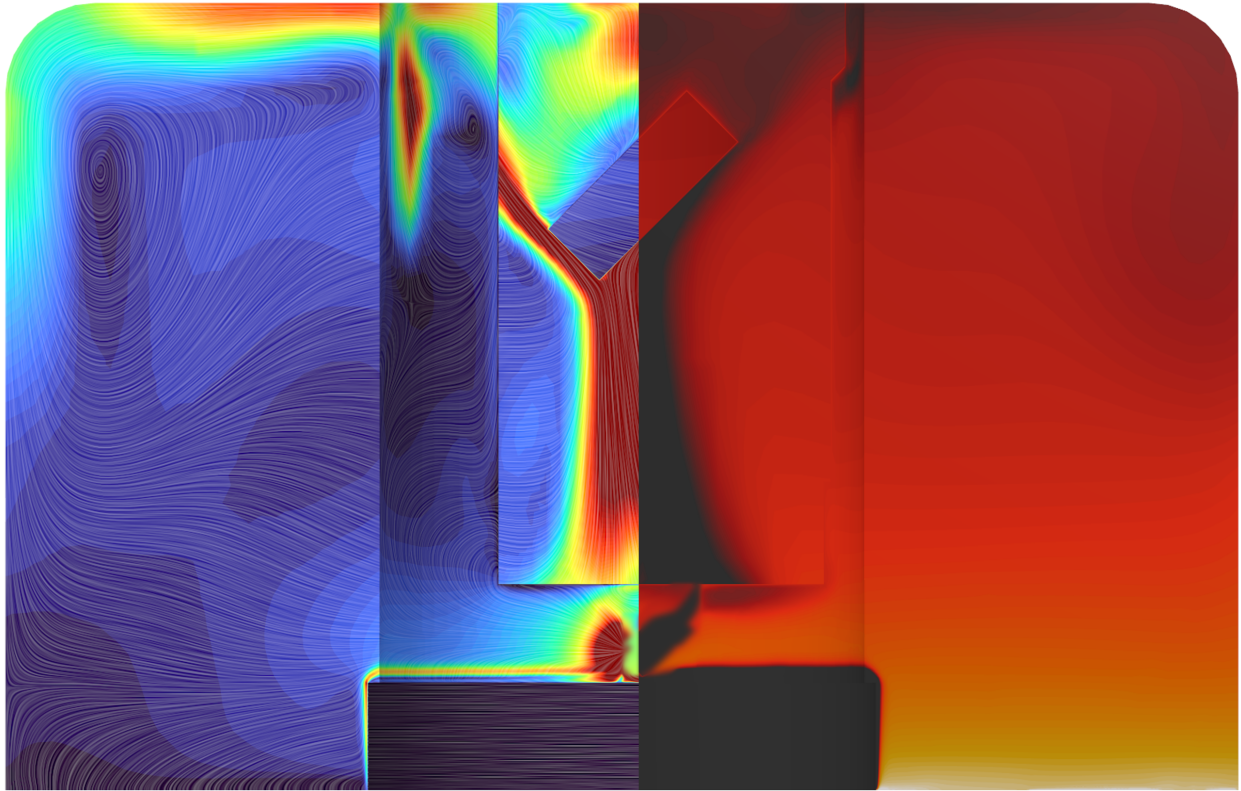




CHALMERS
UNIVERSITY OF TECHNOLOGY



Aero-Thermal Modeling of Laser Metal Deposition with Wire (LMD-w)

Development of a CFD-Based Methodology

Master's thesis in Applied Mechanics

NELLIE KARLSSON

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2026

www.chalmers.se

MASTER'S THESIS IN APPLIED MECHANICS

Aero-Thermal Modeling of Laser Metal Deposition with Wire (LMD-w)

Development of a CFD-Based Methodology

NELLIE KARLSSON



CHALMERS
UNIVERSITY OF TECHNOLOGY

Department of Mechanics and Maritime Sciences
Division of Fluid Mechanics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026

Aero-Thermal Modeling of Laser Metal Deposition with Wire (LMD-w)
Development of a CFD-Based Methodology
NELLIE KARLSSON

© NELLIE KARLSSON, 2026.

Supervisor: Jonas Bredberg and Yasser Alrifai, GKN Aerospace
Examiner: Niklas Andersson, Department of Mechanics and Maritime Sciences

Master's Thesis 2026
Department of Mechanics and Maritime Sciences
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone +46 31 772 1000

Cover: Full-domain visualization from the coupled CHT analysis of the LMD-w process. The left half shows the velocity field, while the right half presents the temperature distribution. See Chapter 6 for further details.

Typeset in L^AT_EX
Gothenburg, Sweden 2026

Aero-Thermal Modeling of Laser Metal Deposition with Wire (LMD-w)
Development of a CFD-Based Methodology
NELLIE KARLSSON
Department of Mechanics and Maritime Sciences
Division of Fluid Mechanics
Chalmers University of Technology

Abstract

In Laser Metal Deposition with Wire (LMD-w), components are built layer by layer by continuously melting and depositing wire material, typically using alloys such as Ti-6Al-4V. The process involves complex interactions between heat input, build geometry, and surrounding gas flow, which determine the aero-thermal conditions in the build zone. These conditions and governing mechanisms are challenging, and the available methods are often limited in terms of calibration and validation. As a result, phenomena such as thermal accumulation and flow behavior largely rely on trial-and-error in process development. To address this limitation, a CFD-based methodology for analyzing the build zone environment is developed. Firstly, a calibration case is used to assess and validate the thermal modeling of the solid, addressing aspects such as laser heat input, phase change, and convective and radiative heat transfer, providing a basis for subsequent simulations. Secondly, the modeling assumptions established in the calibration study are applied to the real geometry to assess the suitability of simulation methods. The results suggest that steady-state modeling captures the asymptotic thermal behavior of the solid and offers a computationally efficient alternative to long transient simulations while still providing relevant temperature data. Finally, the methodology is extended to coupled conjugate heat transfer (CHT) simulations, enabling analysis of fluid-solid interaction and the thermal response of the solid under realistic flow conditions. In addition, a forced cooling strategy is investigated, where argon is injected through tubes directly onto the surface, resulting in a significant improvement in the temperature distribution of the solid. The developed methodology provides a foundation for analyzing build zone conditions and enables future studies on parameter sensitivity, cooling strategies, and process optimization.

Keywords: Additive Manufacturing, LMD-w, CFD, Heat Transfer Calibration, Thermal Modeling, CHT, Cooling Strategy, Argon Cooling.

Preface

This report presents the outcome of my master's thesis project carried out at GKN Aerospace in collaboration with the Department of Mechanics and Maritime Sciences at Chalmers University of Technology during the spring of 2026.

Acknowledgements

I would like to express my sincere gratitude to my supervisors Jonas Bredberg and Yasser Alrifai at GKN Aerospace for their continuous support, valuable guidance, and many insightful discussions throughout this thesis. I am especially grateful for the time they have taken to support and help me throughout the work. Their expertise has been invaluable and has greatly shaped this thesis. I would also like to extend my gratitude to Sakari Tolvanen at GKN Aerospace for generously sharing extensive knowledge about the LMD-w process. The insights into its operation, associated challenges, and the data provided have been essential for this thesis. Furthermore, I would like to thank GKN Aerospace and the Future Concepts team for giving me the opportunity to carry out this thesis. I am very grateful for the welcoming and supportive environment. I am also thankful for all my fellow thesis students, both for interesting discussions and for making this period such a fun and memorable time. A special thank you to all my friends at Chalmers University of Technology for their encouragement, support, and for all the laughter during these five years. I would also like to acknowledge all teachers and staff at Chalmers for providing me with a strong educational foundation that has made this work possible. Finally, I would like to thank my family for their constant support, encouragement, and belief in me throughout my studies.

Nellie Karlsson, Gothenburg, June 2026

List of Acronyms

Below is the list of acronyms that have been used throughout this thesis listed in alphabetical order:

AHC	Apparent Heat Capacity
AM	Additive Manufacturing
CCA	Continuity convergence accelerator
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy number
CHT	Conjugate Heat Transfer
DED	Directed Energy Deposition
DNS	Direct Numerical Simulation
DoE	Design of Experiments
FEM	Finite Element Method
FVM	Finite Volume Method
MEX	Material extrusion
MJT	Material jetting
MRF	Moving Reference Frame
LES	Large Eddy Simulation
LMD	Laser Metal Deposition
LMD-p	Laser Metal Deposition with Powder
LMD-w	Laser Metal Deposition with Wire
PBF	Powder bed fusion
RANS	Reynolds-Averaged Navier-Stokes
RMSE	Root Mean Square Error
SHL	Sheet lamination
SST	Shear Stress Transport
TC	Thermocouples
URF	Under-Relaxation Factor
VPP	Vat photopolymerization

Nomenclature

Below is the nomenclature of parameters that have been used throughout this thesis.

Parameters

h	Enthalpy
k	Thermal conductivity
c_p	Sensible heat capacity
c_p^{lat}	Latent heat capacity
c_p^{app}	Apparent heat capacity
H_m	Latent heat
ρ	Density
μ	Dynamic viscosity
ν	Kinematic viscosity
α	Thermal diffusivity
β	Thermal expansion coefficient
ε	Emissivity
R	Specific gas constant
T	Temperature
T_{surf}	Surface temperature
T_{∞}	Ambient temperature
T_{solidus}	Solidification temperature
T_{liquidus}	Liquidification temperature
T_m	Melting temperature
T_{film}	Film temperature

$\mathbf{u}$	Fluid velocity vector
U	Characteristic velocity
u_τ	Friction velocity
$\mathbf{v}_s$	Convective velocity in solids
p	Pressure
e	Total energy per unit mass
$\mathbf{g}$	Gravitational acceleration
$\mathbf{f}$	Body forces
σ	Stress tensor
τ_w	Wall shear stress
S_e	Energy source per unit volume
$\dot{Q}$	Heat transfer rate
q''	Heat flux
h_{conv}	Convective heat transfer coefficient
h_{screw}	Contact conductance
E	Emissive power
G	Irradiative power
σ	Stefan-Boltzmann constant
Re	Reynolds number
Ra	Rayleigh number
Nu	Nusselt number
Pe	Peclet number
Fo	Fourier number
y^+	Dimensionless wall distance
t	Time
A	Area
A_s	Surface area
P	Perimeter
L	Characteristic length
d_{wire}	Wire diameter

P_{laser}	Laser power
η_{laser}	Laser absorption
$v_{\text{transverse}}$	Printing velocity
v_{wirefeed}	Wire feeding velocity
$\dot{m}_{\text{wire}}$	Wire feed rate
P_{wire}	Wire heating power
η_{wire}	Wire heating efficiency

Contents

List of Acronyms	ix
Nomenclature	xi
List of Figures	xix
List of Tables	xxiii
1 Introduction	1
1.1 Background	1
1.1.1 Additive Manufacturing	1
1.2 Objectives	2
1.3 Limitations and Assumptions	3
1.3.1 Neglecting Melt Pool Fluid Dynamics	3
1.3.2 Constant Geometry Assumption	4
2 Theory	5
2.1 Additive Manufacturing	5
2.1.1 LMD-w Process Parameters	6
2.2 Titanium and Titanium Alloys	8
2.2.1 Microstructure	9
2.2.2 Material Reactivity and Shielding Gas	10
2.3 Melting and Solidification	10
2.3.1 Enthalpy Method	11
2.3.2 Apparent Heat Capacity Method	12
2.4 Governing Equations	13
2.4.1 Continuity Equation	14
2.4.2 Momentum Equation	14
2.4.3 Energy Equation	14
2.5 Heat Transfer	15
2.5.1 Conduction	15
2.5.2 Convection	16
2.5.3 Radiation	17
2.5.4 Dimensionless Numbers	18
2.6 Fluid Flow	19
2.6.1 Flow Regimes	19
2.6.2 Boundary Layers	20

2.6.3	Density Modeling	20
2.7	Computational Fluid Dynamics	21
2.7.1	Finite Volume Method	21
2.7.2	Turbulence Modeling	22
2.7.2.1	Standard k - ε Model	23
2.7.2.2	k - ω Model	23
2.7.2.3	SST k - ω Model	23
2.7.2.4	Wall Treatment	24
2.7.3	Mesh Generation	24
2.7.3.1	Surface Meshes	25
2.7.3.2	Volume Meshes	25
2.7.3.3	Prism Layer Generation	25
2.7.3.4	Conformal and Non-Conformal Meshes	26
2.7.4	Conjugate Heat Transfer	26
2.7.5	Energy Equation in CFD	26
3	Methods	27
3.1	Material Properties	29
3.1.1	Solid Modeling and Phase Change Modeling	29
3.1.2	Fluid Modeling	31
4	Thermal calibration model	33
4.1	Geometry and Mesh	34
4.1.1	Mesh and Time-Step Sensitivity Analysis	36
4.2	Experimental Data	37
4.3	Solver Settings	38
4.4	Boundary Conditions	39
4.4.1	Heat Input Modeling	39
4.4.2	Convective Heat Transfer	41
4.4.3	Radiative Heat Transfer	42
4.4.4	Conductive Heat Transfer	44
4.5	Results and Discussion	45
4.5.1	Mesh and Time-Step Sensitivity Analysis	45
4.5.2	Sensitivity Analysis and Calibration	46
4.5.2.1	Evaluation Criteria	47
4.5.2.2	Radiation	47
4.5.2.3	Screw Conduction	49
4.5.2.4	Convection	51
4.5.3	Assessment of Transient Behavior	53
5	Assessment of the Validity of Steady-State Modeling for an Industrial Drum	59
5.1	Solver Settings	61
5.1.1	Transient Simulation	62
5.1.2	Steady-State Simulation	62
5.2	Boundary Condition Treatment	63
5.2.1	Treatment of Wire Melting and Energy Balance	63

5.2.2	Conduction to Bottom Fixture	64
5.2.3	Radiation	66
5.3	Geometry and Mesh	66
5.3.1	Mesh Generation	67
5.3.2	Mesh Topology Evaluation	70
5.3.3	Mesh Sensitivity Analysis	71
5.4	Results and discussion	71
5.4.1	Transient behavior	71
5.4.2	Mesh Topology and Sensitivity Evaluation	74
5.4.3	Assessment of Steady-State Assumption	78
6	Coupled CFD Analysis and Forced Cooling Evaluation	81
6.1	Geometry and Computational Domain	82
6.1.1	Geometry Pre-Processing	82
6.1.2	Forced Cooling Setup	83
6.2	Mesh Generation	84
6.2.1	Surface Mesh	84
6.2.2	Boundary Layer Treatment	85
6.2.3	Volume Mesh	85
6.3	Physical and Numerical Models	87
6.3.1	Solver Settings	87
6.3.2	Turbulence Modeling	88
6.3.3	Convergence Criteria	89
6.4	Boundary Conditions	89
6.4.1	Momentum Boundary Conditions	89
6.4.2	Thermal Boundary Conditions	90
6.4.3	Forced Cooling Configuration	90
6.4.4	Initial Conditions	90
6.5	Simulation Cases and Experimental Data	90
6.6	Results and Discussion	91
6.6.1	CFL Sensitivity and Transient Flow Behavior	91
6.6.2	Comparison of Solid-Only and Coupled CHT Models	94
6.6.3	Effect of Forced Cooling	94
6.7	Future Work	97
7	Conclusion	99
	Bibliography	101

List of Figures

2.1	Schematic illustration of the LMD-w process. The figure shows the substrate (i.e., the base material or previously deposited layer) on which new material is added. The newly deposited material, also referred to as the bead, is formed as the process progresses. A localized melt pool is generated by the laser beam, into which the wire is continuously fed. The wire is supplied through a wire feeder and melts upon interaction with the laser, contributing to the growth of the geometry.	7
2.2	Illustration of the relationship between temperature, enthalpy and heat capacity for phase change. The left plot shows the enthalpy-temperature relation for isothermal and non-isothermal phase change. The right plot shows the corresponding heat capacity, where the latent heat is represented as an artificial contribution c_p^{lat} (shaded in blue). .	12
2.3	Development of velocity and thermal boundary layers over a flat plate.	20
3.1	Overview of the modeling workflow. The methodology begins with a small-scale thermal model for calibration and sensitivity analysis, followed by application to the industrial drum geometry (annular cylinder) to assess key modeling assumptions. Once confidence in the solid thermal response is established, the approach is extended to a coupled CFD-CHT model with direct fluid-solid interaction. The model can then be used for process understanding and optimization.	28
3.2	Temperature-dependent material data for Ti-6Al-4V, including original experimental data, interpolated data, solidus temperature, and liquidus temperature. The mushy zone between the solidus and liquidus temperatures, denoted as T_s and T_l , is indicated by the shaded gray region. In the case of heat capacity, the latent heat contribution is included through a modified heat capacity curve, illustrated by the blue shaded region.	30
3.3	Temperature-dependent material data for Ti-6Al-4V, including experimental data, solidus temperature, and liquidus temperature. The mushy zone between the solidus and liquidus temperatures is indicated by the shaded gray region.	31
4.1	Geometry of the experimental setup, showing the bottom plate, deposited material and the thermocouples. The clamping screws are not shown.	33

4.2	Geometry of the calibration model and computational domain used in the simulations. Dimensions are omitted due to confidentiality, while the overall geometry and placement of thermocouples TC1-TC6 are shown.	35
4.3	Overview of the mesh with refinements at the deposition surface and close to the screws.	35
4.4	Experimentally determined temperatures at thermocouples TC1-TC6 during three experiments. An average function of three experimental datasets was also calculated.	38
4.5	Experimentally determined temperatures of the argon gas inside the tent and at the fixture, measured by thermocouples from the three experiments. Representative curves were constructed from the three experimental datasets to approximate the overall trends.	41
4.6	Emissivity curve for Ti-6Al-4V. The black markers are experimental data obtained from Yang et al. [38] and the blue curve is the equation fitted to the data.	43
4.7	Mesh sensitivity results for different time step sizes. Three time-step combinations and 5 mesh sizes were investigated. The obtained temperatures at the six thermocouples TC1-TC6 indicate convergence with mesh and timestep refinement.	46
4.8	Temperatures at TC1-TC6 at the end of the test for different emissivity values, illustrating the effect of radiation.	48
4.9	Temperatures at thermocouples TC1-TC6 at the end of the simulation, for different screw contact conductances and contact areas. The contact area is indicated by color, while the contact conductance is shown along the x-axis.	50
4.10	Comparison of the root mean square error (RMSE) and the corresponding standard deviation of the temperature error for each parameter combination. The RMSE is shown in the left subplot, while the standard deviation is presented in the right subplot.	50
4.11	Measured data points at thermocouples TC1-TC6 from parametric sweep when HTC for convection was varied. A polynomial fit, shown in Table 4.10, is also illustrated as well as calculated values of h_{Conv}	52
4.12	Temperature contours at selected time points for increasing number of layers, with time defined as $t = (t_{weld} + t_{pause}) \times 2 \times n_{layers}$	54
4.13	Comparison of simulated and experimental temperature evolution over time at all thermocouple locations for the 17 layer configurations.	56
4.14	Comparison of average heat transfer during one cycle of both heat addition through the heat source and heat losses through convection, radiation and conduction, for the 17 layer configurations.	56
5.1	Geometry for the drum with dimensions given in mm. The rotational direction, laser location and interpass measurement location are also shown.	59

5.2	Normalized experimental data at the interpass location and multiple possible temperature evolutions representing different heat buildup scenarios.	60
5.3	Thermal response in a vertical cross-section of the drum from the interpass location down to the bottom, along with the corresponding temperature gradient.	65
5.4	Mesh of the unstructured mesh generated by the polyhedral function in STAR-CCM+.	68
5.5	Mesh of the unstructured mesh generated by the tetrahedral function in STAR-CCM+.	68
5.6	Mesh of the structured mesh generated by the trimmed-cell function in STAR-CCM+.	69
5.7	Structured mesh topology used for both the STAR-CCM+ directed mesher and the ANSA mapped mesh. The figure also illustrates the cell divisions in the radial, circumferential, and axial directions, which were varied during the mesh sensitivity study.	70
5.8	Mesh of the structured mesh generated by the HexaInterior function in ANSA.	70
5.9	Simulation results for the transient simulation of the drum.	73
5.10	Temperature distributions for the different cases in a vertical cross-section from the interpass location down to the bottom. All temperatures are normalized using the transient reference simulation.	76
5.11	Temperature fields of the drum for different mesh setups, and a transient reference simulation.	77
6.1	Representative schematic of the computational domain, showing the drum, machine, wire feeder, wire, laser, pyrometer, gas extraction outlet, and enclosing tent. The geometry is simplified due to confidentiality.	81
6.2	Drum geometry and selected sectional planes used for flow field analysis.	82
6.3	Representative illustration of three cooling jets directing cold argon gas onto the newly deposited, hot surface downstream of the laser.	83
6.4	Mesh of a selected region of the geometry, illustrating the solid domain, prism layers, and volume mesh. The solid region is represented in pink, while the fluid region is shown in light grey.	87
6.5	Temperature, temperature gradient, velocity, and velocity gradient for all four simulation cases, evaluated at two probe locations positioned 1 cm and 5 cm above the melt pool.	92
6.6	Comparison between natural and forced cooling cases. Temperatures at the two solid probe locations (interpass and downstream of the melt zone) are shown together with the heat transfer coefficient, including both values averaged over the entire drum as well as averaged values for the top and side surfaces.	93
6.7	Temperature and velocity distribution for CFL 5 with and without forced cooling, shown in a combined cylindrical surface view and cross-sectional view through the center of the drum.	95

