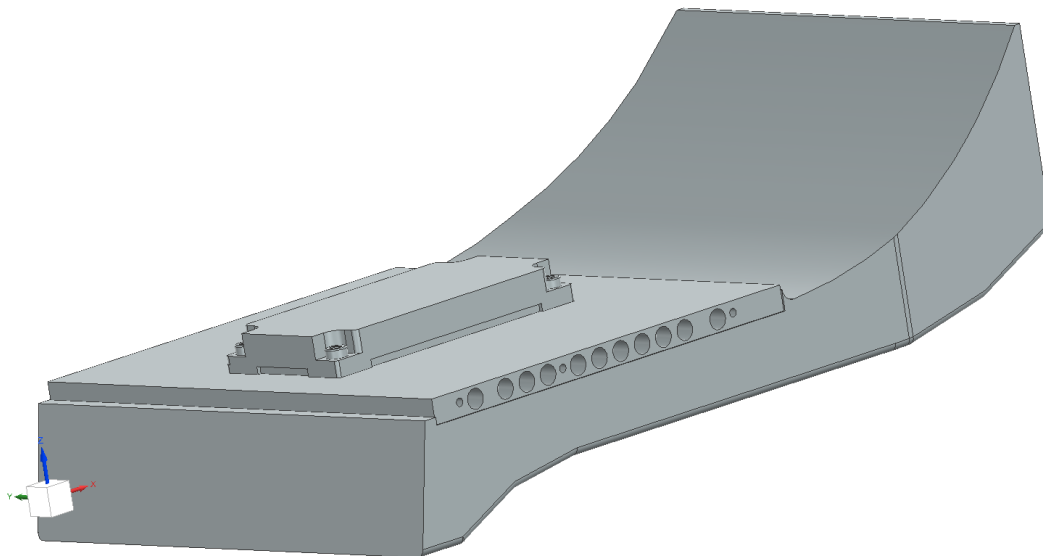


Figure 3.9: Complete original design from NX

All components were initially designed, assembled, and spatially constrained in Siemens NX to ensure correct alignment and mechanical integration within the airframe. The complete assembly was then imported into Siemens Star CCM+ as individual Parasolid bodies, preserving geometric accuracy and relative positioning. This workflow ensures that the CFD model faithfully represents the real installation

conditions and captures the influence of packaging constraints on the cooling performance.

3.3.1 Numerical Implementation and Initial Setup

After importing the geometries into Simcenter STAR-CCM+, dimensional and positional accuracy was verified. Proper thermal and mechanical continuity between components was ensured through the definition of face-to-face contacts, with imprints applied where necessary to generate conformal interfaces and avoid discontinuities.

The conjugate heat transfer (CHT) setup in Simcenter STAR-CCM+ required the definition of multiple fluid–solid interfaces through the creation of corresponding shadow surfaces. This process is inherently user-dependent and time-consuming, as it requires careful manual verification of correct interface pairing to ensure accurate thermal coupling between the solid and fluid domains.

Three separate continua were defined to represent the fluid and solid materials:

- **Air (Fluid Continuum):** Three-dimensional, steady-state RANS with the realizable $k-\varepsilon$ two-layer turbulence model and two-layer all- y^+ wall treatment. Coupled flow and energy equations were enabled to capture conjugate heat transfer.
- **Aluminium (Solid Continuum):** Three-dimensional steady solid energy model with constant material properties.
- **Copper (Solid Continuum):** Three-dimensional steady solid energy model with constant material properties.

Four computational regions were created and assigned the corresponding continua:

1. **Flow Domain:** Assigned the fluid continuum. The domain was initialized at 300 K with a prescribed outlet static pressure of -500 Pa.
2. **Heat Pipes:** Assigned the copper solid continuum.
3. **Heat Sink:** Assigned the aluminium solid continuum.
4. **Heat Source:** Assigned the copper solid continuum with a volumetric heat source corresponding to a total power input of 1500 W.

To evaluate solution stability and mesh sensitivity, a line probe containing four measurement points along the centerline of the flow domain was introduced. Temperature, velocity, and pressure data extracted from this probe were used in the subsequent mesh independence study.

3.3.2 CFD Simulation Workflow

To ensure reproducibility, the full simulation procedure followed in this study is summarised in a structured workflow. The conjugate heat transfer (CHT) analysis in Simcenter STAR-CCM+ was implemented according to the following sequence:

1. Import of CAD geometry from Siemens NX as Parasolid bodies.
2. Verification of geometric integrity and dimensional consistency.
3. Definition of computational regions (fluid and solid domains).
4. Assignment of material properties to each region.
5. Creation of fluid–solid interfaces using face-to-face contacts and shadow surfaces.
6. Application of conjugate heat transfer physics models and turbulence modelling (RANS, realizable $k-\varepsilon$).
7. Generation of the computational mesh using polyhedral elements with prism layers near walls.
8. Specification of boundary conditions (velocity inlet, pressure outlet, thermal loads).
9. Solution initialization and steady-state solving.
10. Monitoring of residuals and key physical quantities until convergence criteria are satisfied.

This structured workflow ensures that the simulation setup can be reproduced consistently for all cases presented in this study.

3.3.3 Mesh Independence Study (MIS) - Full 3D

A mesh independence study was conducted to ensure the accuracy, reliability, and numerical stability of the heat sink system's CFD simulations. The purpose of this study is to determine the point at which further refinement of the computational mesh does not lead to significant changes in key simulation results, such as temperature distribution, velocity profiles, and pressure drops. Achieving mesh independence is critical to confirm that the simulation results are primarily influenced by the physics of the system rather than numerical artifacts caused by inadequate spatial resolution.

The study followed a structured, component-wise approach, performed in the following sequence: **flow domain**, **heat sink**, **heat pipes**, and **heat source**. The **flow domain** was prioritized first because it governs the convective heat transfer and overall airflow behavior within the motor-controller system. Accurate resolution of the flow field is

essential, as errors in this domain propagate to downstream components and can significantly affect temperature and pressure predictions. After the flow domain mesh was optimized, refinement efforts focused on the **heat sink**, with special attention to areas with sharp thermal gradients, such as thin fins and heat pipe embedment regions. Subsequently, the **heat pipes** were meshed, concentrating on the interface with the heat sink to capture lateral heat conduction accurately. Finally, the **heat source** was refined, as it acts as the system's thermal input. Its meshing depends on the previously established accuracy of the flow and solid components to ensure that the resulting heat flux and temperature distribution are represented correctly. This sequential approach minimizes computational redundancy, ensures physical consistency, and guarantees that mesh independence is achieved efficiently for each component.

Mesh refinement within each region concentrated on areas where significant fluid dynamic and thermal gradients were anticipated, particularly near solid boundaries and critical interfaces. Local enhancements, such as prism layers along walls and targeted refinements near inlets and outlets, were applied to accurately capture boundary layer development and near-wall velocity and temperature gradients. Mesh independence was evaluated by monitoring key parameters at defined probe locations along the centerline of the flow domain, including temperature, velocity, and pressure, as well as maximum temperatures in the solid components. The mesh was considered independent when further refinement produced negligible changes in these monitored results.

Figure 3.10 illustrates the initial coarse mesh with a base size of 50 mm. At this resolution, the lack of geometric detail is apparent, especially in areas requiring finer resolution, such as part intersections, edges, and regions with high thermal gradients. Local mesh enhancements, which will be detailed in subsequent sections, are necessary to capture these critical phenomena accurately. The initial mesh serves as a baseline to guide the progressive refinement of each component, ensuring that the final simulation results are both accurate and reliable.

Simcenter STAR-CCM+

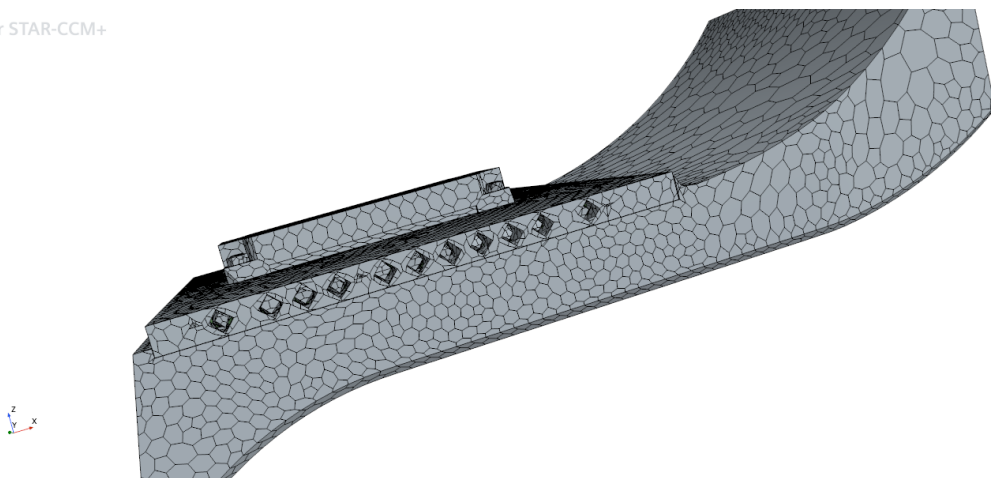


Figure 3.10: Original starting mesh with base size 50 mm, showing coarse resolution and limited geometric detail.

Flow Domain

As previously discussed, the Flow Domain was the first region to undergo mesh refinement in the full 3D CFD study. Table 3.1 summarizes the relationship between the base mesh size, the total number of cells, and the maximum temperature observed during the simulation. The following meshing techniques and refinements were applied to ensure high-fidelity results:

- **Surface Remesher:** This tool improves surface mesh quality prior to volume meshing by redistributing nodes, aligning edges to geometric features, and refining regions with sharp curvatures. High-quality surface meshes are essential for accurately capturing boundary layers and interfaces, which is critical in conjugate heat transfer (CHT) simulations.
- **Polyhedral Mesher:** Polyhedral meshes replace traditional tetrahedral elements with cells that have multiple faces. This results in higher numerical stability, faster convergence, and smoother gradients. Increased connectivity between polyhedral cells reduces numerical diffusion, which is particularly beneficial for complex flow and heat transfer simulations.
- **Prism Layer Mesher:** High-aspect-ratio prism layers are generated near walls to resolve boundary layers accurately. Correct boundary-layer resolution ensures accurate calculation of wall shear stress, heat transfer coefficients, and near-wall turbulence. Maintaining appropriate y^+ values is critical for the reliability of RANS-based turbulence models.
- **Optimize Boundary Vertices:** This option improves the surface mesh by adjusting vertex positions to reduce skewness and improve orthogonality while maintaining geometric fidelity. Optimized surface meshes contribute to better solver convergence and more accurate fluid-solid interaction modeling.
- **Optimize Cell Topology:** This feature refines volume mesh connectivity, improving element shape and aspect ratio. Optimized cell topology reduces numerical artifacts and ensures stable and accurate simulations in regions with steep thermal or velocity gradients.

Boundary Conditions: For the mesh independence study, all physical boundary conditions were kept constant across refinement levels. A velocity inlet was prescribed at the upstream boundary corresponding to the nominal fan-induced volumetric flow rate. A pressure outlet with fixed ambient static pressure was applied at the downstream boundary. All solid–fluid interfaces were treated as no-slip walls with conjugate heat transfer coupling enabled. Steady-state RANS simulations were performed using the realizable $k-\varepsilon$ two-layer turbulence model.

The results shown in Table 3.1 demonstrate that the maximum temperature stabilizes as the base mesh size decreases from 3 mm to 2 mm, indicating that mesh independence has been achieved for the Flow Domain.

Base size (mm)	Number of cells	Max Temperature (°C)
50	40217	87.27
25	154725	88.80
10	680673	87.64
5	1526525	87.41
4	1904266	87.30
3	2788002	87.10
2	4341500	87.11

Table 3.1: Flow Domain Mesh Independence Study (MIS) - 3D

Figure 3.13 visualizes the percentage difference relative to the previous simulation based on the total number of cells. The metrics plotted include ΔP (pressure change), ΔV (velocity change), ΔT (temperature change), and ΔTotal (combined variation). Within the range of 2.5 to 3 million cells, all metrics stabilize with a variance of approximately $\pm 1\%$, indicating that further refinement produces negligible changes in key simulation results. Minor fluctuations beyond this point are attributed to unrefined regions elsewhere in the model. Based on these observations, a base size of 2 mm was selected for subsequent refinements and analyses.

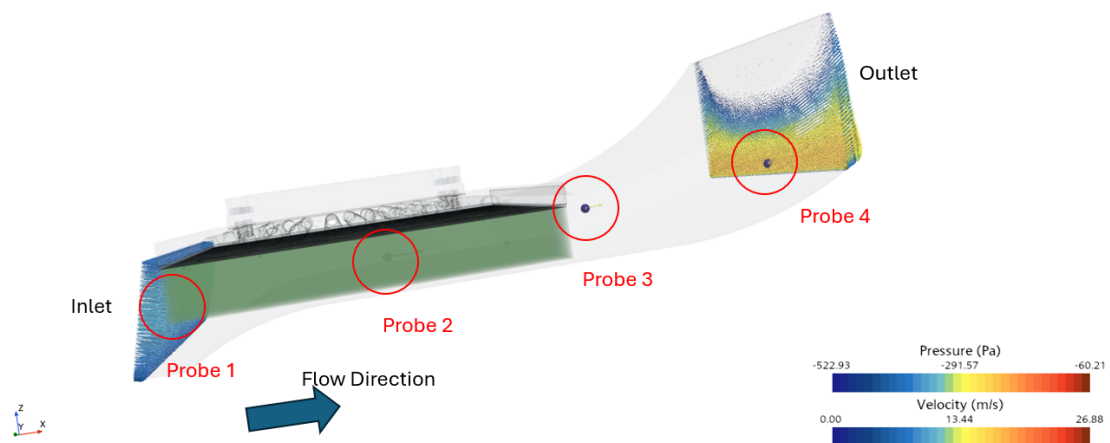


Figure 3.11: Location of Pressure and Velocity measurements

Simcenter STAR-CCM+

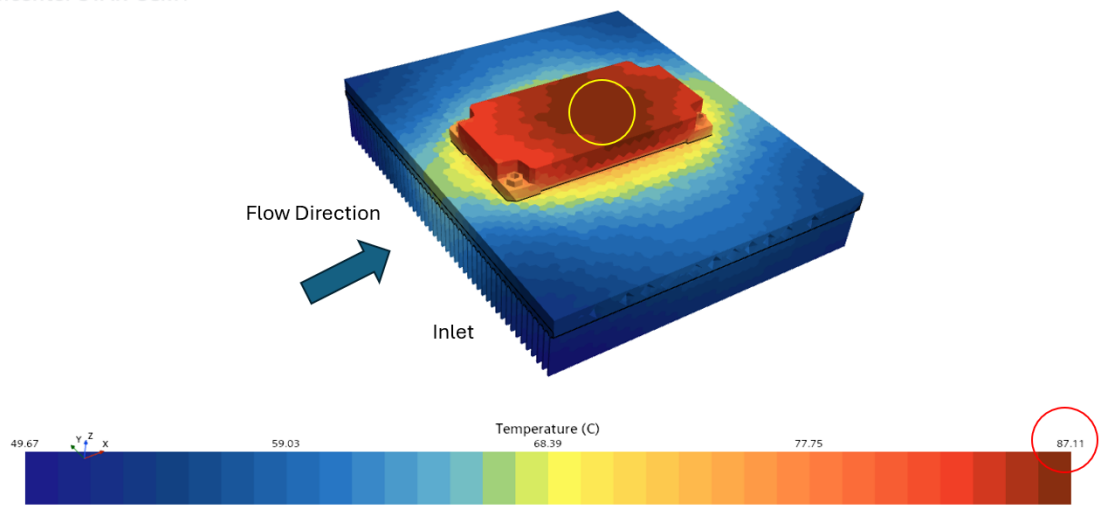


Figure 3.12: Maximum Temperature measurement

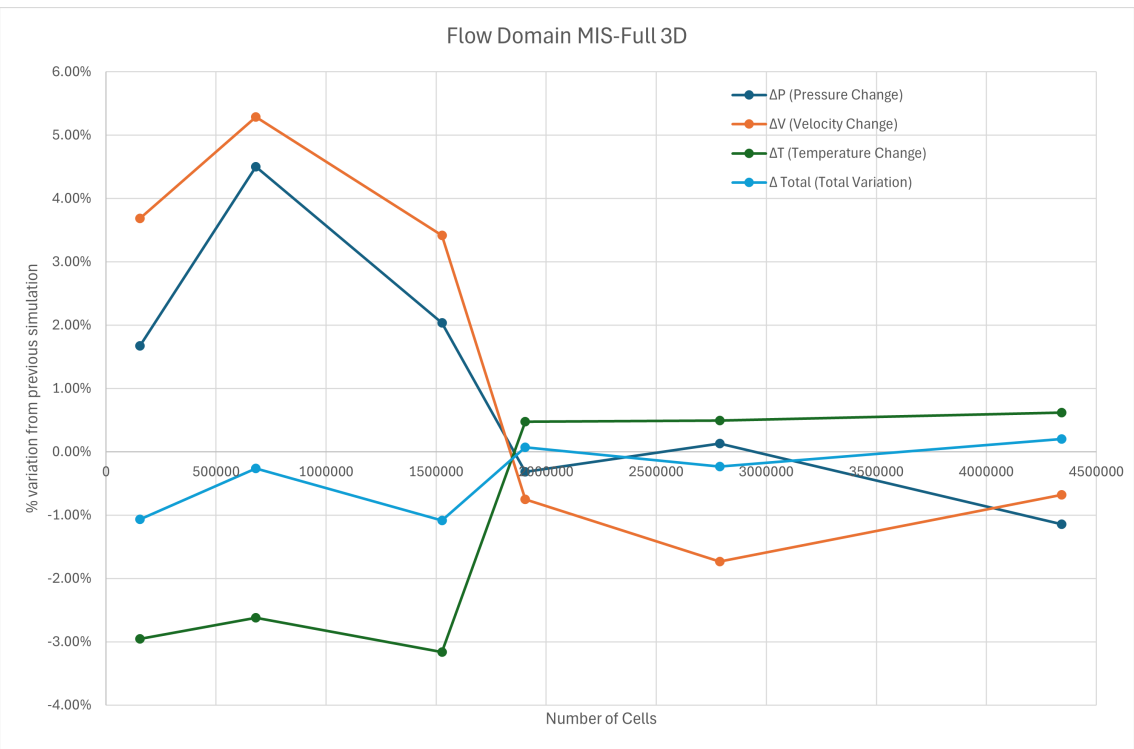


Figure 3.13: Flow Domain MIS - 3D: Percentage differences in key parameters relative to previous mesh refinement iterations.

Figure 3.11 shows the CFD setup with the line probes used to monitor velocity and pressure changes. Four probes were positioned along the center of the flow domain: Line Probe 1 at the inlet, Line Probe 4 at the outlet, and Probes 2 and 3 spaced equally between them. For the mesh independence study, measurements were taken between Probe 1 (just upstream of the heat sink at the inlet) and Probe 3 (just downstream of the heat sink) to capture the airflow response through the heat sink fins.

Figure 3.12 illustrates the maximum temperature within the domain. This measurement captures both the peak temperature in the heat source/heat sink assembly and the overall temperature rise. At this stage of the study, the heat pipes were not yet modeled, resulting in pronounced lateral temperature gradients along the heat sink surface. This emphasizes the importance of accurately representing heat spreading for later simulations.

Figure 3.13 summarizes the percentage differences in pressure (ΔP), velocity (ΔV), temperature (ΔT), and the combined variation (ΔTotal) relative to the previous mesh iteration. Between 2.5 and 3 million cells, all metrics stabilize within approximately $\pm 1\%$, confirming mesh independence. Minor fluctuations beyond this range are attributed to unrefined regions elsewhere in the domain. Based on these results, a base mesh size of 2 mm was selected for subsequent analyses to ensure accurate resolution of flow behavior, heat transfer, and boundary layer effects.

Figure 3.14 shows the final mesh configuration for the Flow Domain, highlighting the increased mesh detail, particularly near intersections with the heat sink fins. This resolution ensures accurate capture of flow behavior, heat transfer, and boundary layer effects essential for reliable full 3D CFD simulations.

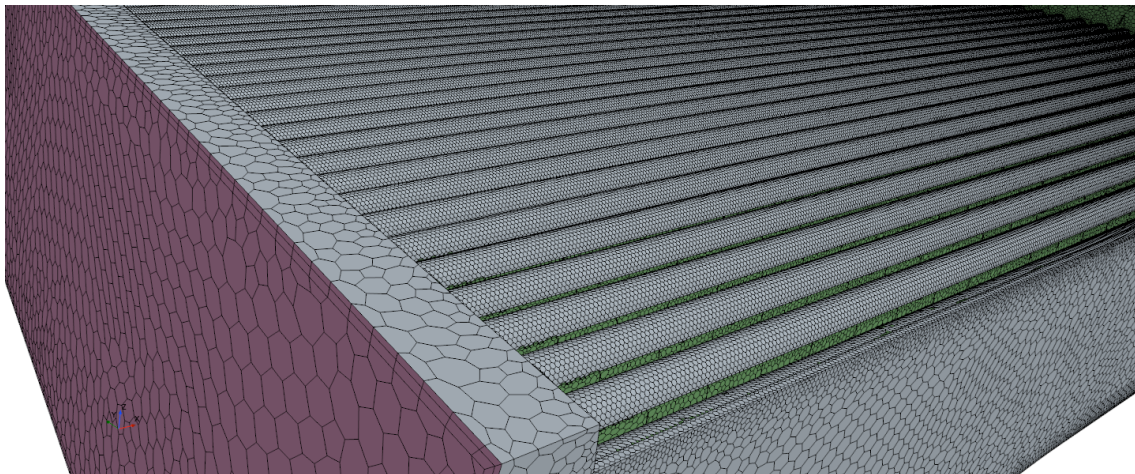


Figure 3.14: Flow Domain with final base mesh size of 3 mm, showing refined regions near heat sink fins.

Heat Sink

The heat sink was the second region to undergo mesh refinement in the full 3D CFD study. The same meshing techniques and refinements applied to the Flow Domain were employed here to ensure high-fidelity results:

- **Surface Remesher:** Improves surface mesh quality, redistributes nodes, and better captures sharp edges and thin fin geometries.
- **Polyhedral Mesher:** Generates polyhedral volume cells for enhanced numerical

stability, faster convergence, and reduced numerical diffusion in complex fin geometries.

- **Prism Layer Mesher:** Resolves boundary layers along the fin surfaces accurately, maintaining proper y^+ values for turbulence modeling and heat transfer calculation.
- **Optimize Boundary Vertices:** Smooths and adjusts surface vertices to improve element quality while preserving the geometric fidelity of the fins and heat sink body.
- **Optimize Cell Topology:** Refines the volume mesh structure to minimize skewness and improve element aspect ratios, ensuring accurate predictions of temperature and flow gradients.

Table 3.2 presents the results of the mesh independence study for the heat sink. It shows the relationship between the base mesh size, the number of cells in the heat sink, and the maximum temperature recorded in the simulation. As the base size decreases to 5 mm, the maximum temperature rises significantly. Beyond this point, further refinement results in only minor changes, indicating that mesh independence is being approached.

Base Size (mm)	Number of Cells	Max Temperature (°C)
50	39639	87.30
25	120447	88.75
10	902081	90.90
5	3485981	93.36
4	4059544	93.27
3	5838928	93.31

Table 3.2: Heat Sink Mesh Independence Study (MIS) - 3D

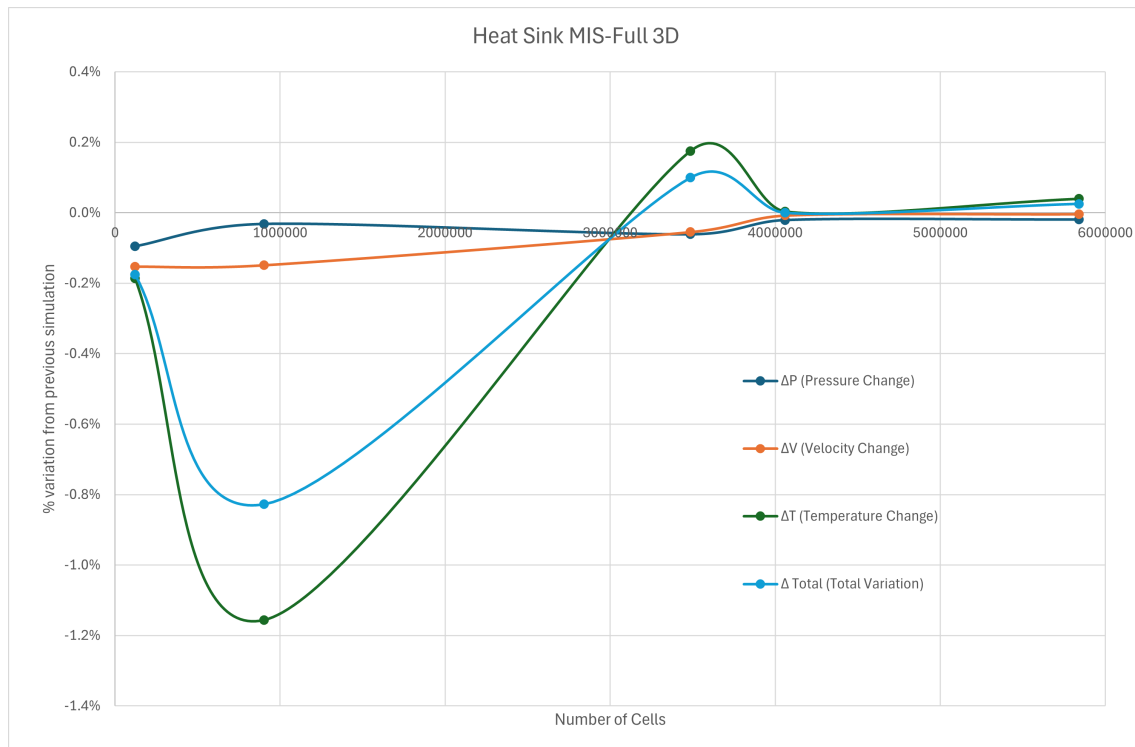


Figure 3.15: Heat Sink MIS - 3D: Maximum temperature variations with mesh refinement.

Figure 3.15 further illustrates that around 4 million cells, temperature values stabilize, indicating that additional refinement does not produce significant changes in the results. Therefore, a base mesh size of 3 mm, along with the meshing refinements described above, is considered sufficient to achieve mesh independence for the heat sink.

Simcenter STAR-CCM+

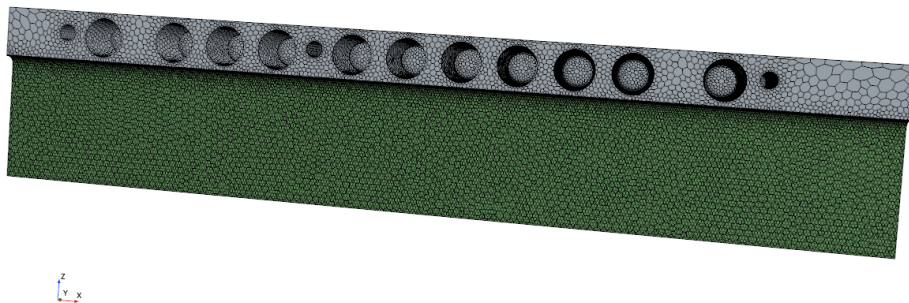


Figure 3.16: Heat Sink with final mesh base size of 2 mm, showing refined fin and body details.

Figure 3.16 demonstrates the increased level of detail in the final heat sink mesh compared to the original coarse mesh (Figure 3.10). This refined mesh more accurately captures the geometric features of the fins and the heat pipe embedment, improving the

reliability of thermal predictions and interactions between the components.

Heat Pipes

The third region to undergo mesh refinement in the full 3D CFD study was the heat pipes. The system contains a total of 11 heat pipes, which are critical for lateral heat distribution within the heat sink. To ensure accurate representation of heat transfer, the same meshing techniques and refinements applied to the Flow Domain and Heat Sink were used:

- **Surface Remesher:** Enhances surface mesh quality and captures the cylindrical geometry of the heat pipes with improved node distribution.
- **Polyhedral Mesher:** Generates volume cells with high connectivity, improving numerical stability and convergence in narrow cylindrical domains.
- **Prism Layer Mesher:** Resolves boundary layers along the heat pipe surfaces, ensuring accurate calculation of convective heat transfer to the surrounding air.
- **Optimize Boundary Vertices:** Adjusts the position of surface vertices to improve mesh quality while maintaining geometric fidelity of the heat pipes.
- **Optimize Cell Topology:** Refines volume cell connectivity to minimize skewness and enhance element aspect ratios, ensuring accurate thermal predictions.

Table 3.3 summarizes the mesh independence study results for the heat pipes, showing the relationship between base mesh size, number of cells, and maximum temperature. The maximum temperature decreases as the mesh is refined from a base size of 50 mm down to 10 mm. Beyond 10 mm, temperature variations become minimal, indicating that the mesh is approaching independence.

Base Size (mm)	Number of Cells	Max Temperature (°C)
50	6450	93.36
25	19302	90.11
10	126251	86.70
5	332574	86.74

Table 3.3: Heat Pipes Mesh Independence Study (MIS) - 3D

Figure 3.17 illustrates the variation of the key simulation parameters across the mesh refinement study. It confirms that the maximum temperature stabilizes as the base size reaches approximately 5–10 mm. Minor fluctuations beyond this point are negligible, demonstrating mesh independence for the heat pipes.

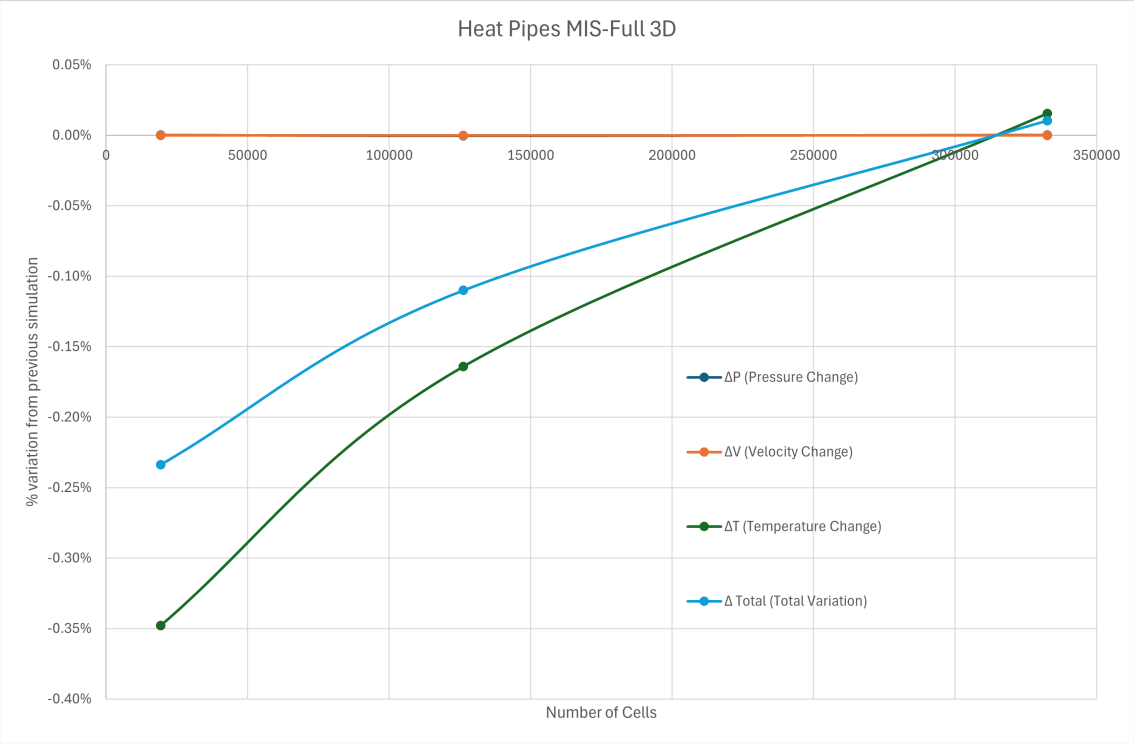


Figure 3.17: Heat Pipes MIS - 3D: Maximum temperature variations with mesh refinement.

Figure 3.18 shows the final mesh for the heat pipes with a base size of 5 mm. The higher resolution ensures accurate simulation of heat transfer along the pipes and their interfaces with the heat sink, providing a reliable basis for further analyses and optimization.

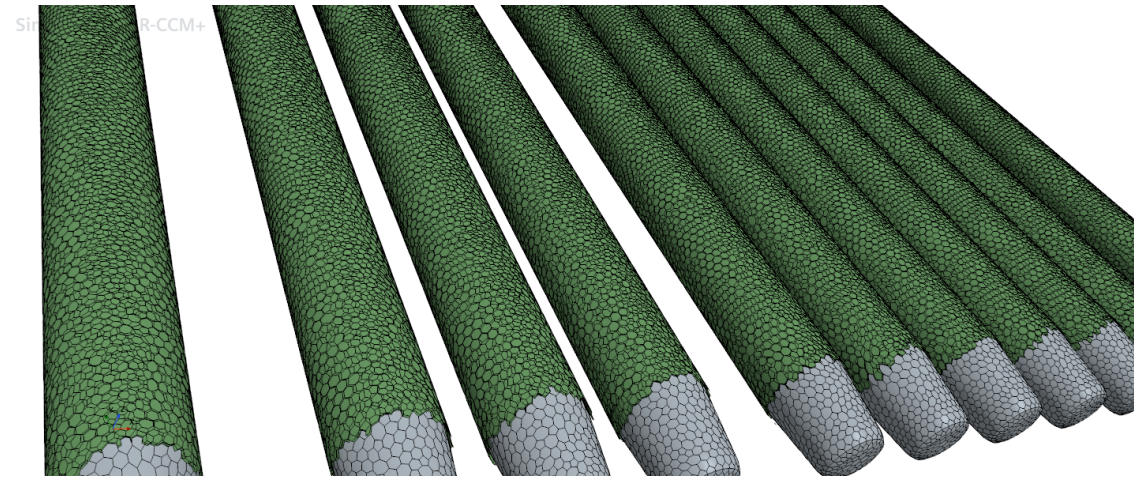


Figure 3.18: Heat Pipes with final mesh base size of 5 mm.

Heat Source

The heat source was the final component to undergo mesh refinement in the full 3D CFD study. Since the heat source primarily transfers heat through its bottom contact surface, it was initially assumed that high-resolution boundary layers were unnecessary. Therefore, the Prism Layer Mesher was not applied for this region.