6.8	Temperature distribution of the drum for CFL 5.	96
6.9	Heat transfer coefficient for CFL 5.	97

List of Tables

2.1	Empirical Nusselt number correlations for natural convection over a horizontal plate.	17
4.1	Process parameters for the thermal-only simple calibration model. . .	34
4.2	Mesh and timestep settings used in STAR-CCM+ for the thermal-only simple calibration model.	36
4.3	Investigated time step combinations for heating and cooling phases. .	37
4.4	Experimentally determined temperatures at thermocouples TC1-TC6 at the end of the test.	37
4.5	Thermophysical properties of argon evaluated at the film temperature, $T_{\text{film}} = 87.5^\circ\text{C}$, used in the empirical correlations for natural convection. . . .	42
4.6	Evaluated screw radii and corresponding contact areas.	44
4.7	Results for the mesh and timestep sensitivity study, normalized with the smallest mesh and smallest time step.	45
4.8	Simulated thermocouple temperatures for different emissivity values, presented relative to experimental measurements at the end of the test.	48
4.9	Simulated thermocouple temperatures for varying heat transfer coefficients for convection, h_{Conv} , presented relative to experimental measurements at the end of the test.	52
4.10	Summary of polynomial fits performed for each thermocouple for varying heat transfer coefficient h_{Conv} . For each thermocouple, a corresponding calculated value of h_{Conv} was determined such that the polynomial representation matches the experimentally measured temperature of TC1-TC6. An average value of h_{Conv} was then determined.	53
4.11	Results from the 17 layer-specific simulations, evaluated at the final simulation time, including temperature data at the thermocouple locations. The table also shows normalized average heat losses, normalized by the average heat input, and the corresponding contributions from each heat loss mechanism.	55
5.1	Experimentally determined interpass temperatures from pyrometer, normalized with the temperature after 39 minutes.	60
5.2	Process parameters for the drum.	61
5.3	Summary of the prescribed thermal boundary conditions applied to the drum in the solid-only simulation.	63

5.4	Interpass temperatures obtained for different bottom temperatures used in the conductive boundary condition are presented together with the corresponding heat transfer through conduction. The bottom temperatures and the interpass temperature is normalized with the reference temperature, and the conduction is normalized with the value corresponding to the lowest tested bottom temperature.	65
5.5	Table containing mesh type results for transient simulations for the polyhedral mesh and ANSA-generated mapped mesh.	72
5.6	Table containing mesh type results categorised by mesh type for steady-state compared to the corresponding transient results. The temperature difference is normalized with the transient reference solution.	75
6.1	Surface mesh element sizes for the different parts.	84
6.2	Prism layer settings for different regions.	86
6.3	Overview of simulated cases with varying CFL numbers and cooling type.	91
6.4	Normalized experimental data for natural and forced cooling configurations at different times.	91
6.5	Average values of the parameters over the selected normalized iteration interval.	92

1

Introduction

1.1 Background

GKN Aerospace is a global supplier of advanced aerospace components, providing critical structures and engine components to major aircraft manufacturers worldwide. The company has a long history of manufacturing high-performance parts for aerospace applications, where strict requirements on quality, reliability, and efficiency must be met.

To address these demands and remain competitive in a rapidly evolving industry, GKN Aerospace continues to build on its strong heritage in advanced manufacturing technologies. In this context, additive manufacturing (AM) has emerged as an increasingly important area of interest due to its potential to enable greater manufacturing flexibility and design freedom.

One of the main drivers for adopting AM is the dependency on external suppliers in conventional manufacturing processes, particularly for large cast structures. Such reliance can lead to increased costs, long lead times, and reduced flexibility when supplier conditions change. By utilizing AM, GKN can move toward more in-house production, thereby reducing this dependency and increasing control over the manufacturing process.

1.1.1 Additive Manufacturing

Additive manufacturing (AM) is a manufacturing technique in which components are built layer by layer through the successive addition of material. In contrast, conventional manufacturing methods are typically subtractive, where a solid workpiece is shaped into the desired geometry by removing material using processes such as milling, turning, or drilling. These methods often generate significant amounts of waste material, increasing production costs and reducing material efficiency [15].

Although minor post-processing using subtractive methods is often required for components produced by AM to achieve the desired surface quality and dimensional accuracy, the overall material waste remains significantly lower [21]. This is because material is primarily added only where needed during the build process, reducing the amount of excess material that must be removed.

Among the various AM techniques, processes suitable for metallic components have gained particular interest. One such technique is Laser Metal Deposition with wire (LMD-w), where material is supplied as a wire that is melted using a laser heat source. This process enables efficient material utilization and is well suited for manufacturing large and complex structures, which has led to increasing interest, particularly within the aerospace industry and at GKN Aerospace [21].

Despite its potential, LMD-w is still an emerging manufacturing method, and several challenges remain before the process can be fully optimized and widely adopted. It is desirable from an efficiency standpoint to manufacture entire components in a continuous manner. However, the high energy input required to sustain melting during deposition introduces a risk of thermal accumulation within the component. If heat is not dissipated sufficiently, temperatures may progressively increase during the build process, potentially leading to process instability and reduced part quality. This challenge can be particularly pronounced for smaller geometries, where the available heat dissipation paths are limited.

Such thermal accumulation is undesirable from both a material and a process perspective. There is an optimal temperature range for deposition, and deviations from this can affect the melt pool behaviour negatively, layer quality, and overall process stability, resulting in deposited layers that deviates from the nominal geometry specification. Elevated temperatures also lead to thermal expansion, which, upon cooling, may introduce stresses and result in defects such as cracking, as observed in practice. From an operational standpoint, continuously rising temperatures pose significant challenges, as handling and operating a system at several hundred degrees Celsius is not feasible, both in terms of safety and the risk of thermal distortion of the part and equipment.

Together, these factors illustrate that uncontrolled temperature rise during continuous deposition could be harmful both operationally and in terms of final material properties. Understanding and mitigating thermal accumulation is therefore essential, which motivates the relevance and necessity of the present thesis work.

1.2 Objectives

The objective of this thesis is to develop and validate a CFD-based methodology for predicting the aero-thermal environment in the build zone of a Laser Metal Deposition with Wire (LMD-w) process. An important part of this work is to understand and predict the thermal response of the deposited material, with particular focus on the heat transfer mechanisms governing the solid temperature. The work therefore aims to investigate how factors such as surrounding flow conditions and applied cooling strategies influence the temperature evolution in the component during the build process.

Based on this, the project aims to assess the conditions prevailing in the build zone and recommend adjustments to process parameters and machine design, particularly with respect to cooling strategies. Ultimately, the developed methodology

should support process optimization, increase manufacturing robustness, and reduce reliance on empirical trial-and-error approaches.

1.3 Limitations and Assumptions

The main assumptions and limitations of the present study are outlined in the following section.

1.3.1 Neglecting Melt Pool Fluid Dynamics

Melt pool behavior in metal additive manufacturing is governed by several coupled thermo-fluid phenomena acting across multiple length scales. Previous studies have shown that melt pool dynamics are strongly influenced by temperature-dependent surface tension gradients driving Marangoni convection, buoyancy-driven flow caused by density variations, and recoil pressure induced by local vaporization of the material [4, 20]. Under sufficiently high laser energy densities, intense vaporization can lead to keyhole formation, characterized by the development of a deep and narrow vapor cavity within the melt pool. The presence of a keyhole, together with associated vapor-plume interactions, can significantly modify laser energy absorption and alter melt pool morphology [20]. All these mechanisms generate highly transient, three-dimensional flow fields that strongly influence melt pool geometry, thermal gradients, solidification behavior, and defect formation.

Accurately resolving these phenomena requires high-fidelity CFD approaches that solves the Navier-Stokes equations together with phase change, free-surface deformation, evaporation, radiation, and pressure coupling between the melt pool and surrounding gas [4, 20]. Such simulations are computationally demanding due to the required micrometer-scale spatial resolution, sub-microsecond time stepping, and the strong nonlinear coupling between thermal and fluid-dynamic effects. Furthermore, many of the governing parameters, such as absorptivity, recoil-pressure coefficients, and temperature-dependent material properties, often require experimental calibration.

In the present work, the melt pool is modeled solely from a thermal perspective, and all fluid-dynamic behavior within the molten region is neglected. Heat transport within the melt pool is therefore assumed to be purely conductive using an embedded phase-change formulation. This simplification substantially reduces computational cost and model complexity while still enabling investigation of the macroscopic thermal behavior and aero-thermal conditions of the build environment. A consequence of this simplification is that flow-driven heat redistribution within the molten metal is omitted, which may affect the accuracy of predicted local temperature gradients, melt pool dimensions, and solidification behavior. Therefore, the model is not intended for detailed melt pool characterization but rather for analysis of the larger-scale thermal behavior of the process. Since the objective of the study is not to resolve detailed melt pool dynamics, the assumption is considered appropriate for the present work.

1.3.2 Constant Geometry Assumption

In a typical LMD-w process, material addition plays a significant role. The incoming wire alters the geometry, affects the energy balance, and influences melt pool size, shape, and cooling rates. The addition of mass also involves complex interactions such as wire melting dynamics, fluid flow disturbances caused by the incoming material, and continuous change of the free surface and deposition profile. Capturing these effects requires dynamic mesh updates, free-surface tracking, and modeling of wire-melt pool interaction, significantly increasing computational complexity.

In the present work, material addition is not considered. Consequently, the geometry of the computational domain remains constant throughout the simulation, implying that the wire does not interact with the melt pool. Although the machine physically feeds wire into the melt pool during real manufacturing, this effect is intentionally omitted in the model to simplify the thermal analysis.

2

Theory

This chapter presents an overview of the theoretical concepts and principles central to LMD-w, heat transfer, fluid dynamics, and computational fluid dynamics (CFD).

2.1 Additive Manufacturing

Depending on the underlying joining mechanism and feedstock delivery method, AM processes can be classified into several categories. ASTM International has standardized AM technologies into seven main process groups based on shared process characteristics [2].

- **Binder jetting (BJT)**: A process in which a liquid bonding agent is selectively deposited to join powder materials.
- **Directed energy deposition (DED)**: A process in which focused thermal energy is used to fuse materials by melting as they are being deposited.
- **Material extrusion (MEX)**: A process in which material is selectively dispensed through a nozzle or orifice.
- **Material jetting (MJT)**: A process in which droplets of feedstock material are selectively deposited.
- **Powder bed fusion (PBF)**: A process in which thermal energy selectively fuses regions of a powder bed.
- **Sheet lamination (SHL)**: A process in which sheets of material are bonded to form a part.
- **Vat photopolymerization (VPP)**: A process in which liquid photopolymer in a vat is selectively cured by light-activated polymerization.

For the additive manufacturing of metallic materials, two main process groups are predominantly used: Powder Bed Fusion (PBF) and Directed Energy Deposition (DED).

In PBF, a focused energy source, typically a laser or electron beam, selectively melts and fuses powder particles within a powder bed to form the component layer by layer. This process enables the fabrication of highly complex geometries with fine feature

resolution and good surface finish. As a result, PBF is well suited for components requiring high precision, intricate internal structures, and complex design features. However, the process is constrained by relatively low deposition rates and limited build volumes, as the part size is restricted by the dimensions of the build chamber. This limits its applicability for large-scale components. [29]

Directed Energy Deposition (DED), in contrast, uses a focused energy source to create a molten pool on the substrate into which material is continuously fed. The feedstock can be supplied either in powder or wire form, allowing material to be deposited directly where needed [4]. Compared to PBF, DED offers significantly higher deposition rates and enables the manufacturing of larger components, making it particularly attractive for repair, cladding, and large-scale production [29, 21].

A specific type of DED is Laser Directed Energy Deposition (L-DED), commonly referred to as Laser Metal Deposition (LMD), in which a laser is used as the heat source. In this process, metallic feedstock is delivered into a laser-generated melt pool as the laser traverses the surface. The material solidifies upon cooling, forming a metallurgically bonded layer. By repeating this process, components are built row by row, layer by layer [21].

When powder is used as feedstock (LMD-p), the process allows for greater flexibility in material delivery, enabling deposition from multiple directions. This makes LMD-p suitable for producing more geometrically complex features compared to wire-based processes. However, LMD-p is associated with several disadvantages, including lower material efficiency and an increased susceptibility to contamination. This is primarily due to the large surface area of the powder particles and their exposure during handling and reuse [14].

When wire is used as feedstock (LMD-w), the process offers near 100% material utilization [21]. In addition, LMD-w provides cleaner processing conditions, higher deposition rates, and improved process stability. Compared to PBF, LMD-w is not constrained by build chamber size, enabling the production and repair of significantly larger components, which is of particular relevance in aerospace applications. While the use of wire limits the achievable geometric detail compared to powder-based approaches, LMD-w is particularly well suited for the manufacturing of large and relatively simple structures [21]. Figure 2.1 provides an overview of the LMD-w process.

Overall, each process offers distinct advantages depending on the application requirements. PBF is preferred for high-precision and complex geometries, while LMD-p provides greater flexibility within DED processes. However, for large-scale structural components, LMD-w is particularly advantageous due to its high deposition rate, efficient material utilization, and suitability for manufacturing large parts.

2.1.1 LMD-w Process Parameters

The main process parameters in LMD-w are the laser power, the wire feed rate, and the traverse speed. Together, these control the energy input, the deposition rate, and

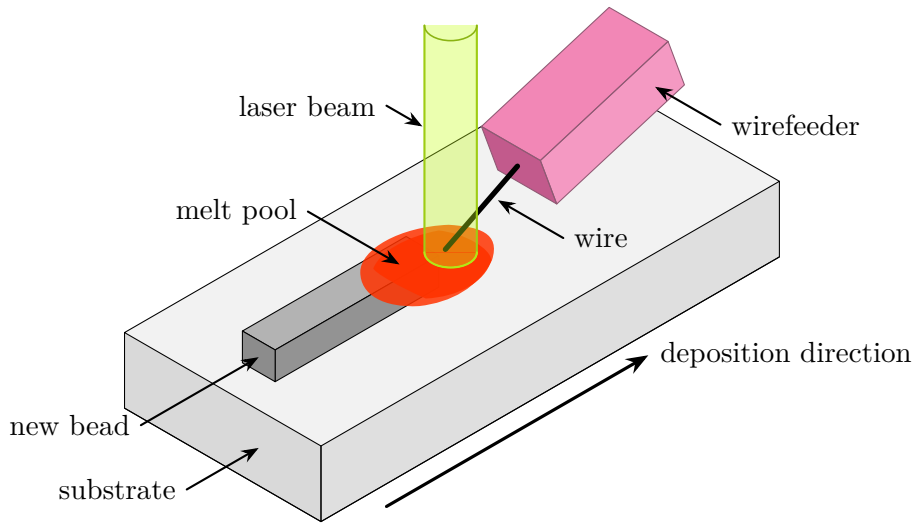


Figure 2.1: Schematic illustration of the LMD-w process. The figure shows the substrate (i.e., the base material or previously deposited layer) on which new material is added. The newly deposited material, also referred to as the bead, is formed as the process progresses. A localized melt pool is generated by the laser beam, into which the wire is continuously fed. The wire is supplied through a wire feeder and melts upon interaction with the laser, contributing to the growth of the geometry.

the resulting bead width and height. For stable deposition, these parameters must be carefully tuned. In practice, the optimal settings depend not only on the material and equipment but also on the specific geometry being built. Different designs, such as thin walls or overhangs, may require different parameter combinations to achieve stable melt pool behavior and the desired final shape. Some of the most important parameters and their roles are listed below.

- **Laser power:** Determines the energy input to the process and, together with the laser spot size and the traverse speed, controls the melt pool geometry and consequently the bead geometry. Different lasers can have different power distributions, such as top-hat or Gaussian profiles, which influence how the energy is delivered to the surface [1].
- **Laser beam size:** The laser power, traverse speed, and the size and shape of the laser beam together control the resulting melt pool geometry. The diameter of the laser spot strongly affects both the melt pool size and the extent of the heat affected zone [31]. Since the melt pool determines the final bead geometry formed during solidification, its dimensions are critical. In practice, the melt pool must at least be large enough to fully melt the fed wire.
- **Laser traverse speed:** Controls the added mass per unit length and the input energy per unit length. Low traverse speeds result in higher local energy input and a more easily controlled deposition process. At higher traverse speeds, however, both path accuracy and the motion system's acceleration capabilities must be carefully considered to maintain stable deposition. [1]

- **Wire feeding:** Controls the added mass per unit time and strongly influences melt pool size and the dimensions of the deposited track. Several parameters can be adjusted, and all of them influence both the process stability and the final deposition [1].
 - **Wire feeding speed:** Process stability requires a balanced ratio between wire feed rate and traverse speed. If the wire feed rate is too low relative to the traverse speed and heat input, droplet formation may occur [31]. Conversely, excessive wire feed can lead to material accumulation beyond the melting capacity of the laser, resulting in unmelted regions.
 - **Wire diameter:** Also affects the amount of material added to the melt pool as more material requires more energy to fully melt. In addition, it interacts with the laser beam size, since the wire can shadow the substrate beneath it.
 - **Wire temperature:** Preheating of the wire reduces the energy required for melting, thereby affecting the process stability window and enabling adjustments in laser power and feed rate to maintain stable deposition.
 - **Feeding direction and angle:** The process is sensitive to how the wire enters the melt pool. Both wire angle and feeding direction (front-, side- or back-feeding) affect process stability by changing shading conditions and the allowable range of wire feed rates [31].
- **Interpass temperature:** The interpass temperature refers to the temperature of previously deposited material between successive deposition passes. In practice, it is typically evaluated slightly upstream of the melt pool, representing the thermal state of the material just before new material is added. This temperature strongly influences melt pool formation and bonding quality. A higher interpass temperature reduces the required input energy, resulting in a larger melt pool and potentially reduced process stability. Conversely, low interpass temperatures lead to rapid heat extraction, which can cause insufficient melting and poor bonding.

2.2 Titanium and Titanium Alloys

Titanium and its alloys are widely used in demanding engineering applications due to their unique combination of low density, high specific strength, excellent corrosion resistance, and good performance at elevated temperatures. Compared to conventional engineering materials such as steels and aluminum alloys, titanium offers a superior strength-to-weight ratio, making it particularly attractive for weight-critical applications.

Commercially pure titanium, cp-Ti ($> 99\%$ Ti), belongs to the α -alloy class and is characterized by excellent corrosion resistance, high ductility, and low density. It also retains good mechanical performance up to approximately $300\text{ }^{\circ}\text{C}$, together with favorable creep and oxidation resistance. However, compared to other tita-

niun alloys, cp-Ti exhibits relatively low strength, which limits its use in highly load-bearing applications. Due to its single-phase α structure, cp-Ti cannot be significantly strengthened through heat treatment, further restricting its achievable mechanical strength [16].

To overcome these limitations, alloying is used to tailor and enhance the material properties. By introducing alloying elements, specific phases can be stabilized and key properties such as strength, ductility, and temperature resistance can be tailored to meet different application requirements. Different classes of titanium alloys provide improved performance depending on the application. For instance, near- α alloys (e.g. Ti-8Al-1Mo-1V) are optimized for high-temperature applications and offer excellent creep resistance. In contrast, metastable β -alloys (e.g. Ti-21S) can achieve very high strength, together with high fracture toughness and good formability. However, these alloys are typically more specialized and may involve trade-offs, such as reduced ductility, higher density, or less favorable performance at elevated temperatures. [16]

Among titanium alloys, α - β alloys stand out as the most widely used class, as they provide a well-balanced combination of properties. The alloy Ti-6Al-4V, commonly referred to as Ti64, consists of a mixture of hexagonal close-packed (α) and body-centred cubic (β) phases, which enables a favorable combination of low weight, high strength, ductility, and toughness. The alloying elements contribute directly to these properties: aluminum stabilizes the α phase and provides solid-solution strengthening, while vanadium stabilizes the more ductile β phase, thereby improving hot workability and supporting processing conditions such as those encountered in LMD-w [38].

As a result, Ti-6Al-4V is widely used in industry and is also employed in the additive manufacturing process considered in this work.

2.2.1 Microstructure

An important factor influencing the performance of Ti-6Al-4V is its thermal history, i.e. the sequence of heating and cooling cycles that occur during layer-by-layer deposition. Each newly deposited layer reheats, melts, and partially remelts the underlying material, resulting in complex thermal cycling and the formation of heat-affected regions. This thermal exposure governs phase transformations, grain morphology, and the development of internal stresses, all of which strongly influence the resulting mechanical properties [39, 3].

Rapid cooling of the newly deposited material promotes the formation of fine microstructures, characterized by small grain sizes and a refined phase distribution. At sufficiently high cooling rates, the material may transform into non-equilibrium structures, which generally result in increased strength and hardness. However, such rapid cooling can also reduce ductility, indicating a trade-off between strength and toughness. In contrast, slower cooling allows the formation of coarser $\alpha + \beta$ microstructures, which typically results in lower strength but improved ductility. [39]

In additive manufacturing, achieving sufficiently high cooling rates is difficult due to repeated thermal cycling and heat accumulation during the build process. Consequently, the concern is typically not excessively rapid cooling, but rather insufficient cooling. The reheating of previously deposited layers effectively reduces the cooling rate and continuously alters the microstructure, leading to the development of coarser grains [3].

2.2.2 Material Reactivity and Shielding Gas

Titanium’s high chemical reactivity introduces additional challenges. An oxygen content of 0.05 wt % has been shown to increase the brittleness of pure titanium [31]. The presence of oxygen can lead to the formation of an oxygen-enriched, hard and brittle “ α -case” layer [31], which can act as a crack initiation site and compromise structural integrity. Oxidation of both the melt pool and surrounding material must therefore be prevented.

LMD systems therefore employ inert shielding gas, typically argon, to isolate the process zone from ambient air. This can be achieved in sealed, argon-filled chambers or under vacuum, which provide robust protection against oxidation but require long purge times and increased cost. Alternatively, local shielding can be applied directly around the melt pool, offering greater flexibility but requiring careful control of the shielding-gas flow to ensure sufficient protection of the heated region. Schulz et al. [31] investigated such a local shielding approach for LMD-w of Ti-6Al-4V and showed that insufficient shielding may lead to oxidation, whereas excessive gas flow can disturb the melt pool and degrade bead quality.

2.3 Melting and Solidification

Melting and solidification are phase-change processes in which a material transitions between solid and liquid states. A key concept in these processes is the latent heat of fusion, which represents the energy required for a material to change phase. During melting, thermal energy is absorbed to break the solid atomic structure, while the temperature remains approximately constant until the phase transition is complete. The presence of latent heat introduces strong nonlinearities in the thermal response and significantly complicates the numerical treatment of the problem.

From a physical perspective, melting and solidification involve the development of a phase boundary separating solid and liquid regions. This boundary evolves over time and its position is not known a priori, but must be determined as part of the solution. In metal additive manufacturing, the evolution of the melt pool is additionally coupled to complex thermo-fluid phenomena such as Marangoni convection, buoyancy-driven flow, recoil pressure, and free-surface deformation. As a result, accurately resolving the melt pool boundary generally requires simultaneous solution of both the thermal and fluid-dynamic behavior of the molten material.

Several numerical approaches have been developed to model moving phase boundaries and melt pool evolution. Common interface-resolving methods include Volume

of Fluid (VOF) [18], level set [27], and front tracking methods [34], where the melt pool interface is tracked during the simulation. Accurately resolving melt pool dynamics with such interface-resolving methods generally requires very fine spatial and temporal resolution together with additional physical models for surface tension, evaporation, recoil pressure, and free-surface motion. Consequently, these approaches are computationally demanding and are often impractical for large-scale thermal simulations or process-level investigations where the primary interest lies in the macroscopic thermal behavior rather than detailed melt pool flow physics.

Previous research has shown that many melting and solidification problems can instead be represented accurately using purely thermal formulations without explicitly resolving melt pool fluid dynamics [10]. Methods such as the enthalpy method and the apparent heat capacity (AHC) method incorporate latent heat directly into the energy equation, allowing phase change to be modeled within a single-phase thermal framework. Although such approaches do not resolve detailed melt pool flow or free-surface evolution, they still provide reliable predictions of temperature fields, cooling rates, and solidification behavior [10, 30]. Simplified thermal models are therefore widely used in additive manufacturing due to their significantly lower computational cost compared to fully resolved multiphase CFD simulations. Several studies have successfully applied conduction-dominated formulations together with temperature-dependent material properties and AHC-based melting models to predict the thermal response in PBF and DED processes [8]. In such approaches, melt pool dimensions are typically inferred from temperature fields and isotherms rather than explicitly resolved through fluid dynamics, which has been shown to provide sufficiently accurate results when the focus is on the overall thermal response of the process.

An additional important aspect is the nature of phase change in alloys. Pure metals undergo melting and solidification at a single, well-defined temperature, resulting in a sharp solid-liquid interface. Alloys, by contrast, melt and solidify over a finite temperature interval bounded by the solidification and liquidification temperatures, T_{solidus} and T_{liquidus} . Within this interval, solid and liquid phases coexist and the local phase fraction changes continuously as temperature evolves. This temperature interval is sometimes referred to as the *mushy zone* [33]. This non-isothermal phase change behavior leads to a gradual transition rather than a well-defined, sharp interface, as illustrated in Figure 2.2. As a consequence, the assumption of a distinct phase boundary becomes less appropriate for alloys such as Ti-6Al-4V, further limiting the applicability of explicit interface-resolving methods. Instead, due to the gradual phase change in Ti-6Al-4V and the complex thermal conditions in additive manufacturing, enthalpy-based or apparent heat capacity methods offer a more practical and efficient modeling approach.

2.3.1 Enthalpy Method

The enthalpy method models phase change by incorporating latent heat directly into the material enthalpy. The energy equation is therefore formulated in terms of enthalpy rather than temperature, allowing phase change to be captured through

the enthalpy-temperature relation. For a one-dimensional conduction problem, the governing equation reads

$$\rho \frac{dh}{dt} = \frac{d}{dx} \left(k \frac{dT}{dx} \right) \quad (2.1)$$

where h denotes the specific enthalpy. The temperature is obtained from the enthalpy using a predefined enthalpy-temperature relation that includes the latent heat contribution.

The method is particularly suitable for problems with strong latent heat effects, where small temperature variations correspond to large enthalpy changes. However, it requires solving for enthalpy and recovering temperature via inversion of the enthalpy-temperature relation, which can increase numerical complexity.

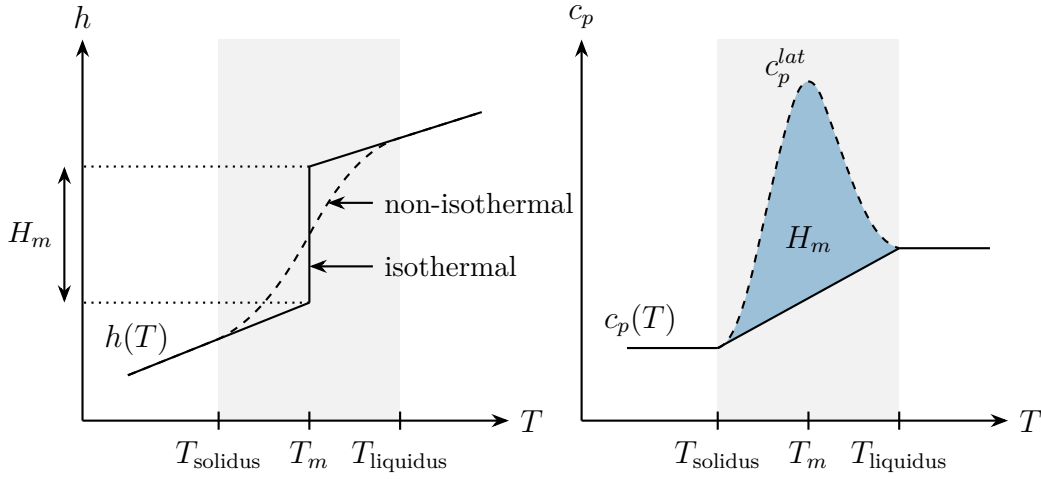


Figure 2.2: Illustration of the relationship between temperature, enthalpy and heat capacity for phase change. The left plot shows the enthalpy-temperature relation for isothermal and non-isothermal phase change. The right plot shows the corresponding heat capacity, where the latent heat is represented as an artificial contribution c_p^{lat} (shaded in blue).

2.3.2 Apparent Heat Capacity Method

The apparent heat capacity (AHC) method provides an alternative formulation of the phase-change problem, in which the governing energy equation is solved directly in terms of temperature. Latent heat effects are incorporated by artificially increasing the heat capacity within a specified temperature interval around the phase change.

Phase change is modeled by augmenting the original (sensible/physical) heat capacity with an additional contribution that accounts for the latent heat of fusion. Outside the phase-change interval, the material retains its original heat capacity, while within the interval $[T_{\text{solidus}}, T_{\text{liquidus}}]$ an extra heat capacity term is introduced. This results in an apparent heat capacity that includes both sensible and latent heat

effects. The apparent (effective) heat capacity is defined as

$$c_p^{\text{app}}(T) = \begin{cases} c_p(T), & T < T_{\text{solidus}}, \\ c_p(T) + c_p^{\text{lat}}(T), & T_{\text{solidus}} \leq T \leq T_{\text{liquidus}}, \\ c_p(T), & T > T_{\text{liquidus}}, \end{cases} \quad (2.2)$$

where $c_p(T)$ is the original temperature-dependent heat capacity, and $c_p^{\text{lat}}(T)$ is an additional term introduced to represent the latent heat during the phase-change interval. The latent heat contribution is constructed such that the total added energy equals the latent heat of fusion, H_m , as

$$\int_{T_{\text{solidus}}}^{T_{\text{liquidus}}} c_p^{\text{lat}}(T) dT = H_m. \quad (2.3)$$

This ensures that the correct amount of energy is absorbed during melting.

Figure 2.2 illustrates the relationship between enthalpy, temperature, and apparent heat capacity for both isothermal and non-isothermal phase change.

For non-isothermal phase-change with a mushy zone, the liquidus isotherm, T_{liquidus} , is often used to indicate the boundary of fully molten material, while isotherms within the interval provide information about the extent of the transition region.

The apparent heat capacity method is mathematically equivalent to an enthalpy formulation but implemented in a temperature-based framework. It is particularly well suited for numerical solvers that use temperature as the primary variable and is therefore widely adopted in industrial FEM and CFD software [8, 28].

2.4 Governing Equations

The governing equations of fluid flow represent mathematical expressions of three conservation laws of physics, namely

- the mass of a fluid is conserved,
- the rate of change of momentum equals the sum of the forces on a fluid particle (Newton's second law),
- the rate of change of energy is equal to the sum of the rate of heat addition to and the rate of work done by a fluid particle on its surroundings (first law of thermodynamics).

Moreover, the fluid is treated as a continuum. This means that at macroscopic scales, the molecular structure and motion of individual molecules can be ignored [35]. Instead, the fluid is described using macroscopic properties such as velocity, pressure, density, and temperature. Within the continuum framework, these quantities are treated as continuous field variables that can be defined at every point in space and time.

2.4.1 Continuity Equation

The continuity equation represents the conservation of mass in a fluid flow. For a general fluid flow, the continuity equation is expressed as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2.4)$$

where ρ is the fluid density and $\mathbf{u}$ represents the velocity of the fluid. Equation (2.4) is the unsteady, three-dimensional mass conservation or continuity equation. The first term describes the rate of change of density with time, while the second term represents the transport of mass due to the fluid motion. Together, these terms ensure that mass is conserved within the flow field, meaning that any change in mass at a given point must be balanced by the flow of mass in or out of that region.

For an incompressible fluid, the density, ρ , is assumed to be constant, meaning that variations in pressure and temperature do not significantly affect the density. This is generally a valid assumption for liquids and for gases at low Mach numbers. Under the incompressible assumption, the continuity equation simplifies to

$$\nabla \cdot \mathbf{u} = 0 \quad (2.5)$$

Nevertheless, temperature-dependent density variations may still be incorporated in the momentum equation through a buoyancy source term, allowing natural convection effects to be captured.

2.4.2 Momentum Equation

Newton's second law states that the rate of change of momentum of a fluid particle is equal to the sum of the forces acting on it. Applying this principle to a fluid continuum leads to the momentum conservation equation, commonly known as the Navier-Stokes equation [35]

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{f} \quad (2.6)$$

where ρ is the fluid density, p is the pressure, μ is the dynamic viscosity, and $\mathbf{f}$ represents body forces acting on the fluid. The most common example is gravity, which can be expressed as $\mathbf{f} = \rho \mathbf{g}$. Other body forces may also be included in this term, such as Coriolis forces, centrifugal forces, or electromagnetic forces.

2.4.3 Energy Equation

The energy equation is derived from the first law of thermodynamics, which states that energy can neither be created nor destroyed, but only transferred or converted between different forms. Applied to a fluid particle, this implies that the rate of change of energy is equal to the rate of heat addition to the particle and the rate of work done on it [5]. In its most general form, the conservation of energy can be written as

$$\frac{\partial(\rho e)}{\partial t} + \nabla \cdot (\rho e \mathbf{u}) = \mathbf{f} \cdot \mathbf{u} + \nabla \cdot (\mathbf{u} \cdot \boldsymbol{\sigma}) - \nabla \cdot \mathbf{q}'' + S_e \quad (2.7)$$

where e is the total energy per unit mass, q'' is the heat flux, and S_e is an energy source per unit volume [32].

For most heat transfer applications, this equation is simplified and expressed in terms of temperature. Neglecting viscous dissipation and mechanical work, the energy equation reduces to

$$\rho c_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) = k \nabla^2 T \quad (2.8)$$

where T is the temperature, c_p the specific heat capacity, and k the thermal conductivity. In solid regions, no fluid motion is present, and the convective term $\mathbf{u} \cdot \nabla T$ therefore vanishes, resulting in purely diffusive heat transport.

2.5 Heat Transfer

Heat transfer refers to the transport of thermal energy driven by temperature differences. It occurs whenever a temperature gradient exists within a medium or between different bodies, and takes place through conduction, convection, and radiation. Heat transfer is commonly quantified using rate equations that describe the transfer of thermal energy per unit time.

2.5.1 Conduction

Heat conduction is the transfer of thermal energy through a material due to microscopic particle interactions. It can be described as the transfer of energy from more energetic to less energetic particles through their interactions, causing heat to flow from regions of higher to lower temperature.

In gases, this mechanism can be understood through molecular kinetic theory in the absence of bulk (macroscopic) motion. Molecules in hotter regions possess higher translational, rotational, and vibrational energy. Due to their random motion, molecules continuously move between regions of different temperature. When more energetic molecules from hotter regions interact with molecules in cooler regions, energy is transferred. As a result, a net energy flux arises from regions of higher temperature to regions of lower temperature. Molecular collisions further enhance this transfer, making conduction effectively a diffusion of energy. [5]

In liquids, the mechanism of heat conduction is similar to that in gases, but the molecules are more closely spaced, resulting in stronger and more frequent interactions. Consequently, energy transfer occurs more efficiently through intermolecular interactions. [5]

In solids, heat conduction is primarily associated with atomic vibrations within the crystal lattice. These vibrations can be viewed as lattice waves that transport energy through the material. In non-conducting materials, heat transfer occurs exclusively through these lattice vibrations, whereas in metals, it is further enhanced by the movement of free electrons, which can transport energy very efficiently. [5]

The conduction process is quantified by Fourier's law, which relates heat flux to the temperature gradient. In one dimension, the heat flux is defined as

$$q''_{\text{conduction}} = -k \frac{dT}{dx} \text{ (W/m}^2\text{)} \quad (2.9)$$

Here, k (W/m·K) is the thermal conductivity and the negative sign indicates heat flows in the direction of decreasing temperature. Note that this equation provides a heat flux, that is, the rate of heat transfer per unit area. The heat rate by conduction, $\dot{Q}$ (W), through a plane wall of area A (m²) is then the product of the flux and the area $\dot{Q} = q''_{\text{conduction}} A$.

2.5.2 Convection

Convection heat transfer occurs through a combination of molecular diffusion and fluid motion. When a fluid flows over a surface at a different temperature, a hydrodynamic boundary layer forms, where velocity increases from zero at the wall to the free-stream value. Simultaneously, a thermal boundary layer develops, in which temperature transitions from the surface temperature T_{surf} to the free-stream temperature T_{∞} . Depending on fluid properties and flow conditions, the thermal boundary layer may be thinner or thicker than the velocity boundary layer. Near the wall, fluid velocity is essentially zero, so heat transfer occurs purely by diffusion (conduction). Further from the wall, the bulk motion of the growing boundary layer transports heat downstream, meaning convection is sustained by the interaction between diffusive and advective processes. Understanding boundary-layer behavior is therefore fundamental to analyzing convection. [5]

A crucial distinction in convection is the nature of the fluid motion [5]:

- Forced convection occurs when external means, such as fans, pumps, pressure gradients, or aerodynamic flow, drive the motion. This typically yields strong advection, higher heat transfer rates, and thin boundary layers.
- Natural (free) convection occurs when buoyancy causes warmer, less dense fluid near a surface to rise, creating a flow in which the rising warm fluid is continuously replaced by cooler ambient fluid.
- Mixed convection arises when both forced flow and buoyancy forces significantly influence the boundary layer.

Regardless of the nature of the convection heat transfer process, the convective heat flux is described by Newton's law of cooling

$$q''_{\text{convection}} = h_{\text{conv}}(T_{\text{surf}} - T_{\infty}) \text{ (W/m}^2\text{)} \quad (2.10)$$

The convective heat flux is hence proportional to the difference between the surface and fluid temperatures, T_{surf} and T_{∞} , respectively, and the heat rate is then defined as $\dot{Q} = q''_{\text{convection}} A$. The parameter h_{conv} (W/m²·K), called the convective heat transfer coefficient and the key parameter for convection as it represents how effectively the fluid can remove or supply heat at the surface. The value of h_{conv}

depends on several factors, including the surface geometry, fluid properties, the presence of buoyancy effects, and any imposed flow velocities. All of these factors can vary strongly from case to case, resulting in large variations in h_{conv} . Hence, determining an accurate value of h_{conv} is one of the most critical steps in any convection analysis.

Approximate values of the convective heat transfer coefficient are often reported in the literature, with the range $2 \text{ W/m}^2\cdot\text{K}$ to $25 \text{ W/m}^2\cdot\text{K}$ being typical for natural convection [5, 17]. However, for more accurate analysis, natural convection heat transfer is commonly described using empirical correlations based on dimensionless parameters. In particular, the Rayleigh number, Ra_L , characterizes the strength of buoyancy-driven flow, defined as

$$Ra_L = \frac{g\beta(T_{\text{surf}} - T_{\infty})L^3}{\nu\alpha} \quad (2.11)$$

where g is the gravitational acceleration, β is the thermal expansion coefficient of the fluid, ν is the kinematic viscosity and α is the thermal diffusivity. For horizontal plates, characteristic length scales are typically based on surface area, with $L = A_s/P$, where A_s and P are the plate surface area and perimeter, respectively [5]. Additionally, all thermophysical properties evaluated at the film temperature, $T_{\text{film}} = (T_s + T_{\infty})/2$ [5].

Moreover, the Nusselt number, Nu_L , relates the convective heat transfer coefficient to the thermal properties of the fluid. A range of empirical correlations for the average Nusselt number, Nu_L , have been developed. Some correlations for natural convection for a horizontal plate are shown in Table 2.1.

Table 2.1: Empirical Nusselt number correlations for natural convection over a horizontal plate.

Lloyd and Moran [19]	$Nu_L = 0.540 Ra_L^{0.250}$	$1 \times 10^4 \lesssim Ra_L \lesssim 1 \times 10^7$
Wei, Yu, and Kawaguchi [37]	$Nu_L = 0.823 Ra_L^{0.201}$	$1 \times 10^5 \lesssim Ra_L \lesssim 1 \times 10^7$
Martorell, Herrero, and Grau [23]	$Nu_L = 1.280 Ra_L^{0.167}$	$8 \times 10^2 \lesssim Ra_L \lesssim 2 \times 10^6$

2.5.3 Radiation

Thermal radiation is the transfer of energy by electromagnetic waves emitted by any matter at a temperature above absolute zero. Unlike conduction and convection, radiation does not require a medium, and it occurs most efficiently in a vacuum. Radiation originates from changes in the electron energy states of atoms or molecules, and all solids, liquids, and gases can emit heat via radiation. The following relations and derivations are based on the formulations presented in [5].