The following meshing techniques were used:

- **Surface Remesher:** Improves surface mesh quality and captures small geometrical features of the heat source.
- **Polyhedral Mesher:** Generates volume cells with high connectivity to improve convergence and accuracy.
- **Optimize Boundary Vertices:** Adjusts surface vertices to enhance mesh quality while preserving geometry.
- **Optimize Cell Topology:** Refines the volume cell structure to reduce skewness and maintain stable numerical solutions.

Table 3.4 presents the results of the mesh independence study for the heat source, showing the relationship between base mesh size, number of cells, and maximum temperature. The maximum temperature remains relatively stable across the refinement steps, indicating early mesh independence.

Base Size (mm)	Number of Cells	Max Temperature (°C)
50	2444	86.70
25	3816	86.56
10	9292	86.59

Table 3.4: Heat Source Mesh Independence Study (MIS) - 3D

Figure 3.19 illustrates the variation in maximum temperature with mesh refinement. The minor differences observed confirm that the mesh has reached independence. Given the relatively low number of cells compared to other components, a base size of 10 mm was chosen for the final heat source mesh to ensure sufficient accuracy without unnecessarily increasing computational cost.

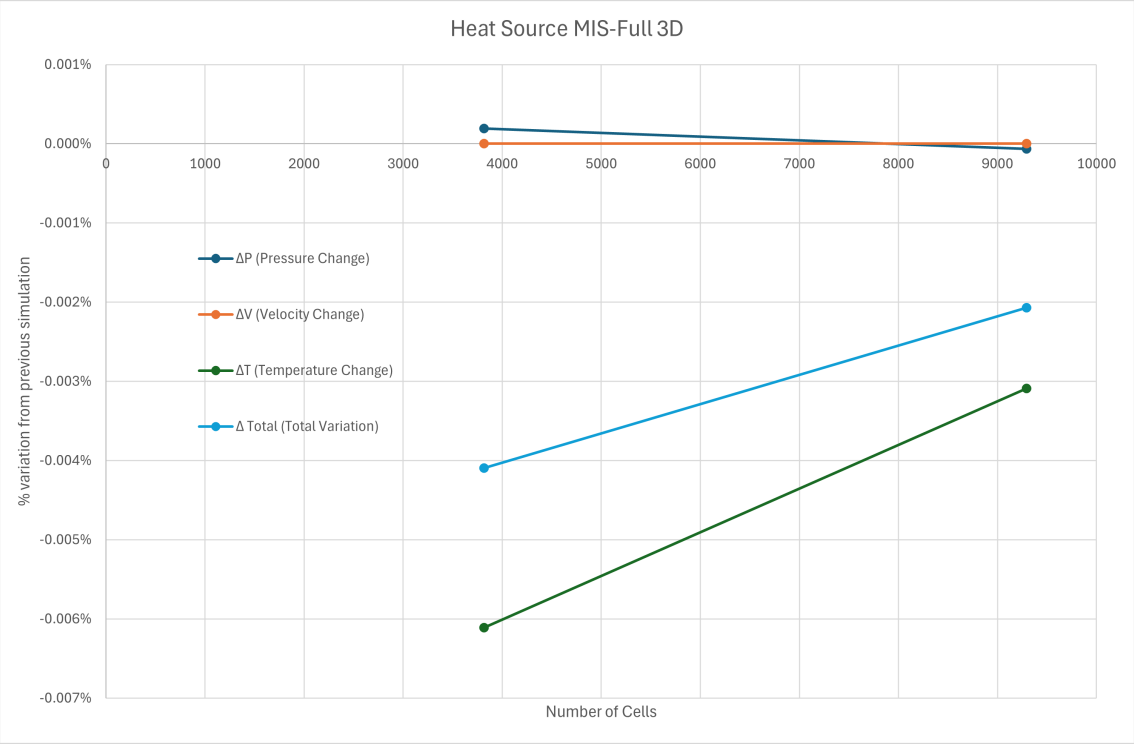


Figure 3.19: Heat Source MIS - 3D: Maximum temperature variation with mesh refinement.

Figure 3.20 shows the final heat source mesh with a base size of 10 mm. The difference in mesh resolution between the top (non-contact) and bottom (contact) surfaces ensures accurate representation of heat transfer to the heat sink, preserving the fidelity of the thermal simulation.

Simcenter STAR-CCM+

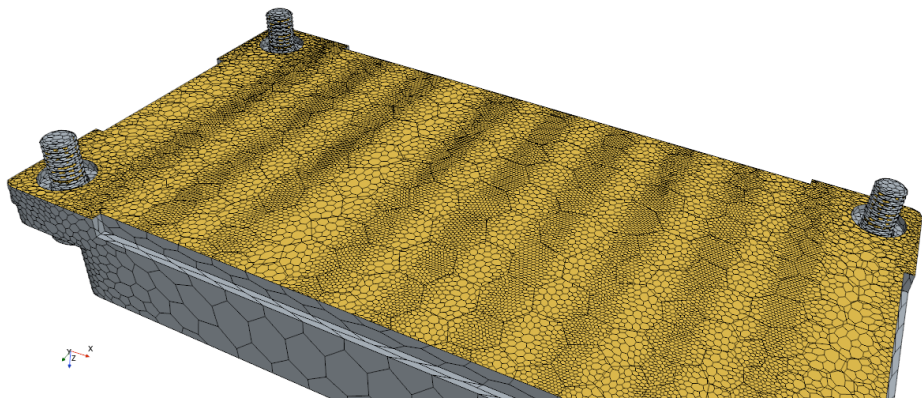


Figure 3.20: Heat Source with final mesh base size of 10 mm.

3D MIS Conclusion

Part	Base Size (mm)	Number of Cells
Flow Domain	2	4341500
Heat Sink	3	5838928
Heat Pipes	5	332574
Heat Source	10	9292

Table 3.5: Final base sizes and corresponding cell counts for the 3D MIS.

Table 3.5 summarizes the final base sizes and the associated number of cells for each component, which will be used in all subsequent simulations. The selection of these base sizes is based on the detailed mesh refinement studies, which considered maximum temperature variation, velocity, and pressure changes across probe locations. The meshing techniques and optimizations employed—including surface remeshing, polyhedral meshing, prism layers, and cell topology optimization—are described in Chapter 3.3.3.

All simulations were run until convergence was achieved. Due to computational constraints, mesh density and solver settings were selected to ensure feasible runtimes on a laptop equipped with an Intel i7 processor and 16 GB of RAM. A representative simulation with approximately 10.5 million cells required approximately 16–20 hours to reach convergence.

For the mesh independence study (MIS), simulations were initially performed on the same laptop system. At a later stage, a higher-performance workstation became available, providing significantly improved computational resources; however, this system was not available during the MIS phase.

The numerical results obtained on the laptop remain valid, as the solution is independent of wall-clock computation time and hardware performance, provided that convergence is achieved and consistent solver settings are maintained. Within these constraints, further mesh refinement was not pursued, as the selected mesh resolution already demonstrated mesh-independent and numerically stable behaviour.

Simcenter STAR-CCM+

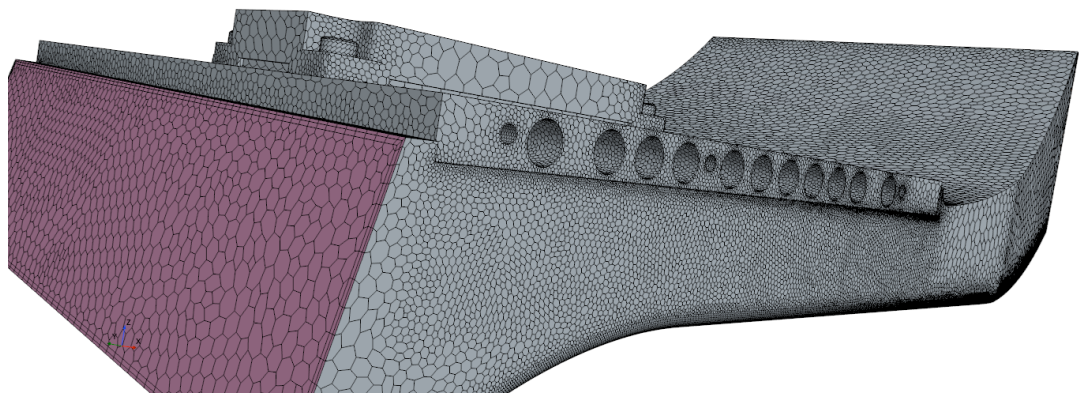


Figure 3.21: Final 3D MIS with all components.

Figure 3.21 illustrates the high level of detail captured in the final mesh. The results demonstrate that the Full 3D model provides sufficient accuracy for evaluating the thermal performance of the JS3 RES motor-controller system. While alternative lower-fidelity methods, such as the Thin-Wall approach for heat sink fins or air gaps, could potentially reduce computational costs, these methods may fail to accurately capture turbulence and detailed heat transfer within the system. Therefore, the full 3D model remains the preferred approach for precise thermal analysis.

3.3.4 Material Properties and Thermal Interactions

The thermal performance of the JS3 RES motor-controller system is strongly influenced by the properties of the materials used in its components. Table 3.6 summarizes the key thermal properties of each material.

Material	Density (kg/m ³)	Specific Heat Capacity (J/kg·K)	Thermal Conductivity (W/m·K)
Air (25 °C)	1.225	1005	0.025–0.026
Aluminium 6061-T6	2700–2710	875–900	160–180
Copper (Heat Source)	8960–8970	380–390	385–400
Evacuated Copper Heat Pipes	8900–8960	380–390	2000–80000

Table 3.6: Material properties of JS3 RES components used in the CFD model.

Thermal Interactions

In this system, heat transfer occurs via three primary mechanisms:

1. **Conduction within solids:** The heat source, heat sink, and heat pipes conduct heat through their respective materials. The heat sink fins and heat pipe interfaces provide pathways for efficient lateral and axial heat distribution.
2. **Interface conduction:** Thermal contact resistances occur at the interfaces between components due to surface roughness and imperfect contact. These were represented in the CFD model using equivalent thermal resistance values. The press-fit interface between the heat pipes and heat sink and the screwed interface between the heat source and heat sink were modelled accordingly:

$$R_{th} = \frac{\Delta T}{Q} \quad [\text{K}\cdot\text{m}^2/\text{W}] \quad (3.1)$$

Typical values are:

- Heat pipes to heat sink (press fit, copper–Al): $R_{th,HP-HS} \approx 1 \times 10^{-6} - 5 \times 10^{-6} \text{ K}\cdot\text{m}^2/\text{W}$
 - Heat source to heat sink (screwed connection, copper–Al with thermal interface material): $R_{th,HS-HS} \approx 1 \times 10^{-5} - 5 \times 10^{-5} \text{ K}\cdot\text{m}^2/\text{W}$
3. **Convection to air:** Heat transfer from the heat sink fins to the surrounding air governs dissipation into the environment. The convective thermal resistance can be estimated as:

$$R_{conv} = \frac{1}{hA} \quad [\text{K}/\text{W}] \quad (3.2)$$

where h is the convective heat transfer coefficient and A is the surface area of the fins. Typical values for forced convection are:

$$R_{conv} \approx 0.005 - 0.02 \text{ K}\cdot\text{m}^2/\text{W},$$

depending on airflow velocity, fin spacing, and geometry.

Thermal Interface Material (Thermal Paste)

The thermal interface material (thermal paste) between the heat source and heat sink was not modelled as a geometric layer in the CFD model due to its extremely small thickness relative to the computational mesh scale. Instead, its effect was incorporated using an equivalent thermal contact resistance within Simcenter STAR-CCM+, representing the reduction in interfacial thermal resistance provided by the material.

Thermal paste improves contact between mating surfaces by filling microscopic surface irregularities and reducing trapped air gaps. Typical thermal conductivities range from 1–8 W/m·K depending on composition (ceramic, metal-based, or carbon-based formulations). Its effect in the model is therefore captured implicitly through a reduced interface resistance rather than explicit geometric representation.

3.3.5 Environmental Conditions and Atmospheric Modelling

Following completion of the verification and validation process, the CFD model was adjusted to replicate the environmental conditions recorded during the experimental and flight-testing campaigns. These corrected boundary conditions enabled direct comparison between the simulated and measured thermal performance of the heat-sink system.

Only the experimentally relevant measurements were incorporated into the CFD validation process:

- Ambient atmospheric temperature ($T[1]$)
- Air temperature behind the fans ($T[2]$)
- Air temperature after the heat sink ($T[5]$)
- Heat-sink outlet temperature ($T[6]$)
- Heat-sink inlet temperature ($T[7]$)
- Maximum temperature at the heat source ($T[8]$)
- Pressure drop across the heat sink (ΔP)
- Thermal power input ($Q = 445.6 \text{ W}$)

To ensure consistency between the CFD model and the experimental setup, line probes were positioned within the simulation at locations corresponding to the physical sensor positions used during testing. Since the exact sensor coordinates were unavailable, the probe locations were approximated using photographs of the experimental setup together with information provided by the study supervisor and the student responsible for the original testing campaign.

Adjusted Simulation Parameters The following environmental and operating conditions were applied to the corrected CFD model:

- Heat input: $Q = 445.6 \text{ W}$
- Ambient air temperature: $T_{\infty} = 25.9 \text{ }^{\circ}\text{C}$
- Air density: $\rho = 1.0064 \text{ kg/m}^3$

The reduced air density relative to standard sea-level conditions is a direct consequence of the elevated test altitude and has a measurable influence on mass flow rate, pressure drop, and convective heat-transfer performance. Accurate representation of these atmospheric conditions is therefore essential for realistic CFD predictions.

Air Density Calculation

The air density used in the simulation was calculated using the ideal gas law:

$$\rho = \frac{p}{RT} \quad (3.3)$$

where

- ρ [kg m^{-3}] is the air density,
- p [Pa] is the atmospheric pressure,
- R [$\text{J kg}^{-1} \text{K}^{-1}$] is the specific gas constant for dry air ($R = 287.058$),
- T [K] is the absolute air temperature.

The use of corrected atmospheric density values ensures accurate prediction of airflow behaviour, pressure losses, and convective heat transfer within the CFD model. Assuming standard sea-level conditions would introduce errors into the calculated cooling performance and flow characteristics.

Atmospheric Pressure at Altitude

The atmospheric pressure at the test altitude ($h = 1374$ m) was determined using the standard barometric relation:

$$p = 101325 \left(1 - \frac{0.0065 h}{288.15} \right)^{5.2559} \quad (3.4)$$

where

- p [Pa] is the atmospheric pressure,
- h [m] is the altitude above sea level,
- 101325 Pa is the ISA sea-level reference pressure,
- 0.0065 K m^{-1} is the standard atmospheric lapse rate,
- 288.15 K is the ISA sea-level reference temperature,
- 5.2559 is the exponent derived from the hydrostatic and ideal-gas relations.

This correction accounts for the reduction in atmospheric pressure and air density at elevated altitude. Since convective cooling performance is strongly dependent on air density and mass flow rate, inclusion of these atmospheric corrections is necessary to accurately reproduce the experimental operating conditions within the CFD simulations.

3.3.6 Summary

The combination of conduction within solids, low-resistance interfaces (press-fit and screwed connections), and convective heat transfer to air determines the overall thermal behavior of the JS3 RES system. Proper modeling of these mechanisms in CFD ensures accurate predictions of temperature distributions, enabling validation against benchmark and/or experimental data.

Chapter 4

Simulation Setup, Implementation, and Comparison

In this chapter, the thermal performance of the JS3 RES motor-controller heat sink system is investigated through a combination of computational simulations and analytical assessments. The analysis begins with the evaluation of the thermal behavior of individual components, taking into account the material properties and initial conduction characteristics of the flow domain, heat sink, heat pipes, and heat source, as previously discussed in Chapter 3.3.3 and the material properties section. Special attention is given to the thermal interfaces, such as the press-fit connection between the heat pipes and the heat sink, as well as the screwed interface between the heat source and the heat sink, which play a critical role in the overall heat transfer performance.

Following the component-level analysis, the chapter presents a comparison of the CFD simulation results with benchmark data from literature or experimental studies to evaluate the accuracy and reliability of the model. Any discrepancies are identified and discussed, along with the steps taken to refine the CFD setup for improved fidelity.

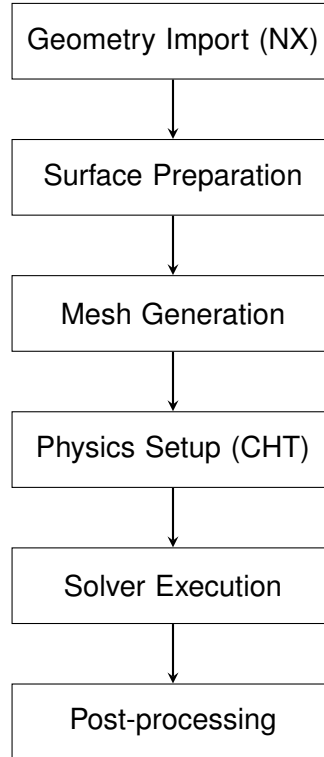


Figure 4.1: Numerical simulation workflow implemented in STAR-CCM+.

4.1 Initial Material Property Selection and Conduction Analysis

The initial material property selection is critical for establishing the thermal conduction baseline for the JS3 RES motor-controller heat sink system. In this section, the thermal properties of the flow domain, heat sink, heat pipes, and heat source are defined, and their role in heat transfer is discussed.

These properties serve as input for the CFD simulations of airflow and convective heat transfer, and they represent the initial selection, which may be revised based on validation against benchmark results.

4.1.1 Flow Domain

The flow domain is filled with air, which is modelled as an incompressible fluid. The following properties were selected at 38 °C (311.15 K):

- Density: $\rho = 1.225 \text{ kg/m}^3$
- Specific heat capacity: $c_p = 1005 \text{ J/(kg} \cdot \text{K)}$

- Thermal conductivity: $k = 0.026 \text{ W}/(\text{m} \cdot \text{K})$
- Dynamic viscosity: $\mu = 1.81 \times 10^{-5} \text{ Pa} \cdot \text{s}$
- Turbulent Prandtl number: $Pr_t = 0.85$
- Outlet pressure: $\Delta P = -260 \text{ Pa}$

The outlet pressure of approximately -260 Pa was determined from experimental measurements performed in a previous final-year project by Griessel (2024). Convection between the air and the heat sink fins dominates heat removal from the system. The selected outlet pressure serves as an initial condition for the CFD simulations and may be slightly adjusted later to ensure consistency and comparability with benchmark test data.

4.1.2 Heat Sink

The heat sink is made from Aluminium 6061-T6, chosen for its high thermal conductivity and lightweight. The selected properties are:

- Density: $\rho = 2700 \text{ kg}/\text{m}^3$
- Specific heat capacity: $c_p = 896 \text{ J}/(\text{kg} \cdot \text{K})$
- Thermal conductivity: $k = 167 \text{ W}/(\text{m} \cdot \text{K})$

The heat sink conducts heat from the heat pipes and heat source to the fin surfaces, enabling efficient dissipation to the surrounding air.

4.1.3 Heat Pipes

The heat pipes are constructed from evacuated copper (C1020) with de-ionized water as the working fluid and a sintered wick along the inner wall. The sintered wick provides capillary-driven return of the liquid to the evaporator section, enabling continuous heat transport. Each heat pipe is 200 mm long and has a maximum input power rating of 61 W, as specified in the datasheet.

For the CFD simulations, the heat pipes are represented as solid copper cylinders with an effective axial thermal conductivity to capture the combined effects of the copper casing, working fluid, and wick. This approach allows the simulation to approximate the high-efficiency heat transport of the heat pipes without explicitly modeling the internal fluid flow and phase change. The selected properties are:

- Effective axial thermal conductivity: $k_{\text{axial}} = 20,000 \text{ W/m}\cdot\text{K}$
- Density (copper): $\rho = 8960 \text{ kg/m}^3$
- Specific heat capacity: $c_p = 385 \text{ J/kg}\cdot\text{K}$
- Length: $L = 200 \text{ mm}$

The effective thermal conductivity value provides a reasonable approximation of the heat transport capacity, consistent with the maximum rated input power of 61 W. The heat pipes are press-fitted into the aluminium heat sink to ensure intimate contact and low thermal resistance at the interface, promoting efficient lateral heat distribution to the fins.

4.1.4 Heat Source

The heat source is constructed from copper and is mechanically attached to the heat sink with screws. Selected properties are:

- Density: $\rho = 8960 \text{ kg/m}^3$
- Specific heat capacity: $c_p = 385 \text{ J/kg}\cdot\text{K}$
- Thermal conductivity: $k = 385 \text{ W/m}\cdot\text{K}$

The screwed interface between the heat source and heat sink ensures conduction of heat into the heat sink.

4.2 Initial Thermal Interface Analysis (Thermal Resistance)

Accurate modeling of thermal interfaces is critical for the JS3 RES motor-controller system, as the efficiency of heat transfer between components strongly depends on the quality of these contacts. This section presents the thermal resistance considerations for the interfaces between the heat pipes and the heat sink, as well as between the heat source and the heat sink.

4.2.1 Heat Pipes to Heat Sink

The heat pipes are press-fitted into the aluminium heat sink, ensuring intimate contact along the embedment surfaces. This interface is primarily conductive and plays a significant role in the lateral distribution of heat from the heat pipes into the fins of the heat sink.

For simulation purposes, the interface is modeled as a very low-resistance contact with a thermal contact resistance of

$$R_{th,HP-HS} = 1 \times 10^{-6} \text{ K}\cdot\text{m}^2/\text{W},$$

which reflects the high-quality metal-to-metal contact achieved in a press-fit configuration with clean surfaces. This low contact resistance ensures efficient conduction of heat from the heat pipes into the heat sink while avoiding artificially limiting the heat transfer in the simulation. The thermal contact resistance is assumed constant, neglecting minor variations due to surface roughness or thermal expansion. This assumption is conservative and adequately represents the effective heat spreading in the heat sink for the purposes of the CFD study.

4.2.2 Heat Source to Heat Sink

The copper heat source is mechanically fastened to the aluminium heat sink using screws, creating a direct metal-to-metal contact. In practical applications, a thin layer of thermal paste is applied to fill microscopic surface irregularities and enhance thermal conduction across the interface.

In the simulation, the thermal contact resistance of the screwed interface is modeled as

$$R_{th,HS-HS} = 1 \times 10^{-5} \text{ K}\cdot\text{m}^2/\text{W},$$

which represents a realistic value for a copper-to-aluminium interface with good surface contact and moderate clamping pressure. This low contact resistance ensures accurate heat transfer from the heat source into the heat sink while maintaining the structural integrity of the assembly in the simulation. The thermal contact resistance is assumed constant, neglecting minor variations due to surface roughness or thermal expansion.

4.3 Verification and validation

Before proceeding with the analysis and results, it is essential to define the concepts of Verification and Validation (V&V), outline their key questions, and highlight their importance in CFD modeling.

- **Verification** ensures that the CFD model is correctly implemented and that the numerical methods accurately solve the governing equations. It focuses on identifying and minimizing numerical errors through assessments of discretization accuracy, mesh independence, and convergence behavior.
- **Validation** ensures that the verified model adequately represents physical reality by comparing simulation results with reliable experimental or analytical data under equivalent conditions.

In summary:

- **Verification:** “Did we solve the equations correctly?”
- **Validation:** “Did we solve the correct equations?”

In this study, the practical test results obtained during the undergraduate project are unfortunately not suitable for quantitative validation due to measurement inaccuracies, high fluctuations, and procedural inconsistencies. As an alternative, the CFD model is verified using established benchmark cases from literature. This approach allows confirmation that the mesh generation strategy, region coupling, and solver settings are correctly implemented and numerically sound. Following successful verification, limited validation is performed by comparing the model inputs and trends against reliable real-world data, such as material properties and boundary conditions obtained from manufacturer datasheets and literature.

4.4 Benchmark Verification

A comprehensive mesh independence study was already conducted in Chapter 3; consequently, the next stage in the verification process focuses on evaluating the accuracy of the modeling techniques and the solver implementation. As previously discussed, a peer-reviewed experimental study by Ventola et al. (2016) serves as the physical benchmark for this assessment. This same study was also utilized by SimScale AG (2025) to validate and verify the SimScale CFD platform. In the present work, the STAR-CCM+ modeling setup is verified using the same benchmark methodology. This approach allows the model to be assessed not only against experimental results but also against theoretical predictions and previous simulation data, providing a robust verification of the modeling approach.

The first step in this process is to reproduce the geometry and boundary conditions described by Ventola et al. (2016) and implemented in the SimScale validation case. This ensures consistency and enables a direct comparison of results. The same meshing strategy and solver settings established during the mesh independence study are applied to maintain numerical consistency. After completing the simulation, key performance quantities—such as base plate temperature, fin surface temperature distribution, and overall thermal resistance—are compared with the experimental data of Ventola et al. (2016) and the numerical results reported by SimScale AG (2025).

If significant deviations are observed, the model setup (e.g., mesh density in critical regions, near-wall resolution, or solver tolerances) will be adjusted iteratively until the discrepancy between the present results and the benchmark data falls within acceptable uncertainty limits. This approach confirms that the numerical model and solution

methodology are correctly implemented before proceeding to the validation of the actual heat sink configuration.

4.4.1 Selection of Appropriate Benchmark Test

The benchmark test selected for the verification of the CFD model is based on the study conducted by Ventola et al. (2016), which investigates conjugate heat transfer in rectangular fin heat sinks. This study was chosen for several reasons. Firstly, it provides a well-documented experimental dataset with clearly defined boundary conditions, material properties, and geometric parameters, allowing accurate model reproduction. Secondly, the problem configuration is representative of the current application, as it involves air-cooled heat sinks operating under forced convection, which closely resembles the conditions of the present study.

The geometry and configuration used for the Benchmark Verification are closely aligned with the actual heat exchanger setup. In both cases, the system consists of a straight, finned heat sink with a heat source mounted opposite the fins. The air channel, representing the flow domain, surrounds the fins but does not enclose the heat sink base or the heat source itself. Both heat sinks are constructed from aluminium, and a fixed heat input (in watts) is applied to the heat source to represent the thermal load. The boundary conditions, including inlet air temperature, air density, and inlet/outlet flow parameters (such as mass flow rate, volumetric flow rate, inlet velocity, or pressure difference), are defined and known for both configurations. This arrangement provides a controlled and representative environment for verifying the accuracy of the selected solvers and their respective numerical settings, as well as evaluating the suitability of the meshing strategies employed in the simulation.

Additionally, the study has been successfully used by SimScale AG (2025) as a benchmark for CFD verification, demonstrating that it is a reliable and widely recognized reference case for conjugate heat transfer simulations. By using this benchmark, the verification process can focus on confirming that the numerical setup, meshing strategy, and solver settings are correctly implemented, ensuring that the model provides physically meaningful and numerically consistent results.

4.4.2 CFD Simulation Results vs. Benchmark Data

The following figures are taken directly from the SimScale AG (2025) website and are therefore used courtesy of SimScale. The first and most significant component is the heat sink model utilized for the verification study. When compared to Figure 3.2, which depicts the heat sink design used for the motor controller, the geometric similarities between the two configurations become evident. The dimensions of the SimScale heat sink are summarized in Table 4.1.

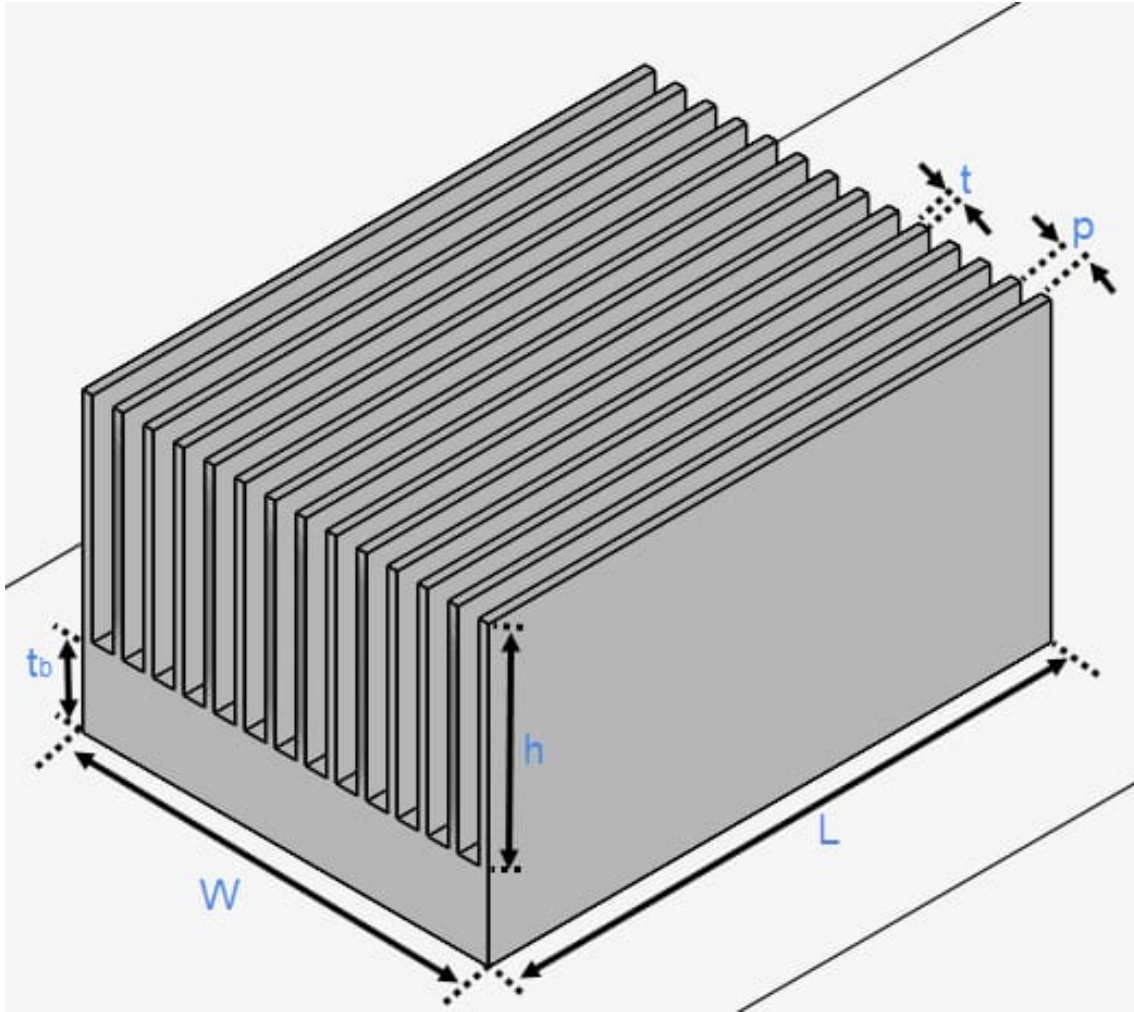


Figure 4.2: SimScale Heat Sink

Dimension	Value [mm]
h (height)	21.8
W (width)	41.4
L (length)	57.2
t_b (base height)	8.4
t (thickness of fin)	1.0
p (spacing between adjacent fins)	2.1
N (number of fins)	14

Table 4.1: SimScale Heat Sink Dimensions

The heat source used in the benchmark case closely resembles the one implemented in the JS3 RES motor-controller system, both in geometry and in its functional purpose. It serves as a uniform thermal load applied to the base of the heat sink, replicating the heat generation of an electronic component mounted on the cooling surface. The design and corresponding dimensions of the heat source are presented in Figure 4.3 and Table 4.2.

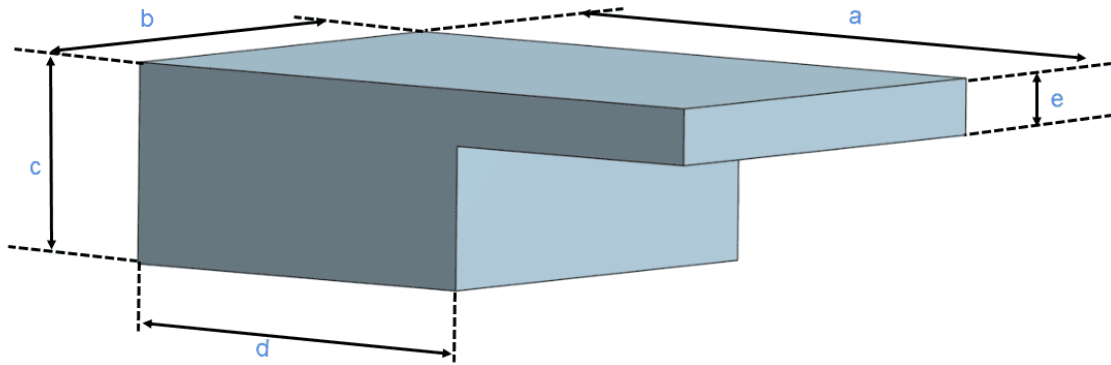


Figure 4.3: SimScale Heat Source

Dimension	Value [mm]
a	15.5
b	10.0
c	4.5
d	9.0
e	1.3

Table 4.2: SimScale Heat Source Dimensions

The flow domain used for the bench mark verification is presented in Figure 4.4 with dimensions presented in Table 4.3.

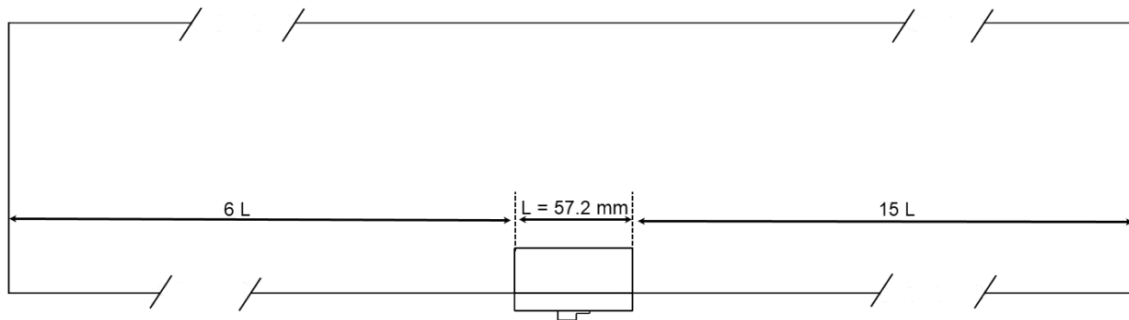


Figure 4.4: SimScale Flow Domain

Dimension	Value [mm]
Height	130.7
Width	74.7
Length before Heat Sink	$6 \times L$
Length after Heat Sink	$15 \times L$
Total Length	$22 \times L$

Table 4.3: SimScale Flow Domain Dimensions

Due to the geometric symmetry of the system, only one half of the domain was modeled in the simulation, consistent with the configuration employed in the SimScale case. To maintain equivalence, the total power input applied to the heat source was halved accordingly.