The rate at which a surface emits radiant energy per unit area is called the emissive power, E . The maximum possible emission is given by the Stefan-Boltzmann law,

$$E_b = \sigma T_{\text{surf}}^4 \quad (2.12)$$

which describes an ideal emitter known as a blackbody. Here, T_{surf} is the surface temperature and $\sigma = 5.670 \times 10^{-8} \text{ W/m}^2 \cdot \text{K}^4$ is the Stefan-Boltzmann constant.

However, real surfaces emit less radiation, and their emissive power is described as

$$E = \varepsilon \sigma T_{\text{surf}}^4 \quad (2.13)$$

where ε is the emissivity, ranging from 0 to 1, and depends strongly on surface material and finish, and temperature.

In addition to emitting radiation, a surface also receives radiation from its surroundings. The total incident radiation per unit area is referred to as the irradiation, G , of which a fraction α is absorbed by the surface. A common assumption is that a small surface exchanges radiation with large, isothermal surroundings, characterized by a uniform temperature T_{∞} . In this case, the irradiation can be approximated as blackbody radiation, i.e. $G = \sigma T_{\infty}^4$. The absorbed radiation is therefore given by

$$G_{\text{abs}} = \alpha G = \alpha \sigma T_{\infty}^4, \quad (2.14)$$

where α is the absorptivity of the surface, ranging from 0 to 1.

If the surface is assumed to be gray, its radiative properties are considered independent of wavelength. Under this assumption, the absorptivity equals the emissivity ($\alpha = \varepsilon$), which allows the radiation exchange to be expressed in a simplified form. The net radiative heat flux is defined as the difference between emitted and absorbed radiation, described by

$$q''_{\text{radiation}} = \varepsilon \sigma T_{\text{surf}}^4 - \alpha \sigma T_{\infty}^4 = \varepsilon \sigma (T_{\text{surf}}^4 - T_{\infty}^4) \text{ (W/m}^2\text{)}. \quad (2.15)$$

This formulation assumes that the medium between the surface and the surroundings does not significantly participate in radiation, which is equivalent to conditions in a vacuum. In many applications, surrounding gases can be approximated as transparent to thermal radiation, and the expression remains valid. However, in the presence of participating media, such as gases that absorb or emit radiation, the radiative heat transfer may be altered.

2.5.4 Dimensionless Numbers

In systems where both thermal diffusion and bulk motion contribute to heat transfer, dimensionless parameters are commonly used to assess the relative importance of different transport mechanisms.

Thermal diffusion is governed by the thermal diffusivity, defined as

$$\alpha = \frac{k}{\rho c_p}, \quad (2.16)$$

where k is the thermal conductivity, ρ the density, and c_p the specific heat capacity.

In systems where both diffusion and bulk motion contribute to heat transport, the relative importance of these mechanisms can be assessed using the Péclet number, defined as

$$Pe = \frac{vL}{\alpha}, \quad (2.17)$$

where v is a characteristic velocity and L is a characteristic length scale [22]. The Péclet number is the ratio between heat transport due to motion (advection) and heat transport due to thermal diffusion (conduction). For large Péclet numbers, $Pe \gg 1$, heat transfer is dominated by advection, and thermal diffusion in the direction of motion becomes negligible compared to motion [22]. In such cases, temperature gradients tend to persist and are transported with the moving material.

The Fourier number, Fo , is defined as

$$Fo = \frac{\alpha t}{L^2}, \quad (2.18)$$

where t is a characteristic time scale. The Fourier number characterizes the extent of thermal diffusion over a given time relative to the size of the system. When $Fo \ll 1$, thermal diffusion is limited and temperature gradients remain largely unchanged. Conversely, when $Fo \sim 1$ or larger, diffusion becomes significant and tends to smooth out temperature variations. [5]

Together, the Péclet and Fourier numbers provide a framework for analyzing the competition between thermal diffusion and transport due to motion, as well as the timescales over which temperature fields evolve.

2.6 Fluid Flow

Fluid flow influences the heat transfer between the solid surface and the surrounding gas by transporting heat away from the surface. Depending on the flow conditions, this transport may be driven by buoyancy effects arising from temperature differences or by imposed flow velocities.

In the present study, both simplified and CFD-based descriptions are considered. While simplified models focus on buoyancy-driven flow, the CFD simulations capture the combined effects of natural and forced convection in the vicinity of the heated region.

The fundamental fluid flow phenomena relevant to this process are briefly outlined in the following sections.

2.6.1 Flow Regimes

The behavior of fluid flow is influenced by the balance between inertial and viscous forces, commonly described by the Reynolds number,

$$\text{Re} = \frac{\rho U L}{\mu} \quad (2.19)$$

where U is a characteristic velocity scale of the flow, L is a characteristic length. Here, U represents a scalar measure of the flow speed, in contrast to the local velocity vector $\mathbf{u}$.

This dimensionless number provides an indication of the flow regime. At low Reynolds numbers, the flow is typically laminar, characterized by smooth and ordered motion. As the Reynolds number increases, inertial effects become more significant and the flow may transition to turbulence, where mixing is enhanced and the flow becomes unsteady.

The flow regime influences heat transfer, as turbulent flows generally promote stronger mixing and therefore higher convective heat transfer compared to laminar flows.

2.6.2 Boundary Layers

When a fluid flows over a solid surface, a boundary layer develops due to viscous effects. Within this region, the fluid velocity increases from zero at the wall to the free-stream value, resulting in strong velocity gradients.

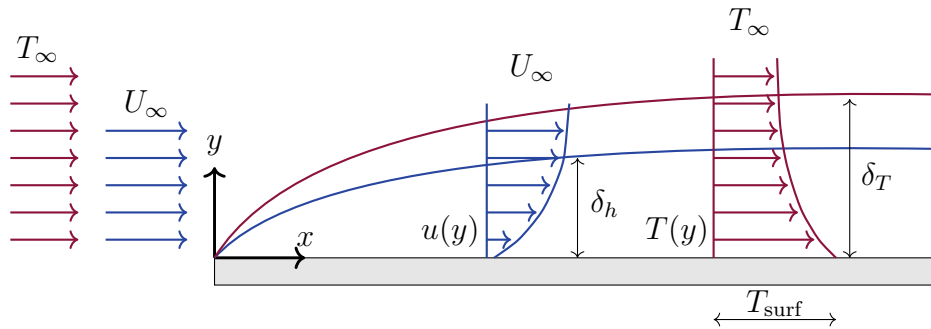


Figure 2.3: Development of velocity and thermal boundary layers over a flat plate.

If there is a temperature difference between the surface and the fluid, a thermal boundary layer also forms. In this region, heat transfer is governed by temperature gradients normal to the wall, and the wall heat flux is directly related to the local temperature gradient.

The thickness and behavior of both the velocity and thermal boundary layers influence the convective heat transfer between the surface and the fluid. While these phenomena are not analysed in detail in this work, they are inherently captured in the CFD solution and affect the resulting heat transfer.

2.6.3 Density Modeling

The fluid density can be treated as either constant or variable depending on the flow conditions. For low-speed flows with small temperature variations, the density

is often assumed constant, corresponding to incompressible flow. For compressible flows, the density is commonly described using the ideal gas law,

$$\rho = \frac{p}{RT} \quad (2.20)$$

where p is the pressure, R is the specific gas constant which is defined as $R = R_u/M$ where $R_u = 8314.4621 \text{ J/kmol} \cdot \text{K}$ is the universal gas constant and M is the molecular weight. [32]

When temperature differences give rise to buoyancy effects, density variations can instead be approximated using the Boussinesq approximation. In this approach, density is assumed constant everywhere except in the body force term, where it is expressed as

$$\rho = \rho_0 [1 - \beta(T - T_0)]. \quad (2.21)$$

where β is the thermal expansion coefficient, defined as the relative change in density with temperature. This approximation allows buoyancy-driven flow to be captured while avoiding the need for a fully compressible formulation.

2.7 Computational Fluid Dynamics

The governing equations describing fluid flow and heat transfer are generally too complex to be solved analytically, due to their nonlinear nature and the complexity of realistic geometries and boundary conditions. Instead, numerical methods are used to obtain approximate solutions.

Computational Fluid Dynamics (CFD) refers to the application of such methods to solve the governing equations over a discretized domain. By dividing the physical domain into a finite number of control volumes and integrating the governing equations over each control volume, the continuous equations are transformed into a system of algebraic equations that can be solved numerically. This enables the analysis of complex flow phenomena such as turbulence and heat transfer.

2.7.1 Finite Volume Method

The finite volume method (FVM) is a commonly used discretization technique in CFD due to its conservative formulation. In this approach, the computational domain is divided into a finite number of control volumes, and the governing equations are integrated over each control volume.

Starting from a general transport equation [35]

$$\frac{\partial(\rho\phi)}{\partial t} + \nabla \cdot (\rho\mathbf{u}\phi) = \nabla \cdot (\Gamma\nabla\phi) + S_\phi \quad (2.22)$$

which is integrated over an arbitrary control volume V , resulting in

$$\int_{CV} \frac{\partial(\rho\phi)}{\partial t} dV + \int_{CV} \nabla \cdot (\rho\mathbf{u}\phi) dV = \int_{CV} \nabla \cdot (\Gamma\nabla\phi) dV + \int_{CV} S_\phi dV. \quad (2.23)$$

For transient problems, the governing equations must also be considered over a finite time interval, meaning that the equations are effectively integrated in both space and time.

By applying Gauss' divergence theorem, the volume integrals of the divergence terms are transformed into surface integrals, allowing the governing equations to be expressed in terms of fluxes across the control volume faces. These fluxes are approximated using numerical schemes, resulting in a system of algebraic equations of the form

$$a_P \phi_P = \sum_N a_N \phi_N + b \quad (2.24)$$

where the coefficients depend on the chosen discretization schemes and mesh properties.

The resulting system is solved numerically using either segregated or coupled approaches. In a segregated method, the governing equations are solved sequentially, with variables such as pressure and velocity updated iteratively. In contrast, a coupled method solves the equations simultaneously as a single system, which can improve convergence for strongly coupled problems at the expense of higher computational cost.

The accuracy and stability of the solution are influenced by the choice of discretization schemes. Lower-order schemes are generally more robust but introduce higher numerical diffusion, which can smear sharp gradients, while higher-order schemes improve accuracy by reducing numerical diffusion, but may be less stable [35]. For transient simulations, time discretization can be handled using explicit or implicit schemes. Explicit methods are conditionally stable and require small time steps, while implicit methods allow larger time steps at increased computational cost [35]. The choice of discretization therefore represents a trade-off between accuracy, stability, and computational cost.

2.7.2 Turbulence Modeling

Turbulent flows are characterized by chaotic, time-dependent fluctuations over a wide range of length and time scales. Resolving all scales directly is computationally prohibitive for most applications, and turbulence models are therefore required.

Common modeling approaches include Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and Reynolds-Averaged Navier-Stokes (RANS). DNS resolves all relevant scales but is computationally too expensive for many applications. LES resolves the larger turbulent structures while modeling the smaller scales, offering improved accuracy at a higher computational cost. Instead, the Reynolds-Averaged Navier-Stokes (RANS) approach is commonly used, where the governing equations are time-averaged and only the mean flow is solved using Reynolds decomposition

$$\phi = \bar{\phi} + \phi' \quad (2.25)$$

where $\bar{\phi}$ represents the time-averaged component and ϕ' the fluctuating part.

Applying this decomposition to the Navier-Stokes equations introduces additional unknown terms, known as the Reynolds stresses, $-\rho \overline{u'_i u'_j}$. These terms represent the effect of turbulence on the mean flow and require modeling to close the system of equations. A common closure model for this is the Boussinesq hypothesis, which relates the Reynolds stresses to the mean velocity gradients, as

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (2.26)$$

where μ_t is the turbulent viscosity and k is the turbulent kinetic energy. Models based on this assumption are referred to as eddy-viscosity models, among which two-equation models are commonly employed. These involve solving transport equations for the turbulent kinetic energy and an additional variable associated with its dissipation.

2.7.2.1 Standard k - ε Model

The standard k - ε model is a widely used two-equation turbulence model, where transport equations are solved for the turbulent kinetic energy k and its dissipation rate ε [13].

The turbulent viscosity is computed as

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (2.27)$$

where C_μ is a model constant.

The model is known for its robustness and relatively low computational cost, and is commonly applied to fully turbulent flows. However, it has limitations in predicting near-wall flows, strong pressure gradients, and flow separation.

2.7.2.2 k - ω Model

The k - ω model solves transport equations for the turbulent kinetic energy k and the specific dissipation rate ω . The turbulent viscosity is defined as

$$\mu_t = \frac{\rho k}{\omega} \quad (2.28)$$

A key advantage of the k - ω model is its ability to accurately resolve near-wall regions without the need for empirical wall functions. However, the model is sensitive to free-stream boundary conditions, which can reduce its robustness in certain applications.

2.7.2.3 SST k - ω Model

The Shear Stress Transport (SST) k - ω model, introduced by Menter [24], combines the strengths of both the k - ω and k - ε models. It utilizes the k - ω formulation in

the near-wall region to accurately capture boundary layer effects, while gradually transitioning to a k - ε formulation in the free stream.

In addition, the SST model introduces a limiter on the turbulent shear stress, improving the prediction of flow separation under adverse pressure gradients. This makes the model particularly suitable for complex engineering flows involving separation, strong pressure gradients, and wall-bounded turbulence. Due to its robustness and accuracy across a wide range of flow conditions, the SST k - ω model is widely regarded as one of the most reliable RANS turbulence models for engineering applications.

2.7.2.4 Wall Treatment

The treatment of the near-wall region is critical in turbulence modeling, as steep gradients exist within the boundary layer. The near-wall resolution is commonly characterized using the non-dimensional wall distance, y^+ , defined as

$$y^+ = \frac{u_\tau y}{\nu}, \quad u_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (2.29)$$

where u_τ is the friction velocity, τ_w is the wall shear stress, y is the distance to the wall, and ν is the kinematic viscosity.

Two main approaches are typically employed: wall-resolved modeling (low-Reynolds number approach) and wall function-based modeling (high-Reynolds number approach).

- **Wall functions**, which model the near-wall region using empirical relations. In this case, a coarser mesh is used, typically with $y^+ > 30$.
- **Wall-resolved modeling**, which requires a fine mesh such that the first cell centre satisfies $y^+ \approx 1$. This allows the viscous sublayer to be directly resolved.

The region $5 < y^+ < 30$, known as the buffer layer, should generally be avoided in simulations, as neither wall functions nor low-Reynolds formulations perform reliably in this range.

The SST k - ω model is particularly well suited for low-Reynolds number modeling, as it can integrate through the viscous sublayer without relying on wall functions. For accurate boundary layer resolution, a sufficiently fine mesh near the wall is therefore recommended when using this model.

2.7.3 Mesh Generation

The computational mesh defines how the governing equations are spatially represented and directly influences both accuracy and convergence. The domain is discretized into a finite number of cells, where each cell represents a control volume in the finite volume formulation.

2.7.3.1 Surface Meshes

In many meshing approaches, the volume mesh is generated from an underlying surface mesh that represents the geometry. The quality of the surface mesh is therefore critical, as it directly affects the resulting volume mesh. Poor surface resolution, distorted elements, or gaps in the surface representation may lead to low-quality volume cells and reduced solution accuracy. This is particularly important for complex geometries, where careful surface discretization is required to ensure reliable results.

2.7.3.2 Volume Meshes

The volume mesh is generated from the surface mesh and defines the discretization of the computational domain. Meshes are commonly classified as structured or unstructured.

Structured meshes consist of regularly connected cells, typically hexahedral, with a well-defined indexing pattern. This allows for accurate gradient evaluation and low numerical diffusion. However, generating structured meshes for complex geometries is often difficult.

Unstructured meshes consist of irregularly connected cells and offer greater flexibility for complex geometries, at the cost of increased computational effort and potentially higher numerical diffusion.

Common mesh types in industrial CFD (e.g., STAR-CCM+) include:

- **Tetrahedral mesh:** Easy to generate and fit complex geometries, however their limited number of faces and irregular orientation reduces flux accuracy per cell, often requiring a larger number of cells.
- **Polyhedral mesh:** Cells with many faces, typically around 12–14, which improve flux evaluation and gradients which gives higher accuracy per cell than tetrahedral meshes.
- **Trimmed cell mesh:** Generated from a Cartesian background grid intersected with the geometry. The resulting cells are predominantly hexahedral and aligned, which reduces numerical diffusion and improves stability. This approach is also less sensitive to surface mesh quality.
- **Directed meshing:** Generates structured or semi-structured meshes by sweeping cells between surfaces, based on the underlying geometry. This produces high-quality hexahedral cells with controlled alignment, which can improve numerical accuracy and convergence.

2.7.3.3 Prism Layer Generation

In addition to the core mesh, prism layer meshes are commonly used near solid boundaries to accurately resolve boundary layers. These consist of layers of prismatic (hexahedral) cells extruded normal to the wall.

Since velocity and temperature gradients are steep near walls, prism layers are essential for resolving boundary layer effects and accurately predicting wall shear stress and heat transfer. The resolution is often characterized using the dimensionless wall distance, y^+ .

2.7.3.4 Conformal and Non-Conformal Meshes

In multi-region simulations, the interface between regions can be treated using either conformal or non-conformal meshes.

A conformal mesh has matching cell faces at the interface, ensuring direct flux continuity between neighbouring regions. This generally improves numerical accuracy and reduces interpolation errors.

Non-conformal meshes allow different mesh resolutions on either side of an interface and require interpolation between non-matching faces. While this provides greater flexibility in mesh generation, it may introduce additional numerical errors.

2.7.4 Conjugate Heat Transfer

Conjugate heat transfer (CHT) describes the coupled heat transfer between fluid and solid domains, where convection in the fluid and conduction in the solid are solved simultaneously. At the fluid-solid interface, continuity of temperature and heat flux is enforced, ensuring energy conservation between the domains [11]. This allows the heat transfer to be determined by the local flow conditions rather than prescribed boundary coefficients.

2.7.5 Energy Equation in CFD

In CFD, the energy equation is solved for both fluid and solid regions. In fluid domains, both convective and conductive heat transfer are included through the velocity field.

In solid regions, heat transfer is typically governed purely by conduction. However, STAR-CCM+ employs a generalized energy transport formulation in which a convective velocity, $\mathbf{v}_s$, can also be defined in solid domains. This allows energy transport due to prescribed solid motion, such as translation or rotation under steady-state conditions, to be represented when required. The energy equation solved by the software for solid regions is given by

$$\frac{\partial}{\partial t} \int_V \rho c_p T \, dV + \oint_A \rho c_p T \mathbf{v}_s \cdot d\mathbf{a} = - \oint_A \mathbf{q}'' \cdot d\mathbf{a} + \int_V s \, dV \quad (2.30)$$

where $\mathbf{v}_s$ is the prescribed solid convective velocity [32]. For stationary solids, $\mathbf{v}_s = 0$, and the formulation reduces to purely conductive heat transfer.

3

Methods

Numerical simulations, such as CFD and thermal analyses, are not exact representations of physical systems but predictive models based on mathematical formulations, discretization, and simplifying assumptions. Their accuracy depends on material data, boundary conditions, and modeling choices, and is further limited by uncertainties in experimental data and numerical implementation. This is particularly critical in method development, where simulations support design and optimization. Without validation, models may yield physically misleading results despite appearing reasonable. Establishing confidence in the modeling approach is therefore essential.

In coupled fluid-solid simulations, CFD provides boundary conditions at the fluid-solid interface, primarily in the form of heat transfer coefficients and fluid temperatures. While the fluid determines the heat transfer conditions, the system behavior is governed by the solid's thermal response. The fluid domain thus defines the driving mechanisms for heat transfer, whereas the solid model determines the response. As a result, inaccuracies in the solid model will directly propagate to the predicted temperatures and can dominate the overall error, regardless of CFD fidelity. A validated thermal model of the solid is therefore a prerequisite for meaningful CFD-based heat transfer analysis. Such a model must accurately represent material properties, heat conduction, and thermal storage, and respond consistently to variations in boundary conditions.

Modern CFD tools, such as STAR-CCM+, solve the energy equation in both fluid and solid domains, enabling conjugate heat transfer (CHT) within a unified framework. All simulations in this work were conducted in STAR-CCM+, allowing consistent treatment of both solid-only and coupled cases. While it is possible to couple separate CFD and FEM solvers through iterative exchange of boundary conditions, such approaches introduce additional complexity and potential numerical inconsistencies. By using a single multi-physics tool, the interaction between fluid and solid can be handled directly, simplifying the workflow and ensuring consistency between simulation stages.