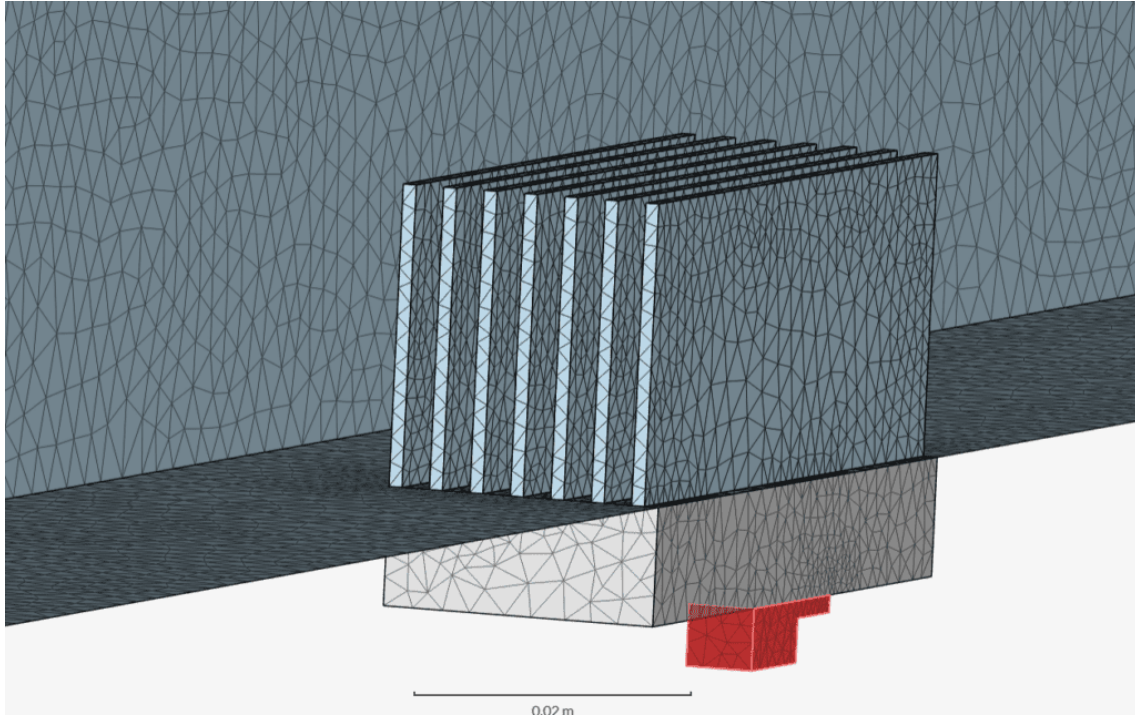


Figure 4.5: SimScale setup showing the half-domain assembly with internal mesh and conjugate interfaces

Figure 4.5 shows the assembled half-domain model used in SimScale. The image illustrates the internal mesh distribution across both the fluid and solid regions, including the heat sink, heat source, and surrounding flow domain. The symmetry plane is visible at the cut surface, confirming that only one half of the physical geometry is represented in the simulation.



Figure 4.6: External view of the SimScale flow domain with mesh and local refinement in the heat sink region

Figure 4.6 presents the exterior of the computational flow domain. The global polyhedral mesh can be observed along the outer boundaries, while a visibly refined mesh region is applied locally in the vicinity of the heat sink to better resolve velocity and temperature gradients within the fin array. This refinement strategy ensures adequate spatial

resolution in the region of strongest conjugate heat transfer interaction.

The mesh generation in SimScale utilized standard 3D polyhedral elements, resulting in a total of 603,883 cells. The simulation was conducted as a Conjugate Heat Transfer (CHT) analysis using OpenFOAM®.

For the purposes of this verification study, the meshing algorithms and base sizes were kept consistent with the methodology outlined in Chapter 3. The final mesh generated for the present study consisted of 802,548 cells, with the distribution among the individual components summarized in Table 4.4.

Although SimScale does not provide an exact breakdown of cell counts for each component, it is evident that—in the current setup—the flow domain alone contains more cells than the total number reported by SimScale. This indicates that the present mesh is more refined, providing improved spatial resolution of the dominant flow and thermal gradients.

Part	Number of Cells
Flow Domain	630750
Heat Sink	171560
Heat Source	238
Total	802548

Table 4.4: Number of cells for each component in the benchmark verification.

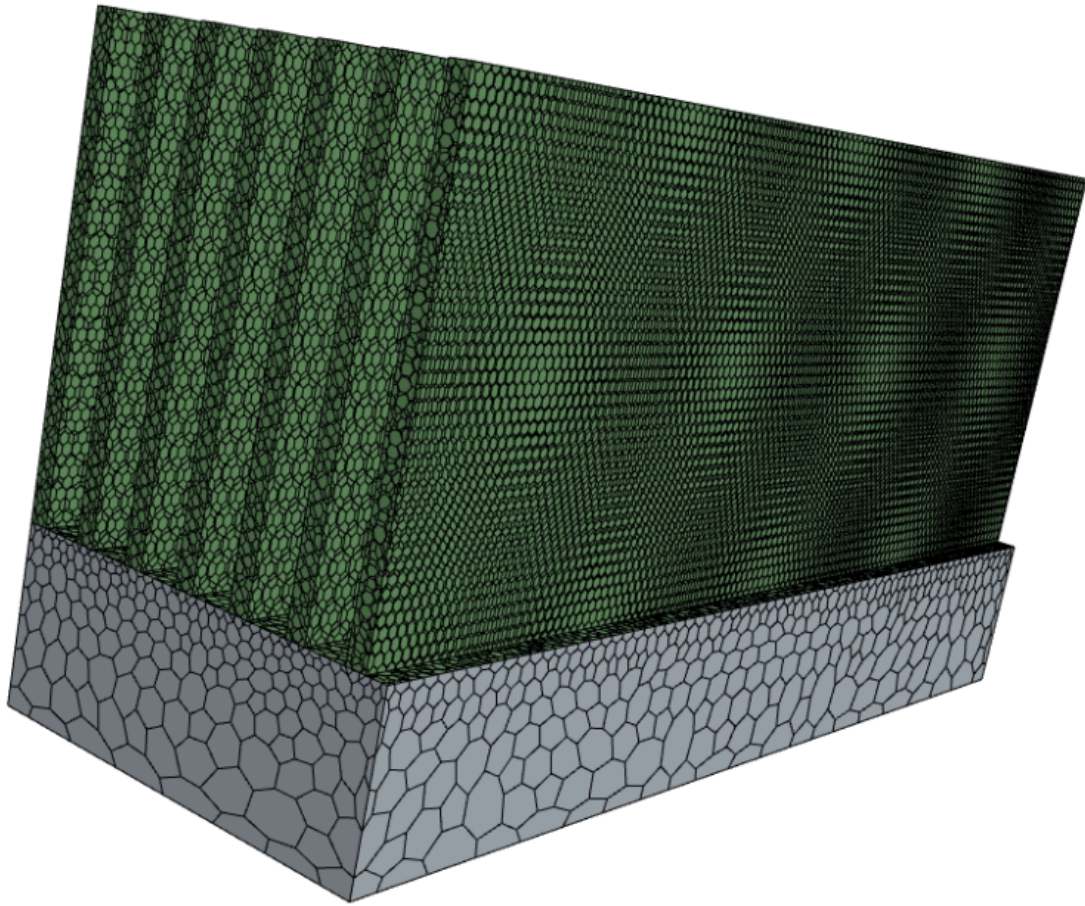


Figure 4.7: Mesh of the heat sink used for the verification study, highlighting the refined discretization on the fins.

The heat sink mesh shown in Figure 4.7 exhibits substantial refinement around the fins, where steep thermal and flow gradients are expected. In contrast, the base region is meshed more coarsely, as the thermal gradients there are less pronounced. This strategy ensures accurate resolution of conduction and convection effects while maintaining computational efficiency. The same meshing approach is applied in the main simulation campaign.

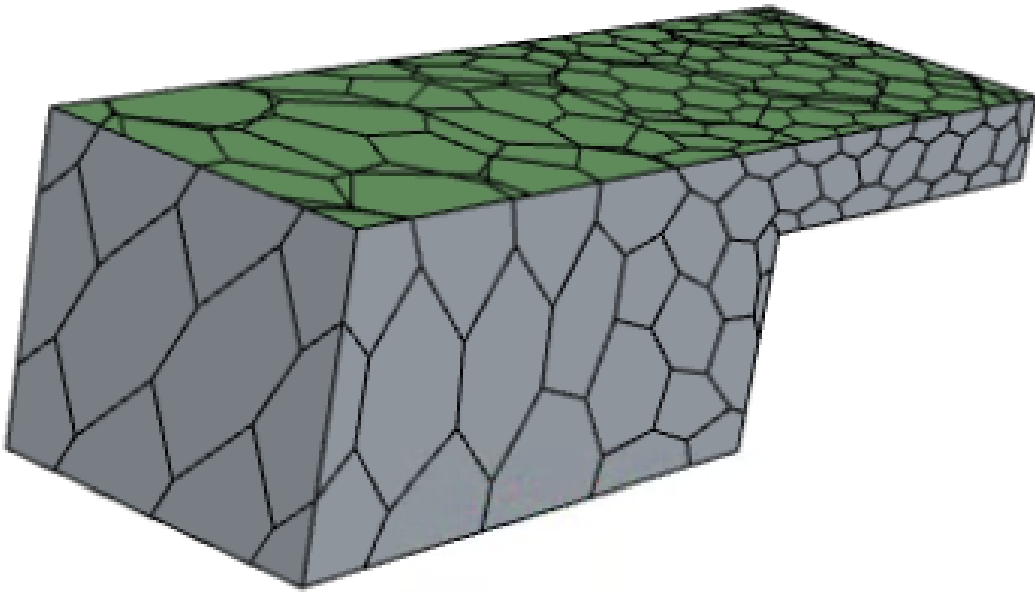


Figure 4.8: Mesh of the heat source block used in the verification study.

The heat source, illustrated in Figure 4.8, is discretized using a comparatively coarse mesh. This is sufficient because the heat source interacts with the heat sink only through a single flat interface, and no complex geometrical or flow features are present that would require additional refinement.

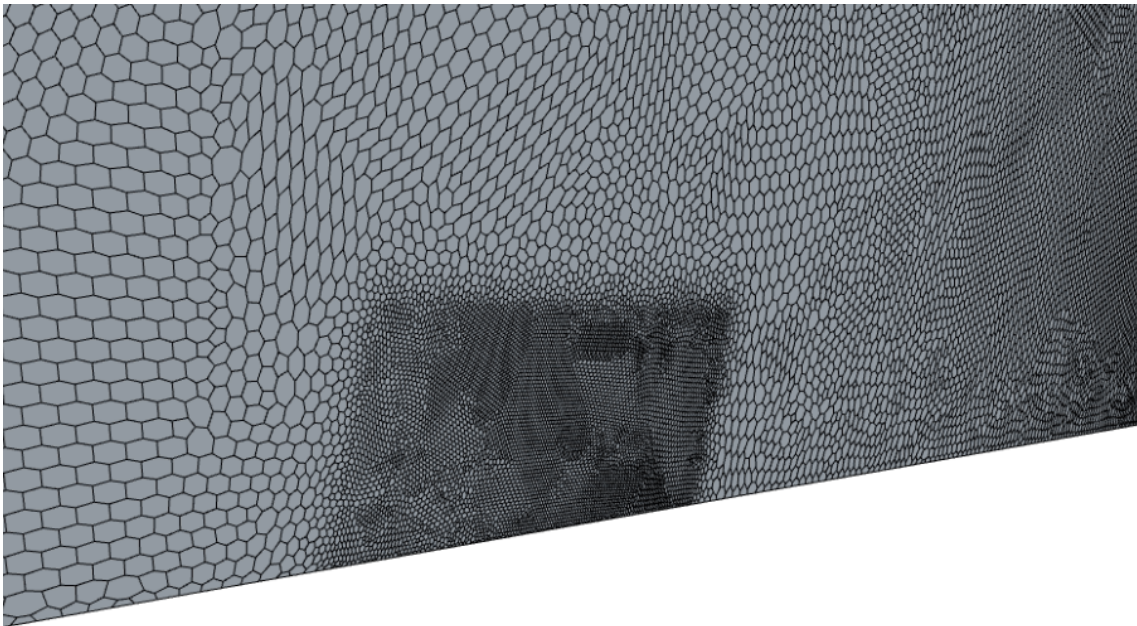


Figure 4.9: External view of the flow domain mesh showing local refinement around the heat sink region.

As shown in Figure 4.9, the mesh is locally refined in the region surrounding the heat

sink. This ensures accurate resolution of conjugate heat transfer phenomena, including boundary-layer development, turbulence effects, wall interactions, and thermal gradients. Refinement is limited to areas where physical accuracy is required, keeping the overall cell count manageable by allowing coarser cells in regions with less flow complexity.

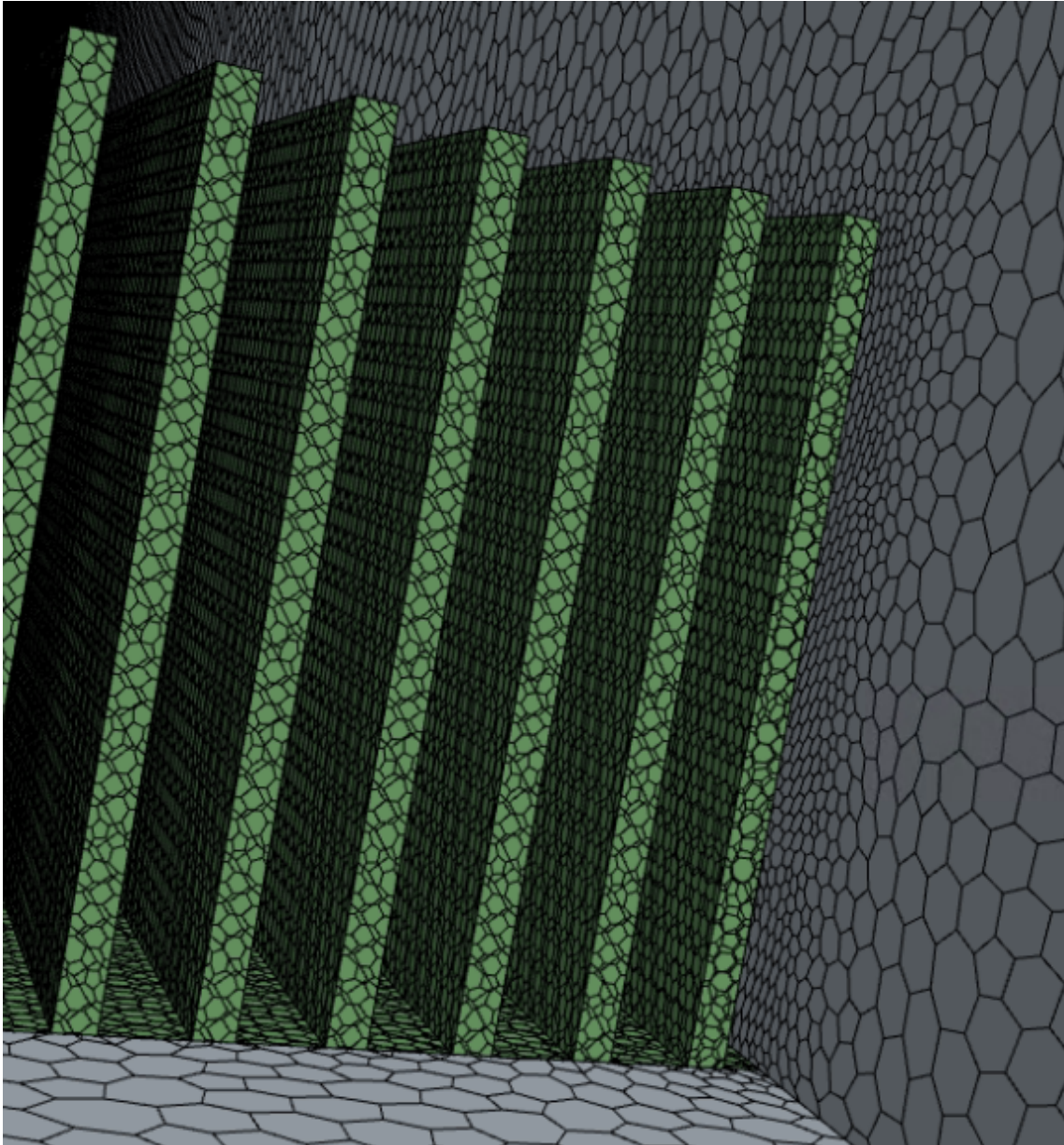


Figure 4.10: Internal view of the flow domain mesh illustrating refined regions near the heat sink.

Figure 4.10 provides an internal view of the domain. It clearly shows increased mesh density on the right-hand side of the flow domain, corresponding to the region where heat transfer and interaction with the heat sink occur.

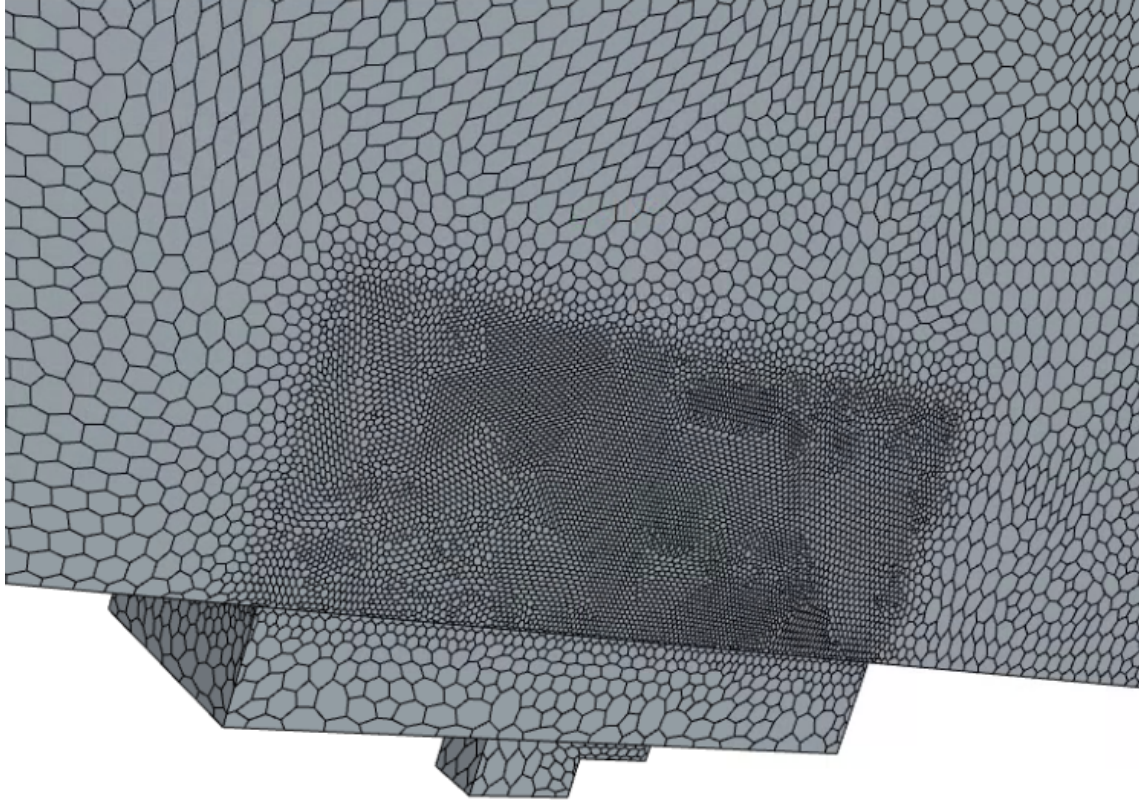


Figure 4.11: External view of the verification-case mesh with visible refinement zones.

Comparison between Figure 4.11 and the SimScale mesh (Figure 4.12) shows that the present study applies significantly smaller base cell sizes and more pronounced refinement zones. This is essential for accurately capturing conjugate heat transfer—a requirement previously identified during the mesh independence study.

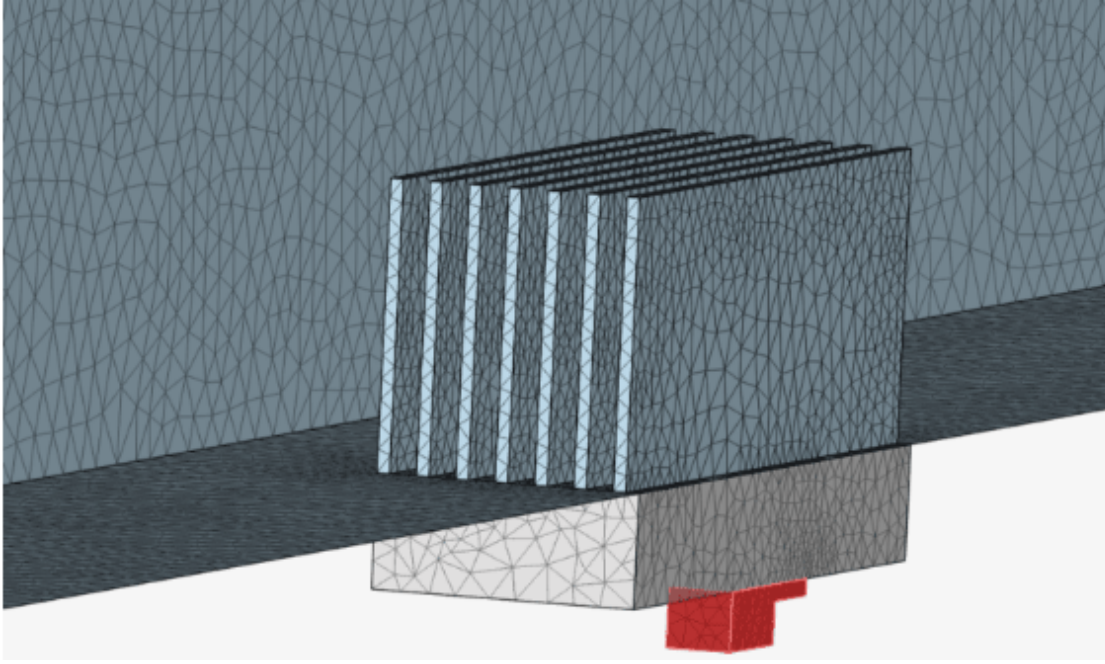


Figure 4.12: Original mesh used by SimScale for the benchmark study.

The following parameters were chosen for the verification study, coinciding with the theoretical, experimental, and SimScale simulation results. Upon reviewing the SimScale setup, it was found that the thermal conductivity of the heat source had been calculated incorrectly. The thermal conductivity is determined using the relation

$$\kappa_{hs} = \frac{\frac{1}{2}a}{R_{jc}A_s} \quad (4.1)$$

where a is the side length of the heat source (15.5 mm), A_s is the contact surface area (155 mm²), and R_{jc} is the thermal contact resistance between the heat source and the heat sink (0.5 K/W). This calculation yields a thermal conductivity of 100 W/m·K, rather than the 38.6 W/m·K reported on the SimScale website. The lower thermal conductivity used in the original model results in predicted temperatures that are approximately 20–25 K higher than those obtained from corrected calculations, experimental measurements, or simulations. When the correct thermal conductivity is applied, the simulated temperatures align significantly better with both experimental and theoretical expectations.

The physical properties used for the verification are summarized in Tables 4.5, 4.6, and 4.7.

Physical Property	Value
Density, ρ	1.179 kg/m ³
Turbulent Prandtl Number, Pr_{turb}	0.85
Specific Heat, c_p	1013 J/(kg·K)

Table 4.5: Thermophysical properties of air used in the verification study

Physical Property	Value
Thermal Conductivity, κ	209 W/(m·K)
Specific Heat, c_p	897 J/(kg·K)
Density, ρ	2700 kg/m ³

Table 4.6: Thermophysical properties of the heat sink material (aluminium) used in the verification study

Physical Property	Value
Thermal Conductivity, κ	100 W/(m·K)
Specific Heat, c_p	705 J/(kg·K)
Density, ρ	2330 kg/m ³

Table 4.7: Thermophysical properties of the heat source used in the verification study

The exact material of the heat source is not specified; however, assigning the aforementioned thermophysical properties to any material does not significantly affect the results of the conjugate heat transfer (CHT) simulation.

The results of the verification study are summarized in Tables 4.9 and 4.10. Due to the geometric symmetry of the system, only half of the domain was modeled, and accordingly, the inlet mass flow represents 50% of the actual mass flow. The inlet temperature, T_a , was applied directly, while the input heat flux, P , was halved to account for the reduced model domain.

Table 4.9 presents the calculated junction temperatures and junction-to-air thermal resistances (R_{ja}) for the different verification cases. The table also reports the average thermal resistance across all setups. Subscripts are used to distinguish between the different sources of data: t for theoretical calculations, e for experimental measurements, s for the SimScale CFD results, and v for the current STAR-CCM+ simulation being verified. The junction temperature, T_j , was measured on the bottom face of the heat source, which is opposite to the heat sink, and represents the primary thermal interface for the system.

Table 4.10 reports the corresponding percentage errors, where Sim refers to the SimScale simulation, Anal to the theoretical analysis, and Exp to the experimental data. From these results, as well as Figure 4.13, it is evident that the STAR-CCM+ CFD

simulation consistently demonstrates improved agreement with theoretical predictions compared to both the experimental data and the SimScale simulation.

During the verification process, a discrepancy was identified in the experimental thermal resistance value corresponding to an inlet velocity of 9.96 m/s (0.05871 kg/s mass flow). This value was corrected and used in the present verification, and the corrected data are reported in the tables and figures.

Finally, Table 4.8 summarizes the inlet velocities and volumetric flow rates used in the SimScale simulation, along with the corresponding mass flow rates applied in the STAR-CCM+ simulation. This ensures consistent boundary conditions across the different modeling approaches and facilitates direct comparison of the results.

Inlet Velocity [m/s]	Volumetric Flow Rate $\dot{U}$ [m ³ /s]	Mass Flow Rate $\dot{m}$ [kg/s]
5.47	0.0270	0.03183
7.03	0.0352	0.04150
8.59	0.0430	0.05064
9.96	0.0498	0.05871
11.23	0.0562	0.06626
12.50	0.0620	0.07310
13.57	0.0678	0.07997

Table 4.8: Inlet velocities, volumetric flow rates, and corresponding mass flow rates for the verification cases.

Mass Flow Rate $\dot{m}$ [kg/s]	T_a [K]	P [W]	$T_{j,e}$ [K]	$T_{j,s}$ [K]	$T_{j,v}$ [K]	$R_{ja,t}$ [K/W]	$R_{ja,e}$ [K/W]	$R_{ja,s}$ [K/W]	$R_{ja,v}$ [K/W]
0.03183	296.95	56.64	371.25	370.84	370.324	1.203	1.312	1.305	1.295
0.04150	297.35	71.40	384.15	384.03	381.831	1.142	1.216	1.214	1.183
0.05064	297.95	82.36	391.85	392.89	389.569	1.100	1.140	1.153	1.112
0.05871	298.25	97.32	395.05	406.40	402.086	1.071	0.995	1.111	1.067
0.06626	298.95	85.07	391.15	391.149	386.893	1.050	1.084	1.084	1.034
0.07310	299.25	76.30	377.45	380.435	376.218	1.031	1.025	1.064	1.009
0.07997	299.65	60.24	357.55	362.87	359.193	1.017	0.961	1.049	0.988
Average						1.088	1.105	1.140	1.098

Table 4.9: Mass flow rate, temperatures, power, and thermal resistances for different inlet conditions.

Mass Flow Rate $\dot{m}$ [kg/s]	Err Sim→Anal [%]	Err Sim→Exp [%]	Err V→Anal [%]	Err V→Exp [%]	Err V→Sim [%]
0.03183	8.442	-0.552	7.685	-1.246	-0.698
0.04150	6.305	-0.138	3.608	-2.672	-2.537
0.05064	4.795	1.108	1.129	-2.429	-3.498
0.05871	3.761	11.725	-0.378	7.269	-3.989
0.06626	3.219	-0.001	-1.546	-4.617	-4.616
0.07310	3.203	3.817	-2.158	-1.575	-5.194
0.07997	3.193	9.188	-2.809	2.838	-5.816
Average	4.703	3.790	2.759	3.235	3.764

Table 4.10: Percentage errors for different inlet conditions, comparing simulation, analytical, and experimental results.

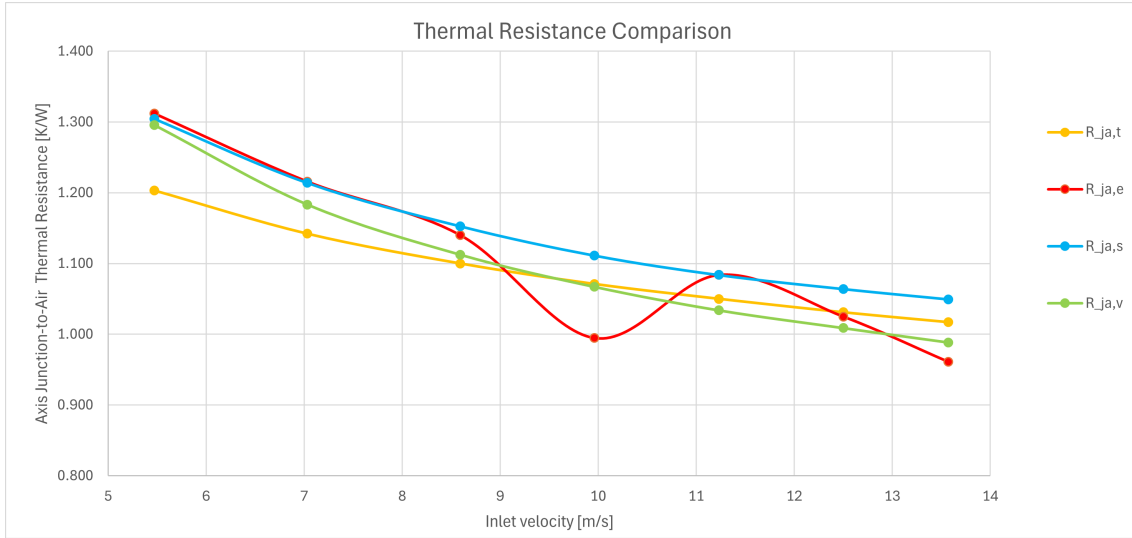


Figure 4.13: Thermal Resistance Comparison

Each row in Table 4.9 corresponds to a distinct inlet mass flow rate $\dot{m}$, representing a different cooling intensity. The inlet air temperature T_a and applied heat input P define the thermal boundary conditions for that case. The junction temperature T_j is measured at the interface between the heat source and the heat sink, representing the primary thermal reference point of the system.

The subscripts denote the data source:

- e — experimental measurement,
- s — SimScale CFD simulation,
- v — present STAR-CCM+ verification simulation,
- t — theoretical analytical calculation.

The junction-to-air thermal resistance is defined as

$$R_{ja} = \frac{T_j - T_a}{P}$$

and provides a normalized measure of cooling performance independent of absolute temperature levels. Lower values of R_{ja} indicate improved heat dissipation capability.

4.4.3 Refinement of CFD Simulation Based on Benchmark

Based on the results presented in the previous subsection, no further meshing or solver refinement is required, as the simulated outcomes fall within predefined acceptance

criteria. In this study, a deviation of $\pm 5\%$ in junction-to-air thermal resistance (R_{ja}) relative to the theoretical reference solution is defined as the acceptable tolerance range for verification. All STAR-CCM+ results fall within this limit across the investigated operating conditions, with a maximum observed deviation of 3.76%.

In fact, the STAR-CCM+ CFD model demonstrates improved predictive accuracy when compared to both the experimental data and the SimScale simulation over the full range of test conditions.

Table 4.11 summarizes the average junction-to-air thermal resistances, R_{ja} , obtained from theoretical calculations ($R_{ja,t}$), experimental measurements ($R_{ja,e}$), the SimScale CFD simulation ($R_{ja,s}$), and the current STAR-CCM+ simulation ($R_{ja,v}$). The corresponding percentage deviations from the theoretical values are also provided.

	$R_{ja,t}$ [K/W]	$R_{ja,e}$ [K/W]	$R_{ja,s}$ [K/W]	$R_{ja,v}$ [K/W]
Average values	1.088	1.105	1.140	1.098
% deviation from theoretical	-	1.55%	4.81%	0.98%

Table 4.11: Summary of average junction-to-air thermal resistances and deviations from theoretical predictions.

From Table 4.11, it can be observed that the STAR-CCM+ simulation produces an average thermal resistance of 1.098 K/W, corresponding to only a 0.98% deviation from the theoretical value. This is a marked improvement over the SimScale simulation, which exhibits a 4.81% deviation. The experimental data show a 1.55% deviation, indicating that the CFD model not only captures the theoretical behavior accurately but also surpasses the experimental measurements in terms of consistency with the expected thermal performance. These results confirm that the modeling approach, boundary conditions, and solver settings used in the STAR-CCM+ simulation are highly appropriate for the verification study and can be confidently used for subsequent analyses.

4.5 Validation with physical data

In this section, the verified CFD model is assigned the correct physical data for each component of the system. The rationale behind the selection of these properties is based on a combination of manufacturer-provided datasheets, verified literature sources, and certification-relevant data, as discussed earlier in this chapter. The use of accurate thermophysical properties is crucial to ensure that the model replicates realistic heat transfer and flow behavior under operating conditions. The following tables summarize the physical data used for the flow domain, heat sink, heat pipes, and heat source.

Physical Property	Value
Inlet temperature T_{in}	311.15 K
Density ρ	1.225 kg/m ³
Thermal conductivity k	0.026 W/m·K
Dynamic viscosity μ	1.81×10^{-5} Pa·s
Specific heat capacity c_p	1005 J/kg·K
Turbulent Prandtl number Pr_t	0.85
Outlet pressure ΔP	−260 Pa

Table 4.12: Physical values assigned to the flow domain.

Physical Property	Value
Density ρ	2700 kg/m ³
Specific heat capacity c_p	896 J/kg·K
Thermal conductivity k	167 W/m·K

Table 4.13: Physical values assigned to the heat sink.

Physical Property	Value
Density ρ	8960 kg/m ³
Specific heat capacity c_p	385 J/kg·K
Effective axial thermal conductivity k_{axial}	20000 W/m·K

Table 4.14: Physical values assigned to the heat pipes.

Physical Property	Value
Density ρ	8960 kg/m ³
Specific heat capacity c_p	385 J/kg·K
Thermal conductivity k	385 W/m·K

Table 4.15: Physical values assigned to the heat source.

The properties defined in Tables 4.12–4.15 ensure that each material behaves realistically within its respective domain. The flow domain values correspond to air at an ambient temperature of approximately 38 °C, representing the CS22 requirements. These physical definitions form the foundation for accurate conjugate heat transfer analysis within the CFD model.

The thermal interface resistances between key components, as previously discussed at the beginning of this chapter, are summarized again here for completeness. These resistances represent the interfacial thermal impedances between solid parts, accounting for surface roughness, contact pressure, and interface materials such as thermal paste. Accurate representation of these resistances is critical for achieving realistic temperature

gradients and junction temperatures in the simulation, particularly in high-power electronic assemblies where interfacial losses can dominate the overall heat transfer path.

Part–Part Interface	Physical Property	Value
Heat Source–Heat Sink	$R_{th,HS-HS}$	$1 \times 10^{-5} \text{ K}\cdot\text{m}^2/\text{W}$
Heat Pipes–Heat Sink	$R_{th,HP-HS}$	$1 \times 10^{-6} \text{ K}\cdot\text{m}^2/\text{W}$

Table 4.16: Thermal interface resistances between the primary conduction paths.

The total energy input into the system is assumed to be 1500 W at a maximum battery power output of 40 kW. This value corresponds to the estimated power loss through the motor controller under full electrical load. A linear relationship between battery power and generated heat was assumed, decreasing at a rate of 37.5 W per kilowatt of battery power, as expressed by

$$Q = 37.5 P_{bat} \quad (4.2)$$

where Q is the heat input in watts and P_{bat} is the battery power in kilowatts. Consequently, at a battery output of 25 kW, the system experiences an effective heat influx of 937.5 W. This linear scaling provides a simplified yet sufficiently accurate representation of the system's thermal behavior under varying operating loads, ensuring that both maximum and partial load conditions are adequately captured in the CFD simulations.

More detailed information on the motor controller as well as all other parts, including datasheets and engineering drawings, can be found in Chapter B.

All four parts were meshed within a single meshing operation to ensure consistent meshing parameters and proper alignment of shared interfaces between the solid and fluid regions. Executing the meshing process simultaneously allows STAR-CCM+ to generate a conformal mesh at contact surfaces, eliminating interpolation errors across non-conformal interfaces and improving numerical accuracy in the conjugate heat transfer analysis. Although the flow domain contained the largest number of cells, it completed meshing before the smaller solid components due to its relatively simple geometry and smooth surfaces. In contrast, the smaller solid parts, despite using identical mesher settings, required more time because of finer local surface refinements, higher surface-to-volume ratios, and the need to maintain conformal interfaces and prism layers at shared boundaries. This behavior highlights that meshing time is influenced not only by part size or total cell count but also by geometric complexity and interface requirements. Even though the mesher processes regions sequentially under the hood, the benefits of a unified meshing operation—such as conformal interfaces, consistent global meshing parameters, and optimized transition zones—are fully preserved. Additionally, the unified meshing approach optimizes overall efficiency and reduces total

computational time compared to performing separate meshing operations for each part.

After applying the selected material properties, boundary conditions, and interface parameters, the simulation was run until convergence was achieved. The results demonstrate stable and physically consistent behavior, indicating that the model accurately represents the thermal interactions between the heat source, heat pipes, and heat sink.

This concludes the Validation and Verification process, confirming that the simulation model is reliable and can be confidently used for further investigations. With a functioning and trustworthy model established, it is now possible to perform parametric studies, evaluate design modifications, and compare the original configuration with the "corrected" or optimized simulation setups. The detailed methodology, assumptions, and parameter values have been presented in the preceding sections and chapters.

Chapter 5

Results and Discussion

5.1 Interpretation of Results

This chapter presents the analysis and interpretation of the CFD simulation results obtained for the motor controller heat sink system. The simulations were performed under steady-state conditions for multiple heat input levels representing different operational modes of the system. Particular focus is placed on the Maximum Power (MP) and Maximum Continuous Power (MCP) operating conditions, which correspond to short-duration peak operation and sustained continuous operation, respectively.

The 1500 W heat-input case represents the short-duration Maximum Power (MP) operating condition. In practical operation, the system remains at this power level for approximately 90 seconds and therefore does not reach thermal steady state during flight. Consequently, the 25 kW battery case, corresponding to a heat input of 937.5 W, is considered more representative of long-term operating behaviour and continuous thermal performance assessment. All simulations presented in this chapter are based on the previously validated CFD model described in Chapter 3, ensuring that the predicted thermal and flow behaviour remain physically consistent and numerically reliable.

5.1.1 Simulation with 40 kW and 25 kW Battery Power

The simulation corresponding to a 40 kW battery power input, equivalent to a 1500 W heat generation rate, produced physically realistic and numerically stable results. As shown in Figure 5.2, the maximum temperature at the heat source reached approximately 100.5 °C under steady-state conditions.

Recent flight-test measurements showed that the motor controller reached an average corrected temperature of approximately 84 °C after 88 seconds of operation at an outside air temperature of 38 °C. At this stage of operation, thermal steady state had not yet been

reached. Furthermore, the tested aircraft incorporated the latest RES door configuration, in which the doors remain partially open when the motor is fully extended, thereby improving airflow through the motor compartment and increasing cooling performance relative to earlier configurations.

It should also be noted that the recorded flight-test temperature represents an average measured heat-source temperature rather than a localized peak hotspot temperature. Considering the transient nature of the flight-test condition, the enhanced cooling associated with the modified RES door geometry, and the difference between average and maximum temperature measurements, the CFD simulation is considered to provide realistic and representative thermal predictions.

Achieving thermal steady state during flight operation is not practically possible, as the maximum allowable motor-controller temperature is limited to 85 °C. Under elevated ambient conditions of approximately 38 °C, this temperature limit would be reached before steady-state thermal equilibrium could develop. In addition, controlled experimental comparison using the original air-cooled configuration is no longer possible, as Jonker Sailplanes has transitioned fully to a water-cooled RES system. A direct comparison would therefore require a dedicated laboratory or ground-based test setup using the original air-cooled configuration, which is currently unavailable.

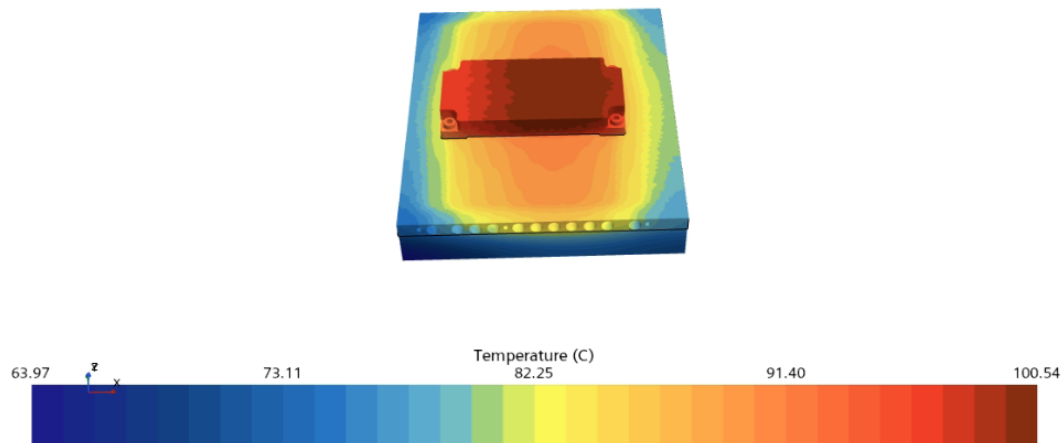


Figure 5.1: Temperature distribution for the 1500 W heat influx simulation before flow-domain extension. Airflow direction is from left to right.

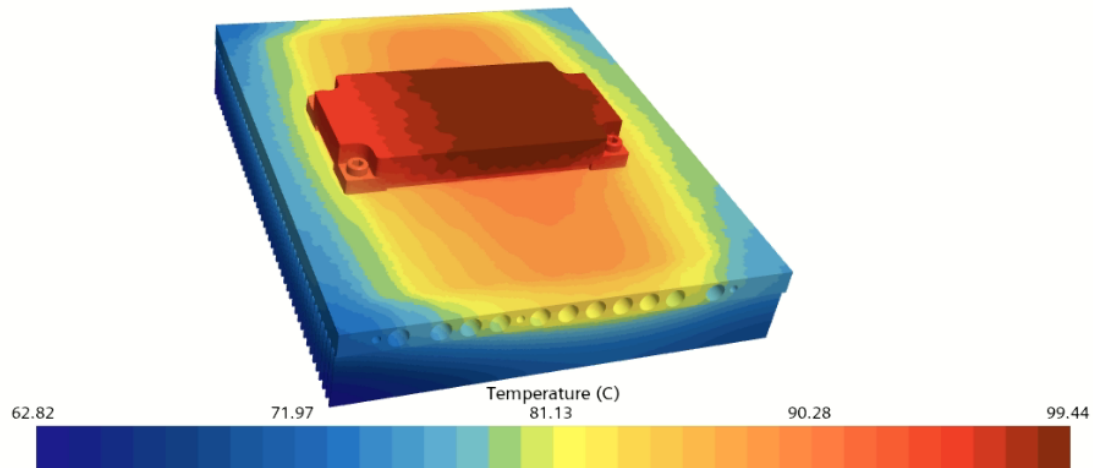


Figure 5.2: Temperature distribution for the refined 1500 W heat influx simulation. Airflow direction is from left to right.

Figure 5.3 illustrates the solver residuals for the original 1500 W simulation case. Although several residuals did not fully reach the desired target levels and later exhibited periodic oscillatory behaviour, the simulation was initially considered sufficiently converged for preliminary 3D conjugate heat transfer analysis. This assessment was based primarily on stabilization of the monitored thermal quantities, with the maximum and minimum temperatures varying by less than 0.01 °C over more than 200 consecutive iterations.

The oscillatory residual behaviour likely resulted from a combination of conservative relaxation settings, partially developed inlet and outlet flow regions, and minor mesh inconsistencies between adjacent computational regions. The selected solver settings prioritised numerical stability and practical computational time over achieving extremely low residual magnitudes. Nevertheless, the residual behaviour indicated that further refinement of the numerical setup was required to improve overall solution stability and confidence in the predicted flow development.

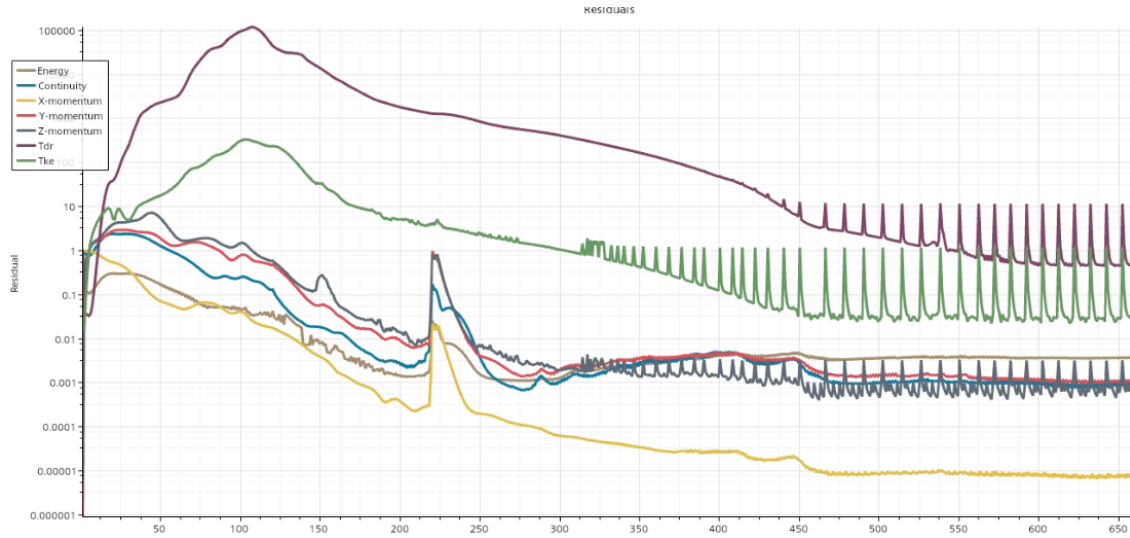


Figure 5.3: Residuals during the original 1500 W heat influx simulation before flow-domain extension

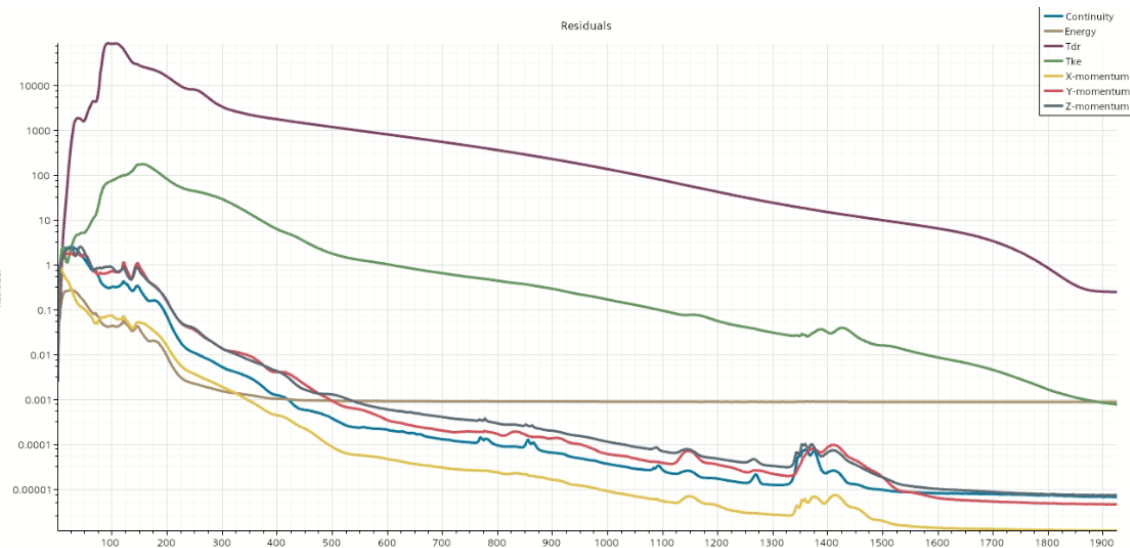


Figure 5.4: Residuals during the refined 1500 W heat influx simulation

5.1.2 Model Refinements to Address Oscillating Residuals

During the initial simulation campaign, oscillatory behaviour was observed in several solver residuals, indicating that the numerical solution had not yet reached a fully stable state. In CFD analysis, residuals represent the imbalance remaining in the governing conservation equations after each iteration. These include the continuity equation (mass conservation), momentum equations, turbulence transport equations, and the energy equation. Stable and consistently decreasing residuals are therefore commonly used as indicators of numerical convergence and overall solution stability.

In general CFD practice, a solution is typically considered converged when residuals

decrease by at least three to four orders of magnitude from their initial values and subsequently reach a stable plateau. For steady-state conjugate heat transfer simulations, commonly accepted residual levels are generally on the order of 10^{-3} to 10^{-5} for continuity and momentum equations, while the energy equation is typically expected to converge to approximately 10^{-6} or lower, depending on solver settings and model complexity. However, residual values alone are insufficient to confirm convergence. Additional monitoring of physically meaningful quantities, such as pressure drop, heat transfer rate, and temperatures at critical locations, is required to ensure that the numerical solution has stabilised and that iteration-to-iteration variation has become negligible.

The continuity residual represents the degree to which mass conservation is satisfied throughout the computational domain. Large oscillations or divergence in this residual may indicate issues related to mesh quality, boundary conditions, or unstable flow development. The momentum residuals correspond to convergence of the Navier–Stokes equations in the three spatial directions and provide an indication of how well the velocity field has stabilised. Turbulence residuals, including turbulent kinetic energy (k) and specific dissipation rate (ω), reflect convergence of the turbulence model and are particularly important for accurate prediction of convective heat transfer behaviour. The energy residual represents conservation of thermal energy and is generally the final residual to converge due to its dependence on the flow and turbulence fields.

To obtain a more physically consistent and numerically stable solution, several targeted refinements were introduced to the computational setup.

The inlet region was extended by approximately $1L$ (180 mm), while the outlet region was extended by approximately $3L$ (540 mm). These extensions allowed the inlet velocity profile, turbulence quantities, and outlet pressure field to develop more fully before interacting with the heat sink region. Importantly, these modifications were restricted to the external flow domain and did not alter the solid geometry, thermal boundary conditions, or material properties of the cooling system itself.

In addition, all components were meshed simultaneously using a consistent dependency order to improve interface conformity between solid and fluid regions. This reduced interpolation errors across conjugate interfaces and improved the stability of the coupled fluid-solid solution process.

Solver relaxation factors, ramping parameters, and iterative controls were also adjusted to improve numerical stability during the solution process. Although these refinements increased the computational time required per simulation, they eliminated the previously observed oscillatory behaviour and produced smoother residual convergence.

Following implementation of these refinements, the 1500 W simulation exhibited significantly improved numerical stability and smoother convergence behaviour. As a result of the improved flow development and more stable coupled solution, the predicted

maximum temperature decreased from 100.54 °C to 99.44 °C, while the minimum temperature decreased from 63.97 °C to 62.82 °C. The reduction of approximately 1.1 °C reflects improved numerical representation of the flow field rather than any change in the underlying physical modelling assumptions. All material properties, environmental conditions, and thermal boundary conditions remained unchanged between the original and refined simulations.

The improved convergence behaviour is illustrated in Figure 5.4, which contrasts clearly with the oscillatory residual behaviour observed in the original solution shown in Figure 5.3. In addition to improved residual stability, the refined model produced more stable temperature predictions and improved consistency in the calculated pressure drop and thermal field distribution.

5.1.3 937.5 W Heat Influx Case

For a battery discharge power of 25 kW, the estimated heat influx into the motor controller corresponds to approximately 937.5 W. This value represents a slightly conservative estimate, as detailed thermal loss data for both the baseline and modified controller configurations were not available.

The original simulation predicted a maximum temperature of approximately 77.1 °C. This prediction is considered physically realistic when compared with available flight-test data, which indicate an average corrected controller temperature of approximately 64 °C at 0% state of charge. The lower temperatures observed during flight operation are attributed primarily to the improved airflow associated with the modified RES door configuration and the additional cooling effects generated by cockpit ventilation airflow.

Ground-test data corrected to an outside air temperature of 38 °C consistently showed peak temperatures above 70 °C, further supporting the plausibility of the simulated temperature predictions.

Applying the same modelling refinements used for the 1500 W case resulted in a reduction of the predicted maximum temperature from 77.15 °C to 76.46 °C, while the minimum temperature decreased from 54.27 °C to 53.55 °C. This behaviour is consistent with the improved numerical stability and more physically representative flow development achieved through the refined computational setup.

The 937.5 W simulation was initiated as a “hot-start” case, meaning that a previously converged flow solution was used as the initial condition, with only the applied heat input modified. This significantly reduced the computational effort required to achieve convergence. As illustrated in Figure 5.8, a temporary increase in the energy residual occurs at approximately 1900 iterations, corresponding directly to the change in thermal loading. The remaining residuals remain largely unchanged, confirming that the flow field had already converged and that only the thermal solution required adjustment.

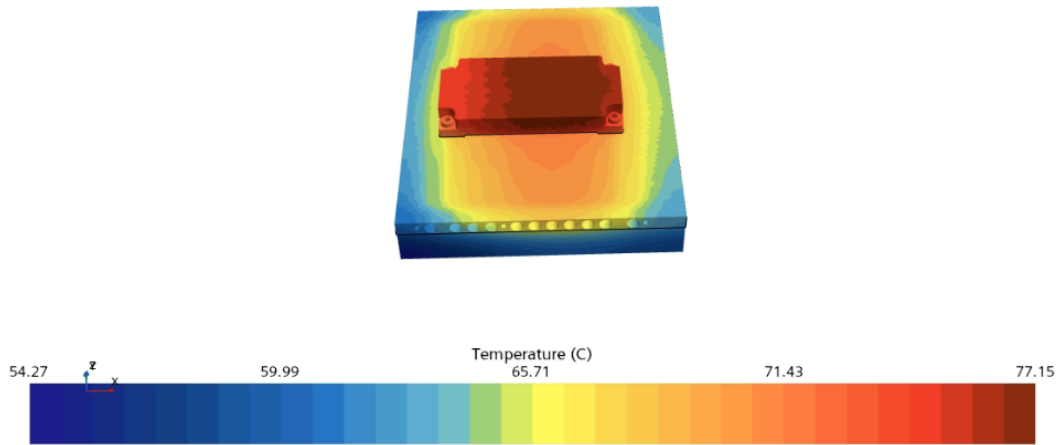


Figure 5.5: Temperature distribution for the original 937.5 W heat influx simulation before flow-domain extension. Airflow direction is from left to right.

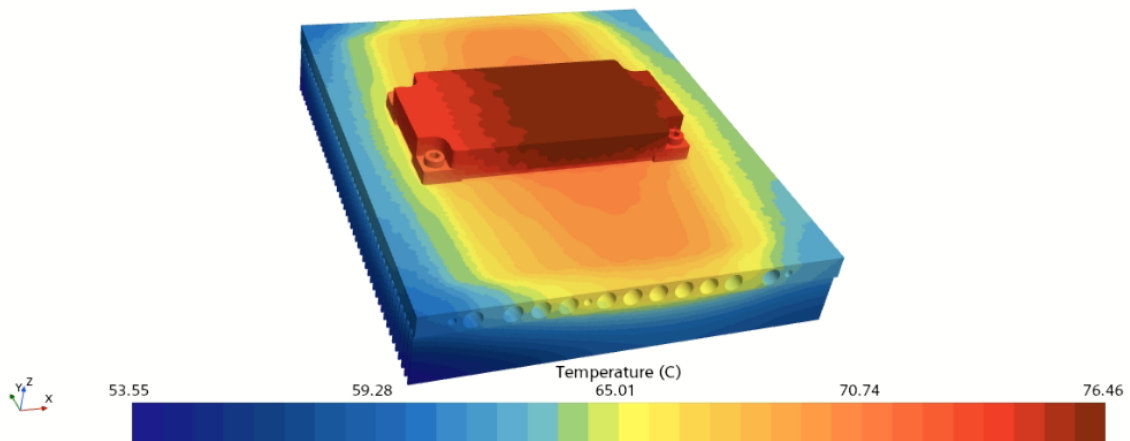


Figure 5.6: Temperature distribution for the refined 937.5 W heat influx simulation. Airflow direction is from left to right.

Figure 5.7 shows the residual behaviour for the original 937.5 W simulation case. Similar to the original 1500 W simulation, several residuals exhibit oscillatory behaviour and do not fully stabilise. After implementation of the numerical refinements discussed previously, significantly smoother convergence behaviour and improved thermal stability were achieved, as shown in Figures 5.6 and 5.8.

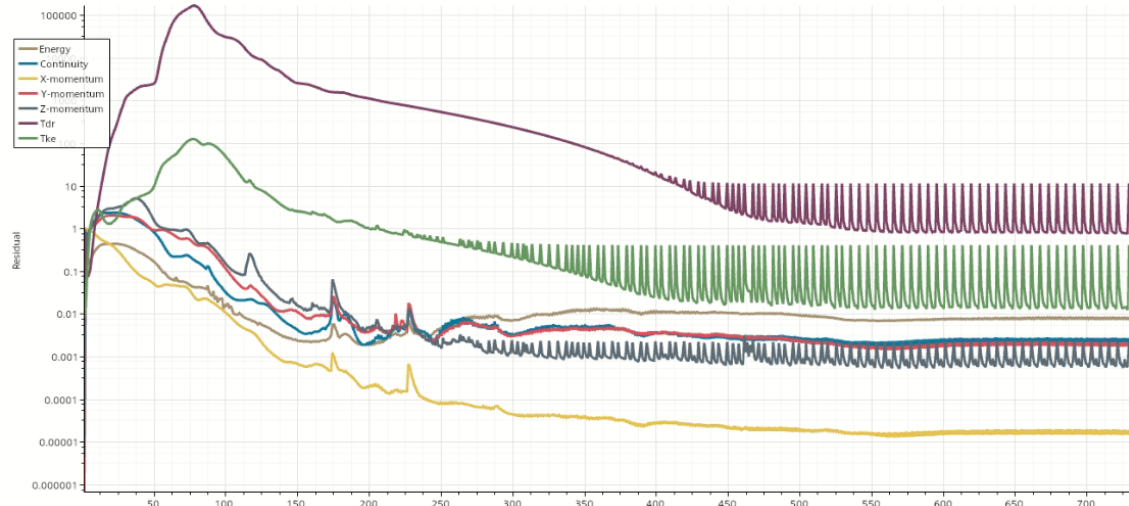


Figure 5.7: Residuals during the original 937.5 W heat influx simulation before flow-domain extension

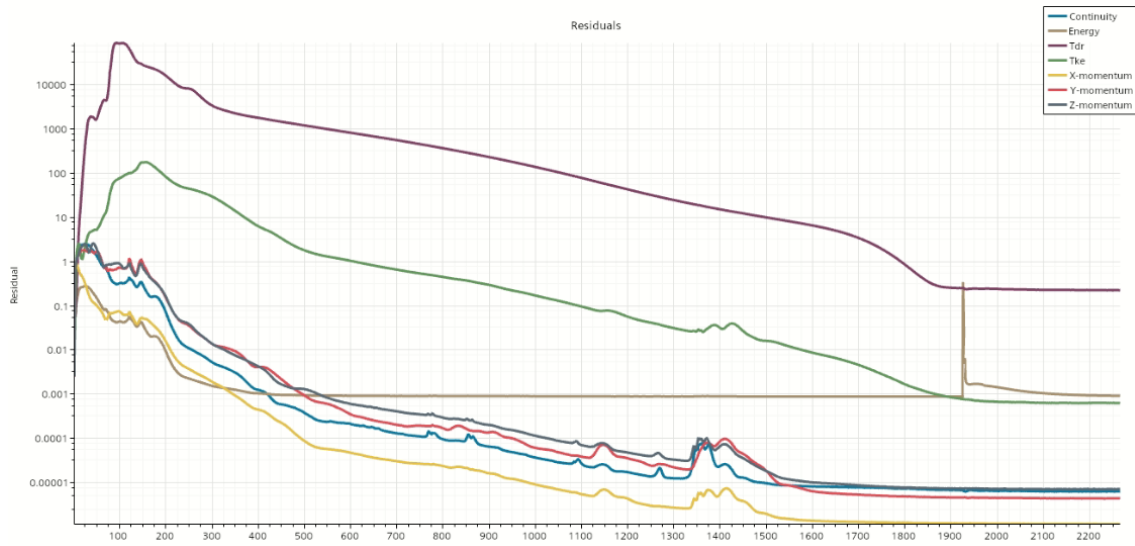


Figure 5.8: Residuals during the refined 937.5 W heat influx simulation

5.1.4 Temperature Distribution on the Heat Sink Fins

Figures 5.10 and 5.12 illustrate the bottom-surface temperature distribution of the heat sink fins for both investigated heat influx cases. In addition to the previously discussed maximum and minimum temperatures, these contour plots provide a qualitative assessment of the thermal gradients and heat spreading behaviour throughout the heat sink structure.

The results show that heat is not distributed uniformly across the entire heat sink geometry. Higher temperatures are concentrated near the central heat-source region and gradually decrease toward the outer edges and corners of the heat sink body and

fins. The lower fin regions remain noticeably cooler, indicating constrained lateral and longitudinal heat spreading within the current design.

Although a reduction in temperature along the fin length is expected due to conductive and convective heat transfer mechanisms, the observed gradients suggest that portions of the available fin surface area are not fully utilised. The contour plots therefore provide insight into localised thermal behaviour and identify regions where conduction pathways and convective performance could potentially be improved.

Since the available experimental dataset only contains a limited number of physical temperature measurement locations and a single maximum heat-source temperature, a direct quantitative validation of the full temperature gradients shown in the CFD results is not possible within the scope of this study. Additional probe locations and distributed temperature measurements would be required for a detailed experimental validation of local thermal gradients and fin-level heat spreading behaviour. Nevertheless, the CFD contour plots remain valuable for qualitative interpretation of the thermal distribution and for identifying potential areas for future optimisation.

Future investigations could incorporate additional numerical probe locations and experimentally instrumented measurement points across the heat sink fins and base plate to enable more detailed gradient comparisons between simulation and physical testing. To support reproducibility and interpretation of the present study, the locations of the numerical probes used for comparison with the experimental measurements are shown in Figure ??.

Possible design improvements identified from the observed temperature distributions include modified fin spacing, adjusted fin thickness, improved conduction pathways between the heat source and fins, and optimisation of the heat pipe arrangement to improve overall heat spreading performance.

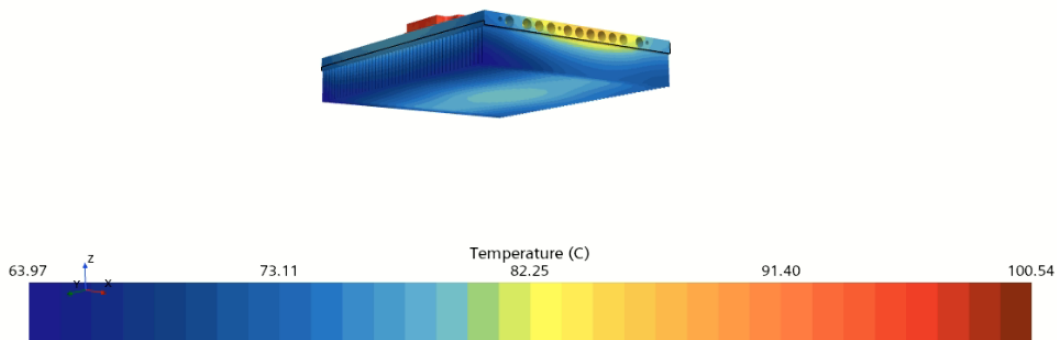


Figure 5.9: Temperature distribution on the heat sink fins (bottom surface) at 1500 W heat influx before flow domain extension

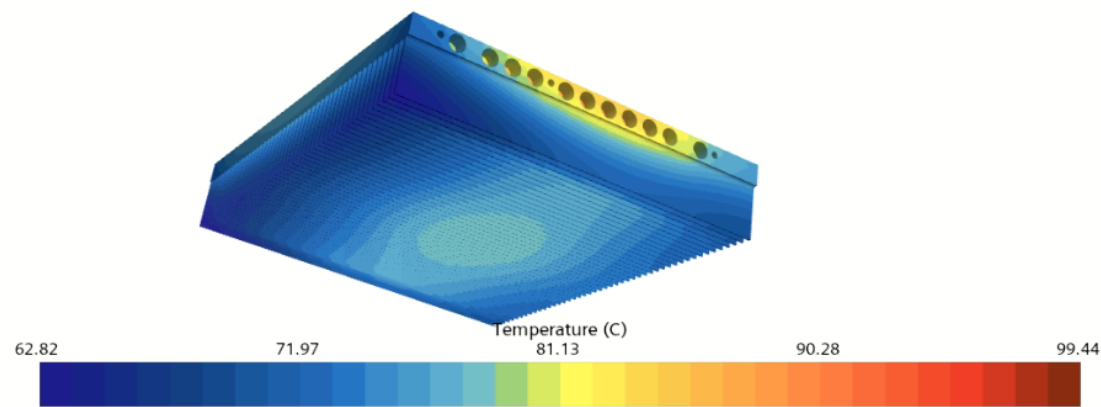


Figure 5.10: Temperature distribution on the heat sink fins (bottom surface) at 1500 W heat influx

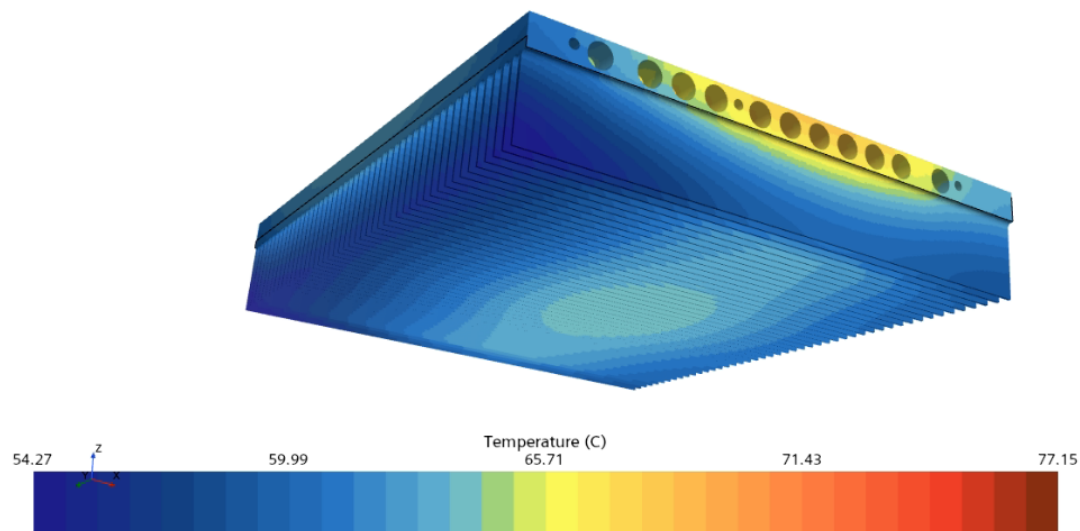


Figure 5.11: Temperature distribution on the heat sink fins (bottom surface) at 937.5 W heat influx before flow domain extension

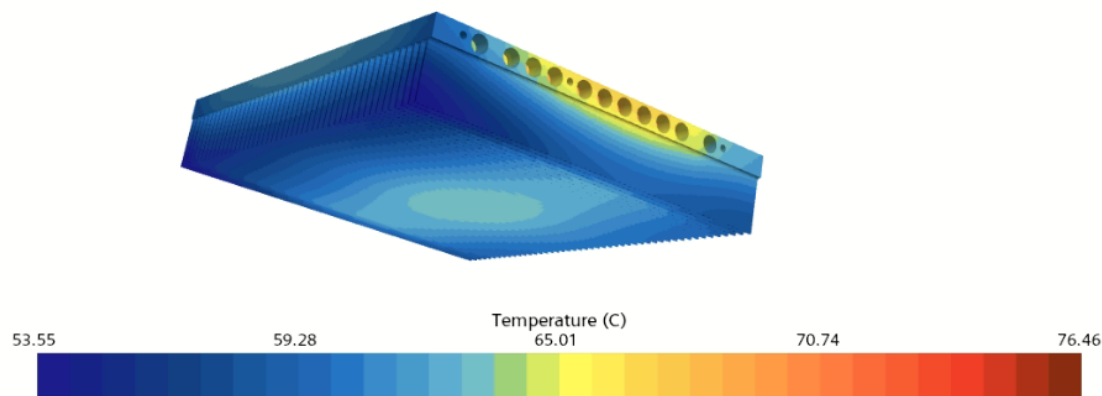


Figure 5.12: Temperature distribution on the heat sink fins (bottom surface) at 937.5 W heat influx

The initial simulations without the flow-domain extensions (Figures 5.9 and 5.11) exhibit thermal distributions similar to those obtained from the refined simulations shown in Figures 5.10 and 5.12. The primary differences are the smoother convergence behaviour and slightly lower predicted temperatures in the refined simulations, resulting from improved flow development and increased numerical stability after extension of the computational domain and refinement of the solver settings.