The ultimate objective of this work is to develop a predictive CFD-CHT model of the LMD-w process for a drum component that is currently under development at GKN. However, direct validation of such a model is challenging due to the limited availability of measurement data for the full-scale system. In addition, several boundary conditions and model parameters are either unknown or only approximately known,

resulting in significant uncertainty in the simulation setup. Under such conditions, agreement between simulations and measurements becomes difficult to interpret, as discrepancies may originate from multiple sources simultaneously.

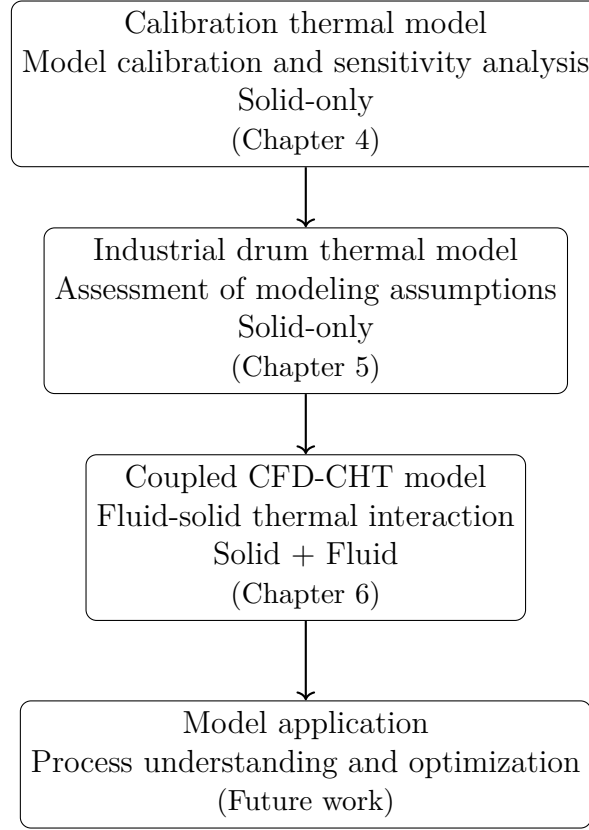


Figure 3.1: Overview of the modeling workflow. The methodology begins with a small-scale thermal model for calibration and sensitivity analysis, followed by application to the industrial drum geometry (annular cylinder) to assess key modeling assumptions. Once confidence in the solid thermal response is established, the approach is extended to a coupled CFD-CHT model with direct fluid-solid interaction. The model can then be used for process understanding and optimization.

To establish confidence in the modeling approach, a stepwise methodology was adopted, as shown in Figure 3.1. Rather than directly applying a full CFD-CHT model to the industrial drum case, a simpler thermal model was first developed using a dedicated calibration experiment with richer measurement data and better-defined boundary conditions. Once confidence in the thermal modeling approach had been established, the methodology was applied to the industrial drum geometry to assess the validity of the underlying modeling assumptions under more realistic conditions. The model was then extended to include fluid flow through a coupled CFD-CHT formulation, allowing the influence of the flow field on the thermal response to be evaluated. This staged methodology builds increased model complexity on a progressively established foundation, improving confidence in the predictive capability of the simulations.

3.1 Material Properties

Depending on the submodel, the simulations included either solid-only heat transfer or coupled fluid-solid modeling.

3.1.1 Solid Modeling and Phase Change Modeling

Material data for Ti-6Al-4V were provided by GKN [6], including temperature-dependent specific heat capacity, thermal conductivity and density. However, the original temperature-dependent data were sparse and exhibited abrupt variations, which are unfavorable for numerical stability in CFD simulations. Therefore, the material data were interpolated to obtain smooth and continuous functions.

In this work, melting and solidification of Ti-6Al-4V are treated within a temperature-based thermal framework. The simulations employ the segregated solid energy model, in which temperature is the primary solved variable and the thermal response of the solid is governed by temperature-dependent material properties. To account for the latent heat associated with melting and solidification, the apparent heat capacity method described in Section 2.3.2 is adopted. By artificially increasing the heat capacity in the mushy zone, the latent heat required for phase change is taken into consideration.

The latent heat contribution is modeled using a squared cosine function over half a period, as proposed by Foteinopoulos et al. [8]:

$$c_p^{\text{lat}}(T) = A \cos^2 \left(\pi \frac{T - T_m}{T_{\text{liquidus}} - T_{\text{solidus}}} \right). \quad (3.1)$$

The coefficient A is determined from the condition that the integrated latent heat equals the enthalpy of fusion:

$$H_m = \int_{T_{\text{solidus}}}^{T_{\text{liquidus}}} c_p^{\text{lat}}(T) dT = \frac{A(T_{\text{liquidus}} - T_{\text{solidus}})}{2}, \quad (3.2)$$

which yields

$$A = \frac{2H_m}{T_{\text{liquidus}} - T_{\text{solidus}}}. \quad (3.3)$$

The apparent heat capacity is then defined as

$$c_p^{\text{app}}(T) = c_p(T) + c_p^{\text{lat}}(T). \quad (3.4)$$

Due to confidentiality, exact values for temperature and the material data shown on the axes cannot be disclosed. The interpolated data compared to the original data for specific heat capacity, thermal conductivity, and density, together with the modified heat capacity including the latent heat, is shown in Figure 3.2. This material data were used for the calibration case.

At a later stage of the work, updated material property data became available, including revised density and thermal conductivity values. In addition, enthalpy-temperature data were provided instead of heat capacity. The updated material

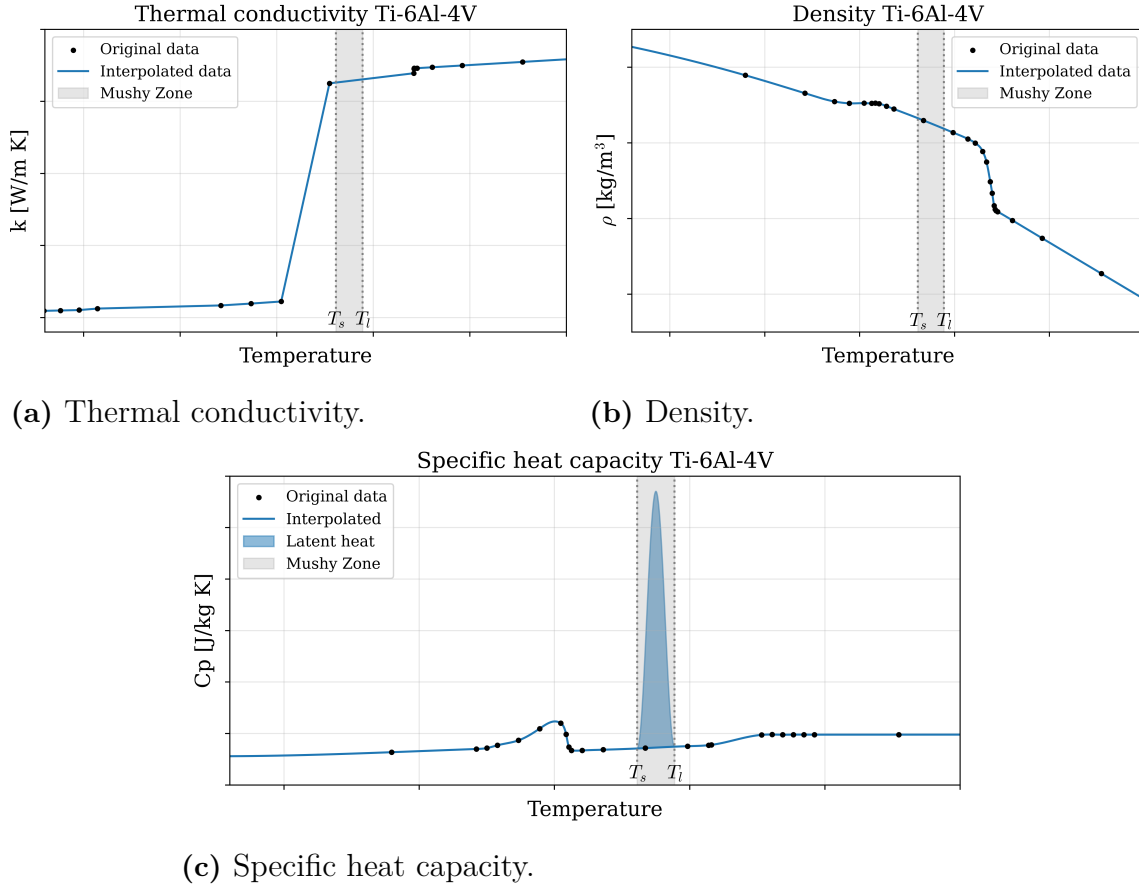


Figure 3.2: Temperature-dependent material data for Ti-6Al-4V, including original experimental data, interpolated data, solidus temperature, and liquidus temperature. The mushy zone between the solidus and liquidus temperatures, denoted as T_s and T_l , is indicated by the shaded gray region. In the case of heat capacity, the latent heat contribution is included through a modified heat capacity curve, illustrated by the blue shaded region.

dataset, including enthalpy data, was provided by GKN [6] and was used for the real geometry model and all subsequent work.

Instead of defining the latent heat through an assumed heat capacity curve, the enthalpy data were used directly as a material property. In this case, the solver internally computes the effective heat capacity as the temperature derivative of the enthalpy curve. As a result, latent heat effects are still incorporated through an apparent heat capacity formulation, but based on enthalpy data rather than an assumed distribution. It is important to note that, although enthalpy data is used as input, the energy equation in the solid domain remains temperature-based. The use of enthalpy data therefore does not imply that an enthalpy formulation is solved numerically. Instead, the enthalpy-temperature relation defines the temperature-dependent energy storage of the material. Both approaches are mathematically equivalent, but the latter provides a more physically accurate representation of the phase change behavior.

The new set of material data, containing enthalpy instead of specific heat capacity, is illustrated in Figure 3.3.

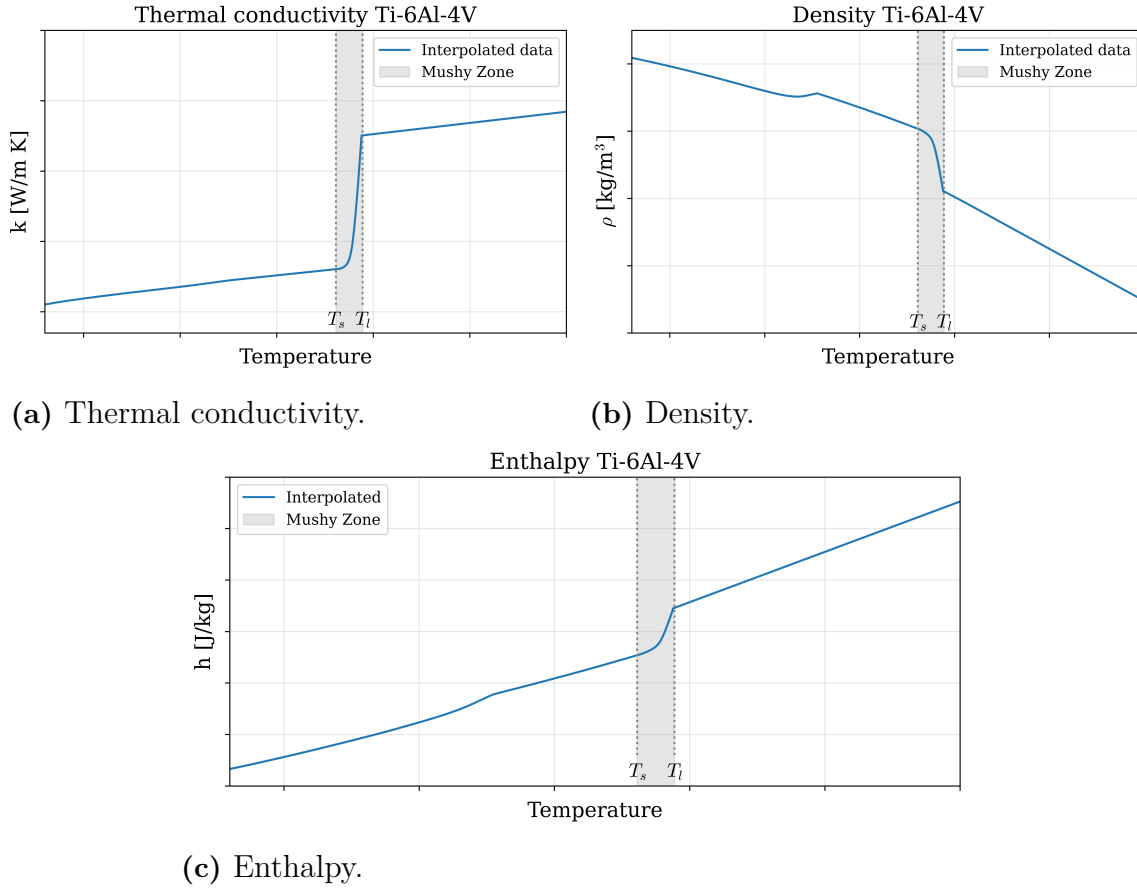


Figure 3.3: Temperature-dependent material data for Ti-6Al-4V, including experimental data, solidus temperature, and liquidus temperature. The mushy zone between the solidus and liquidus temperatures is indicated by the shaded gray region.

3.1.2 Fluid Modeling

The gas phase was modeled using the Boussinesq approximation, see Equation (2.21), instead of an ideal gas formulation. This choice was motivated by initial simulations, which showed unstable and oscillatory behavior, partly due to the strong coupling between pressure and density in the ideal gas model. In the Boussinesq approach, the fluid density is assumed constant in all governing equations except for the buoyancy term in the momentum equation, where it varies linearly with temperature. This eliminates the coupling between pressure and density, leading to improved numerical stability while still capturing the dominant buoyancy-driven flow effects. Standard material properties for argon, as defined in STAR-CCM+, were used in the analysis.

This modeling approach is suitable for the present case, where temperature differ-

ences drive the flow but the resulting density variations remain relatively small. As a result, the Boussinesq approximation provides a stable and physically reasonable representation of natural convection within the tent.

4

Thermal calibration model

This chapter focuses on a calibration case used to assess whether the chosen boundary conditions and solver settings provide a physically reasonable description of the thermal behavior. The analysis is performed as a thermal solid-only simulation, with fluid effects represented through boundary conditions. By comparing simulations with detailed experimental data, the model can be validated and tuned before being applied to the full-scale geometry.

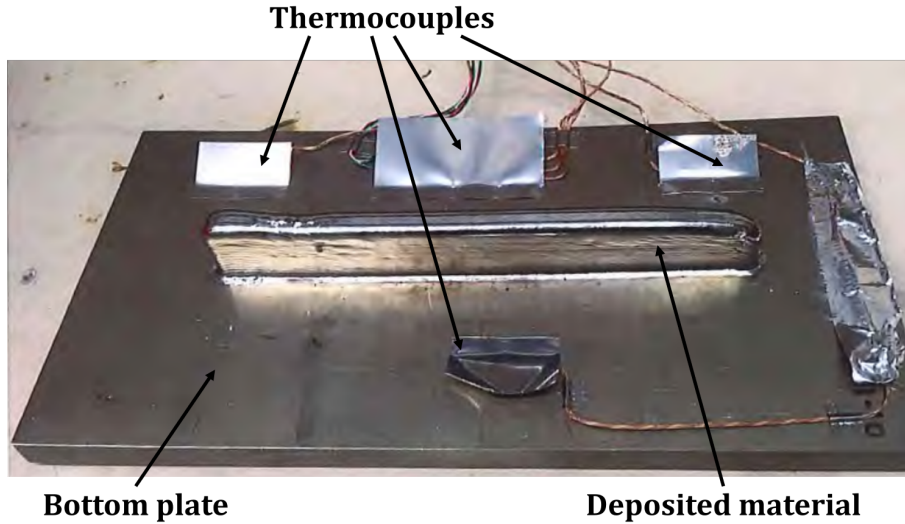


Figure 4.1: Geometry of the experimental setup, showing the bottom plate, deposited material and the thermocouples. The clamping screws are not shown.

The calibration setup consists of a straight build where material is deposited in two parallel rows over multiple layers. Each row is deposited during a relatively short time due to the small build length. After completing a row, the process is paused and the machine, including the laser, is turned off. During this pause, no active cooling is applied, instead, the entire plate is allowed to cool down naturally over a relatively long time period. This sequence of deposition followed by a pause can be considered as one cycle. Each layer is formed by repeating this cycle twice, once for each row, after which the process continues with the next layer. By repeating this procedure, the geometry is gradually built up from an initial base plate to a simple block-like structure. During deposition, the machine moves back and forth along the

build direction, building each row in the same direction. The experimental setup can be seen in Figure 4.1.

To measure the temperature, six thermocouples (TC) are placed at different locations on the base plate, and the experimental data used for calibration is obtained from these measurements. In addition, one thermocouple is placed on the fixture and one is used to measure the argon temperature inside the tent.

The plate is clamped to a fixture using eight screws, with one screw placed above and one below the plate in each corner. This keeps the plate fixed while exposing it to surrounding argon on the top, bottom, and side surfaces.

The process parameters used in this study were obtained from internal documentation at GKN [6]. Due to confidentiality, the exact values cannot be disclosed. An overview of the variables used in the simulations are shown in Table 4.1.

Table 4.1: Process parameters for the thermal-only simple calibration model.

Parameter	Value	Parameter	Value
Printing velocity, v_{traverse}	- mm/s	Wire diameter, d_{wire}	- mm
Laser power, P_{laser}	- kW	Wire feeding velocity, v_{wirefeed}	- m/min
Laser absorption, η_{laser}	-	Wire feed rate, $\dot{m}_{\text{wire}}$	- kg/s
Time laser on, t_{weld}	- s	Wire heating power, P_{wire}	- W
Time laser off, t_{pause}	- s	Wire heating efficiency, η_{wire}	-
Number of weld rows	2		
Number of layers	17		

4.1 Geometry and Mesh

A parameterized geometry was created in STAR-CCM+, see Figure 4.2c, allowing key geometric dimensions, such as the size of the screws, to be varied between simulations. The geometry itself is assumed to be constant in time. Although the simulations are transient, no material addition is modeled explicitly. As a consequence, the amount of material present in the model is not representative at the beginning of the simulation, while it better represents the final state toward the end of the simulation. This assumption is accepted as a modeling simplification, consistent with the focus on the thermal response rather than the exact deposition sequence.

The domain was discretized using a polyhedral cell topology due to its robustness and ease of use within STAR-CCM+. To better capture the melt zone and the associated high thermal gradients, local mesh refinements were applied at the top surface where deposition occurs. Additional refinements were introduced around the small screws to ensure sufficient spatial resolution in these regions. Furthermore, relatively low growth rates were used to maintain a smooth transition in cell size

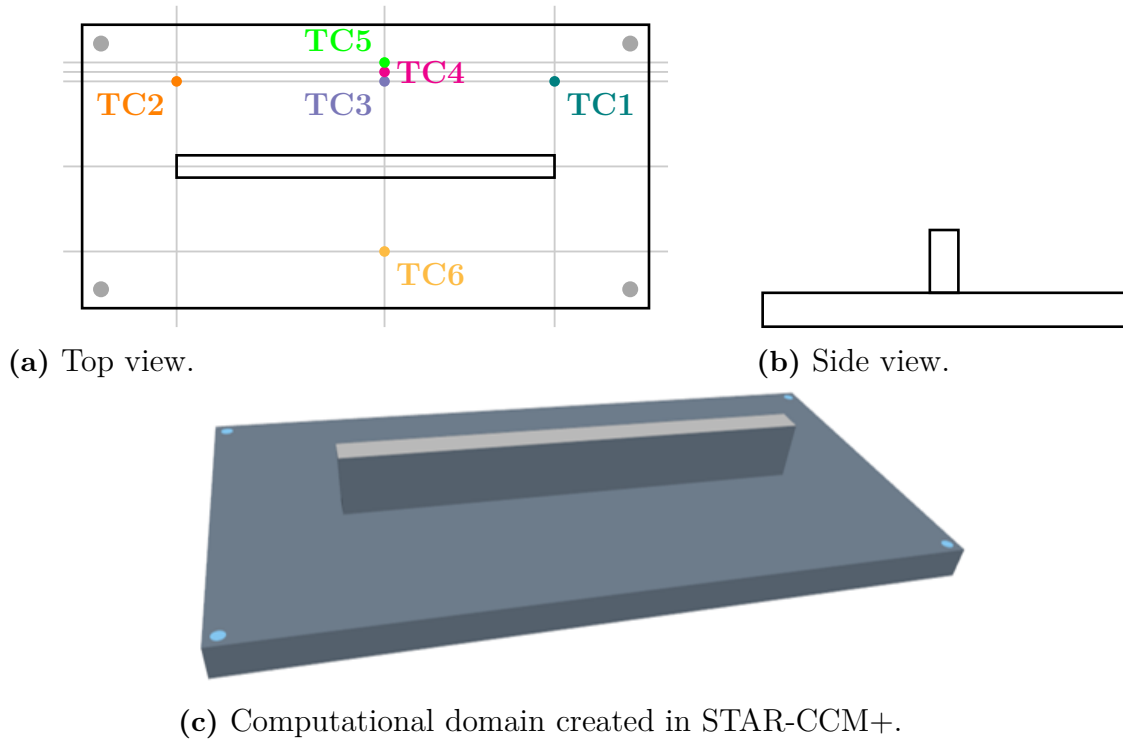


Figure 4.2: Geometry of the calibration model and computational domain used in the simulations. Dimensions are omitted due to confidentiality, while the overall geometry and placement of thermocouples TC1-TC6 are shown.