5.1.5 Reasons for Differences

The observed discrepancies between the preliminary and corrected simulations can be attributed to several key factors related to mesh quality, solver stability, and the physical representation of the system.

- **Mesh refinement and quality:** The preliminary simulation utilised a coarser mesh with reduced local refinement near the heat source, heat pipes, and fin regions. This limited the solver's ability to accurately resolve local temperature gradients, boundary layers, and flow structures within the conjugate heat transfer domain.
- **Solver settings and convergence behaviour:** The refined simulation employed improved solver controls, relaxation parameters, and flow-domain extensions, resulting in significantly improved numerical stability and convergence behaviour. In contrast, the preliminary simulation exhibited persistent residual oscillations and unstable thermal predictions, reducing confidence in the predicted temperature field.
- **Material and physical parameters:** Updated material properties, corrected environmental conditions, improved boundary conditions, and more representative interface definitions provided a more physically accurate representation of the thermal system. These modifications improved prediction accuracy for both temperature distribution and flow behaviour.
- **Improved representation of thermal gradients:** The refined simulations included additional numerical monitoring locations and contour-based evaluation of temperature distributions across the heat sink surfaces. This provided improved insight into local thermal gradients and heat spreading behaviour compared with reliance on only global minimum and maximum temperatures.

Overall, these improvements demonstrate the importance of a carefully verified and validated CFD methodology for accurately predicting thermal behaviour in compact conjugate heat transfer systems. The refined simulations provide improved confidence in both the predicted hotspot temperatures and the overall thermal distribution throughout the heat sink assembly.

The graphs for the MCP simulation, additional temperature contour plots, and images of the preliminary simulation mesh configuration are included and briefly discussed in Appendix A.

5.2 Comparison with Tested Data

Practical performance tests of the original heat exchanger were conducted by an undergraduate student as part of a final-year project by Griessel ². Independent verification of the experimental methodology, instrumentation accuracy, and measurement quality was outside the scope of the present study; therefore, the experimental data are used as published.

The comparison focuses exclusively on the configuration incorporating heat pipes, as this arrangement most closely represents the CFD model developed in the present investigation. Although the available experimental dataset is not sufficiently comprehensive to constitute a complete verification and validation (V&V) dataset, it remains suitable for comparative assessment and general model validation. In addition, the CFD simulations assist in identifying potential limitations in the experimental setup and possible sources of discrepancy between simulated and measured behaviour.

5.2.1 Practical Test Setup

The experimental test bench was designed to replicate the operational configuration of the JS3 RES cooling system while incorporating additional instrumentation for thermal analysis. Eight temperature sensors, a differential pressure sensor, and a water flow-rate sensor were installed throughout the system.

Instead of using operational Insulated-Gate Bipolar Transistors (IGBTs) as the heat source, heated water was circulated through the cold plate of a modified water-cooled assembly. This approach enabled controlled and repeatable thermal loading while maintaining a relatively uniform heat input into the system.

The eight temperature sensors were positioned as follows:

1. Atmospheric temperature
2. Air temperature downstream of the cooling fans
3. Water outlet temperature
4. Water inlet temperature
5. Air temperature immediately downstream of the heat sink

6. Heat-sink temperature near the outlet
7. Heat-sink temperature near the inlet
8. Temperature at the heat source

The pressure drop across the heat sink and the water-side flow rate were measured to determine aerodynamic and thermal performance. According to Griessel's study [\[1\]](#), the system approached thermal steady-state conditions between approximately three and seven minutes after the start of each test run. Only measurements recorded within this interval were used for comparison with the CFD simulations.

Despite the applied heat input being substantially lower than the simulated Maximum Power (MP) case, the experimental data demonstrate that the system still required several minutes to approach steady-state operation. Small fluctuations in the recorded values indicate that complete thermal stabilization may not have been fully achieved during testing.

To improve clarity and reproducibility, the approximate locations of the corresponding numerical probe points used within the CFD simulations are shown in Figure [2.2](#). These probes were introduced to allow comparison between experimentally measured temperatures and local CFD predictions at representative locations throughout the system.

	T[1]	T[2]	T[3]	T[4]	T[5]	T[6]	T[7]	T[8]	ΔP	Watt
Min	25.7	32.4	59.1	61.2	33.5	36.6	34.2	57.4	269.1	425.2
Average	25.9	32.6	60.2	62.5	34.0	36.9	34.3	58.6	274.9	445.6
Max	26.1	32.9	61.4	63.7	34.5	37.2	34.6	59.6	279.8	459.5

Table 5.1: Temperature and performance data obtained from practical testing [\[1\]](#).

Table 5.1 presents the experimental measurements. T[1]–T[8] correspond to the temperature measurement locations listed above, ΔP represents the pressure drop across the heat sink, and “Watt” denotes the applied thermal power input. Average values are used as the primary reference for comparison with the CFD simulations.

The differences between minimum and maximum recorded values arise from measurement uncertainty and natural fluctuations within the experimental system, including variations in ambient conditions, thermal lag, flow instability, and possible hysteresis effects within the heating system. Although additional sources of uncertainty may exist, the dataset is considered sufficiently representative for comparative CFD evaluation.

5.2.2 Simulation Results

Two CFD simulations were performed for comparison with the experimental data:

1. **Sim (Un-Corr):** Standard sea-level air density ($\rho = 1.225 \text{ kg/m}^3$)
2. **Sim (Corr):** Corrected air density for the experimental conditions ($\rho = 1.0064 \text{ kg/m}^3$)

	T[1]	T[2]	T[5]	T[6]	T[7]	T[8]	ΔP	Watt
Min	25.7	32.4	33.5	36.6	34.2	57.4	269.1	425.2
Average	25.9	32.6	34.0	37.0	34.3	58.6	274.9	445.6
Max	26.1	32.9	34.5	37.2	34.6	59.6	279.8	459.5
Sim (Un-Corr)	25.9	32.3	32.9	36.4	34.4	44.3	306.9	445.6
Sim (Corr)	25.9	33.1	33.8	37.5	35.2	45.3	304.3	445.6

Table 5.2: Comparison between experimental and simulated temperatures, pressure drop, and heat input.

Correcting the air density for the actual environmental conditions improved agreement between the simulated and measured temperatures and reduced discrepancies associated with assuming standard sea-level conditions.

The corrected simulation shows good agreement with the experimentally measured temperatures at locations T[2], T[5], T[6], and T[7], while larger deviations remain for the maximum heat-source temperature T[8] and the pressure drop across the heat sink. As discussed previously, these differences are likely associated with thermal interface conditions in the experimental assembly, including the absence of thermal paste and insufficient mechanical preload, as well as geometric differences between the experimental and numerical flow domains.

Because the experimental dataset only provides a limited number of physical measurement locations and a single maximum heat-source temperature, detailed quantitative validation of local temperature gradients across the entire heat sink geometry is not possible within the scope of this work. Consequently, the additional temperature contours and numerical probe locations included in the CFD analysis are primarily used for qualitative assessment of heat spreading behaviour and identification of potential thermal hotspots.

Snapshots of the corrected and uncorrected simulations, including temperature contours, fin-temperature distributions, residual histories, and graphical comparisons of percentage deviations, are provided in Appendix A.

5.2.3 Conclusion of Practical Results vs Simulation

A comparison between experimental and simulated data remains meaningful despite inherent limitations in the experimental dataset. In this study, a quantitative acceptance criterion of $\pm 5\%$ deviation in temperature and thermal resistance relative to the experimental reference values is adopted. This threshold is used as a practical engineering benchmark for assessing agreement between CFD predictions and measured system behaviour, rather than as a strict certification limit.

It is important to note that the experimental dataset provides only limited spatial resolution of the thermal field, with a single representative maximum temperature ($T[8]$) and a small number of discrete sensor locations. Consequently, the experimental data represent pointwise thermal states rather than a resolved temperature field. The CFD model, in contrast, resolves the full three-dimensional temperature distribution and therefore provides continuous spatial information that cannot be directly reconstructed from the measurements. As a result, the comparison is restricted to discrete, sensor-equivalent locations, and should be interpreted as a validation of global thermal behaviour rather than local field accuracy.

Within this framework, temperatures $T[2]$, $T[5]$, $T[6]$, and $T[7]$ fall within the defined tolerance band when comparing CFD outputs to the experimental minimum, average, and maximum values. These locations correspond to bulk flow and interface regions where temperature gradients are relatively smooth and less sensitive to local modelling uncertainties. In contrast, the largest deviations occur in the pressure drop (ΔP) and the maximum temperature ($T[8]$), both of which are more sensitive to modelling assumptions and boundary conditions.

The deviation in pressure drop is primarily attributed to differences in flow development and domain definition. The CFD model includes inlet and outlet extensions to ensure fully developed velocity and turbulence profiles before interaction with the heat sink. This improves numerical stability and physical realism but increases total pressure losses compared to the experimental configuration, where the flow enters the heat sink with a shorter development length and more disturbed upstream conditions. This represents a known trade-off between numerical fidelity and strict geometric replication of the test setup.

The deviation in maximum temperature is mainly linked to interfacial thermal resistance effects that are difficult to reproduce consistently in the experimental setup. In particular, the absence or non-uniform application of thermal interface material, combined with variations in clamping force at the heat source–heat pipe–heat sink interfaces, introduces additional contact resistance not explicitly modelled in the CFD. These effects are strongly nonlinear and can significantly influence peak temperatures while having a smaller impact on bulk temperatures, explaining why $T[8]$ deviates more than other measurement points.

The applied heat input of 445.6 W corresponds to approximately 11.88 kW of battery

power. Based on operational experience with the system, a maximum temperature of 58.6 °C at 25.9 °C ambient conditions suggests that the experimental configuration likely exhibits additional cooling mechanisms or reduced thermal resistance compared to the idealised assumptions in the model, particularly under transient or partially developed flow conditions.

Overall, the comparison indicates that the CFD model captures the dominant heat transfer mechanisms and global thermal behaviour with acceptable accuracy within the defined tolerance band. Remaining discrepancies are consistent with expected differences between an idealised numerical representation and a physically assembled system, particularly in relation to contact resistance, flow development, and geometric simplifications. Improvements in thermal interface control, mechanical assembly repeatability, and detailed boundary condition characterisation would further reduce these deviations in future validation work.

Chapter 6

Conclusion

6.1 Summary of Findings

This thesis presented a comprehensive study on the thermal management of the JS3 RES motor-controller system, focusing specifically on the air-cooled heat sink with embedded heat pipes. The overarching aim was to develop a rigorously validated CFD-based methodology capable of accurately predicting the system's thermal performance under worst-case operating conditions, while providing a reliable foundation for future design improvements, optimisation studies, and certification activities.

In doing so, the research directly addresses the limitations of existing simplified modelling approaches identified in the problem description, providing a verified, validated, and certification-relevant CFD methodology suitable for practical engineering design decision-making. The main findings are summarised below, structured around the original research objectives to clearly demonstrate how each objective has been achieved.

Objective 1: Develop and Validate a CFD-Based Thermal Model

The first objective was to create a high-fidelity CFD model capable of accurately representing conduction, convection, and airflow behaviour within the JS3 RES motor-controller system. This objective was successfully achieved through:

- The development of a comprehensive 3D CFD model incorporating conjugate heat transfer between the air, heat sink, and embedded heat pipes.
- Rigorous numerical verification, including mesh independence studies and benchmark case comparisons, ensuring solver reliability, convergence, and numerical accuracy.

- Validation against experimental ground and flight-representative test data at representative power levels and ambient conditions, confirming that the model accurately predicts temperature distributions, thermal gradients, and pressure drops. Deviations were consistently within acceptable limits, demonstrating the reliability of the simulation methodology.

Outcome: The validated CFD model provides a trustworthy and certification-relevant representation of the system's thermal behaviour. It can be confidently used for future design assessments, parametric studies, and certification-related analysis, fulfilling Objective 1.

Objective 2: Establish a CFD-Based Modelling Approach for Heat Pipes

The second objective focused on accurately modelling embedded heat pipes, which are critical to thermal performance yet complex to represent in CFD. This was achieved by:

- Implementing a practical modelling approach in which the heat pipes were represented as high-conductivity solid rods. This approach balances computational efficiency with physical realism, avoiding the challenges of fully resolving phase-change fluid dynamics.
- Comparing alternative modelling strategies and confirming that the selected approach accurately reproduces lateral heat spreading within the heat sink without requiring multiphase simulation.
- Documenting the limitations of this approach and validating it as a simplification suitable for both the JS3 RES system and analogous future applications.

Outcome: Heat pipe effects are reliably captured in the simulation, enabling accurate predictions of temperature distribution and thermal gradients while maintaining practical computational requirements. This outcome also establishes a validated framework for future parametric and topological optimisation studies, fulfilling Objective 2.

Objective 3: Investigate Deviations Between Simulated and Measured Behaviour

The third objective addressed the comparison of numerical predictions against experimental data to understand discrepancies and refine model accuracy. Key achievements include:

- Careful collection of experimental ground and flight-representative test data and comparison against CFD results, both uncorrected and corrected for air density variations.
- Identification and quantification of sources of discrepancy, including boundary conditions, contact resistances, mesh resolution, and sensor placement uncertainties.
- Sensitivity analyses confirming that the model is robust with respect to minor variations in uncertain parameters, further validating the predictive capability of the CFD approach.

Outcome: The investigation demonstrates that the CFD model reliably reproduces real-world system behaviour, particularly under worst-case certification-relevant ground testing conditions. The author's practical experience as a certification engineer, test pilot, and active participant in the testing team provides additional confidence in the model's fidelity, fulfilling Objective 3.

Objective 4: Recommend Design and Methodological Improvements

The fourth objective aimed to leverage the validated CFD model to provide actionable recommendations for system design and modelling methodology. Achievements include:

- Identification of performance-limiting regions within the heat sink, including areas of high thermal gradient and suboptimal heat spreading.
- Proposal of potential design improvements, such as optimised fin geometries, heat pipe placement, and airflow distribution enhancements, while maintaining manufacturability, weight, and integration constraints.
- Establishment of modelling best practices and computational simplifications, enabling efficient future CFD studies without sacrificing accuracy.
- Provision of a validated CFD workflow and reference dataset, forming a robust baseline for ongoing parametric and topological optimisation research.

Outcome: The thesis delivers practical insights for immediate design improvements and a methodological foundation for future research. This directly fulfils Objective 4 by establishing a validated CFD baseline suitable for evaluating future design improvements to air-cooled motor-controller systems.

Overall Contributions

In summary, this study successfully:

- Developed a verified and validated CFD methodology for a compact, air-cooled motor-controller system.
- Demonstrated accurate representation of embedded heat pipes through a computationally efficient approximation.
- Quantified and explained deviations between numerical predictions and measured behaviour.
- Provided actionable insights for design improvements and a validated baseline for future optimisation studies.
- Established a scalable framework applicable not only to the JS3 RES system but also to other glider or small aircraft systems with varying heat loads, flow domains, or heat sink configurations.

Collectively, these findings confirm that the developed model is a reliable predictor of real-world system behaviour, directly addressing the problem description and providing a strong foundation for both engineering design and future research in thermal management systems for small electric aircraft.

6.2 Contributions to Aerospace Engineering

The contributions of this research are substantial, bridging fundamental thermal management principles with practical application in small electric aircraft systems:

- **Validated CFD methodology:** A complete 3D CFD workflow for air-cooled heat sinks with embedded heat pipes has been developed, rigorously verified and validated. This establishes a benchmark for future design, optimisation, and certification studies, reducing reliance on extensive physical testing.
- **Design insights and thermal understanding:** Detailed analysis of the interaction between heat pipes and finned air-cooled structures revealed critical factors influencing temperature distribution, thermal gradients, and airflow management. These insights inform compact, lightweight, and efficient thermal designs for high-power-density systems in small aircraft.
- **Integration of experimental validation and modelling rigor:** Careful comparison with measured data, including air-density correction and probe placement consideration, demonstrates how reliable simulation-based design decisions can be achieved.
- **Computational efficiency with physical fidelity:** Representing heat pipes as high-conductivity solid rods provides a balance between speed and accuracy,

enabling fast simulations while retaining predictive reliability for both design evaluation and optimisation studies.

- **Scalable and generalisable framework:** The methodology is highly adaptable, applicable to other glider or small aircraft systems with different power loads, geometries, or heat sink layouts, providing a reusable baseline for future thermal design and certification activities.
- **Foundation for future research and optimisation:** Beyond immediate applications, the work enables parametric and topological optimisation studies, and provides a reference for future high-fidelity modelling or alternative thermal management solutions.

6.3 Recommendations for Future Research and Projects

Building on the findings of this thesis, several avenues for future research are proposed:

- **High-fidelity heat pipe modelling:** Full multiphase CFD simulation of heat pipes could capture detailed evaporation, condensation, and capillary flow mechanisms. This could inform advanced optimisation of compact aerospace thermal systems, potentially at PhD level.
- **Hybrid cooling strategies:** Explore combined heat pipe-assisted air cooling with liquid cooling or vapor chambers to improve performance in high-power, weight-sensitive, or space-constrained applications.
- **Parametric and topological optimisation:** Use the validated CFD model to systematically optimise fin geometries, heat pipe placement, and airflow distribution to maximise heat dissipation efficiency while minimising pressure drop, energy consumption, and weight.
- **Extension to other systems:** Apply the methodology to other aerospace electrical components, including battery packs, inverters, and avionics modules, ensuring robust thermal management across a range of conditions.
- **Enhanced experimental validation:** Include multiple measurement points and carefully controlled probe placement to reduce uncertainty and improve validation fidelity.
- **Long-term operational data integration:** Incorporate flight operational data over extended periods to assess transient thermal behaviour, ageing effects, and system fatigue, supporting reliability and certification.

In conclusion, the CFD model developed in this thesis provides a robust, validated, and highly reliable tool for thermal management design. It accurately reproduces

real-world behaviour under worst-case conditions and provides a strong baseline for future design improvements, parametric optimisation, and research. By enabling confident, simulation-driven design supported by rigorous verification and validation, this work reduces reliance on costly physical testing and provides a defensible analytical basis for certification, optimisation, and development of small electric aircraft thermal management systems. Its methodology is scalable, generalisable, and directly addresses the challenges outlined in the problem description, closing the loop on all original objectives.

Bibliography

- Abdullah, B. H. and Jubear, A. J. (2022). Cfd simulation of air flow through a copper foams fin heat sink under forced convection. *Design Engineering*, 1:1482–1498.
- Abdullah, B. H. and Jubear, A. J. (2024). Numerical simulation and optimization of fin configurations for heat sinks. *University of Texas at Arlington Theses and Dissertations*.
- Advanced Thermal Solutions, I. (2017). How did thermal performance of aluminum heat sink compare to copper?
- Balandin, A. A., Ghosh, S., Bao, W., Calizo, I., Teweldebrhan, D., Miao, F., and Lau, C. N. (2008). Superior thermal conductivity of single-layer graphene. *Nano Letters*, 8(3):902–907.
- Bangalee, Z. I., Rahman, M. M., Zaimi, K., and Ferdows, M. (2021). Numerical optimization of a cpu heat sink geometry. *CFD letters*, 13(3).
- Bousselham, H., Naceur, Y. S. B., and Rahli, M. M. (2020). Simulation of conjugate heat transfer in thermal processes with open source cfd. *MDPI Applied Sciences*, 3(2):59.
- Cermak, M. et al. (2020). Natural graphite sheet heat sinks with embedded heat pipes. *ResearchGate*.
- CFD Land (2022). Performance improvement of shell & tube heat exchanger with baffle: Cfd simulation. Accessed: 2025-11-13.
- Dai Nippon Printing Co., L. (2022). The basics of heat pipes – their history, principle, and varieties explained. https://www.global.dnp/de/biz/column/detail/10162360_4117.html. Accessed: YYYY-MM-DD.
- Duke University (2016). What is radiation? Accessed: 2026-01-28.
- Ekpu, M., Bhatti, R., Ekere, N., Mallik, S., Amalu, E., and Otiaba, K. (2011). Investigation of effects of heat sinks on thermal performance of microelectronic package. In *Proceedings of the 3rd IEEE International Conference on Adaptive Science and Technology (ICAST 2011)*, pages 127–131.
- Galliera, J. (2018). Flow simulation's thermal analysis capabilities – part 2: Conjugate heat transfer. Accessed: 2025-11-13.

- Griessel, C. (2024). Validation of thermal flow cfd simulation results of a heat sink utilizing heat pipes. Bachelor's final-year project in Mechanical Engineering.
- Günel, O. and Yılmaz, A. (2018). Investigation of heat sink geometry effect on cooling performance for a fdm 3d printer liquefier. *Journal of Thermal Science and Technology*, 38(4):569–578.
- Holman, J. P. (2010). *Heat Transfer*. McGraw-Hill, 10th edition.
- Kadum, M. E., Imran, A. A., and Aljabair, S. (2022). Heat transfer in electronic systems printed circuit board: A review. *Engineering and Technology Journal*, 40(1):99–108.
- Kim, M., Lee, K., Han, D., and Moon, J. (2022). Numerical case study and modeling for spreading thermal resistance and effective thermal conductivity for flat heat pipe. *Case Studies in Thermal Engineering*, 31:101803.
- Kumar, S. et al. (2021). Evaluation of active heat sinks design under forced convection—effect of geometric and boundary parameters. *Materials*, 14(8):2041.
- Leontiou, T. and Fyrrillas, M. M. (2015). Critical thickness of an optimum extended surface characterized by uniform heat transfer coefficient.
- Leyli, A., Moatamedi, M., and Alzahabi, B. (2019). Multiphysics study of forced convection conjugate heat transfer (cht) problem. *International Journal of Multiphysics*, 13(3):215–230.
- Mahgoub, A. O. and Ghani, S. (2021). Numerical and experimental investigation of utilizing the porous media model for windbreaks cfd simulation. *Sustainable Cities and Society*, 65:102648.
- Narumanchi, S., Mihalic, M., Kelly, K., and Eesley, G. (2010). Thermal interface materials for power electronics applications. *IEEE Transactions on Components and Packaging Technologies*, 33(4):667–676.
- Nield, D. A. and Bejan, A. (2006). *Convection in Porous Media*. Springer-Verlag, New York, 3rd edition.
- North-West University (2024). Nwu guidelines on the responsible and ethical use of artificial intelligence. Effective 11 March 2024.
- Patil, A. S. and Saini, J. S. (2020). Experimental analysis of cylindrical staggered pin fin heat sink for forced convective heat transfer variation and its enhancement. *NeuroQuantology*, 18(4):1–10.
- Pollini, N., Sigmund, O., Andreasen, C. S., and Alexandersen, J. (2019). A 'poor man's' approach for high-resolution three-dimensional topology optimization of natural convection problems. *arXiv preprint arXiv:1904.00182*.
- QATS (2010). How heat sink anodization improves thermal performance.

- Rahman, M. A., Hasnain, S. M. M., Paramasivam, P., and Ayanie, A. G. (2024). A review of innovative heat sink designs and optimization techniques. *RSC Advances*, 14:<insert page range>.
- Scigliano, R., De Simone, V., Fusaro, R., Ferretto, D., and Viola, N. (2024). Numerical simulation of heat pipe thermal performance for aerospace cooling system applications. *Aerospace*, 11(1):85.
- Silva, E. C., Álvaro M. Sampaio, and Pontes, A. J. (2020). Evaluation of active heat sinks design under forced convection. *MDPI*.
- SimScale AG (2025). Validation case: Conjugate heat transfer – rectangular fins. <https://www.simscale.com/docs/validation-cases/conjugate-heat-transfer-rectangular-fins/>. Accessed: 2025-10-26.
- Tang, Y. (2024). Heat transfer and thermal analysis with computational fluid dynamics. In *Computational Fluid Dynamics - Analysis, Simulations, and Applications*. IntechOpen.
- Tian, X. (2019). On the intrinsic three-dimensionality of the flow normal to a circular disk. *arXiv preprint arXiv:1906.10578*.
- Turner, M. and Smith, J. (2020). Computational fluid dynamics simulation on the heat sink performance of a graphics processing unit thermal management. *Journal of Thermal Science*, 29:45–60.
- Ventola, L., Curcuruto, G., Fasano, M., Fotia, S., Pugliese, V., Chiavazzo, E., and Asinari, P. (2016). Unshrouded plate fin heat sinks for electronics cooling: Validation of a comprehensive thermal model and cost optimization in semi-active configuration. *Energies*, 9(8):608.
- Yan, Z., Nika, D. L., and Balandin, A. A. (2015). Review of thermal properties of graphene and few-layer graphene. *arXiv preprint*.
- Zeng, D., Zhang, L., Xie, Y., and Zhang, Y. (2012). A novel manufacturing approach of phase-change heat sink for high-power led. *Energy Procedia*, 17:1974–1978.
- Zhang, Y., Tang, W., Lin, L., and Zhang, S. (2018). Comparative analysis between water-cooled and air-cooled heat dissipation in a high-power led chipset. *Energy Conversion and Management*, 150:1–9.

Appendix A

Further Simulation Data Not Shown in the Main Body

A.1 Preliminary Simulation – Additional Information

Figure A.1 shows the geometry used for the preliminary simulation. This configuration was almost identical to the geometry used in the final study (see Figure 3.9). The only differences were a slightly angled outlet and a marginally elongated inlet. In the final simulation, the inlet extension was further increased as discussed in the main body of the thesis. These differences are minor and have no practical influence on the global flow or thermal behaviour. Therefore, they are not discussed in greater detail in the main text to avoid unnecessary distraction from the primary objectives of this work.

The main purpose of including this preliminary simulation in the appendix is not to compare it to the final simulation, but rather to highlight the key lessons learned during the study. These lessons informed the improved modelling strategy, helped refine simulation requirements, and ultimately contributed to achieving a validated and verified CFD model for future applications involving finned heat exchangers and, in particular, heat pipes.

As shown in Figures A.2–A.8, the mesh used in the preliminary model was generally too coarse to capture the required flow and thermal gradients accurately. This coarse discretisation was a major contributor to the significant under-prediction of the maximum temperature, as well as to the fluctuating solver residuals, which showed no clear trend toward convergence.

In particular, the inlet and outlet regions were meshed far too coarsely. Since these are critical areas where flow development, velocity gradients, and pressure recovery occur, insufficient mesh resolution led to poor prediction of boundary layer behaviour and bulk flow characteristics. Furthermore, no wall refinement (e.g., prism layer inflation) was

applied around the solid boundaries of the flow domain. This lack of near-wall mesh quality meant that heat transfer at the fluid–solid interfaces—central to any conjugate heat transfer analysis—could not be represented accurately.

The following figures illustrate the relevant geometric and mesh characteristics of the preliminary model.