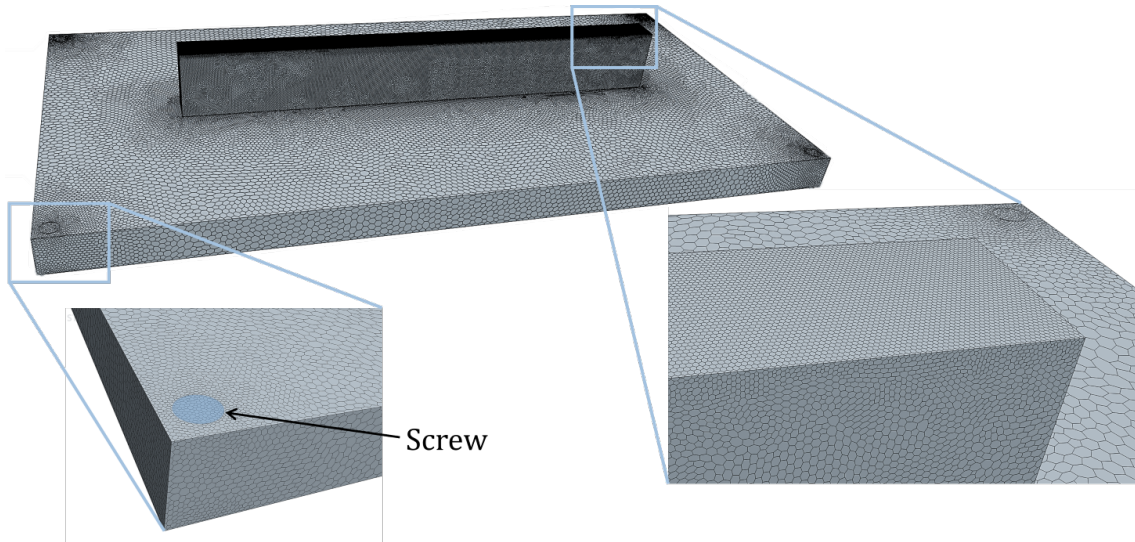


Figure 4.3: Overview of the mesh with refinements at the deposition surface and close to the screws.

and avoid abrupt changes in mesh density. The resulting mesh is shown in Figure 4.3, while the applied mesh settings are summarized in Table 4.2.

Table 4.2: Mesh and timestep settings used in STAR-CCM+ for the thermal-only simple calibration model.

Parameter	Value
Time step when laser is off	Varied
Time step when laser is on	Varied
Mesh type	Polyhedral
Base size	Varied
Surface target size	100% of Base size
Minimum surface size	50% of Base size
Surface growth rate	1.05
Maximum volume size	200 % of Base size
Volume growth rate	1.1
Surface refinement region	Top surface, where deposition occurs
Refinement size	10% of Base size
Surface refinement region	Screws
Refinement size	20% of Base size

4.1.1 Mesh and Time-Step Sensitivity Analysis

Sensitivity studies were performed to ensure that the simulation results are independent of the spatial and temporal discretization. In transient simulations, both mesh resolution and time step size influence the numerical accuracy. A finer mesh allows for better spatial resolution of temperature gradients, while smaller time steps improve the temporal resolution of rapidly evolving thermal phenomena. However, a finer mesh as well as a smaller time step increases the computational cost significantly.

To evaluate the numerical sensitivity of the simulations to the spatial and temporal discretization, a mesh and time-step sensitivity study was conducted. The analysis was limited to a single-process cycle, consisting of a period of active laser followed by a cooling period, extended by an additional 10 s of heating. This extension was included to ensure that the analysis captures the most critical part of the process, where the temperature evolves most rapidly in time and exhibits the steepest spatial gradients during the heating phase.

For the mesh sensitivity study, the base cell size was varied between 10 mm and 1 mm, corresponding to approximately 30 000 to 3 000 000 cells.

For the time step sensitivity study, different time step sizes were evaluated. A variable time stepping approach was adopted, where smaller time steps were used during the heating phase and larger time steps during the cooling phase. This is motivated by the different time scales in the process, when the laser is active, rapid temperature changes and a moving heat source require high temporal resolution, whereas during cooling, the thermal evolution is significantly slower and can be accurately captured with larger time steps. The investigated combinations of time

steps for the heating and cooling phases are summarized in Table 4.3.

Table 4.3: Investigated time step combinations for heating and cooling phases.

Case	Time step laser on [s]	Time step laser off [s]
Fine	0.001	0.01
Medium	0.01	0.1
Large	0.1	1

The mesh and time step were varied simultaneously, as both spatial and temporal discretization influence the predicted temperature field. The discretization was refined until further refinement in either mesh size or time step resulted in negligible changes in the temperature response. The results of the sensitivity analysis are presented in Section 4.5.1.

4.2 Experimental Data

The experimental data used for validation are presented in Figure 4.4. Three experiments were conducted under identical process parameters and operating conditions. As slight temporal offsets were present between the individual experiments, the datasets were aligned such that the onset of laser heating occurs at the same physical time. This ensures a consistent phase relationship between the measurements and allows for a more meaningful comparison with the transient simulation results. To facilitate comparison with the simulations, an average temperature curve was constructed based on the experimental measurements at thermocouples TC1-TC6.

In addition, temperatures at the end of the process were extracted, shown in Table 4.4. These values are primarily used for quantitative comparison in the subsequent analysis.

Table 4.4: Experimentally determined temperatures at thermocouples TC1-TC6 at the end of the test.

TC	Exp01 [°C]	Exp02 [°C]	Exp03 [°C]	Average [°C]
1	135.7	128.7	123.9	129.4
2	139.8	133.5	138.2	137.2
3	168.9	158.6	160.6	162.7
4	165.7	155.6	157.2	159.5
5	163.2	153.0	154.8	157.0
6	174.4	161.4	162.7	166.2

Although the simulations are fully deterministic and would in principle yield identical results for identical inputs, the experimental measurements exhibit noticeable variations between the three test cases. This introduces a measurement uncertainty,

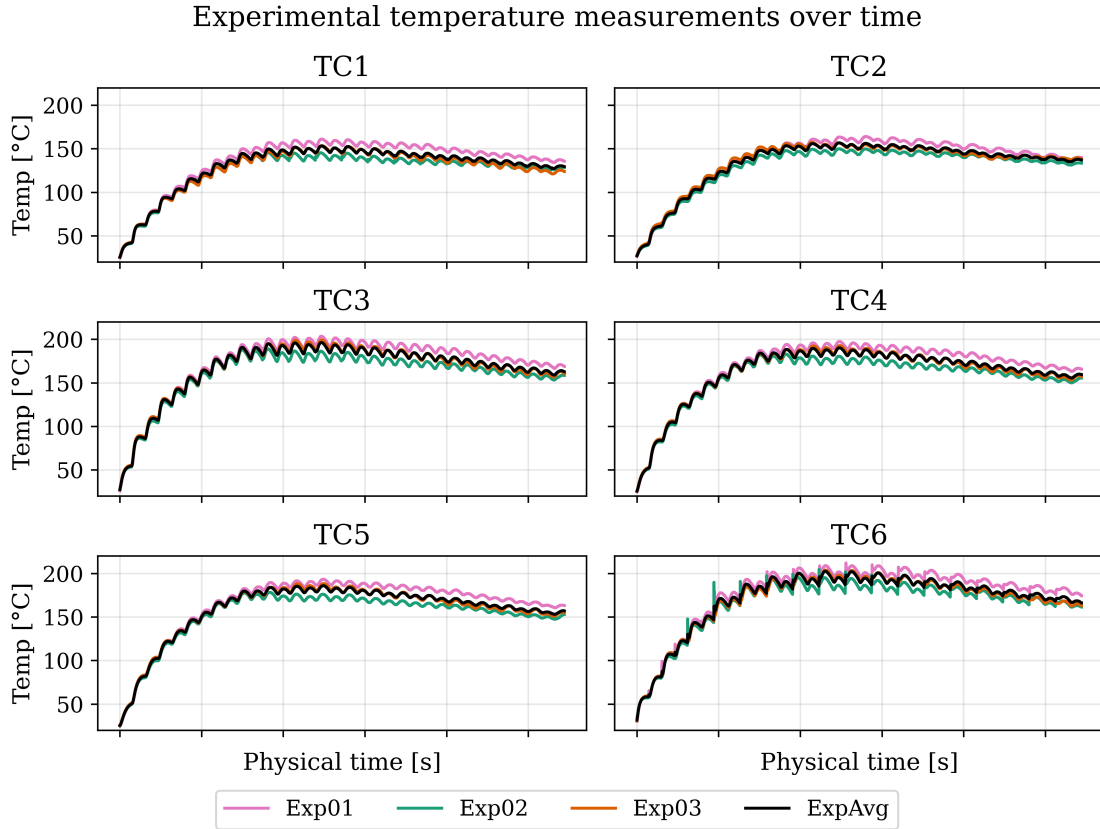


Figure 4.4: Experimentally determined temperatures at thermocouples TC1-TC6 during three experiments. An average function of three experimental datasets was also calculated.

which should be considered when evaluating the agreement between simulations and experiments.

Furthermore, the temperature histories do not follow identical profiles, with some thermocouples showing less smooth behavior, likely due to measurement uncertainties such as variations in thermocouple attachment or the experimental setup. However, this does not affect the overall behavior and is therefore neglected.

4.3 Solver Settings

The simulations were performed using a segregated energy solver, as only heat conduction in solids was considered. A high under-relaxation factor (URF) of 0.99 was used for the energy equation to accelerate the convergence of the residuals [7]. This was considered appropriate, as the solution converged smoothly and remained stable throughout the simulations, without showing any signs of numerical oscillations.

Convergence control was achieved through a combination of stopping criteria. The

overall simulation was terminated based on a prescribed maximum physical time corresponding to the duration of the simulated process. For the inner iterations at each time step, convergence was monitored using the energy residual. An asymptotic stopping criterion was applied, with a normalized tolerance of 0.05 over 10 successive iterations. This criterion ensures that convergence is reached when the variation in the monitored residual becomes sufficiently small over a number of iterations, rather than requiring a strict absolute threshold. The normalized tolerance evaluates the relative variation in the residual by scaling it with respect to its average value over the specified number of samples. This approach is particularly suitable when the exact limiting value of the residual is not known in advance, but where a sufficiently small variation indicates convergence. In addition, a maximum of 100 inner iterations per time step was imposed as a secondary stopping criterion. This ensures that the solver advances in time even in cases where the residual convergence is slow. However, in practice, the energy residual reached convergence within the specified asymptotic criterion for nearly all time steps.

Furthermore, a first-order implicit temporal discretization scheme was employed. While higher-order schemes can provide improved accuracy, a first-order scheme was selected due to its robustness and stable behavior for the present transient thermal problem [35]. The impact of the temporal discretization was evaluated through a dedicated time step sensitivity analysis. The results of the sensitivity analysis indicate that, despite the use of a first-order scheme, the chosen time step provides an accurate representation of the transient thermal response.

For the spatial discretization, the default settings in STAR-CCM+ were used. In the present solid heat conduction problem, which is diffusion-dominated, the spatial accuracy is primarily governed by the mesh resolution. The finite volume formulation provides an accurate representation of the temperature gradients, and no additional discretization settings were required.

4.4 Boundary Conditions

The thermal model requires several boundary conditions to be defined, including heat input from the laser as well as radiative, convective, and conductive heat transfer. Some of these are directly specified from known process conditions, while others are associated with uncertainty and therefore require calibration or sensitivity analysis. In the following sections, the treatment of each boundary condition is described, including which parameters are fixed and which are varied, along with assumptions.

4.4.1 Heat Input Modeling

In the LMD-w process, material is continuously supplied to the melt zone through a wire feed. The wire enters the wire-feeding machine at approximately room temperature, where it is preheated by resistive heating due to an applied current ($P = I \cdot V$). The preheated wire is then fed into the melt pool where it is melted by the laser.

In the present thermal model, neither wire feeding nor melt pool dynamics are modeled explicitly. Consequently, the thermal effect of the incoming wire must be accounted for to maintain a physically consistent energy balance. A straightforward approach would be to subtract the energy required to heat and melt the incoming wire from the heat input. This would yield an effective heat source representing only the energy available to heat the underlying material.

However, this approach assumes that the latent heat associated with wire melting is irreversibly removed from the system. In the present experimental setup, this assumption is not valid as the process does not involve continuous melting. Instead, the deposition and melting process occurs over a short time period, followed by a pause during which the material solidifies and releases latent heat. Consequently, melting and solidification occur sequentially rather than within a continuous melt pool. Treating wire melting as a permanent energy sink would therefore neglect this recovery of latent heat and introduce an artificial energy deficit, resulting in an unphysical energy balance.

Accordingly, the present model is based on the assumption that the material ahead of the laser corresponds to the same volume as the wire being fed into the melt pool. Under this assumption, the applied laser energy heats and melts material that is already present within the computational domain. In this way, phase change effects are captured through the heat source and thermal history, without introducing explicit source or sink terms associated with material addition.

The effective heat source is hence defined as

$$P_{\text{eff}} = \eta_{\text{laser}} P_{\text{laser}} + \eta_{\text{wire heating}} P_{\text{wire heating}} \quad (4.1)$$

Where laser power P_{laser} , laser absorptivity $\eta_{\text{wire heating}}$, wire pre-heating power $P_{\text{wire heating}}$ and wire pre heating efficiency $\eta_{\text{wire heating}}$ are prescribed process parameters. However, the exact values are omitted due to confidentiality.

The heat input was modeled using a volumetric heat source instead of a surface-applied heat flux. A surface heat flux implicitly assumes that the absorbed laser energy is applied directly at the material surface. In practice, laser absorption in metals occurs over a finite optical penetration depth governed by the optical properties of the material [36]. Representing the laser input as a volumetric source therefore provides a more physically realistic description of the heat input.

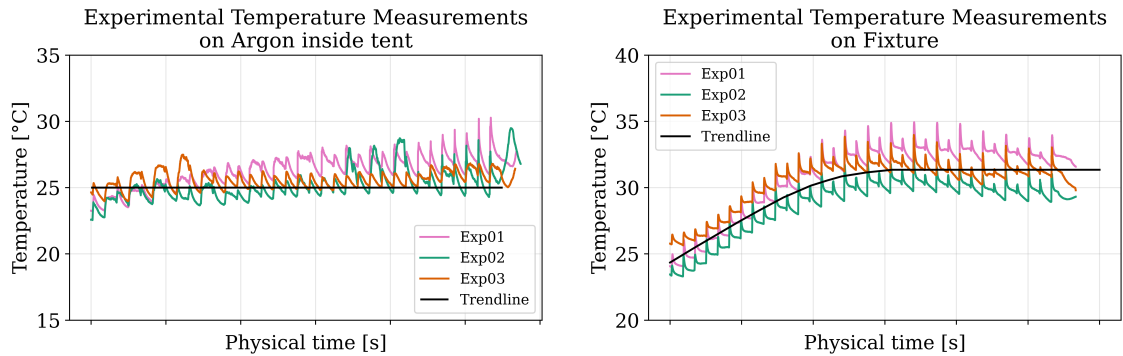
Previous studies have shown that the assumed heat-source formulation can significantly influence the predicted thermal response and melt pool dimensions in thermal AM simulations. More detailed formulations, such as Gaussian or Goldak-type volumetric heat sources [12], are commonly used to improve agreement with experimentally observed melt pool geometries [26, 9]. However, such models generally require additional calibration parameters related to the melt pool shape and energy distribution. Since detailed melt pool dynamics are not resolved in the present work and the primary interest lies in the macroscopic thermal behavior of the domain rather than the exact melt pool geometry, the total applied power is considered more important than the precise heat-source shape.

The heat input was therefore modeled as a moving volumetric heat source with a top-hat radial distribution and an assumed finite penetration depth of 1 mm normal to the irradiated surface. The top-hat profile corresponds to a uniform power distribution across the laser spot and matches the measured intensity distribution of the experimental laser system, while the finite depth represents volumetric absorption within the material rather than a purely surface-based heat input. Combined, this formulation provides a simplified yet physically representative description of the laser-material interaction.

In the present model, the geometry was held fixed in space throughout the simulation. Instead of moving the computational domain, the laser input was represented as a moving volumetric heat source. The heat source was translated over the irradiated surface and, after completing one pass over the defined path, was deactivated to represent a process pause allowing the geometry to cool between heating cycles. The sequence of laser motion followed by a pause was then repeated, forming a cyclic heating pattern.

4.4.2 Convective Heat Transfer

Convective heat transfer occurs at the interfaces between the solid and the surrounding fluid. Therefore, a convective boundary condition was applied to all surfaces, except at the locations where the screws are in contact with the plate. For this boundary condition, a bulk fluid temperature, T_{Ar} , and a convective heat transfer coefficient, h_{conv} , must be prescribed.



(a) Argon temperature inside the tent. (b) Fixture temperature.

Figure 4.5: Experimentally determined temperatures of the argon gas inside the tent and at the fixture, measured by thermocouples from the three experiments. Representative curves were constructed from the three experimental datasets to approximate the overall trends.

Based on experimental measurements, shown in Figure 4.5a, the bulk gas temperature showed minor variations, with some measurements showing slight increases of a few degrees while others remained approximately constant. The bulk gas temperature was therefore assumed to be constant $T_{Ar} = 25\text{ °C}$.

Only natural convection was considered at all convective boundaries, as the surrounding gas was assumed to be quiescent and no forced convection effects were included. An initial baseline value of $h_{\text{conv}} = 5 \text{ W/m}^2\cdot\text{K}$ was adopted. This choice is consistent with commonly reported values in the literature and lies within the typical range for natural convection [5, 17]. Additionally, the same value is frequently used in similar simulations performed at GKN [6]. Additionally, estimates of h_{conv} were obtained using empirical correlations for natural convection. The calculations were based on thermophysical properties evaluated at the film temperature, $T_{\text{film}} = (25^\circ\text{C} + 150^\circ\text{C})/2 = 87.5^\circ\text{C}$. The properties used in the calculations are summarized in Table 4.5.

Table 4.5: Thermophysical properties of argon evaluated at the film temperature, $T_{\text{film}} = 87.5^\circ\text{C}$, used in the empirical correlations for natural convection.

Property	Value
Density, ρ	1.36 kg/m ³
Specific heat, c_p	521 J/kgK
Thermal conductivity, k	0.0207 W/mK
Dynamic viscosity, μ	$2.64 \times 10^{-5} \text{ Pa} \cdot \text{s}$

Using empirical correlations for natural convection, the following values of h_{conv} were obtained:

- Wei [37]: $5.17 \text{ W/m}^2\cdot\text{K}$
- Lloyd [19]: $6.58 \text{ W/m}^2\cdot\text{K}$
- Martorell [23]: $5.08 \text{ W/m}^2\cdot\text{K}$

These results are all close to the initially assumed value of $h_{\text{conv}} = 5 \text{ W/m}^2\cdot\text{K}$, supporting its use as an initial estimate.

However, once the remaining parameters and boundary conditions had been established, the convective heat transfer coefficient was treated as a calibration parameter. The value of h_{conv} was varied between $1 \text{ W/m}^2\cdot\text{K}$ to $10 \text{ W/m}^2\cdot\text{K}$ to achieve agreement between the model predictions and experimental data, while also assessing the sensitivity of the thermal response to its variation.

4.4.3 Radiative Heat Transfer

Radiative heat transfer also occurs at all surfaces exposed to the surroundings. Therefore, a radiative boundary condition was applied to all external surfaces, except at the locations where the screws are in contact with the plate, where heat transfer is instead represented by a conductive heat transfer.

Radiative heat transfer occurs at surfaces exposed to the surroundings. In this work, radiation was incorporated through a linearized approximation, allowing it to

be combined with the convective boundary condition using an effective heat transfer coefficient.

$$q''_{\text{rad}} = \sigma \varepsilon [T_{\text{surf}}^4 - T_{\infty}^4] = \sigma \varepsilon [(T_{\text{surf}}^2 + T_{\infty}^2)(T_{\text{surf}} + T_{\infty})] (T_{\text{surf}} - T_{\infty}) \quad (4.2)$$

$$\begin{aligned} q''_{\text{effective}} &= q''_{\text{conv}} + q''_{\text{rad}} \\ &= h_{\text{conv}}(T_{\text{surf}} - T_{\infty}) + \sigma \varepsilon [(T_{\text{surf}}^2 + T_{\infty}^2)(T_{\text{surf}} + T_{\infty})] (T_{\text{surf}} - T_{\infty}) \\ &= \underbrace{(h_{\text{conv}} + \sigma \varepsilon [(T_{\text{surf}}^2 + T_{\infty}^2)(T_{\text{surf}} + T_{\infty})])}_{h_{\text{effective}}} (T_{\text{surf}} - T_{\infty}) \end{aligned} \quad (4.3)$$

The effective heat transfer coefficient was applied together with the prescribed bulk temperature of the surrounding argon.

However, the emissivity is not known and must therefore be treated as an uncertain parameter. It depends strongly on temperature and surface condition, such as oxidation and roughness. For metallic materials, emissivity can vary over a wide range: polished surfaces may exhibit values below 0.1, whereas oxidized or rough surfaces can exceed 0.8. However, emissivity values in the range 0.2 to 0.4 are commonly reported for Ti-6Al-4V under limited surface oxidation. This is also consistent with previous work at GKN, where an emissivity of 0.3 is commonly adopted.

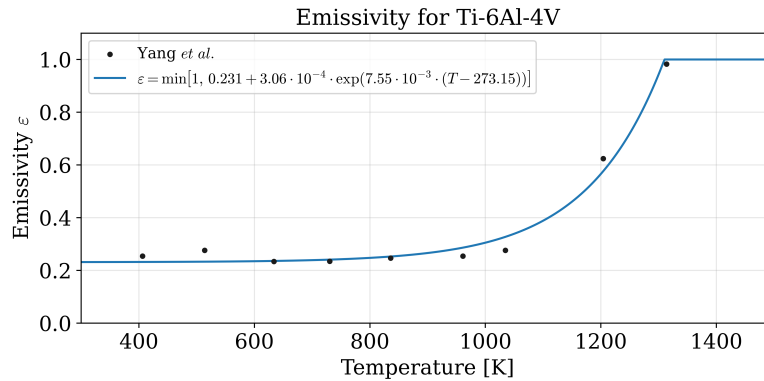


Figure 4.6: Emissivity curve for Ti-6Al-4V. The black markers are experimental data obtained from Yang et al. [38] and the blue curve is the equation fitted to the data.

Yang et al. [38] reported experimentally determined emissivity values for a process comparable to LMD-w. Their measurements showed that the emissivity remained relatively constant up to approximately 800 °C, after which it increased rapidly due to oxidation of the material surface. A curve fitted to their data is shown in Figure 4.6. Since the process studied by Yang et al. [38] did not include argon shielding, the strong increase in emissivity at elevated temperatures is not considered directly representative for the present work, where oxidation is expected to be limited. Furthermore, differences in process conditions and resulting surface finish may also lead to different emissivity values. Their measurements should therefore be regarded as indicative rather than directly transferable.

Based on the uncertainty of the emissivity value, it is treated as a calibration parameter and varied in the range 0.0 to 0.5 in the present study, including the curve fitted to the results of Yang et al. [38].

4.4.4 Conductive Heat Transfer

Heat transfer through solid-to-solid contact, such as the base plate-screw interface, was modeled using a formulation analogous to a convection boundary condition. In this approach, the convective heat transfer coefficient is replaced by a contact conductance, h_{screw} , and the heat transfer is defined relative to the temperature of the adjacent solid, T_{Fixture} .

From the experimental setup, a thermocouple was placed on the fixture onto which the screws are mounted. The measurements from this thermocouple show a continuous increase in temperature for all three experimental cases, as illustrated in Figure 4.5b. At the same time, slight differences in absolute temperature levels are observed between the individual experimental curves. A representative fixture temperature curve was therefore constructed based on the experimental measurements, capturing the overall temporal evolution during operation. This approach reduces the influence of experimental noise while preserving the thermal trend common to all three experiments. The resulting temperature curve was applied as the reference temperature T_{Fixture} for the conductive boundary condition.

The thermal contact conductance however, is not known from the experiments. As it depends on factors such as surface roughness, contact pressure, and material properties, it was therefore varied over a wide interval. Values in the range $500 \text{ W/m}^2\text{K}$ to $500\,000 \text{ W/m}^2\text{K}$ were considered in order to capture both poor and good thermal contact conditions.

In addition to fixture temperature and thermal contact conductance, the effective contact area must also be defined. This area is not reported in the experiments. However, since the clamping screws have pointed tips, the actual contact area is expected to be very small, although its exact value is unknown. As the computational domain is parameterized, the screw contact can be represented by a defined screw radius. The evaluated radii and corresponding contact areas are summarized in Table 4.6.

Table 4.6: Evaluated screw radii and corresponding contact areas.

Radius [mm]	Contact area [mm ²]
1	3.1
2	12.6
3	28.3
4	50.3

4.5 Results and Discussion

This chapter presents and discusses the simulation results. A mesh and time step sensitivity study is first performed to verify the numerical accuracy. This is followed by calibration of unknown model parameters and boundary conditions. Finally, the simulation results are evaluated through comparison with experimental data to assess the model performance.

4.5.1 Mesh and Time-Step Sensitivity Analysis

The sensitivity analysis was performed using fixed model parameters, including $\varepsilon = 0.3$, $h_{\text{screw}} = 500 \text{ W/m}^2\text{K}$, $A_{\text{screw}} = 28.3 \text{ mm}^2$ and $h_{\text{conv}} = 5 \text{ W/m}^2\text{K}$. The results of the sensitivity analysis are summarized in Table 4.7 and illustrated in Figure 4.7.

Table 4.7: Results for the mesh and timestep sensitivity study, normalized with the smallest mesh and smallest time step.

Base size [mm]	Cell count [-]	Time step laser off [s]	Time step laser on [s]	TC1 [%]	TC2 [%]	TC3 [%]	TC4 [%]	TC5 [%]	TC6 [%]
1	2 983 728	0.1	0.01	0.00	0.00	0.00	0.00	0.00	0.00
2	754 521	0.1	0.01	-0.10	0.22	-0.45	-0.19	-0.27	0.48
2.5	472 386	0.1	0.01	-0.11	-0.33	-1.05	0.05	0.30	0.06
5	118 703	0.1	0.01	0.54	-1.78	-1.30	-0.41	0.06	-0.46
10	31 472	0.1	0.01	0.60	-2.90	-0.49	-1.90	-1.98	-1.43
1	2 983 728	1	0.1	-0.06	0.09	-0.06	-0.01	-0.01	0.00
2	754 521	1	0.1	-0.14	0.17	-0.64	-0.24	-0.32	-0.35
2.5	472 386	1	0.1	-0.14	-0.37	-1.09	0.01	0.24	0.11
5	118 703	1	0.1	0.52	-1.95	-1.32	-0.46	0.01	-0.39
10	31 472	1	0.1	1.79	-1.91	0.63	-0.74	-1.27	-0.47
1	2 983 728	10	1	4.42	3.72	3.98	4.53	0.58	2.97
2	754 521	10	1	-1.00	-0.76	-1.53	-1.08	-2.08	-2.20
2.5	472 386	10	1	-1.29	-1.55	-2.25	-1.12	-1.70	-2.03
5	118 703	10	1	-0.62	-3.10	-2.48	-1.59	-2.09	-2.44
10	31 472	10	1	0.72	-2.99	-0.47	-1.80	-3.37	-2.29

The largest time step exhibits the lowest accuracy, with noticeable deviations relative to the reference solution obtained with the finest time step. For all mesh sizes except the finest, the largest time step consistently underpredicts the temperature, whereas the finest mesh results in an overprediction. This inconsistent behavior indicates that the time step is too large to adequately resolve the transient thermal response. In contrast, the difference between the intermediate and smallest time steps is minimal, and the results are nearly identical for all but the coarsest mesh. This suggests that the intermediate time step provides sufficient temporal resolution while maintaining significantly lower computational cost compared to the smallest time step. Based on

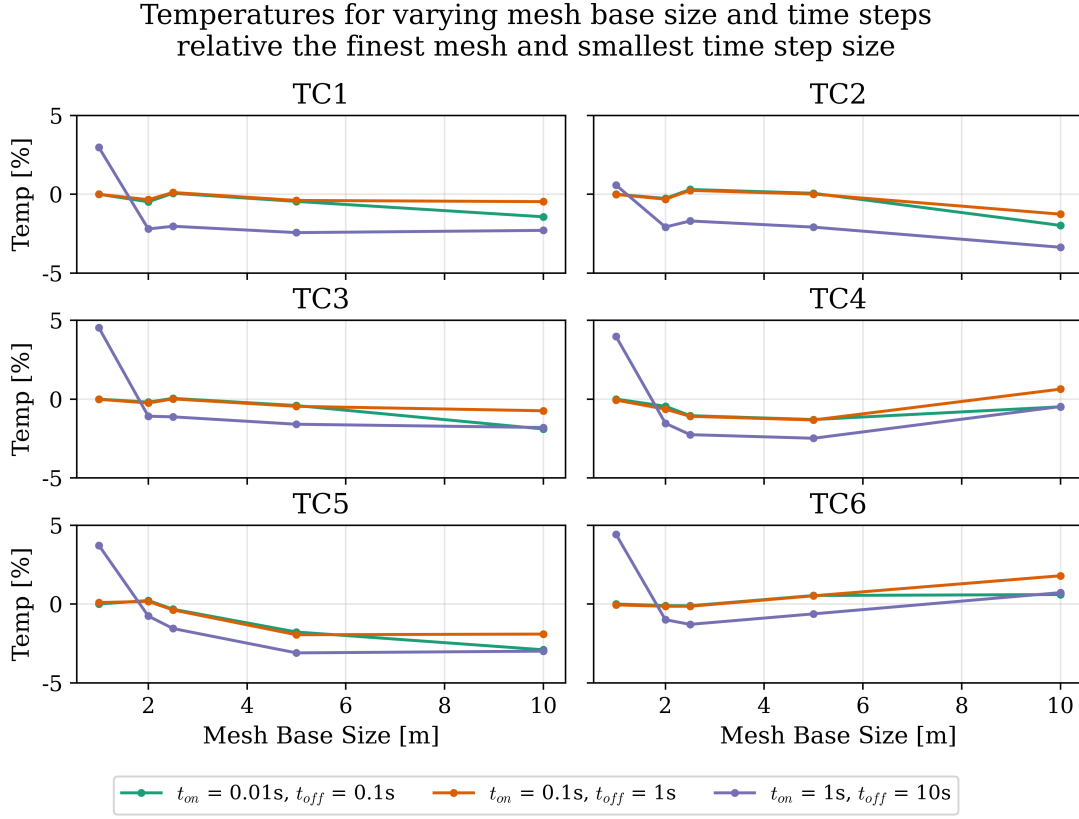


Figure 4.7: Mesh sensitivity results for different time step sizes. Three time-step combinations and 5 mesh sizes were investigated. The obtained temperatures at the six thermocouples TC1-TC6 indicate convergence with mesh and timestep refinement.

this, the intermediate time step of $t_{\text{laser on}} = 0.01\text{ s}$ and $t_{\text{laser off}} = 0.1\text{ s}$ was selected as a suitable compromise between accuracy and computational efficiency.

Considering only the smallest and intermediate time steps, the coarsest mesh exhibits the largest deviation, while the second coarsest mesh still shows a noticeable, though smaller, discrepancy. In contrast, the difference between the intermediate mesh resolutions is negligible. Further refinement to the finest mesh results in only marginal changes in the predicted temperature field, while substantially increasing the number of cells and, consequently, the computational cost. The selection therefore lies between the second finest and third finest mesh resolutions. As the difference between these two is minimal, the third finest mesh, with a base cell size of 2.5 mm, was selected as a suitable compromise between accuracy and computational efficiency.

4.5.2 Sensitivity Analysis and Calibration

This section presents a sensitivity analysis of the boundary condition parameters, where key variables are varied to assess their influence on the simulation results. Based on this, the parameters are calibrated against experimental data to obtain a

representative model of the thermal behavior.

4.5.2.1 Evaluation Criteria

The performance of the developed thermal model was evaluated by comparison with experimental temperature measurements. With data available at six discrete thermocouple locations, both the overall error magnitude and its spatial distribution were considered.

To quantify the overall agreement between simulated and measured temperatures, the Root Mean Square Error (RMSE) was used. RMSE provides a measure of the average prediction error and is defined as

$$\text{RMSE} = \sqrt{\frac{1}{N} \sum_{i=1}^N (T_i^{\text{sim}} - T_i^{\text{exp}})^2} \quad (4.4)$$

where N is the number of measurement points, and T_i^{sim} and T_i^{exp} are the simulated and experimental temperatures at location i , respectively.

While RMSE captures the average temperature error, it does not provide information about the spatial distribution of the error. Certain parameters primarily affect the overall temperature level without significantly changing the spatial pattern. For example, variations in convective boundary conditions tend to shift temperatures uniformly, whereas localized effects, such as heat conduction through the clamping screws, can alter the spatial distribution even if the mean error remains moderate. To account for this, the standard deviation of the temperature error was introduced as a complementary evaluation metric, defined as

$$\sigma_e = \sqrt{\frac{1}{N} \sum_{i=1}^N (e_i - \bar{e})^2} \quad (4.5)$$

where N is the number of measurement points, $e_i = T_i^{\text{sim}} - T_i^{\text{exp}}$ is the temperature error at each measurement location, and $\bar{e}$ is the mean temperature error. A low value indicates a uniform error across all thermocouples, whereas a high value suggests localized discrepancies.

By combining RMSE and standard deviation, it becomes possible to distinguish between global temperature offsets and incorrect spatial behavior, providing a more robust basis for model evaluation and parameter selection.

4.5.2.2 Radiation

Five simulations were conducted in which the emissivity was varied while all other parameters were kept constant, including $h_{\text{screw}} = 500 \text{ W/m}^2\text{K}$, $A_{\text{screw}} = 28.3 \text{ mm}^2$ and $h_{\text{conv}} = 5 \text{ W/m}^2\text{K}$. For each case, the temperature was evaluated at the six thermocouple locations. The results for all simulations and thermocouples are presented in Figure 4.8 and in Table 4.8, where the corresponding RMSE values are also included.

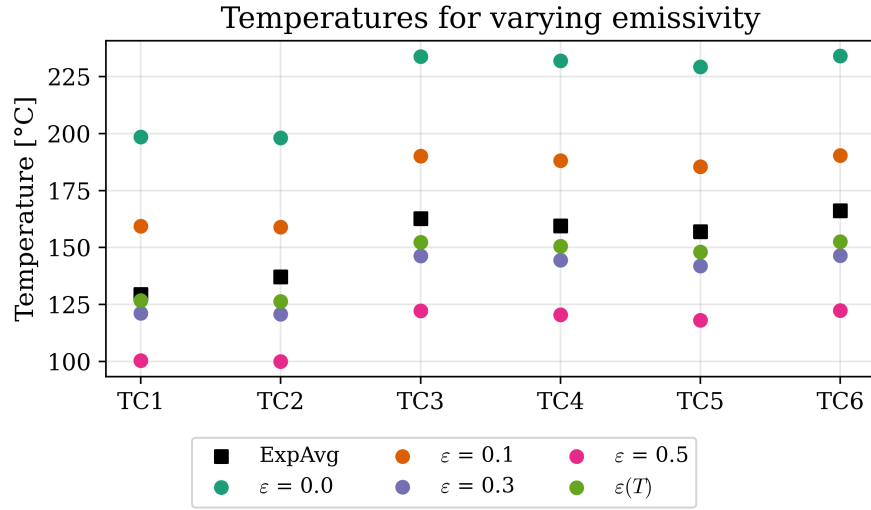


Figure 4.8: Temperatures at TC1-TC6 at the end of the test for different emissivity values, illustrating the effect of radiation.

Table 4.8: Simulated thermocouple temperatures for different emissivity values, presented relative to experimental measurements at the end of the test.

TC	ΔT [°C]				
	$\varepsilon = 0.0$	$\varepsilon = 0.1$	$\varepsilon = 0.3$	$\varepsilon = 0.5$	$\varepsilon(T)$
1	+69.1	+29.9	-8.3	-29.0	-2.7
2	+60.9	+21.8	-16.5	-37.1	-10.8
3	+71.2	+27.5	-16.4	-40.5	-10.3
4	+72.4	+28.7	-15.1	-39.0	-9.0
5	+72.4	+28.5	-15.0	-38.8	-8.9
6	+67.9	+24.2	-19.6	-43.8	-13.6
RMSE	69.1	26.9	15.5	38.3	9.8

The results show that the temperature-dependent emissivity, $\varepsilon(T)$, and the constant emissivity $\varepsilon = 0.3$ give the closest agreement with the experimental data. The slightly better agreement obtained with $\varepsilon(T)$ can likely be explained by the fact that the function yields emissivity values around 0.25 over most of the relevant temperature range. Temperatures above 800 °C occur only locally in the melt zone during active heating and therefore have limited influence on the overall thermal response.

However, the results should not be interpreted independently as the parameters held constant in this study were not yet calibrated to experimental data. Radiation and convection both influence the heat losses in a similar manner, meaning that different combinations of emissivity and convective heat transfer coefficient h_{conv} can yield similar temperature predictions. If one parameter is fixed, the other can be adjusted to compensate, which introduces a non-uniqueness in the calibration.

The parameter sweep should instead be viewed as an evaluation of the sensitivity to emissivity, rather than an attempt to determine a unique physical emissivity value. Increasing the emissivity from $\varepsilon(T) \approx 0.25$ to $\varepsilon = 0.3$ changes the predicted temperature by less than 8 °C, whereas neglecting radiation entirely ($\varepsilon = 0.0$) results in a deviation of nearly 85 °C. This shows that radiation is an important heat transfer mechanism in the process and cannot be neglected, while the exact emissivity value within a reasonable range has a comparatively smaller influence.

Given this limited sensitivity and the coupling to other parameters, the choice of emissivity is not solely based on minimising RMSE in this calibration step. Instead, $\varepsilon = 0.3$ was selected as a consistent value. This choice is supported by its use in the pyrometer measurements as well as in previous work performed at GKN. To determine emissivity with higher accuracy, dedicated measurements accounting for the surface condition of the deposited material would be required. However, this is outside the scope of the present work. Fixing $\varepsilon = 0.3$ therefore provides a practical and robust basis for subsequent calibration of the remaining model parameters.

4.5.2.3 Screw Conduction

The influence of heat conduction through the clamping screws was evaluated by varying both the contact area and the thermal contact conductance, shown in Figure 4.9. To give a better overview of this behavior, the RMSE for all cases is shown in Figure 4.10 where the standard deviation of the error is also included.

The results show that lower values of both parameters result in the closest agreement with the experimental data. However, a closer examination of the temperature distribution reveals that this is not necessarily the case. For the combination with small contact area and low contact conductance, which yields the lowest RMSE, the thermocouples closest to the screws show temperatures similar to or even higher than the experimental measurements. At the same time, thermocouples located further away from the screws exhibit significantly lower temperatures than the experimental data. This indicates a spatially non-uniform error, suggesting that this parameter combination does not accurately represent the physical behavior.

This highlights the importance of considering not only the overall error magnitude but also its spatial distribution. Heat conduction through the screws is a localized effect, acting primarily at the contact regions in the corners. If this mechanism is incorrectly described, the largest deviations are expected near the screws, resulting in a non-uniform error pattern. In contrast, convection acts over the entire surface and therefore introduces a more uniform temperature shift. If convection is incorrectly defined, all thermocouples tend to exhibit similar deviations, corresponding to a global temperature offset.