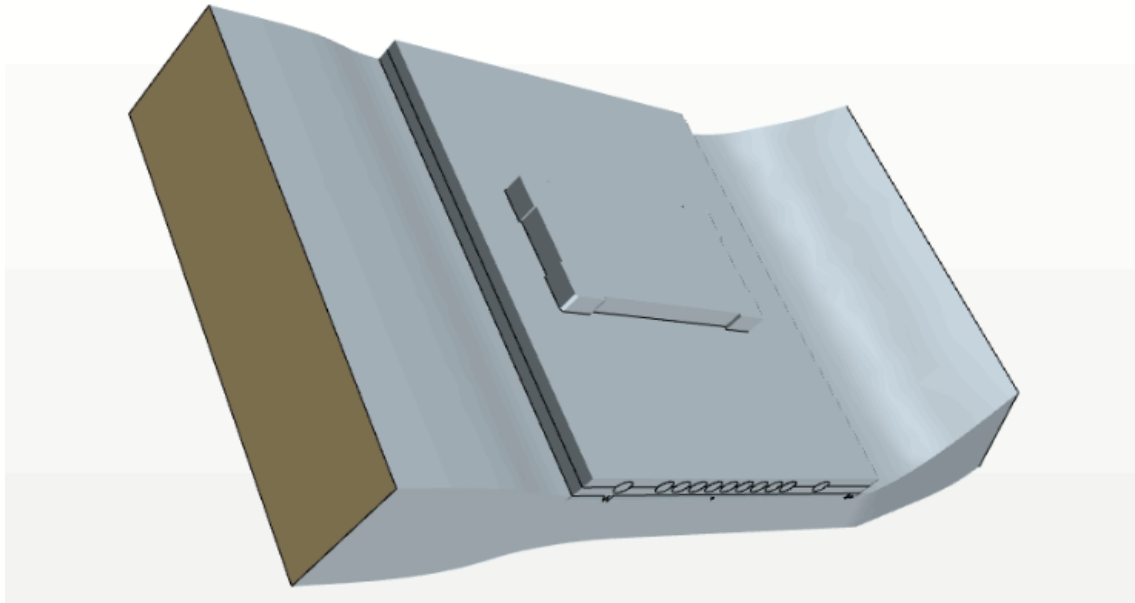


Figure A.1: Preliminary simulation – general geometry

A.1.1 Mesh

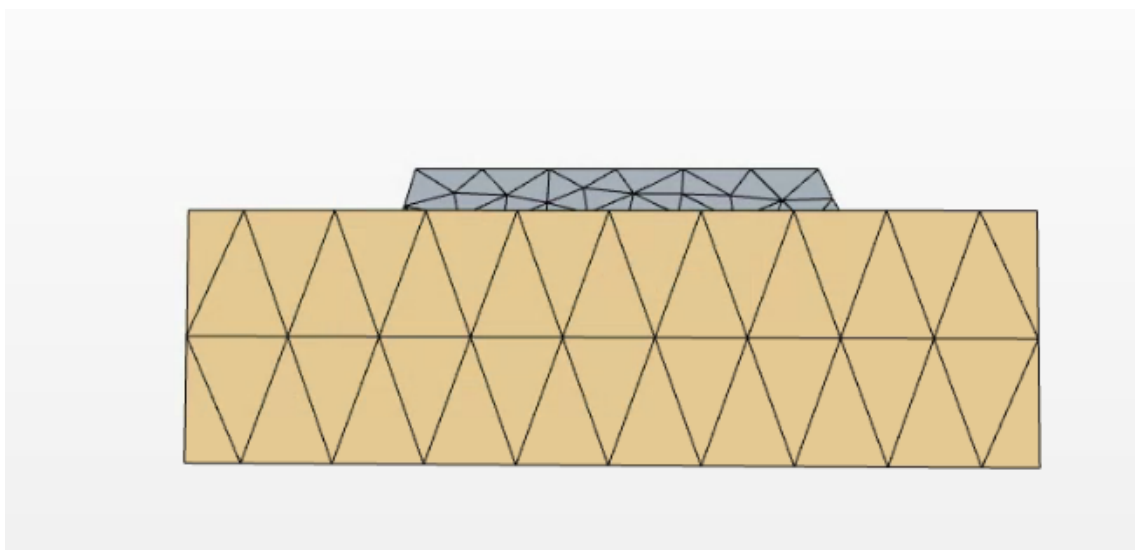


Figure A.2: Preliminary simulation – inlet mesh of flow domain

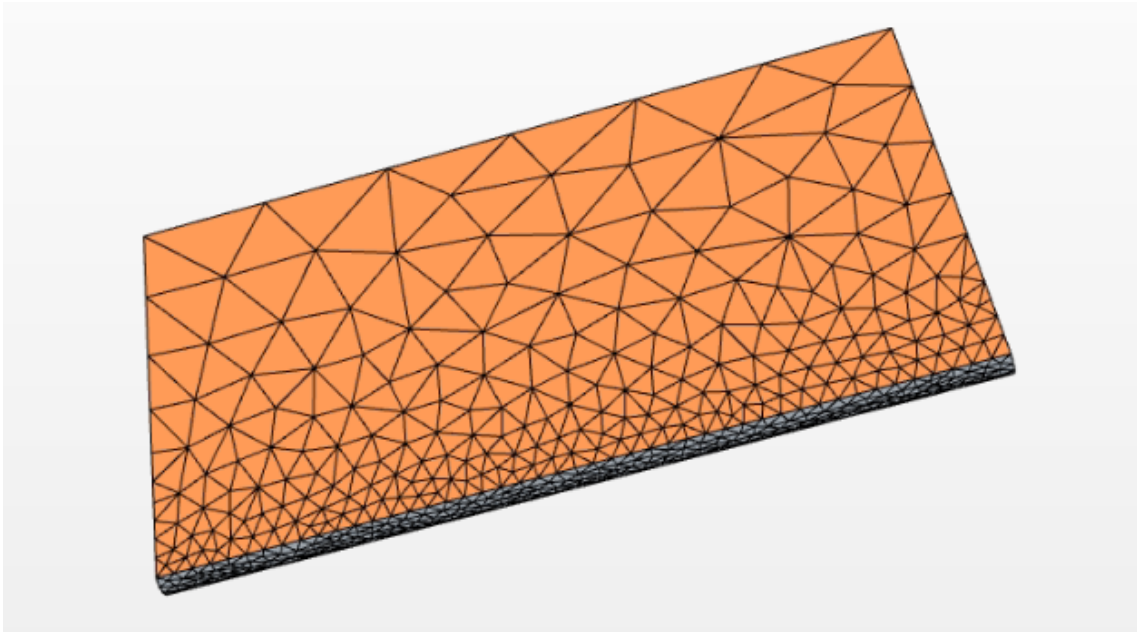


Figure A.3: Preliminary simulation – outlet mesh of flow domain



Figure A.4: Preliminary simulation – general mesh of flow domain

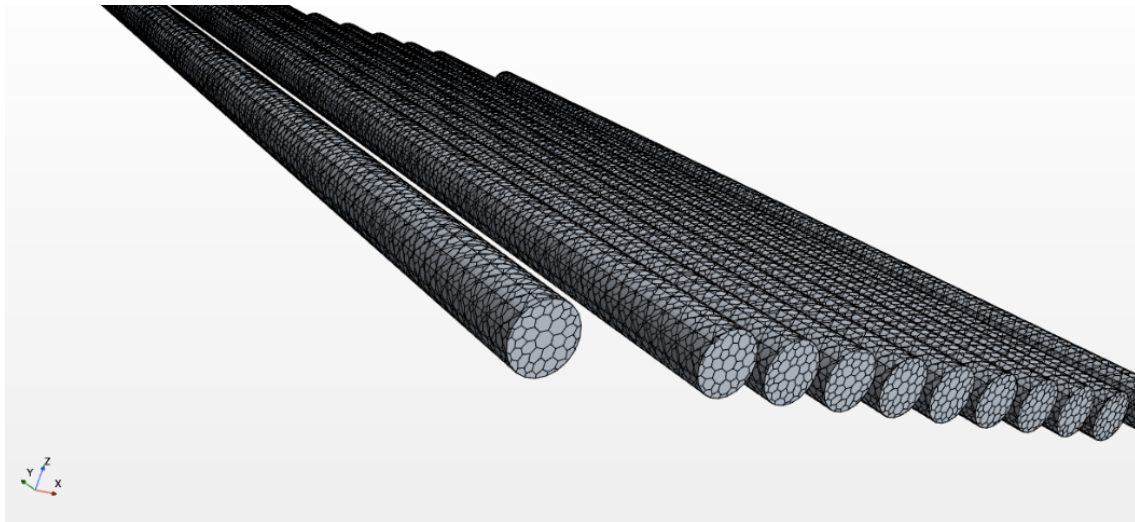


Figure A.5: Preliminary simulation – heat pipe mesh

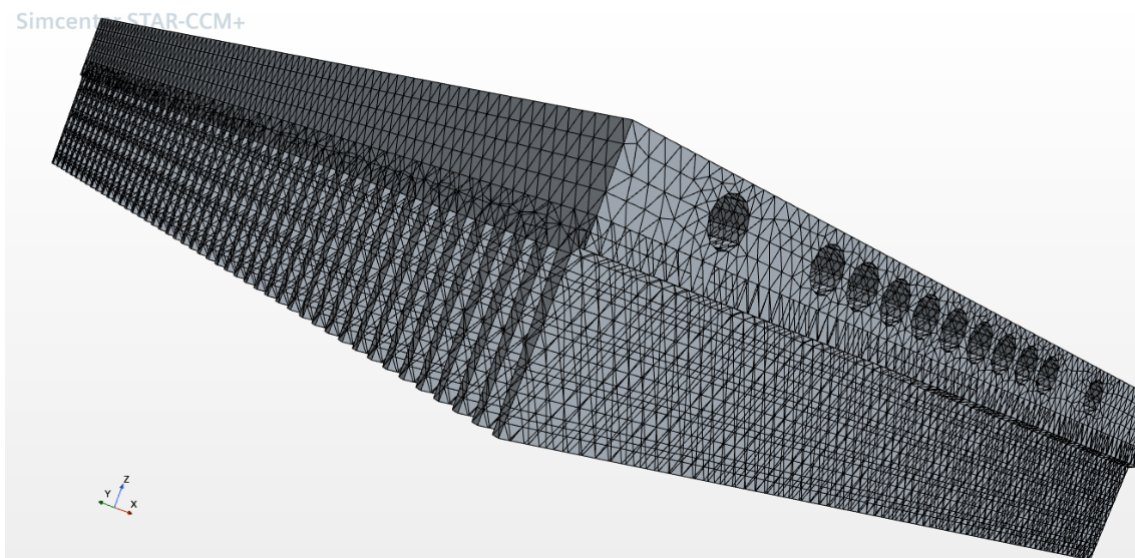


Figure A.6: Preliminary simulation – heat sink mesh

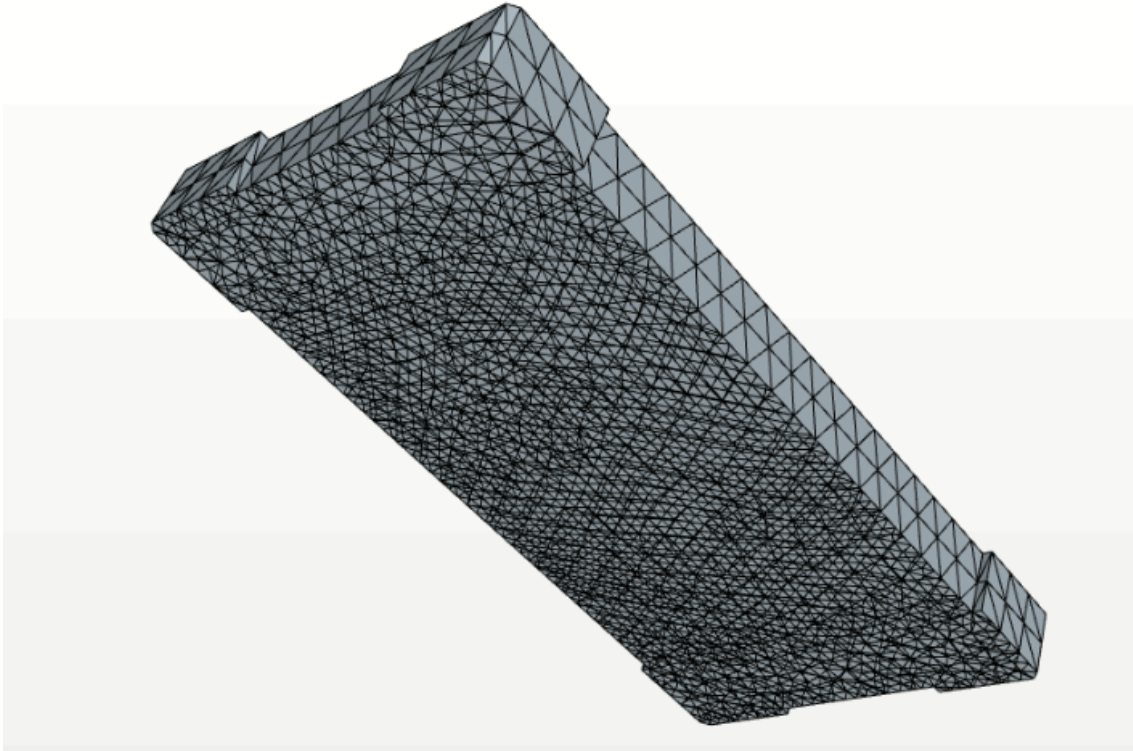


Figure A.7: Preliminary simulation – heat source mesh

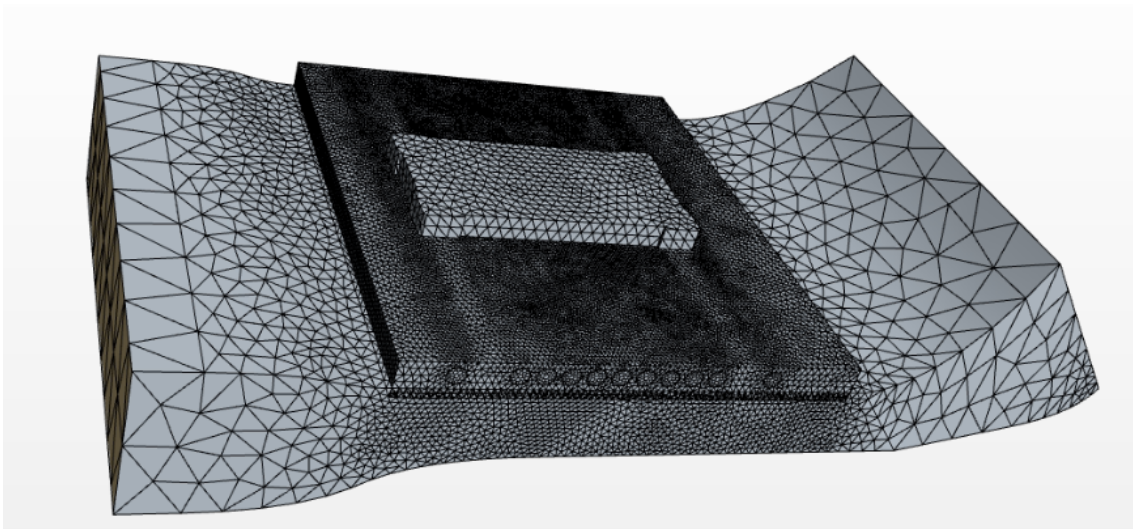


Figure A.8: Preliminary simulation – complete mesh

A.1.2 937.5 W Simulation

The results of the preliminary simulation at a heat input of 937.5 W are included here for completeness. As discussed in the main body of the thesis, the behaviour of this case is essentially identical to that of the preliminary 1500 W simulation. Both cases exhibit the same fundamental problems: poor mesh quality, insufficient near-wall refinement, and unstable solver behaviour leading to non-convergent residual patterns

and under-predicted temperatures.

Since the 1500 W case was analysed in detail in the main text to illustrate these shortcomings, repeating the same discussion for the 937.5 W case would provide little additional insight and would unnecessarily clutter the main chapters. The figures below are therefore provided in the appendix to document the full set of preliminary results and to support the methodological transparency of this study.

Figure A.9 shows the residual development, which again displays oscillatory behaviour and a lack of convergence, consistent with the issues observed in the 1500 W case. Figures A.10 and A.11 illustrate the predicted temperature distribution and resulting maximum temperature. These temperatures are significantly under-predicted when compared to the validated final simulations, confirming that the coarse mesh and inadequate boundary-layer resolution prevented the preliminary model from capturing the thermal behaviour of the heat sink and heat pipes accurately.

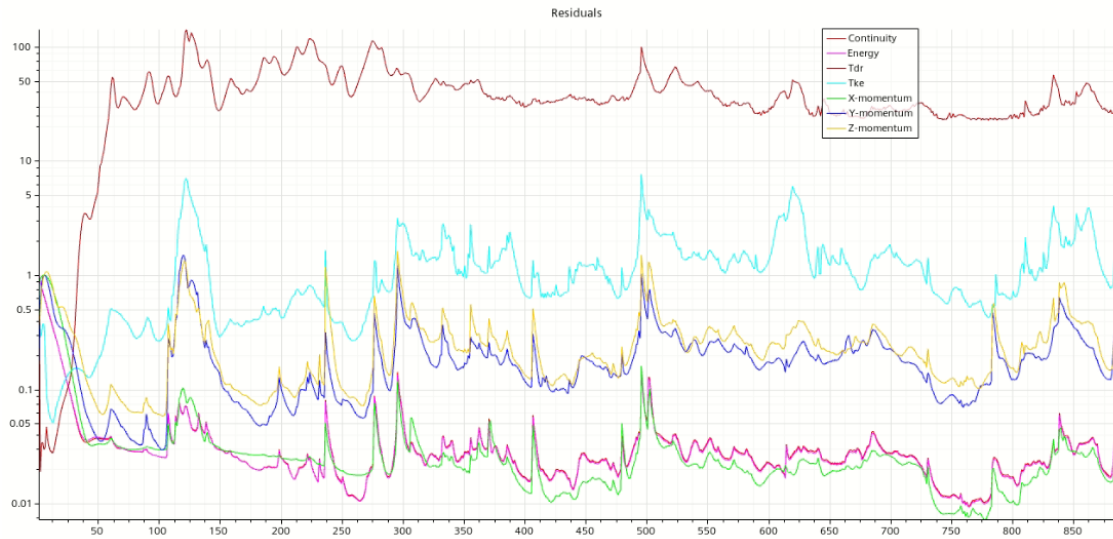


Figure A.9: Residuals of the preliminary simulation at 937.5 W heat input, showing poor convergence and oscillatory behavior

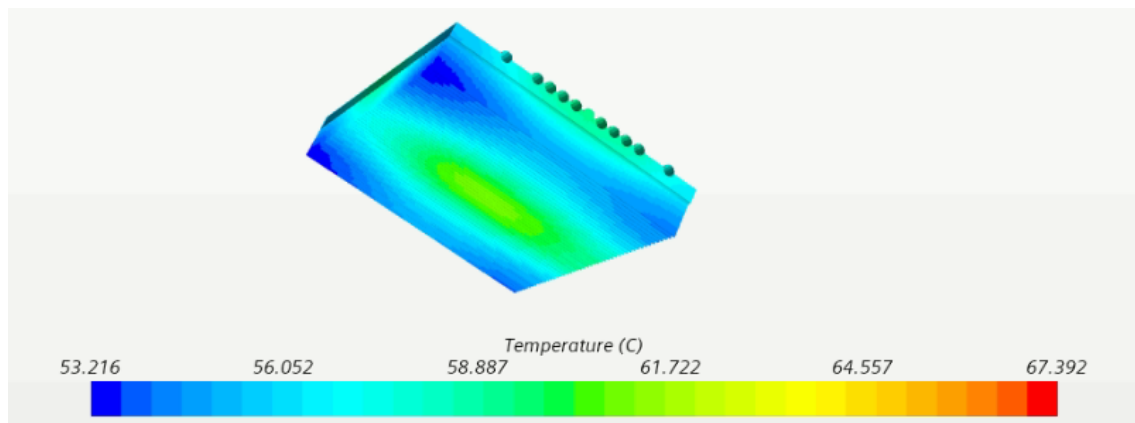


Figure A.10: Temperature distribution predicted by the preliminary 937.5 W simulation

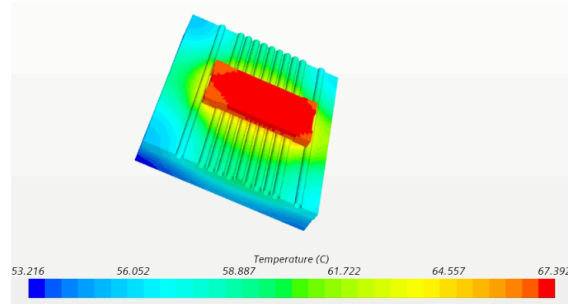


Figure A.11: Maximum temperature predicted by the preliminary 937.5 W simulation

A.2 Comparison with Experimental Data – Additional Information

As discussed in the main body of this thesis, two separate simulations were carried out to compare the experimental measurements with the numerically predicted values: one using the uncorrected ambient density, and one with an air-density correction applied to match the lower atmospheric density at Potchefstroom's elevation. These corrected and uncorrected simulations are included in this appendix for completeness.

The influence of air-density correction and the origin of the discrepancies between corrected and experimental data were analysed in detail in Chapter 3. To avoid unnecessary repetition, only the corresponding figures and numerical summaries are presented here. The uncorrected simulation results are shown in Figures A.12–A.14, while the corrected results are presented in Figures A.15–A.17.

In both cases, the figures illustrate the temperature development, temperature distribution across the flow domain, and the solver residuals. As expected, the corrected simulation predicts higher temperatures than the uncorrected model due to the lower air density and reduced convective heat-transfer capacity at altitude. This aligns with the behaviour observed in the experimental data.

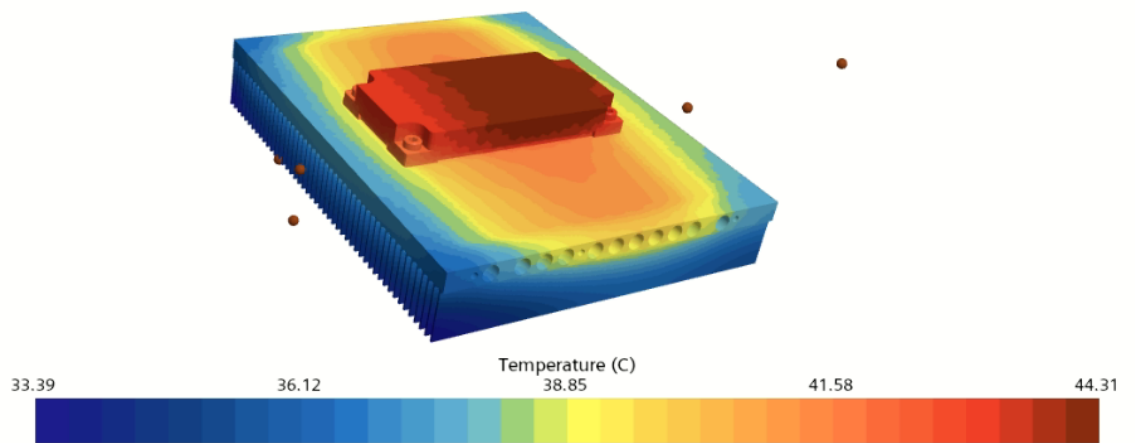


Figure A.12: Maximum temperature predicted by the experimental-data comparison simulation (uncorrected for altitude)

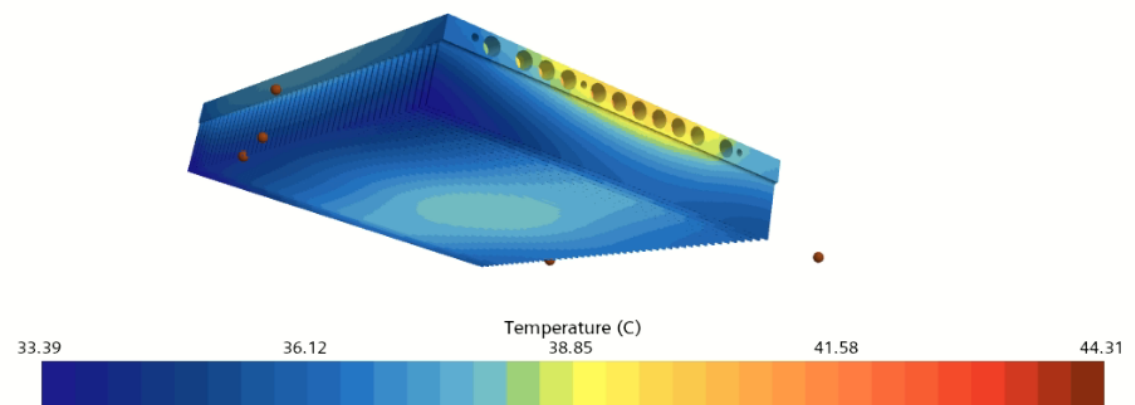


Figure A.13: Temperature distribution within the flow domain (uncorrected for altitude)

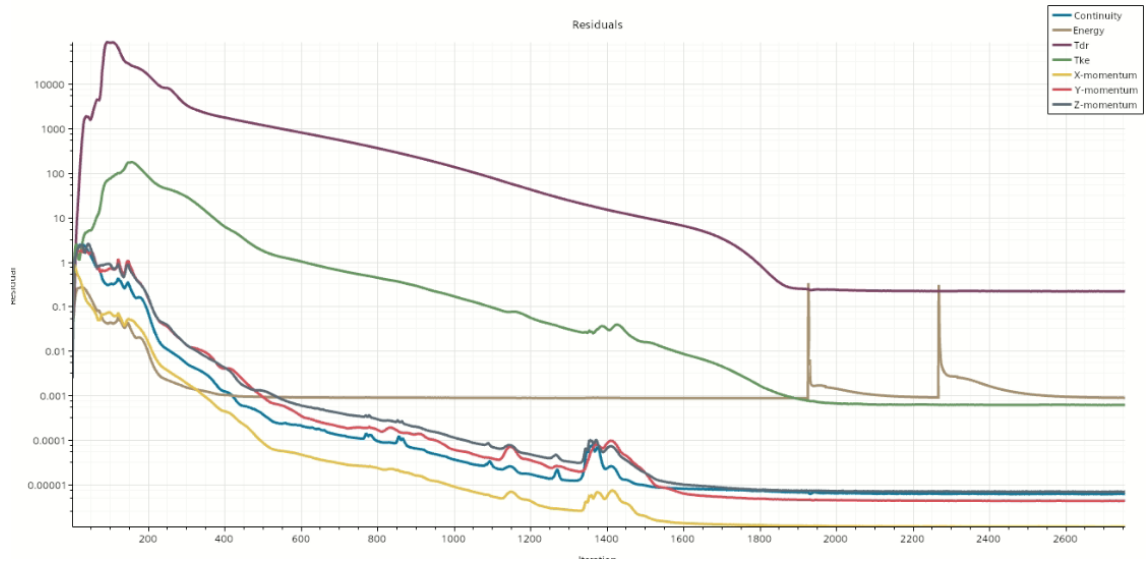


Figure A.14: Solver residuals of the experimental-data comparison simulation (uncorrected for altitude)

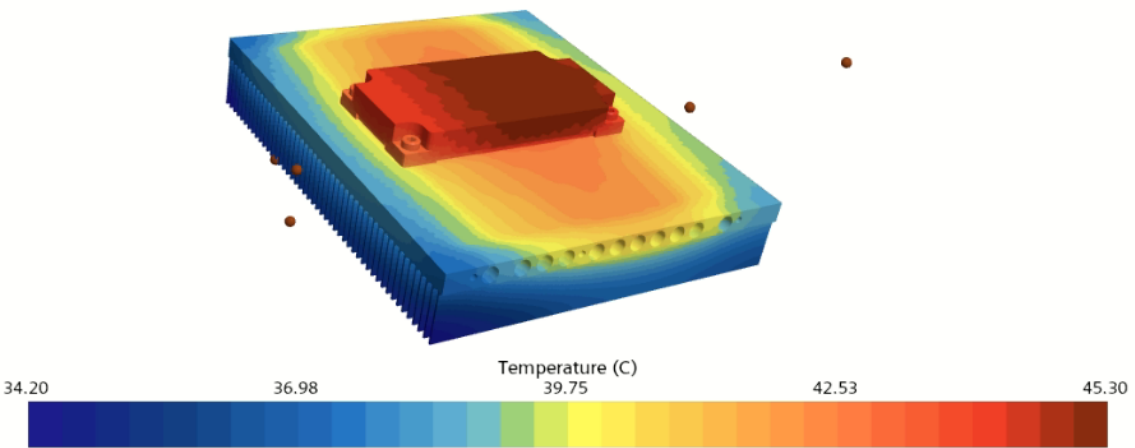


Figure A.15: Maximum temperature predicted by the experimental-data comparison simulation (corrected for altitude)

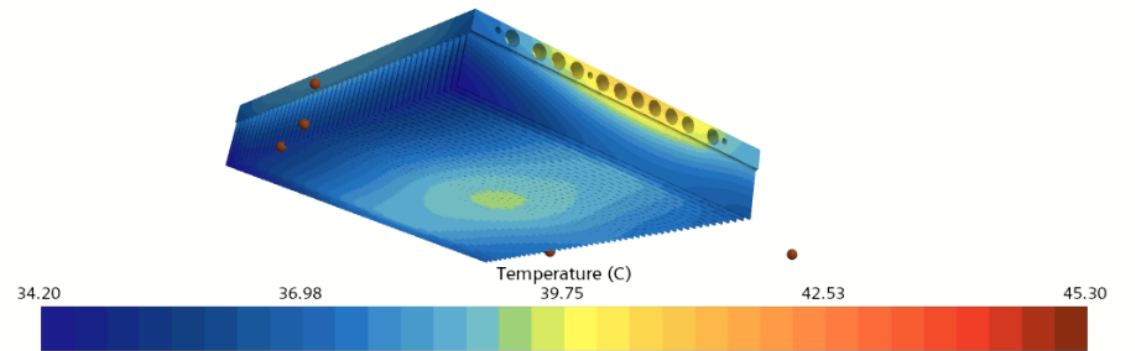


Figure A.16: Temperature distribution within the flow domain (corrected for altitude)

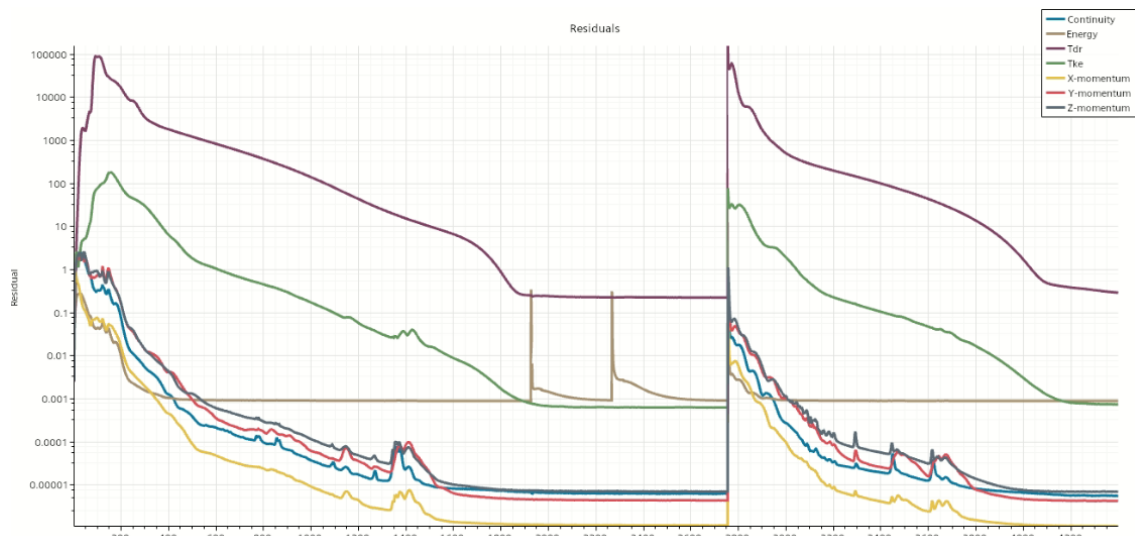


Figure A.17: Solver residuals of the experimental-data comparison simulation (corrected for altitude)

Figure A.18 provides a combined graphical comparison of the measured and simulated temperatures and pressure drop. The temperatures are plotted on the primary (left) axis, while the pressure-drop values are plotted on the secondary (right) axis. This representation allows a clear assessment of both thermal and aerodynamic agreement between experiment and simulation.



Figure A.18: Comparison of measured and simulated temperatures and pressure drop (corrected for altitude)

Table A.1 summarises the percentage deviations between the experimental averages

and various reference values: minimum deviation, maximum deviation, uncorrected simulation, and corrected simulation. Two aggregated measures are included:

- **Average:** the mean absolute deviation across all listed quantities.
- **Average (output):** the mean absolute deviation excluding quantities not directly related to simulation output ($T[1]-T[2]$ and Watt), thus focusing only on CFD-predicted parameters.

This table provides a concise quantitative overview of simulation accuracy and the impact of altitude correction on the agreement with experimental data.

	T[1]	T[2]	T[5]	T[6]	T[7]	T[8]	ΔP	Watt	Average	Average (output)
Min	-0.93%	-0.64%	-1.67%	-0.89%	-0.55%	-2.00%	-2.10%	-4.58%	1.67%	1.31%
Avg	—	—	—	—	—	—	—	—	—	—
Max	0.62%	0.89%	1.26%	0.73%	0.61%	1.76%	1.78%	3.12%	1.35%	1.17%
Sim (un-corr)	0.00%	-1.04%	-3.24%	-1.36%	0.17%	-24.37%	11.67%	0.00%	5.23%	6.97%
Sim (corr)	0.00%	1.51%	-0.70%	1.36%	2.64%	-22.68%	10.70%	0.00%	4.95%	6.60%

Table A.1: Experimental vs. simulated values — percentage deviation.

Figures A.19-A.21 show the experimental setup. The temperature and pressure probe locations were replicated as accurately as possible in the simulation. However, due to limited information regarding the exact physical placement, natural deviations between experimental and simulated results are expected.

For example, $T[8]$ represents the maximum temperature measurement in the experimental setup but is recorded at a single point only. At this location, the cold-plate wall thickness is relatively small, and the measurement lies close to the water outlet, which further influences the recorded temperature. Consequently, the experimental setup cannot be considered fully representative of the physical reality, and the results should be interpreted as comparative rather than absolute.

The photographs shown in Figures A.19-A.21 originate from Charne Griessel's final-year project, and are reproduced here with her permission to illustrate the experimental setup.

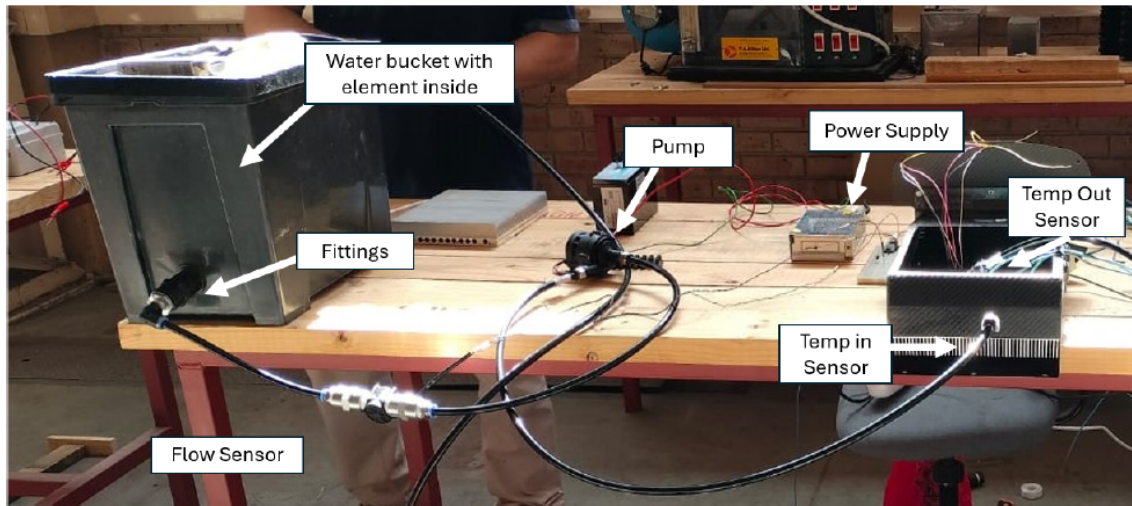


Figure A.19: General Experimental Setup

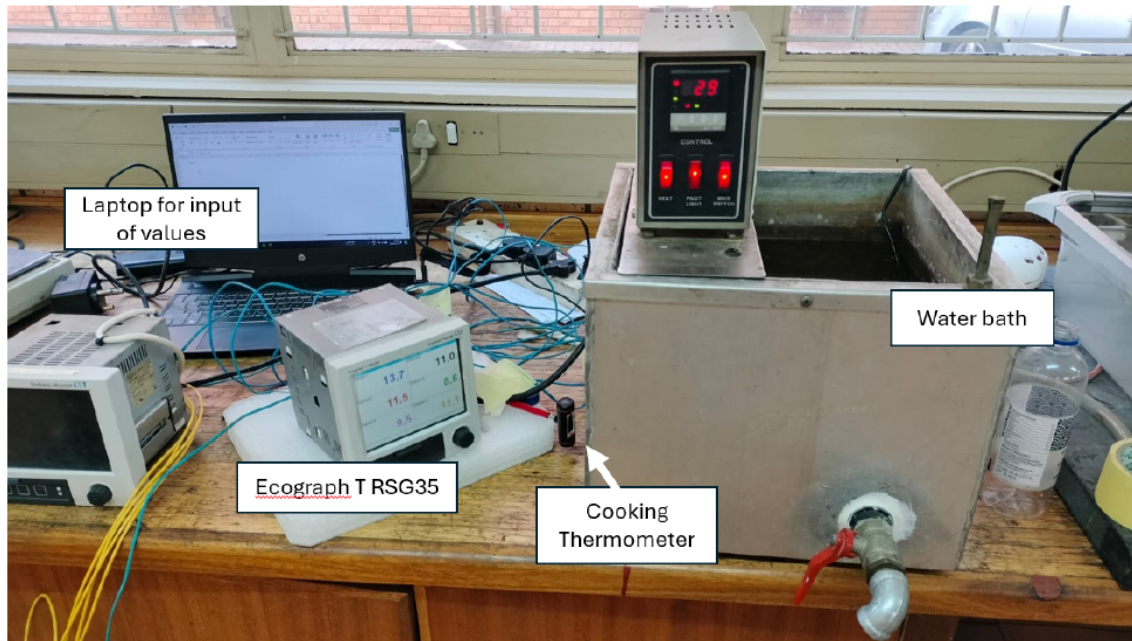


Figure A.20: Water heating of experimental Setup

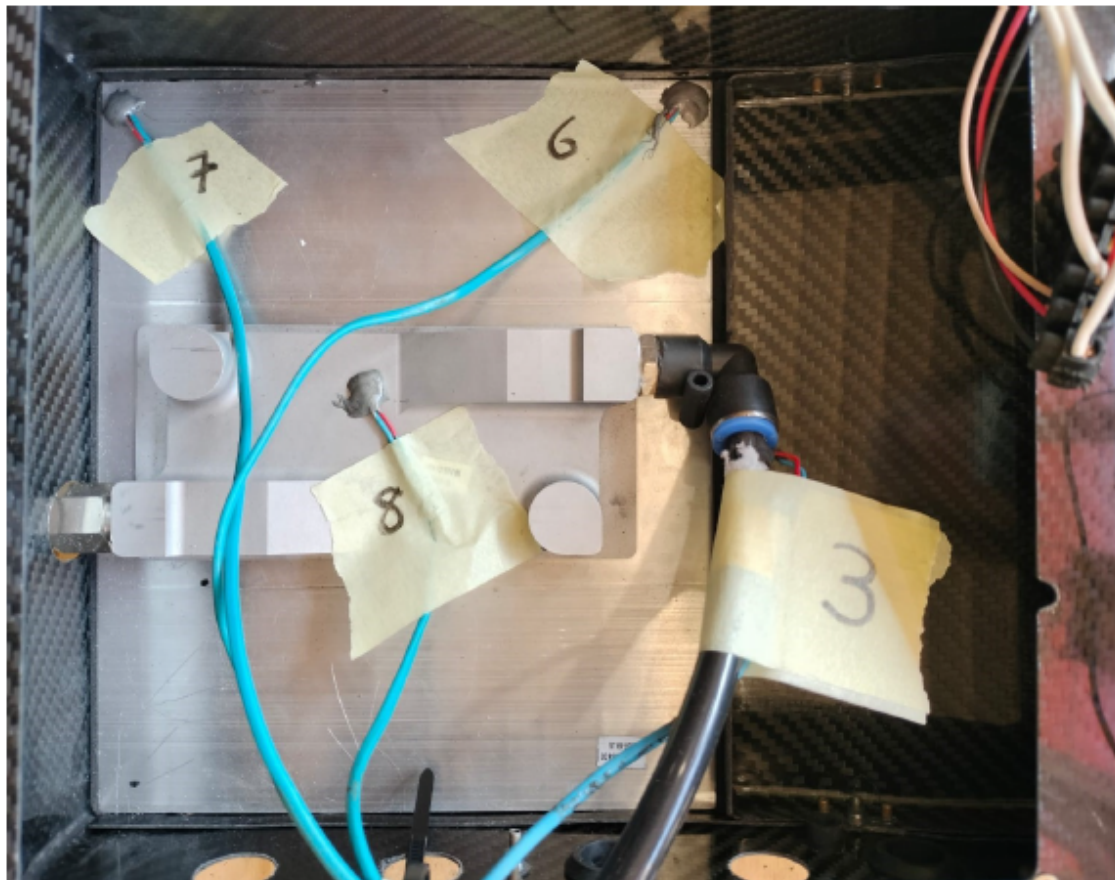


Figure A.21: Temperature probe locations of experimental setup

A.3 Test Flight Data

This appendix presents the available test flight data for both the air-cooled and water-cooled motor controller configurations. Its purpose is to provide traceable raw data and briefly explain how the measurements were obtained and corrected. Detailed analysis is performed in the main body of the thesis.

A.3.1 General Remarks on Data Quality

The data in Tables A.2–A.5 were recorded using the standard onboard RES display. These values are internally averaged from all available temperature sensors for each component group (motor, motor controller, and batteries), with outliers suppressed only if sensor differences exceed a predefined threshold. Consequently, the temperatures should be interpreted with caution, but they are sufficiently representative for operational and comparative purposes.

All temperatures were corrected to a reference outside air temperature (OAT) of 38 °C at mean sea level (MSL), using a standard linear lapse rate of 0.065 °C per 100 m, following EASA guidance for CS-22 certification flight testing.

These measurements illustrate the system's performance under real operational conditions. They are not intended for formal validation or verification at a master's level, and are presented for completeness and context rather than rigorous analysis.

A.3.2 Air-Cooled Motor Controller — Self-Launch Case

Table A.2 summarizes the measurements obtained during a self-launch using the air-cooled motor controller. The data indicate that the controller temperature rises rapidly during the initial phase of operation, reaching approximately 75 °C after roughly 85–90 seconds. Following this peak, a gradual decrease in temperature occurs, with values reducing to around 61.5 °C prior to battery depletion.

A comparison of the corrected temperatures with those recorded from the water-cooled configuration (Tables A.2 and A.5) shows that, under these specific test conditions, the air-cooled system performs on a comparable level.

Furthermore, Tables A.2 and A.3 illustrate the differences between temperatures recorded during the original installation/certification testing done in 2022 and those obtained in the most recent tests (2025). The observed improvements in thermal performance can be attributed to several key factors:

- **Enhanced thermal interface:** High-quality thermal paste was applied between the

heat sink and the heat source, ensuring effective heat transfer and minimal thermal resistance.

- **Optimal mechanical assembly:** The heat sink assembly was torqued according to specifications, achieving a proper press-fit for the heat pipes and further reducing thermal impedance.
- **Improved airflow conditions:** During the climb phase, the RES bay doors were left slightly open, which significantly increased airflow through the heat exchanger and enhanced convective cooling.

These conditions represent a near-best-case scenario for the air-cooled configuration while the simulation represents a near-worst-case scenario. The results demonstrate that, with careful assembly, optimal thermal interface management, and favorable airflow, the air-cooled motor controller can achieve thermal performance comparable to—and, in some aspects, approaching or even exceeding—that of the current water-cooled system.

Temp correction calculation					Motor Temp			Motor Controller			Battery (worst case)			
TP #	Power (kW)	OAT (°C)	Alt (m)	Corr (°C)	Meas	Corr	Limit	Meas	Corr	Limit	Meas	ΔOAT	Corr	Limit
a	0.0	14.3	110	23.0	19	42.0	125	23	46.0	85	21	-6.7	37.3	70
b	41.2	14.4	115	22.9	37	59.9	125	52	74.9	85	23	-6.7	39.2	70
c	39.5	13.1	320	22.8	73	95.8	125	61	83.8	85	29	-6.7	45.1	70
d	25.6	13.0	1000	18.5	93	111.5	125	45	63.5	85	41	-6.7	52.8	70
e	25.8	12.1	1440	16.5	93	109.5	125	45	61.5	85	45	-6.7	54.8	70

Table A.2: Test flight data — Air-cooled motor controller — self-launch (2025)

Temp correction calculation					Motor Temp (°C)			Motor Controller (°C)			Battery (worst case) (°C)			
TP #	Power (kW)	OAT (°C)	Alt (m)	Corr (°C)	Meas	Corr	Limit	Meas	Corr	Limit	Meas	ΔOAT	Corr	Limit
a	0.0	22.0	1375	7.1	15	22.1	125	22	29.1	85	24	-1.5	29.1	70
b	40	22.0	1481	6.4	52	58.4	125	66	72.4	85	29	-1.5	33.9	70
c	25	23.1	1587	4.6	66	70.6	125	60	64.6	85	31	-1.5	34.1	70
d	25	22.8	1703	4.1	75	79.1	125	60	64.1	85	34	-1.5	36.6	70
e	25	22.2	1818	4.0	82	86.0	125	60	64.0	85	37	-1.5	39.5	70
f	25	21.8	1942	3.6	88	91.6	125	61	64.6	85	39	-1.5	41.1	70
g	25	21.1	2069	3.5	92	95.5	125	62	65.5	85	42	-1.5	44.0	70
h	25	21.2	2180	2.6	95	97.6	125	63	65.6	85	45	-1.5	46.1	70
i	25	20.9	2306	2.1	97	99.1	125	63	65.1	85	47	-1.5	47.6	70
j	25	20.7	2440	1.4	99	100.4	125	63	64.4	85	50	-1.5	49.9	70
k	25	20.8	2553	0.6	101	101.6	125	63	63.6	85	53	-1.5	52.1	70
l	25	20.5	2663	0.2	101	101.2	125	62	62.2	85	55	-1.5	53.7	70
m	25	19.5	2754	0.6	100	100.6	125	61	61.6	85	58	-1.5	57.1	70
n	25	18.9	2850	0.6	99	99.6	125	60	60.6	85	61	-1.5	60.1	70
o	25	18.5	2893	0.7	99	99.7	125	59	59.7	85	61.5	-1.5	60.7	70

Table A.3: Test flight data — Air-cooled motor controller — self-launch (2022)

A.3.3 Water-Cooled Motor Controller — Sustainer Case

Table A.4 summarises data from a water-cooled motor controller test performed in sustainer mode at a constant power setting of 13.5kW. As expected, the temperatures increase more gradually compared with the air-cooled self-launch case, due to both the lower power level and the inherent higher thermal mass and heat transport efficiency of the liquid cooling circuit. The corrected motor controller temperatures remain well below the operational limit throughout the run.

Temp correction calculation					Motor Temp (°C)			Motor Controller (°C)			Battery (worst case) (°C)			
TP #	Power (kW)	OAT (°C)	Alt (m)	Corr (°C)	Meas	Corr	Limit	Meas	Corr	Limit	Meas	ΔOAT	Corr	Limit
a	13.5	3.0	1980	22.1	13	35.1	125	17	39.1	85	17	-14.0	25.1	70
b	13.5	2.5	1998	22.5	33	55.5	125	40	62.5	85	23	-14.0	31.5	70
c	13.5	2.3	2032	22.5	40	62.5	125	40	62.5	85	26	-14.0	34.5	70
d	13.5	2.0	2055	22.6	45	67.6	125	40	62.6	85	30	-14.0	38.6	70
e	13.5	2.1	2114	22.2	50	72.2	125	40	62.2	85	34	-14.0	42.2	70
f	13.5	1.7	2136	22.4	53	75.4	125	40	62.4	85	36	-14.0	44.4	70
g	13.5	1.9	2216	21.7	56	77.7	125	40	61.7	85	42	-14.0	49.7	70
h	13.5	1.7	2248	21.7	58	79.7	125	40	61.7	85	45	-14.0	52.7	70

Table A.4: Test flight data — Water-cooled motor controller — Sustainer case

A.3.4 Water-Cooled Motor Controller — Self-Launch Case

Table A.5 presents the corresponding self-launch data for the water-cooled system. As expected, temperatures rise more steeply compared with the sustainer operation, but remain significantly below the limit values when corrected to the reference ambient conditions. The results provide a useful benchmark for comparing self-launch thermal performance across the two cooling concepts.

Temp correction calculation					Motor Temp (°C)			Motor Controller (°C)			Battery (worst case) (°C)			
TP #	Power (kW)	OAT (°C)	Alt (m)	Corr (°C)	Meas	Corr	Limit	Meas	Corr	Limit	Meas	ΔOAT	Corr	Limit
a	0.0	13.4	121	23.8	19	42.8	125	23	46.8	85	25	-11.6	37.2	70
b	40	13.4	124	23.8	33	56.8	125	45	68.8	85	27	-11.6	39.2	70
c	40	13.4	314	22.6	73	95.6	125	53	75.6	85	32	-11.6	43.0	70
d	38.5	13.2	366	22.4	79	101.4	125	54	76.4	85	32	-11.6	42.8	70
e	25.4	13.1	456	21.9	85	106.9	125	49	70.9	85	35	-11.6	45.3	70
f	25.4	13.4	566	20.9	87	107.9	125	49	69.9	85	37	-11.6	46.3	70
g	25.4	13.7	658	20.0	92	112.0	125	50	70.0	85	39	-11.6	47.4	70
h	25.6	14.2	777	18.7	94	112.7	125	50	68.7	85	41	-11.6	48.1	70
i	25.7	13.9	884	18.4	95	113.4	125	51	69.4	85	43	-11.6	49.8	70
j	25.7	13.2	990	18.4	98	116.4	125	50	68.4	85	45	-11.6	51.8	70
k	25.9	12.5	1100	18.4	99	117.4	125	50	68.4	85	49	-11.6	55.8	70
l	25.8	11.8	1210	18.3	100	118.3	125	49	67.3	85	49	-11.6	55.7	70
m	26.1	11.0	1328	18.4	101	119.4	125	49	67.4	85	51	-11.6	57.8	70
n	25.2	10.6	1430	18.1	101	119.1	125	48	66.1	85	54	-11.6	60.5	70
o	26.0	10.3	1514	17.9	102	119.9	125	48	65.9	85	55	-11.6	61.3	70

Table A.5: Test flight data — Water-cooled motor controller — Self-launch case

Appendix B

Datasheets and Drawings

B.1 Heat Pipe Datasheet

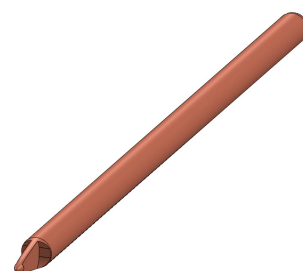
The heat pipe used in this study is the SF-08-200-S. Its key specifications—including thermal conductivity, maximum operating temperature, and physical dimensions—are provided in the datasheet below. These properties were used as input parameters in the CFD simulations presented in Chapter 4.

The full datasheet is included for reference to ensure that all relevant details are available for verification and reproducibility of the simulation results.

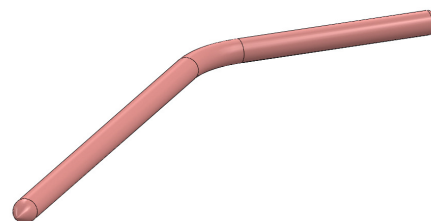
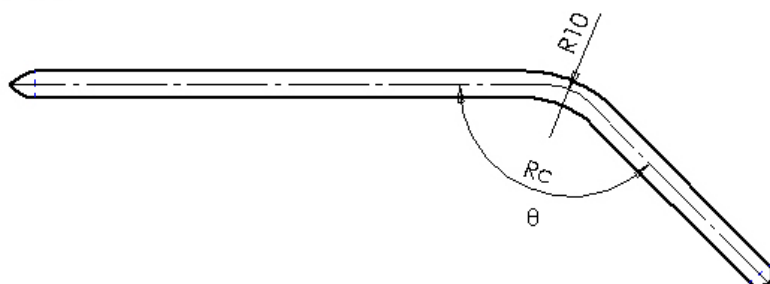
Data sheet

Part number	Diameter [mm]	Length [mm]	Q: input power [W]	Rc [mm]	Head end [mm]	Tail end [mm]
SF-03-100-S	3	100	15	9	5	3
SF-03-150-S	3	150	13	9	5	3
SF-03-200-S	3	200	10	9	5	3
SF-03-250-S	3	250	6	9	5	3
SF-04-100-S	4	100	24	12	7	3
SF-04-150-S	4	150	20	12	7	3
SF-04-200-S	4	200	16	12	7	3
SF-05-150-S	5	150	35	15	10	3
SF-05-250-S	5	250	24	15	10	3
SF-06-100-S	6	100	51	18	11.5	3
SF-06-150-S	6	150	46	18	11.5	3
SF-06-200-S	6	200	36	18	11.5	3
SF-06-300-S	6	300	28	18	11.5	3
SF-08-150-S	8	150	71	24	15	6
SF-08-200-S	8	200	61	24	15	6
SF-08-300-S	8	300	40	24	15	6
SF-08-350-S	8	350	32	24	15	6
SF-10-150-S	10	150	85	30	18	6
SF-10-150-S	10	150	85	30	18	6
SF-10-250-S	10	250	59	30	18	6
SF-12-150-S	12	150	98	36	20	6
SF-12-300-S	12	300	50	36	18	6

Straight



Rounded



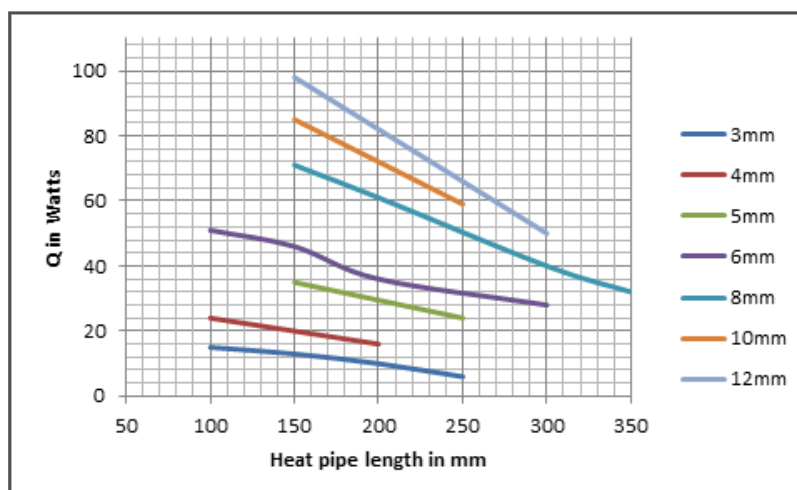
Data sheet

Approvals list

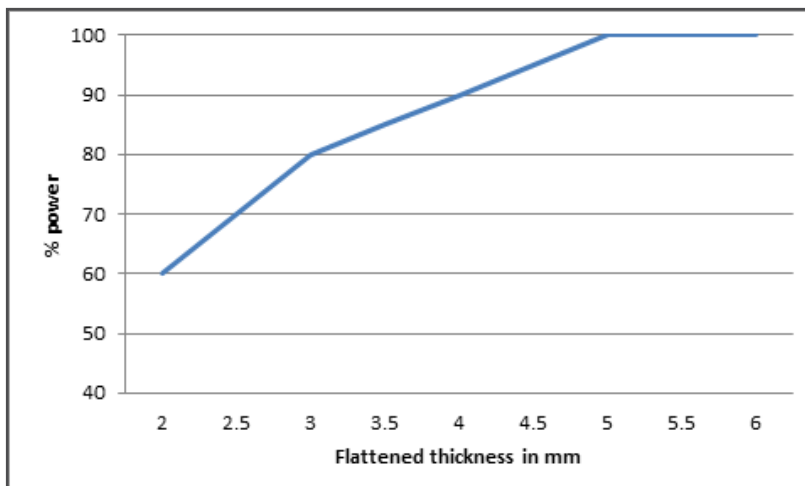
Shape	
Round	3 to 8mm diameter
Bending / shape	Refer to image
Material	Copper C1020
Working fluid	De-ionised water
Wick	Sintered

Fail temp (no cooling)	
Round	250°C leakage
Max heat transfer	0 to 71W
Guarantee	3 years

Heat pipe length (mm) v Q (Watts) of pumping power for each diameter pipe



Compressed thickness v power for 6mm diameter, 150mm length heat pipe



B.2 Electric Motor Datasheet

The propulsion system of the JS3 RES is based on the EMRAX 208 HV AC motor, which is the high-voltage, air-cooled variant of the EMRAX 208 series. This motor was selected for its high power-to-weight ratio, compact design, and proven reliability in aviation applications.

According to the system specification, the peak electrical power supplied by the battery pack to the motor controller is approximately 40 kW. Bench tests conducted by SOLO Aircraft Engines (Figure B.1) demonstrated that the motor is capable of delivering a peak mechanical shaft output power of around 37 kW under maximum load conditions. From these measurements, the overall propulsion system efficiency can be estimated at 92.5%.

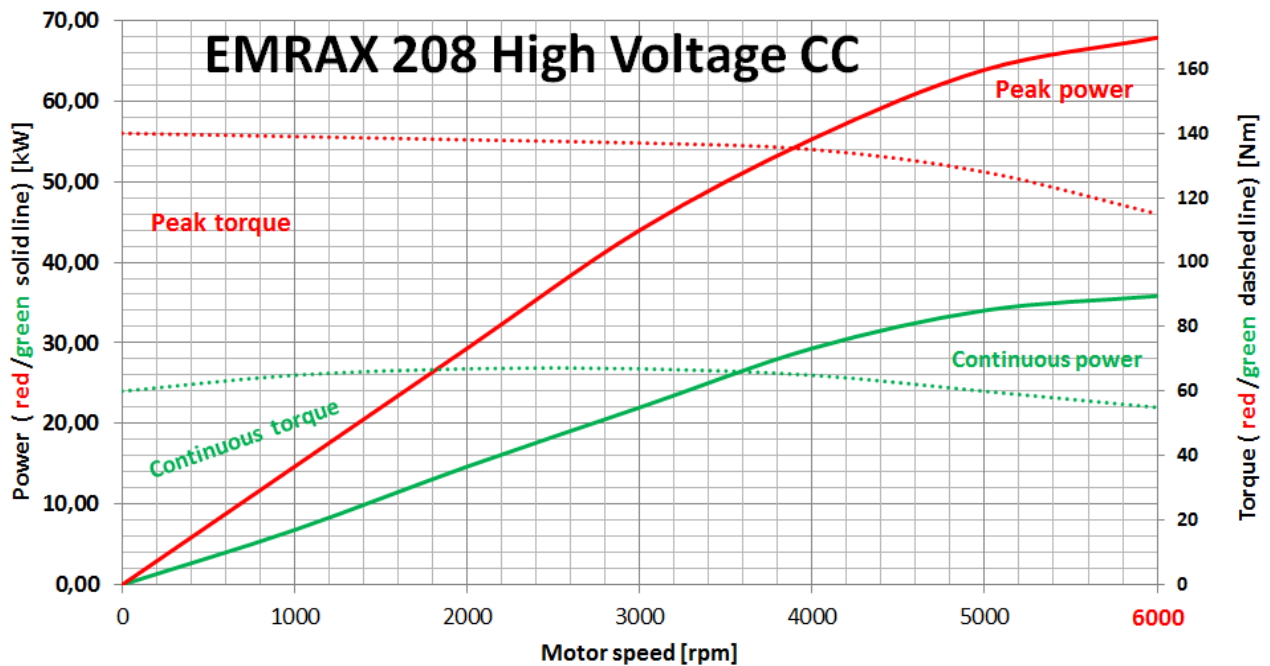
The power losses in the system, approximately 3 kW in total, are attributed to both the motor controller and the motor itself. For thermal management considerations, these losses are assumed to be distributed equally between the two components. Consequently, the motor and the controller each contribute roughly 1.5 kW of thermal load at maximum power, which must be dissipated through their respective cooling systems. This distribution is a reasonable assumption for modeling purposes, given that both components operate under similar electrical and thermal stresses in this configuration.

The complete motor datasheet is included in Figure B.2, while the results from the SOLO Aircraft Engines bench test are provided in Figure B.1 for reference.

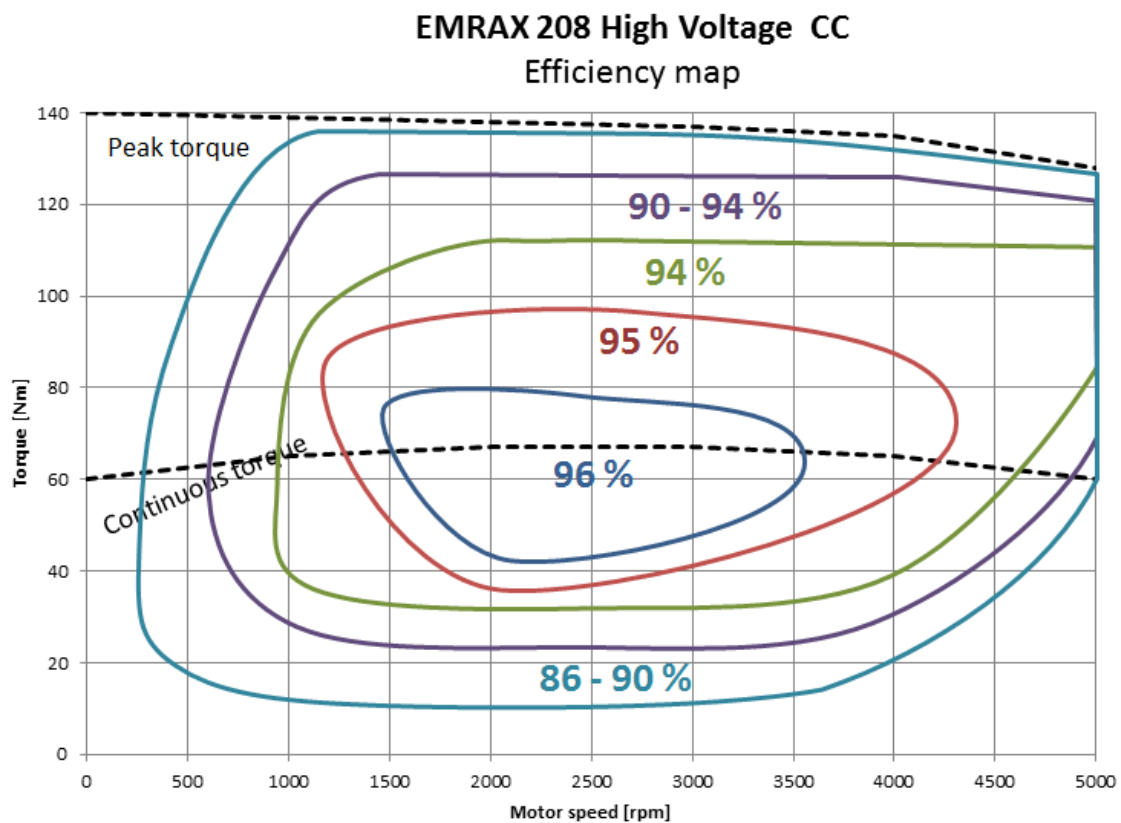
EMRAX 208 Technical Data Table (dynamometer test data)

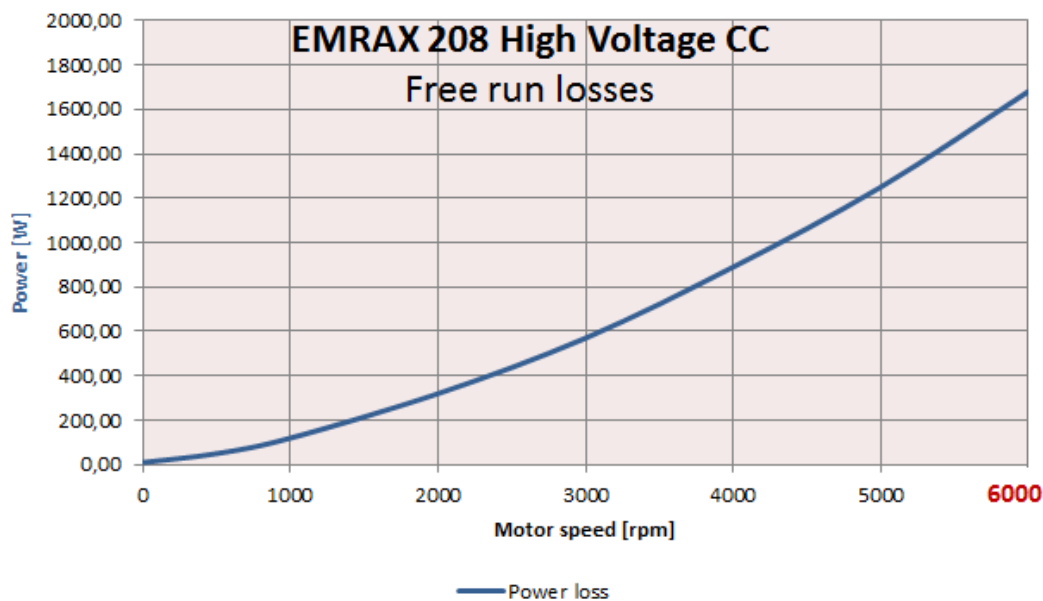
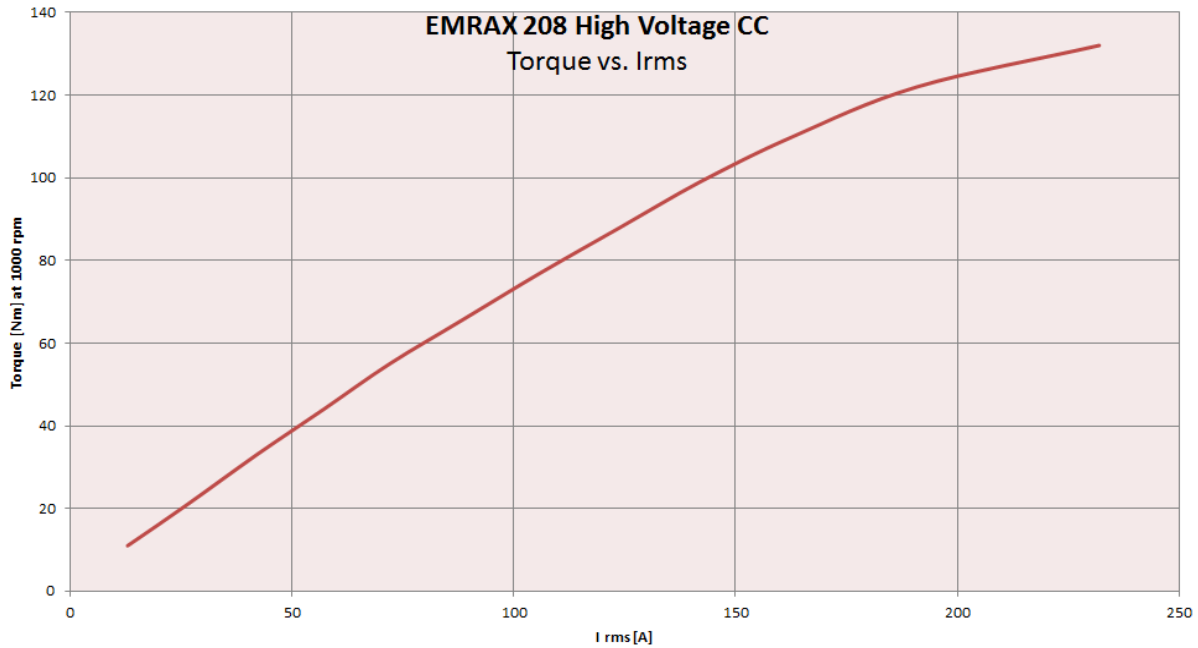
Type	EMRAX 208 High Voltage			EMRAX 208 Medium Voltage			EMRAX 208 Low Voltage		
Technical data	AC	LC	CC	AC	LC	CC	AC	LC	CC
Air cooled = AC Liquid cooled = LC Combined cooled = Air + Liquid cooled = CC									
Ingress protection	IP21	IP65	IP21	IP21	IP65	IP21	IP21	IP65	IP21
Cooling medium specification (Air Flow = AF; Inlet Water/glycol Flow = WF; Ambient Air = AA) If inlet WF temperature and/or AA temperature are lower, then continuous power is higher.	AF=20m/s; AA=25°C	WF=8l/min at 50°C; AA=25°C	WF=8l/min at 50°C; AA=25°C	AF=20m/s; AA=25°C	WF=8l/min at 50°C; AA=25°C	WF=8l/min at 50°C; AA=25°C	AF=20m/s; AA=25°C	WF=8l/min at 50°C; AA=25°C	WF=8l/min at 50°C; AA=25°C
Weight [kg]	9,1	9,4	9,3	9,1	9,4	9,3	9,1	9,4	9,3
Diameter ø / width [mm]	208 / 85								
Maximal battery voltage [Vdc] and full load/no load RPM	470 Vdc (5170/7050 RPM)			320 Vdc (5760/7040 RPM)			125 Vdc (6250/7250 RPM)		
Peak motor power at max RPM (few min at cold start / few seconds at hot start) [kW]	80								
Continuous motor power (at 3000-5000 RPM) depends on the motor RPM [kW]	20 - 32	20 - 32	25 - 40	20 - 32	20 - 32	25 - 40	20 - 32	20 - 32	25 - 40
Maximal rotation speed [RPM]	6000 (7000 peak for a few seconds)								
Maximal motor current (for 2 min if cooled as described in Manual) [Arms]	200			320			800		
Continuous motor current [Arms]	100			160			400		
Maximal peak motor torque [Nm]	150								
Continuous motor torque [Nm]	80								
Torque / motor current [Nm/1Aph rms]	0,83			0,54			0,20		
Maximal temperature of the copper windings in the stator and max. temperature of the magnets [°C]	120								
Motor efficiency [%]	92-98%								
Internal phase resistance at 25 °C [mΩ]	12,0			5,7			0,8		
Input phase wire cross-section [mm²]	10,2			15,2			38		
Wire connection	star								
Induction Ld/Lq [μH]	125/130			52/56			7,2/7,5		
Controller / motor signal	sine wave								
AC voltage between two phases [Vrms/1RPM]	0,0487			0,0319			0,0117		
Specific idle speed (no load RPM) [RPM/1Vdc]	15			22			58		
Specific load speed (depends on the controller settings) [RPM/1Vdc]	11 – 15			18 – 22			50 – 58		
Magnetic field weakening (for higher RPM at the same power and lower torque) [%]	up to 100								
Magnetic flux – axial [Vs]	0,0393			0,0257			0,095		
Temperature sensor in the motor	kty 81/210								
Number of pole pairs	10								
Rotor Inertia (mass dia=160mm, m=4,0kg) [kg*cm²]	256								
Bearings (front:back) - SKF/FAG	6206:6206 (for radial forces) or 6206:7206 (for axial-radial forces; for pull mode; e.g. for air propeller) or 6206:3206 (for axial-radial forces; for pull-push mode; »O« orientation, α=25°); other bearings are possible (exceptionally)								

Graphs valid for EMRAX High Voltage Combined Cooled (CC) motor type:



Note 1: for determining peak or continuous power (kW) you should choose motor speed and then read power from chosen power curve (in the left graph side)
 Note 2: for determining peak or continuous torque (Nm) you should choose motor speed and then read torque from chosen torque curve (in the right graph side)





Graphs of the EMRAX 208 Medium and Low voltage motor type:

Graphs of EMRAX 208 Low Voltage and EMRAX 208 Medium Voltage are similar to graphs of EMRAX 208 High Voltage. The only differences are the DC voltage and motor current. These two parameters can be read from the Technical data table for the EMRAX 208 Low and Medium Voltage motor.

Low Voltage motor needs 4 x higher motor current and 4 x lower DC voltage for the same power/torque and RPM, compared to EMRAX 208 High Voltage motor.

Medium Voltage motor needs 1.52 x higher motor current and 1/3 lower DC voltage for the same power/torque and RPM, compared to EMRAX 208 High Voltage motor.

Graphs of the EMRAX 208 Liquid cooled (LC) and EMRAX 208 Air Cooled (CC):

Continuous power of the liquid cooled or air cooled motor is 20% lower than continuous power of the combined cooled motor. The peak power is the same. Data is presented in the Technical Data Table.



Endprüfung am Flugmotor 80400

Jonker Sailplanes
Tigermoth Street
2520 Potchefstroom
Südafrika, Republik

Typ / Baumuster 80400
Werk-Nr 44
Baujahr 2022
Status Neu

Auftragsnummer H-2022-677
Referenz Kunde Elektrosystem
Datum 15.11.2022
Prüfer G. Gäckle

Offset	139,4	Luftdruck	760,00 Torr	Verbrauch	
		T_Luft	18,00 °C	Laufzeit	15 min
		Korrekturfaktor	1		

Endprüfung am Motorprüfstand

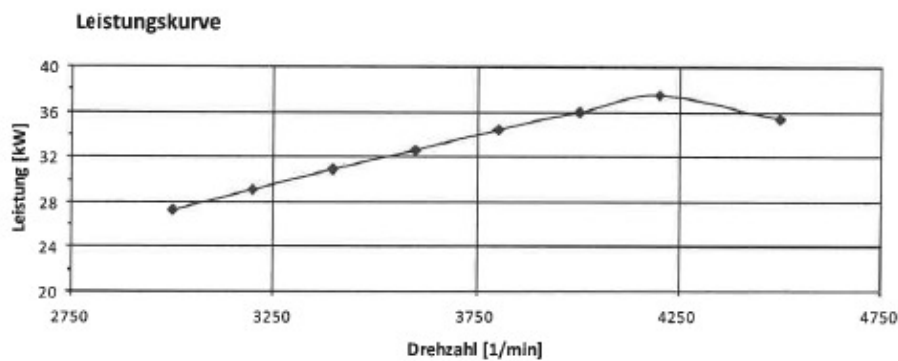


Figure B.1: RES system bench test conducted by SOLO Aircraft Engines.

B.3 Motor Controller Datasheet

The motor controller used in the JS3 RES propulsion system is based on the BAMOCAR-PG-D3, manufactured by UNITEK Industry Electronics GmbH. For the RES application, this controller was adapted and modified by Solo Aircraft Engines into the Solo eControl unit. This customized version retains the robust design and precise motor control capabilities of the BAMOCAR-PG-D3 while being optimized for the specific requirements of aviation applications.

As described in the previous section, the total thermal losses of the propulsion system are estimated at approximately 3 kW under peak load. For modeling purposes, it is assumed that half of this loss (about 1.5 kW) originates from the motor controller. This value was used as the thermal input for the CFD simulations of the cooling system.

The complete controller datasheet is provided in Figure B.3. It should be noted that the attached datasheet corresponds to the original BAMOCAR-PG-D3 from Unitek, as

a detailed datasheet for the Solo eControl is not publicly available. Nevertheless, the electrical and thermal characteristics are directly comparable, since the Solo eControl is derived from and based on the BAMOCAR-PG-D3.

Basic - Information

APPENDIX B. DATASHEETS AND DRAWINGS

145

Environmental conditions	
Protection class	IP 65 (option IP69K)
Standards	EN60204, ISO 16750, EN61800, IEC60146
Overvoltage	400 V and 700 V DC +10 %
Operating temperature range	-30 to +85 °C
Storage, transport	-30 °C to +80 °C EN60721
Installation height	≤ 1000m above sea level 100 %, >1000 m power reduction 2 %/100 m
Cooling	Liquid cooler -30 to max. 65°C, 12 l/min, Pressure max. 6 bar >65 °C Power reduction 2 %/°C
Extended cooling range	Cooling medium 65 to 80 °C Torque reduction (see table)
Mounting position	independent
Pollution	Pollution degree 2 according to EN 61800-5-1
Vibration	10..58 Hz Amplitude 0.075 mm according to IEC 60068-2-358 ..200 Hz 1 g
Shock	15 g for 11 ms
Environmental conditions	Not permissible: Condensation, ice formation, oil mist, salt mist, water
Humidity	Class F Humidity < 85 no internal condensation!

Environmental conditions

The closed housings are resistant to most environmental influences.
Special attention should be paid to the connection plugs.

To prevent internal condensation, pressure equalisation valves with moisture protection are installed.
In case of oil mist or salt spray, the function of the valves may be impaired.
Internal condensation can also occur when the heat sink temperature is colder than the ambient temperature.

Short-term occurrence of impermissible environmental conditions is not critical.

Ice formation can damage the seals.
Operation under water is not permitted.

Humidity

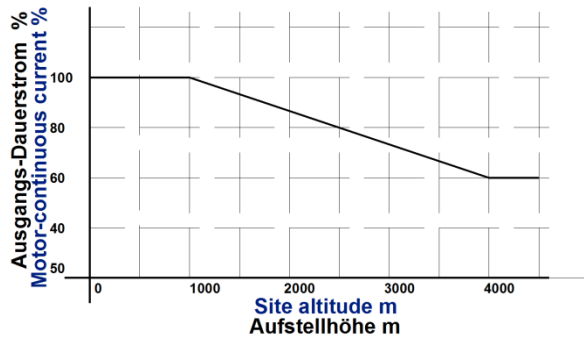
The function of the pressure equalisation valves is overloaded.
Increased humidity in conjunction with salty air can lead to corrosion of the aluminium heat sink.

Basic - Information

APPENDIX B. DATASHEETS AND DRAWINGS

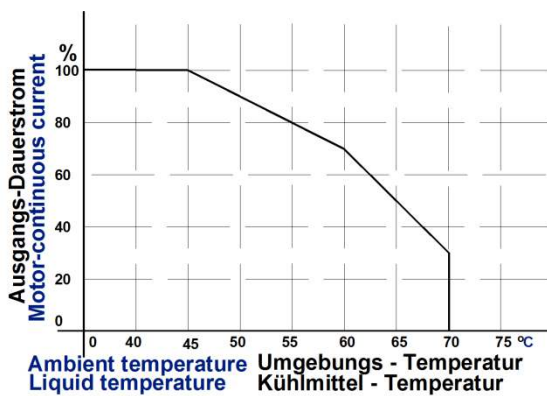
146

Current reduction (torque reduction)



Only with air cooler

Permissible current limit depending on the installation height (not automatic, but must be observed by the user).

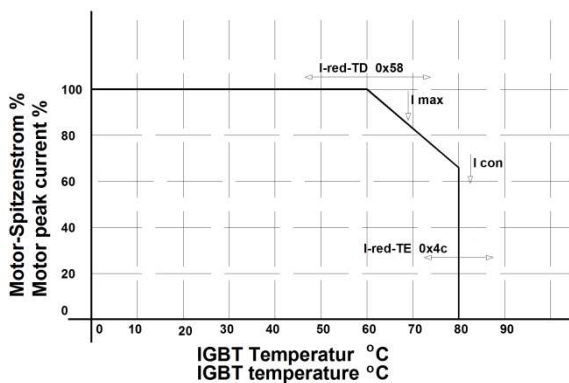


For air cooler

Permissible current limit depending on the ambient temperature.

For liquid coolers

Permissible current limit depending on to the coolant temperature.



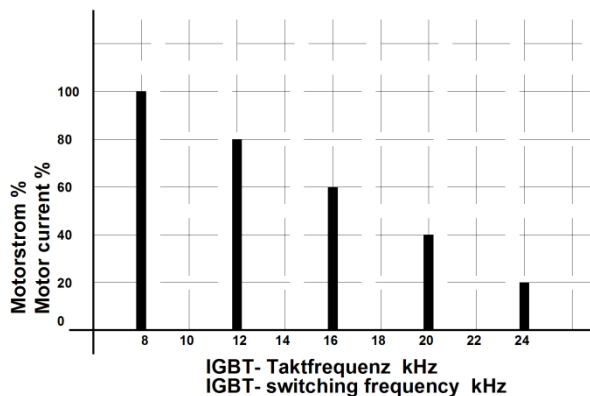
Dependence on IGBT module temperature

Operating temperature of the reduction.
I-red-TD 0x58

Final temperature of the reduction
I-red-TE 0x4C

Protection limits

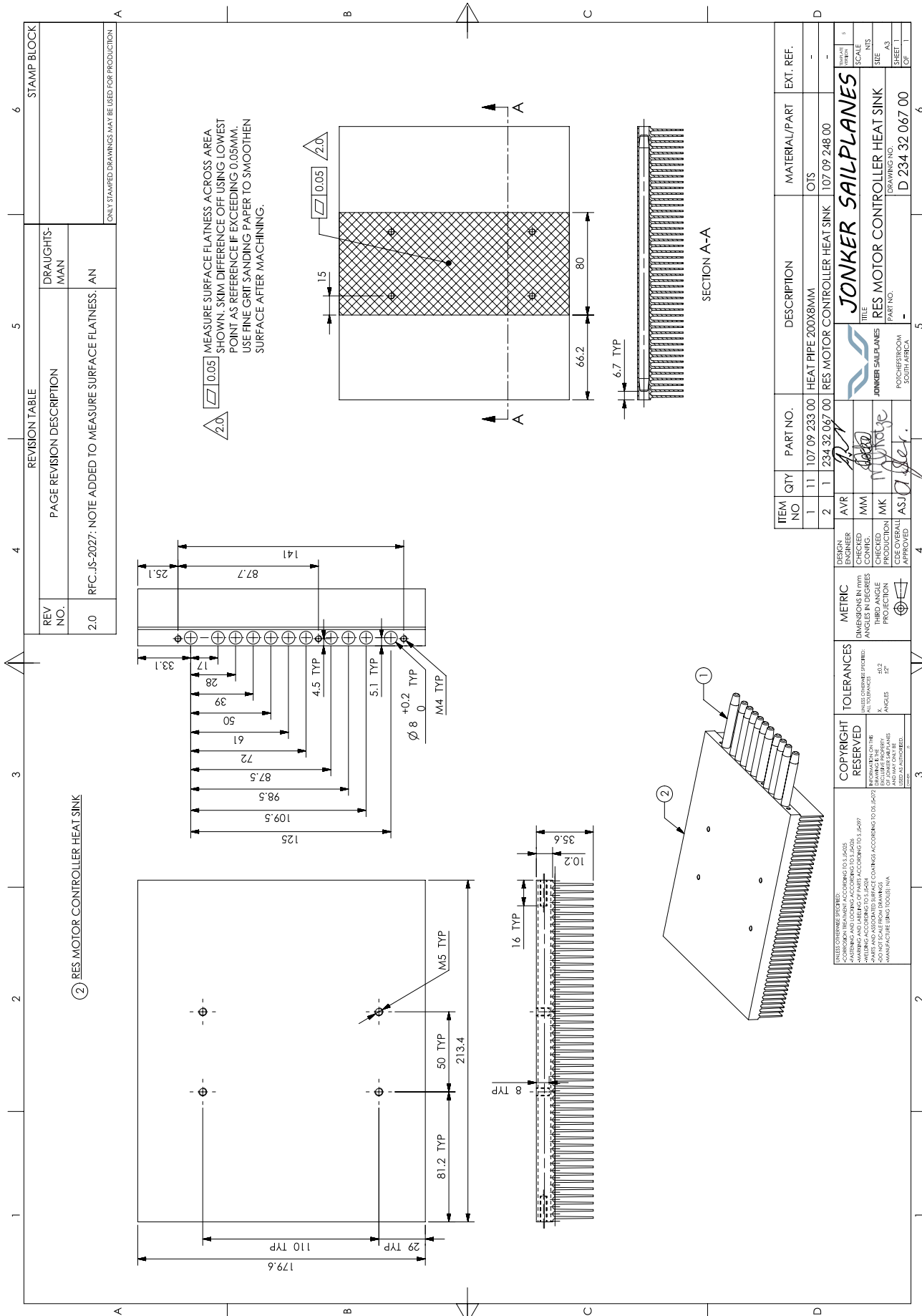
Warning from 0x4A = 24600 Num
Disable at 0x4A = 25200 Num



Automatic current reduction in
Dependence on the clock frequency.

B.4 Heat Sink Drawing

Drawing B.4 presents the technical drawing of the heat sink employed in this study. The document specifies the exact geometry, dimensional tolerances, and structural features of the component, which serve as the foundation for the computational model and subsequent verification against experimental data. This drawing ensures that the simulated heat sink accurately reflects the manufactured part used in the test campaign.



B.5 Motor Controller Box-Top Drawing

Drawing B.5 presents the technical drawing of the motor controller box-top component. This part defines the upper boundary of the flow domain in the simulation setup and therefore directly influences the airflow distribution around the heat sink and associated components. The drawing specifies the geometry, dimensions, and cutout details required for accurate representation in the CFD model. In the configuration shown, the cutout corresponds to the water-cooled motor controller; the only variation in the air-cooled configuration would be the cutout size.

REVISION TABLE		DRAUGHTSMAN	STAMP BLOCK
REV NO.	PAGE REVISION DESCRIPTION		
4.0	RFC.JS-2052: TRIMMING UPDATED.	AN	ONLY STAMPED DRAWINGS MAY BE USED FOR PRODUCTION

COPYRIGHT RESERVED	TOLERANCES UNLESS OTHERWISE SPECIFIED: ANGLES ±0° DIMENSIONS IN MILLIMETERS	METRIC DIMENSIONS IN mm ANGLES IN DEGREES THIRD ANGLE PROJECTION	ENGINEER CHECKED CONFIG.	TVDB	JONKER SAILPLANES	RES MOTOR CONTROLLER BOX TOP	TITLE
NOT TO BE REPRODUCED OR TRANSMITTED IN ANY FORM OR BY ANY MEANS, ELECTRONIC OR MECHANICAL, WITHOUT PERMISSION OF THE COPYRIGHT HOLDER.			MM	MM		PART NO. DRAWING NO. D 234 32 022 02	JONKER SAILPLANES
			MK	Mkaps		A3	SCALE
			CSDE OVERALL APPROVED	ASJ		SHEET 1 OF 1	DRAWN BY

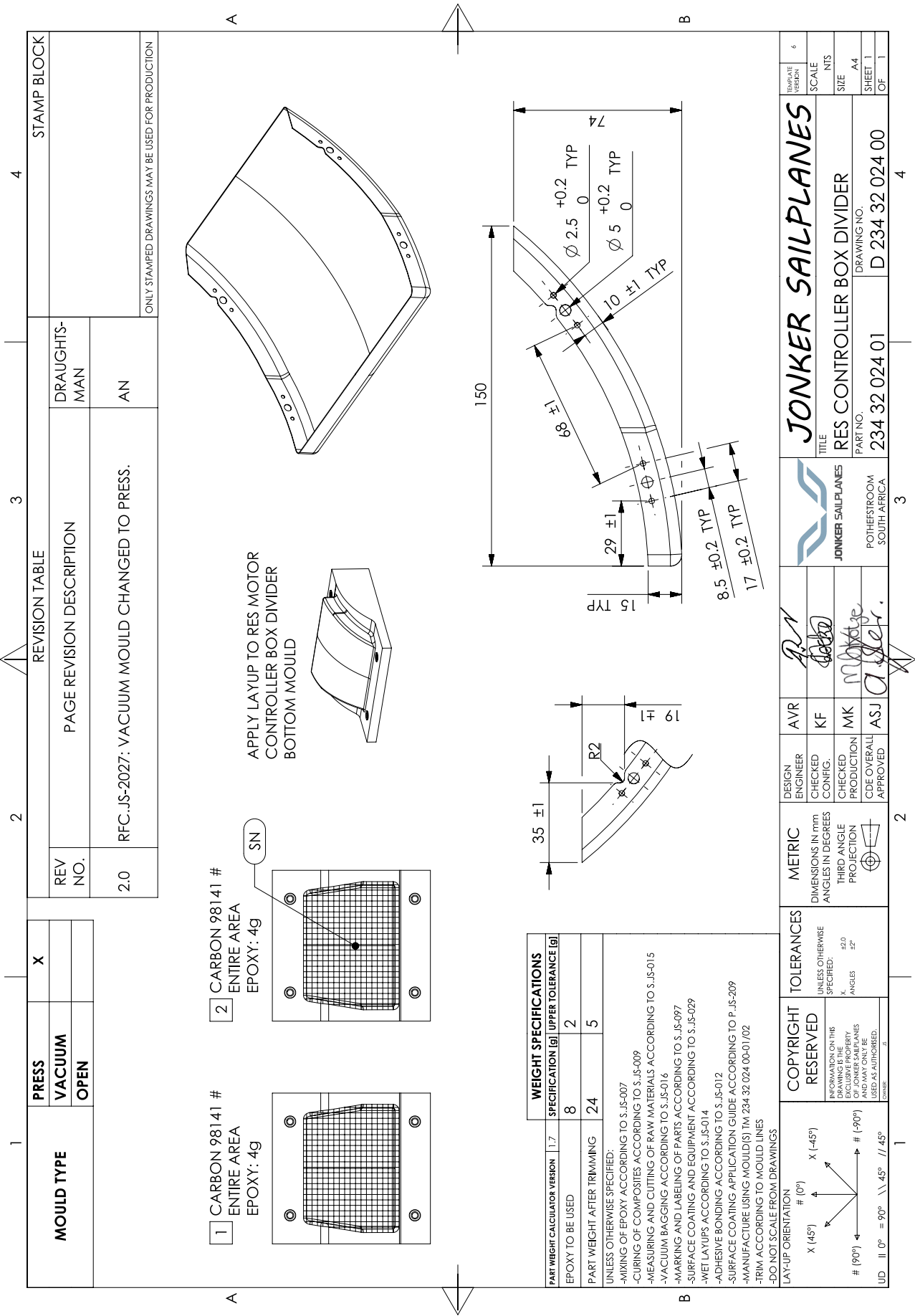
WEIGHT SPECIFICATIONS			
PART WEIGHT CALCULATOR VERSION	1.7	SPECIFICATION [g]	UPPER TOLERANCE [g]
EPOXY TO BE USED		43	9
ONE PART WEIGHT AFTER TRIMMING		132	26

[illegible]

B.6 Motor Controller Box-Bottom drawing

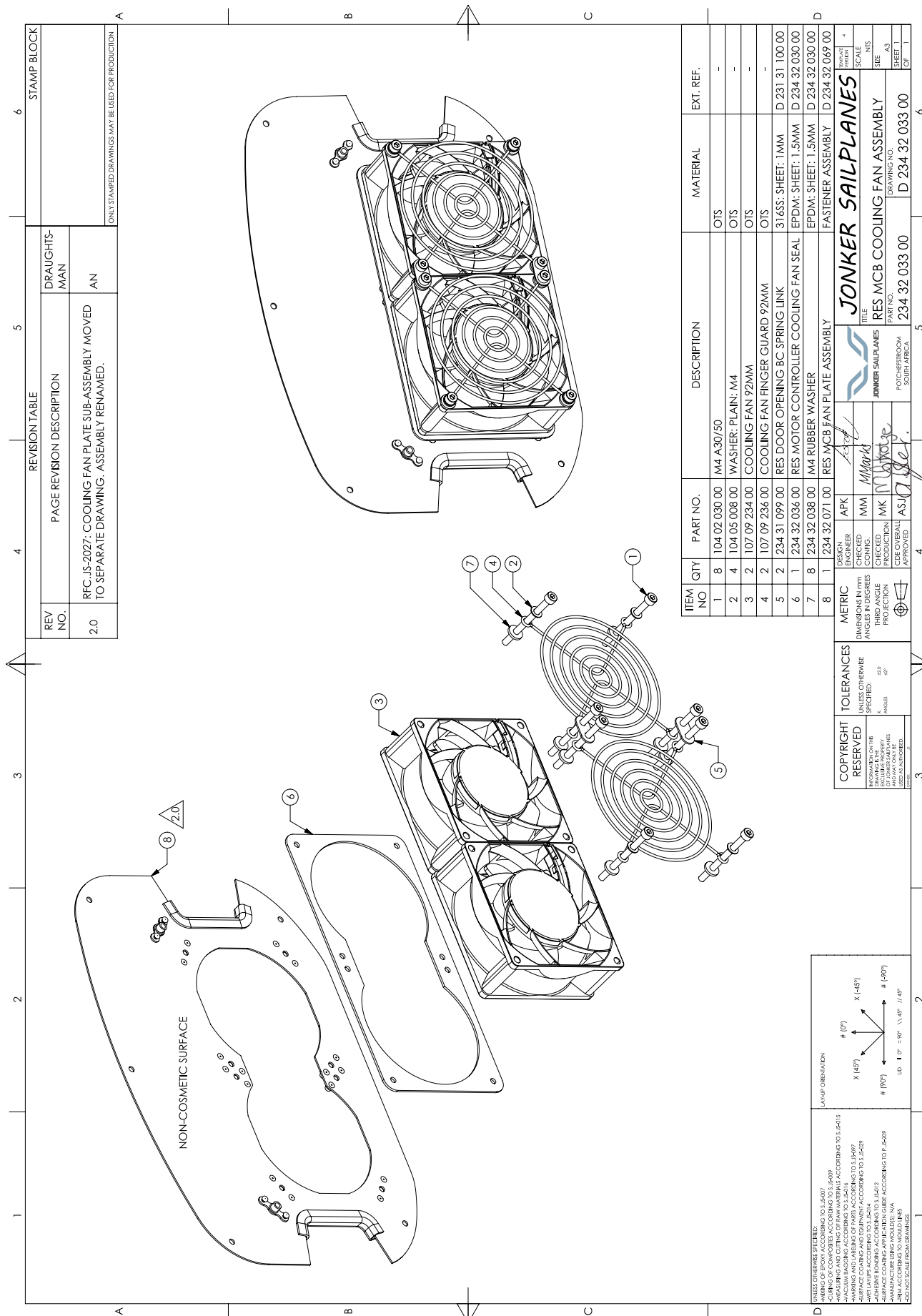
B.7 Motor Controller Box-Divider Drawing

Drawing B.7 shows the technical drawing of the motor controller box-divider component. This part separates the top and bottom sections of the controller box and forms the upper boundary of the flow domain immediately downstream of the heat sink. The drawing provides the geometric dimensions and structural details required for accurate representation in the CFD model, ensuring that the airflow distribution over and beyond the heat sink is properly captured in the simulation.



B.8 Motor Controller Box-Fan Assembly Drawing

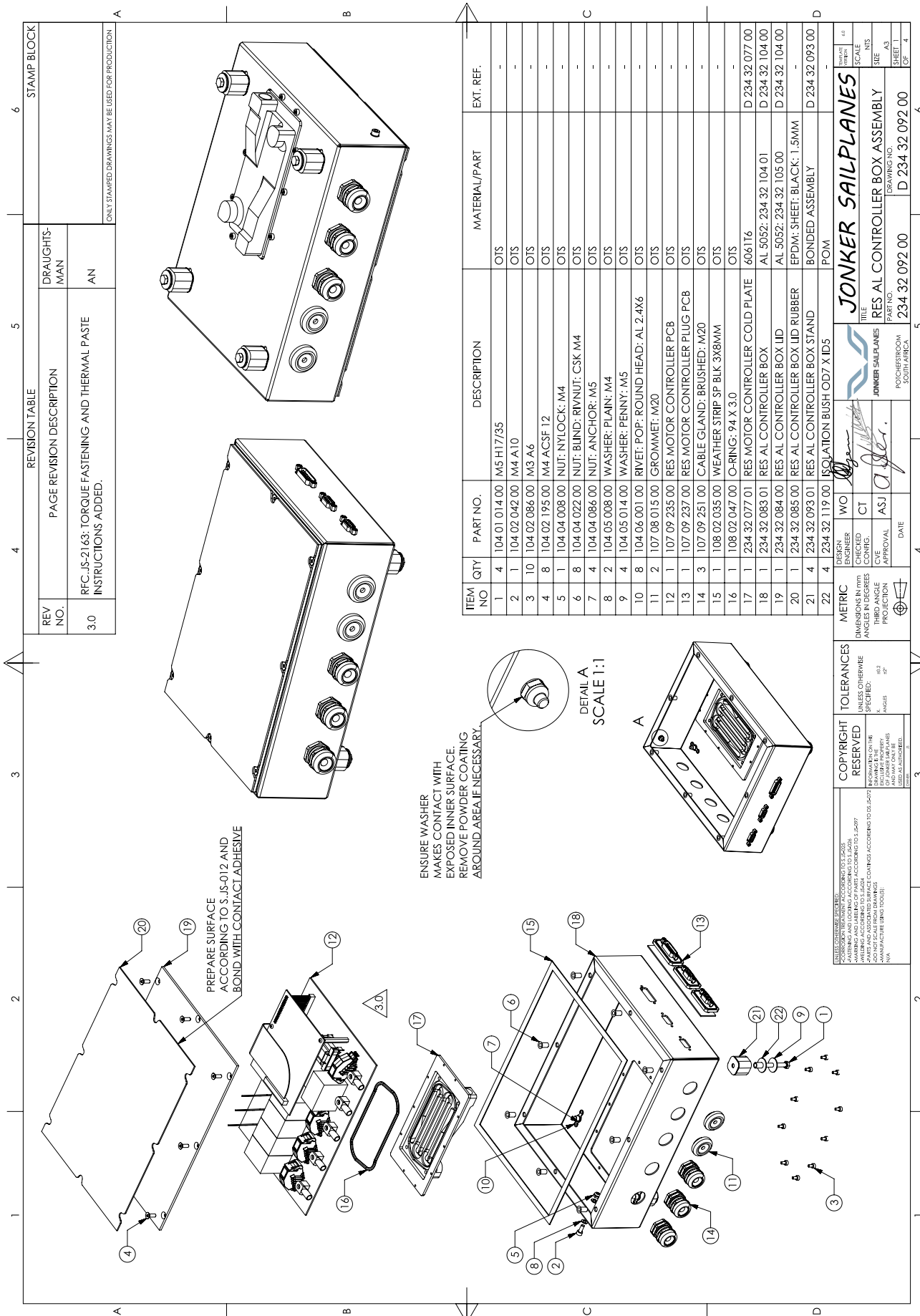
Drawing B.8 shows the technical drawing of the fan assembly used in the air-cooled configuration of the motor controller. This assembly is located at the end of the flow domain and is responsible for inducing airflow through the controller and heat sink, thereby affecting the overall convective heat transfer performance. The drawing specifies the geometry, mounting features, and critical dimensions of the fan assembly, which were considered for accurate representation in the CFD model and for ensuring consistency with the physical hardware.



B.9 Motor Controller Assembly (Air cooled) drawing

B.10 Motor Controller Assembly (Water-Cooled) Drawing

Drawing B.10 presents the technical drawing of the water-cooled motor controller assembly. Although this configuration is not discussed in detail in this thesis, the drawing is included for reference and completeness. It specifies the overall geometry, component placement, and relevant features of the water-cooled assembly, providing a comprehensive overview of the alternative motor controller configuration.



1 2 3 4 5 6

REV NO. 3.0

PAGE REVISION DESCRIPTION

RFC JS-2163: TORQUE FASTENING AND THERMAL PASTE INSTRUCTIONS ADDED.

DRAUGHTSMAN AN

STAMP BLOCK

ONLY STAMPED DRAWINGS MAY BE USED FOR PRODUCTION

THERMAL PASTE (109 03 0143 00) APPLICATION SURFACES

INSTALL O-RING AFTER THERMAL PASTE APPLIED. DO NOT CONTAMINATE O-RING SURFACES THAT SEAL AGAINST MOTOR CONTROLLER PCB.

APPLY LOCTITE 243 TO MOTOR CONTROLLER PCB BOLTS AND BOLT COLD PLATE ONTO PCB WITH TORQUE SPECIFICATION = 8Nm

22 ISOLATION BUSH OD7 X ID5

3.0

3.0

17

16

12

17

11

1

2

3

4

5

6

7

8

9

10

11

12

13

14

15

16

17

18

19

20

21

22

23

24

25

26

27

28

29

30

31

32

33

34

35

36

37

38

39

40

41

42

43

44

45

46

47

48

49

50

51

52

53

54

55

56

57

58

59

60

61

62

63

64

65

66

67

68

69

70

71

72

73

74

75

76

77

78

79

80

81

82

83

84

85

86

87

88

89

90

91

92

93

94

95

96

97

98

99

100

101

102

103

104

105

106

107

108

109

110

111

112

113

114

115

116

117

118

119

120

121

122

123

124

125

126

127

128

129

130

131

132

133

134

135

136

137

138

139

140

141

142

143

144

145

146

147

148

149

150

151

152

153

154

155

156

157

158

159

160

161

162

163

164

165

166

167

168

169

170

171

172

173

174

175

176

177

178

179

180

181

182

183

184

185

186

187

188

189

190

191

192

193

194

195

196

197

198

199

200

201

202

203

204

205

206

207

208

209

210

211

212

213

214

215

216

217

218

219

220

221

222

223

224

225

226

227

228

229

230

231

232

233

234

235

236

237

238

239

240

241

242

243

244

245

246

247

248

249

250

251

252

253

254

255

256

257

258

259

260

261

262

263

264

265

266

267

268

269

270

271

272

273

274

275

276

277

278

279

280

281

282

283

284

285

286

287

288

289

290

291

292

293

294

295

296

297

298

299

300

301

302

303

304

305

306

307

308

309

310

311

312

313

314

315

316

317

318

319

320

321

322

323

324

325

326

327

328

329

330

331

332

333

334

335

336

337

338

339

340

341

342

343

344

345

346

347

348

349

350

351

352

353

354

355

356

357

358

359

360

361

362

363

364

365

366

367

368

369

370

371

372

373

374

375

376

377

378

379

380

381

382

383

384

385

386

387

388

389

390

391

392

393

394

395

396

397

398

399

400

401

402

403

404

405

406

407

408

409

410

411

412

413

414

415

416

417

418

419

420

421

422

423

424

425

426

427

428

429

430

431

432

433

434

435

436

437

438

439

440

441

442

443

444

445

446

447

448

449

450

451

452

453

454

455

456

457

458

459

460

461

462

463

464

465

466

467

468

469

470

471

472

473

474

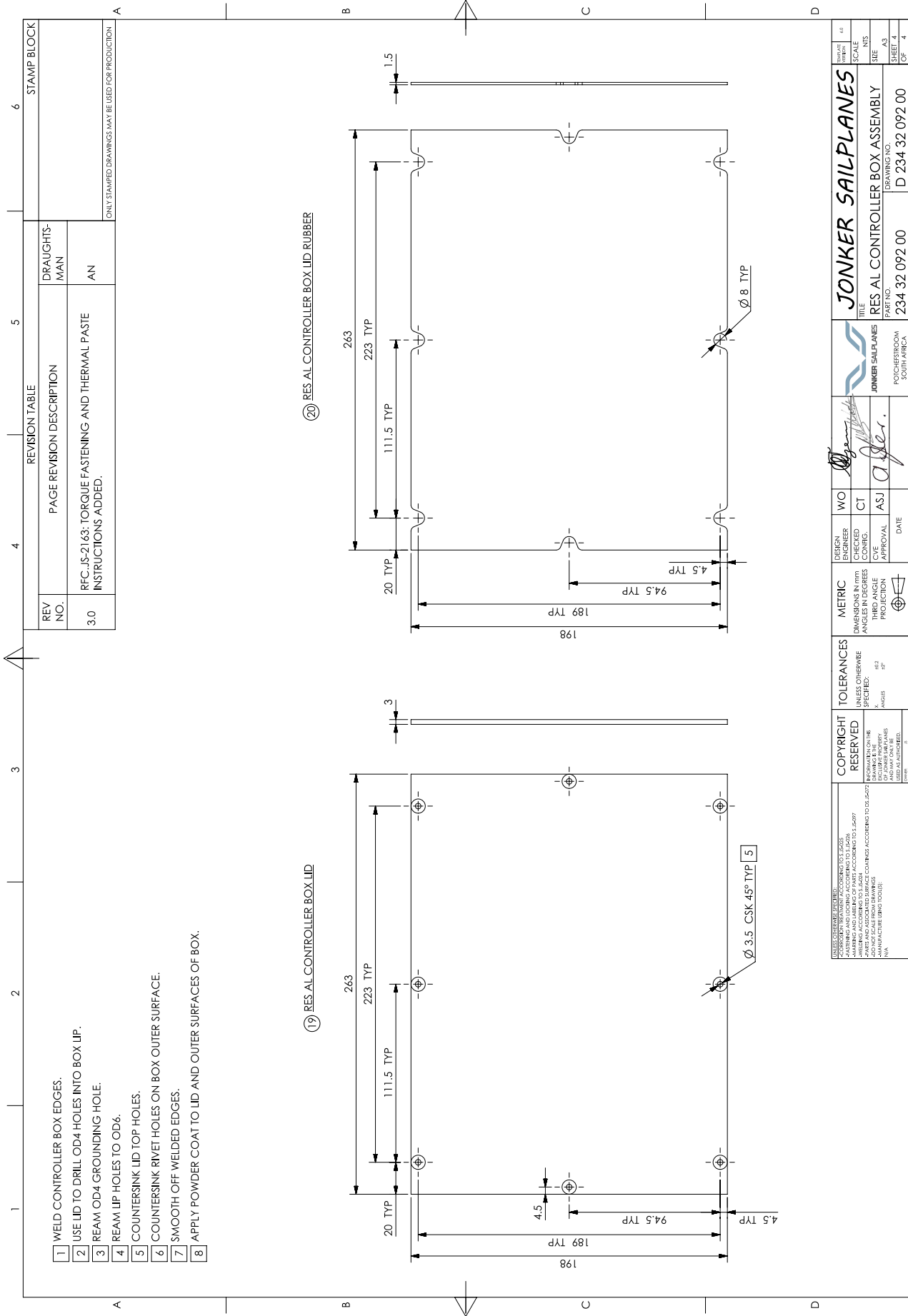
475

476

477

478

[illegible]



B.11 Cooling Fans Datasheet

The cooling fans are shared between both the air-cooled and water-cooled motor-controller systems. In the air-cooled variant, two fans are mounted side by side in a horizontal configuration, whereas the water-cooled system employs a single fan positioned in a push arrangement behind the heat exchanger. Although the fans support variable speed via PWM control, in this application, they operate with only two settings:

1. Off, when the motor-controller temperature is below 40 °C.
2. On, when the motor-controller temperature exceeds 40 °C, running at maximum RPM, which depends on the battery voltage and supply/demand from the DC-DC converter and other electrical loads, ranging between 12.7 and 14.2 V.



Alphacool ES 120mm 1.000-4000rpm Fan V1.1 (120x120x25mm) - Two Ball Bearing - PWM

Alphacool Artikelnummer: 24860



Kurzinformation

Die Alphacool ES Lüfter wurden speziell für den Einsatz in Servern und Workstations entwickelt und bieten, anders als üblicher- weise, keinen 2 oder 3-Pin DC Stromanschluss, sondern eine 4-Pin PWM-Steuerung.

- Über PWM von 1000 bis 4000 rpm regelbar
- Doppelt Kugelgelagertes Rotorlager
- Sehr hoher Luftdurchsatz

Lieferumfang

1 x ES 120 mm Lüfter

Technische Daten

APPENDIX B. DATASHEETS AND DRAWINGS

162

L x B x H	120 x 120 x 25 mm
Drehzahl	1000 - 4000rpm
Lüfterlager	Doppeltes Kugellager
Anschluss	4-Pin PWM
Kabellänge	30 cm
Betriebsspannung	7 - 14V
Startspannung Min.	7V
Stromaufnahme	0,53 - 0,67A (6,36 - 8,04W)
Luftdurchsatz	203,88 m³/h
Statischer Druck	12,67 mm H2O
MTBF	70.000h

Download Links

Produktbilder	24860_Alphacool_ES_120mm_1.000-4.000rpm_Fan__V1.1_(120x120x25mm)_-Two_Ball_Bearing_-_PWM_pics.zip
---------------	---

Verpackungsmaß pro Einheit

L x B x H	125 x 130 x 30 mm
Gesamtgewicht	220 g

Sonstige Daten

Zertifikate	CE, FC, RoHS
EAN	4250197135764
Zoll Nummer	84145915000
Garantie	3 Jahre

APPENDIX B. DATASHEETS AND DRAWINGS

163

Die Alphacool ES Lüfter wurden speziell für den Einsatz in Servern und Workstations entwickelt und bieten, anders als üblicherweise, keinen 2 oder 3-Pin DC Stromanschluss, sondern eine 4-Pin PWM-Steuerung.

PWM-Steuerung

Statt auf die üblichen 2 Kabel oder den 3-Pin Anschluss zu setzen verwendet Alphacool nun auch bei den Enterprise Lüftern einen modernen 4-Pin PWM-Anschluss, der von immer mehr Servermainboards unterstützt wird. Über den 4-Pin PWM Anschluss erhält der Lüfter immer die volle Spannung und wird durch eine Pulsweitenmodulation gesteuert. Dadurch kann der Lüfter über einen deutlich größeren Drehzahlbereich geregelt werden. Durch die Regelung der Lüfter in Serverracks kann eine erhebliche Menge an Energie eingespart werden. Bei einer 0% PWM Ansteuerung können die Lüfter sogar gänzlich gestoppt werden. Der Lüfter bietet auch einen sanften Anlauf, der die Lebensdauer des Lüfters erhöht und das Verletzungsrisiko vermindert. Denn bei Drehzahlen von mehreren Tausend Umdrehungen pro Minuten sollten keine Körperteile zwischen die Lüfterblätter gelangen.

Doppeltes Kugellager

Anders als klassische Gleitlager besitzen die Alphacool ES Lüfter ein doppeltes Kugellager, für eine erhöhte Lebensdauer unter voller Belastung. Doppel-Kugellager gelten allgemein als die hochwertigsten Lager in Bezug auf Langlebigkeit und Qualität. Dadurch ist auch eine hohe Laufruhe in Bezug auf minimale Vibrationen gewährleistet.

Schutzschaltungen

Alle Enterprise Lüfter von Alphacool besitzen mehrere Schutzschaltungen. Wenn die Lüfter blockiert werden, versuchen sie selbständig, nach wenigen Sekunden, einen Neustart mit reduzierter Drehzahl. Durch den sanften Anlauf, inklusive der Notstartfunktion kann der Lüfter für bis zu 72 Stunden blockiert werden ohne das er Schäden erleidet. Bei einer falschen Verpolung beim Anschließen ist ein Durchbrennen der Elektronik für mindestens 5 Sekunden ausgeschlossen. Sollte der Lüfter kurzzeitig falsch angeschlossen werden, ist das also kein Problem.

Die Alphacool ES Lüfter für Server und Workstations bieten einen extrem hohen Luftdurchsatz und einen sehr hohen statischen Druck. Sie eignen sich daher optimal für den Betrieb auf Radiatoren, oder als besonders starke Gehäuselüftertt erstellt werden.