Based on this, the standard deviation of the error was used as a complementary metric to identify parameter sets with a physically reasonable spatial behavior. Three parameter combinations were found to yield low standard deviation:

- $A_{Screw} = 50.3 \text{ mm}^2$ and $h_{Screw} = 500 \text{ W/m}^2\text{K}$

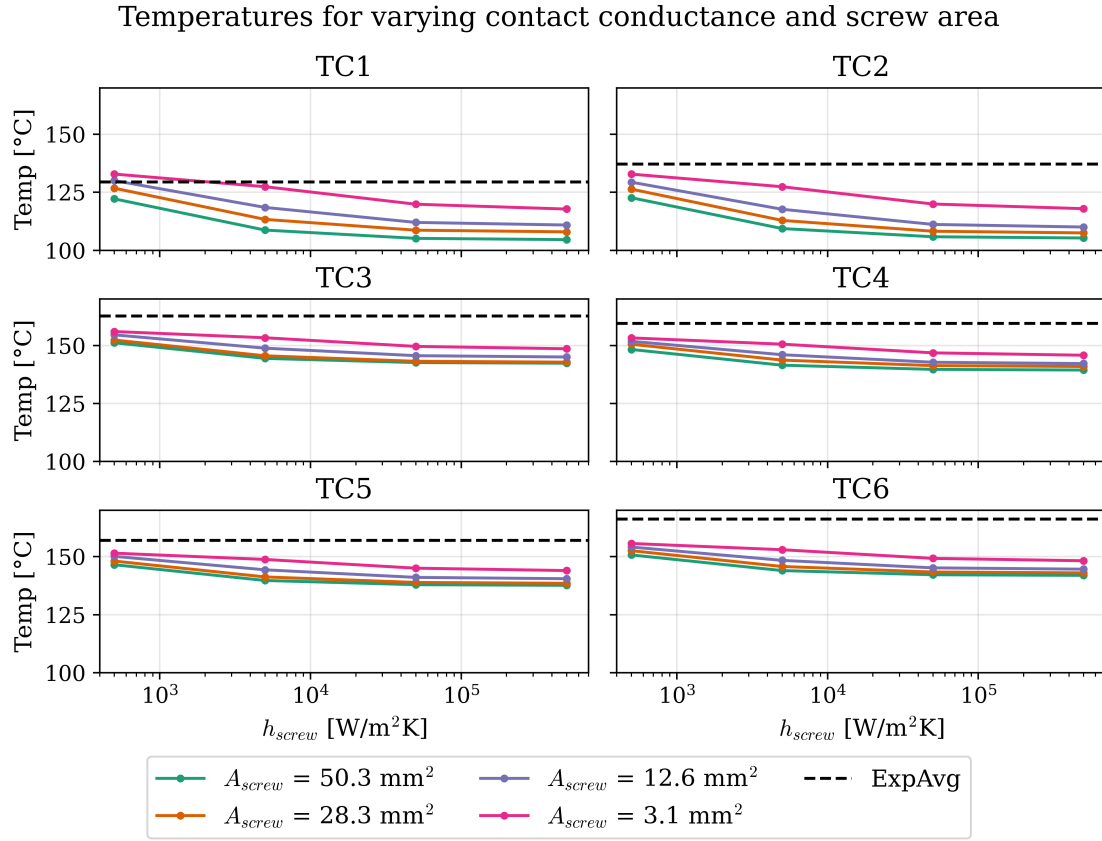


Figure 4.9: Temperatures at thermocouples TC1-TC6 at the end of the simulation, for different screw contact conductances and contact areas. The contact area is indicated by color, while the contact conductance is shown along the x-axis.

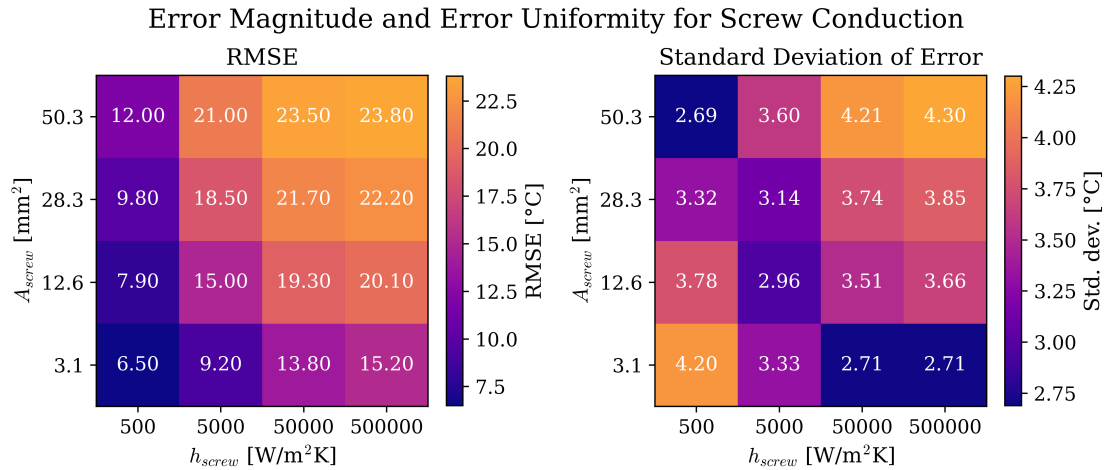


Figure 4.10: Comparison of the root mean square error (RMSE) and the corresponding standard deviation of the temperature error for each parameter combination. The RMSE is shown in the left subplot, while the standard deviation is presented in the right subplot.

- $A_{Screw} = 3.1 \text{ mm}^2$ and $h_{Screw} = 50\,000 \text{ W/m}^2\text{K}$
- $A_{Screw} = 3.1 \text{ mm}^2$ and $h_{Screw} = 500\,000 \text{ W/m}^2\text{K}$

The combination with the largest contact area and lowest contact conductance was not considered physically representative since the screw connections are expected to be tightly clamped, resulting in a high local contact pressure, and consequently a high thermal contact conductance. Among the two remaining combinations, both yielded similar temperature predictions and comparable standard deviations. This indicates that the model response is relatively insensitive to further increases in the conductance. Between these, the combination $A_{Screw} = 3.1 \text{ mm}^2$ and $h_{Screw} = 50\,000 \text{ W/m}^2\text{K}$ was selected. While both candidates provide similar agreement with the experimental data, the lower value of h_{Screw} results in a slightly reduced RMSE. This choice is therefore considered a reasonable balance between physical plausibility and model accuracy.

Despite this, a consistent bias in the predicted temperature levels remains, indicating that other model parameters, most likely convection, are not yet fully calibrated.

A key observation is the magnitude of the temperature variation associated with different screw configurations. Comparing the selected configuration with the case of lowest contact conductance yields temperature differences of $\approx 10^\circ\text{C}$. Furthermore, when comparing the lowest and highest parameter combinations, differences of nearly 30°C are observed at thermocouples closest to the screws (TC1 and TC2). While the latter case is not physically realistic, it highlights the strong influence of the screw boundary condition on the local temperature field.

These findings demonstrate that the modeling of screw connections has a non-negligible impact on the predicted temperatures. Consequently, neglecting the screws entirely would introduce a substantial error in the model. The results therefore indicate that it is important to include the screws in order to obtain a physically representative thermal model, despite their small size.

4.5.2.4 Convection

In the previous sections, the radiative heat transfer has been defined and fixed. The remaining uncertain heat loss mechanism is therefore convection, which is governed by the convective heat transfer coefficient, h_{Conv} . To calibrate this parameter, five simulations were performed where the h_{Conv} was varied. For each case, the temperature was evaluated at the five thermocouple locations. The results for all simulations and thermocouples are presented in Table 4.9, where the corresponding RMSE values are also included.

To determine a representative value of h_{Conv} , a polynomial fit was applied to the simulated temperatures as a function of h_{Conv} for each thermocouple. This results in a relation where h_{Conv} is treated as the independent variable and the corresponding thermocouple temperature as the dependent variable. For each thermocouple, the HTC value that yields the experimental temperature was then identified from the

Table 4.9: Simulated thermocouple temperatures for varying heat transfer coefficients for convection, h_{Conv} , presented relative to experimental measurements at the end of the test.

TC	ΔT [°C]				
	$h_{Conv} = 1.0$	$h_{Conv} = 2.0$	$h_{Conv} = 4.0$	$h_{Conv} = 5.0$	$h_{Conv} = 10.0$
1	+37.9	+21.9	-4.0	-14.4	-48.6
2	+30.2	+14.2	-11.8	-22.1	-56.4
3	+40.1	+22.5	-6.7	-18.6	-59.2
4	+40.5	+22.8	-6.4	-18.3	-58.7
5	+41.3	+23.5	-5.7	-17.6	-57.9
6	+36.2	+18.6	-10.6	-22.4	-63.0
RMSE	37.9	20.8	8.0	19.1	57.5

fitted curve. The final value was obtained by taking the average of these individually determined values across all thermocouples. This is illustrated in Figure 4.11 and in Table 4.10.

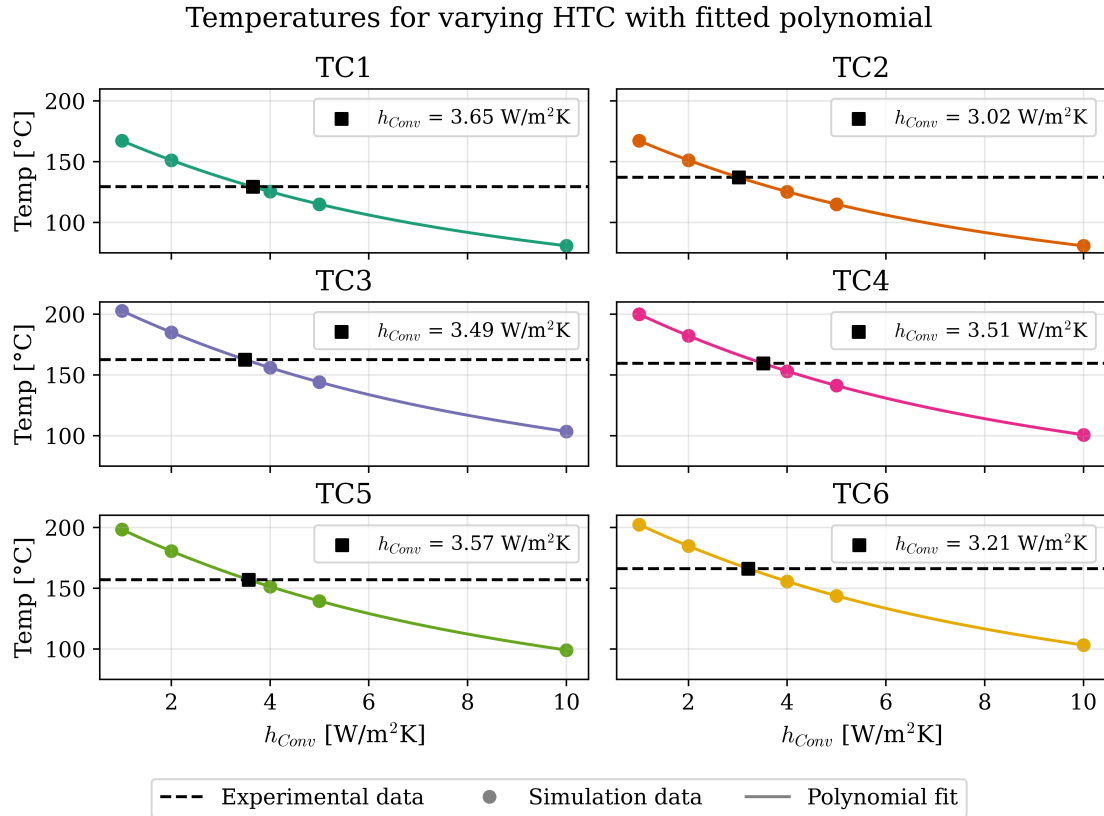


Figure 4.11: Measured data points at thermocouples TC1-TC6 from parametric sweep when HTC for convection was varied. A polynomial fit, shown in Table 4.10, is also illustrated as well as calculated values of h_{Conv} .

The results show a clear sensitivity of the predicted temperature to the choice of

Table 4.10: Summary of polynomial fits performed for each thermocouple for varying heat transfer coefficient h_{Conv} . For each thermocouple, a corresponding calculated value of h_{Conv} was determined such that the polynomial representation matches the experimentally measured temperature of TC1-TC6. An average value of h_{Conv} was then determined.

TC	T [°C] Exp	Fitted polynomial	Calculated h_{Conv} [W/m ² K]
TC1	129.4	$T = 0.0003x^4 - 0.042x^3 + 1.3x^2 - 19.6x + 186$	3.65
TC2	137.2	$T = 0.00029x^4 - 0.042x^3 + 1.3x^2 - 19.6x + 186$	3.02
TC3	162.7	$T = -0.00004x^4 - 0.033x^3 + 1.27x^2 - 21.2x + 223$	3.49
TC4	159.5	$T = 0.00003x^4 - 0.035x^3 + 1.29x^2 - 21.4x + 220$	3.51
TC5	157.0	$T = 0.00006x^4 - 0.036x^3 + 1.31x^2 - 21.4x + 218$	3.57
TC6	166.2	$T = -0.00003x^4 - 0.034x^3 + 1.27x^2 - 21.3x + 222$	3.21
Average			3.41

h_{conv} , where increasing h_{conv} leads to lower temperatures due to enhanced convective heat losses. Compared to the radiation study, variations in h_{conv} have a more pronounced effect, indicating that convection plays a significant role in the overall heat balance.

Based on the results, a calibrated value of $h_{conv} = 3.41 \text{ W/m}^2\text{K}$ was selected for subsequent work. Although this value is lower than the estimates obtained from the empirical correlations, it remains within the range of values reported in the literature for natural convection. The calibrated value can therefore be considered physically reasonable despite the discrepancy with the correlation-based predictions.

4.5.3 Assessment of Transient Behavior

All previous simulations were based on a constant-geometry assumption using the final build geometry consisting of all 17 layers (equivalent to 34 weld passes), meaning that the complete geometry was present from the start of the simulation. In those cases, the transient simulation was run through the entire deposition sequence and the thermal response was primarily evaluated at the final build state.

To assess whether the applied boundary conditions remain valid throughout the build process, an additional study was performed in which the same constant-geometry approach was applied to multiple intermediate build geometries. In total, 17 separate simulations were conducted, each corresponding to a fixed geometry containing between 1 and 17 deposited layers. For each case, the selected geometry remained unchanged throughout the simulation.

Each simulation was initialized at $t = 0 \text{ s}$ using identical initial conditions, but with the geometry corresponding to the selected number of layers. Each layer was modeled as two consecutive weld passes, where each pass consisted of a deposition period followed by a cooling pause, resulting in a weld-pause-weld-pause sequence

for each layer. The simulation was then run transiently only up to the deposition time associated with the selected number of layers, after which the results were extracted. The total simulation time for a given case is therefore

$$t = (t_{\text{weld}} + t_{\text{pause}}) \times 2 \times n_{\text{layers}}, \quad (4.6)$$

where the factor 2 accounts for the two weld passes per layer.

Figure 4.12 shows the resulting temperature distributions for selected cases (2, 7, 12, and 17 layers) at their respective final times. Since thermal data was extracted only at the end of each simulation, the detailed temperature evolution within individual weld and cooling cycles was not captured.

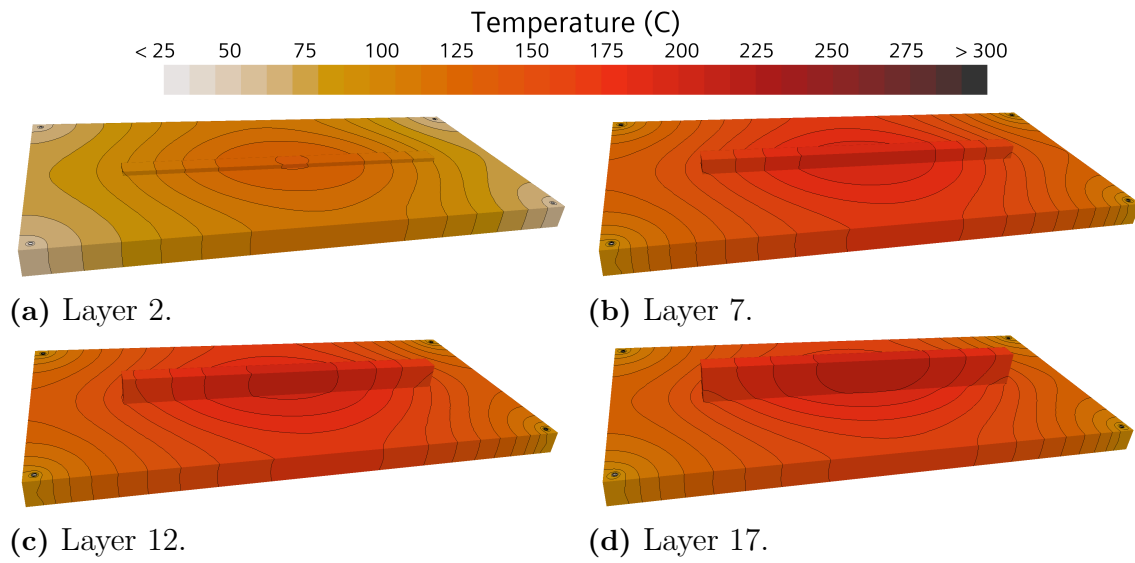


Figure 4.12: Temperature contours at selected time points for increasing number of layers, with time defined as $t = (t_{\text{weld}} + t_{\text{pause}}) \times 2 \times n_{\text{layers}}$.

The resulting temperature evolution over time is shown in Figure 4.13. In addition, the average heat transfer over one cycle, for each simulation, is presented in Figure 4.14. The average heat input during one cycle is defined as

$$\dot{Q}_{\text{Avg heat input}} = \frac{P_{\text{heat source}} t_{\text{weld}}}{t_{\text{weld}} + t_{\text{pause}}}. \quad (4.7)$$

The average heat losses are obtained by averaging the instantaneous heat transfer over the final cycle for each simulation, resulting in a corresponding heat loss rate.

From Table 4.11, the relative contributions of the different heat transfer mechanisms can be evaluated. Convection is the dominant mode, consistent with its strong influence on the temperature when the heat transfer coefficient was varied. Radiation is the second largest contribution and remains important to include, as neglecting radiation leads to significantly underestimated temperatures. Heat transfer through the screws accounts for approximately 15%, confirming their non-negligible impact

Table 4.11: Results from the 17 layer-specific simulations, evaluated at the final simulation time, including temperature data at the thermocouple locations. The table also shows normalized average heat losses, normalized by the average heat input, and the corresponding contributions from each heat loss mechanism.

Layer	Temperatures [°C]						Avg heat losses [W]	Conv- ection	Radi- ation	Cond- uction
	TC1	TC2	TC3	TC4	TC5	TC6				
1	54.7	54.2	78.1	76.2	74.7	77.7	0.22	58%	29%	13%
2	84.7	83.9	115.4	113.6	112.2	115.1	0.44	53%	31%	15%
3	106.1	105.8	139.3	137.5	136.0	139.0	0.62	51%	33%	16%
4	120.4	120.9	154.7	152.8	151.4	154.4	0.75	49%	35%	16%
5	129.5	130.4	164.3	162.5	161.0	164.0	0.84	48%	35%	16%
6	135.3	136.2	170.2	168.2	166.8	169.8	0.90	48%	36%	16%
7	138.5	139.9	173.9	172.2	170.7	174.0	0.94	48%	36%	16%
8	139.9	141.9	175.4	174.0	171.9	176.0	0.97	47%	37%	16%
9	141.5	142.2	175.8	174.5	172.1	175.8	0.98	48%	37%	16%
10	141.7	142.9	175.3	174.2	172.0	175.9	0.99	48%	37%	16%
11	141.0	142.1	175.5	173.1	171.7	174.8	0.99	48%	37%	16%
12	140.9	141.9	173.9	171.8	170.3	174.4	0.99	48%	37%	15%
13	140.7	140.1	173.7	171.2	169.4	173.4	1.00	48%	37%	15%
14	139.0	139.3	173.0	170.0	167.8	171.7	1.00	48%	37%	15%
15	138.8	137.1	171.3	168.6	167.4	171.1	1.00	48%	37%	15%
16	136.6	136.9	169.8	167.2	165.3	169.6	1.00	48%	37%	15%
17	136.1	135.8	168.7	165.9	164.1	168.4	1.00	48%	37%	15%

observed in previous sections. Overall, all heat transfer modes contribute meaningfully and must be accurately modeled to capture the thermal behavior of the solid.

The simulated temperature evolution shows strong agreement with experimental trends at all thermocouple locations, indicating that the model setup and boundary conditions provide a consistent representation of the thermal behavior.

However, it should be noted that each simulation assumes a constant geometry over time. While the correct amount of material is represented at the final simulation time for a given number of layers, this is not the case during the earlier part of the simulation, where the domain contains more material than is physically present in the experiment. This introduces a simplification, as it affects the thermal history and heat accumulation.

The impact of this depends strongly on the thermal regime. During the early stages of the build, approximately from 1 to 9 layers, the system undergoes a significant heat build-up, where the heat input exceeds the heat losses, as shown in Figure 4.14. In this regime, the temperature evolution is highly transient and strongly influenced by the thermal history. Since the simulations overestimate the thermal mass throughout the entire simulation time (from $t = 0$ s to the layer-specific end-time), the temperature increase becomes slower, which leads to deviations from the experimental behavior even if the correct number of layers is represented at the final time.

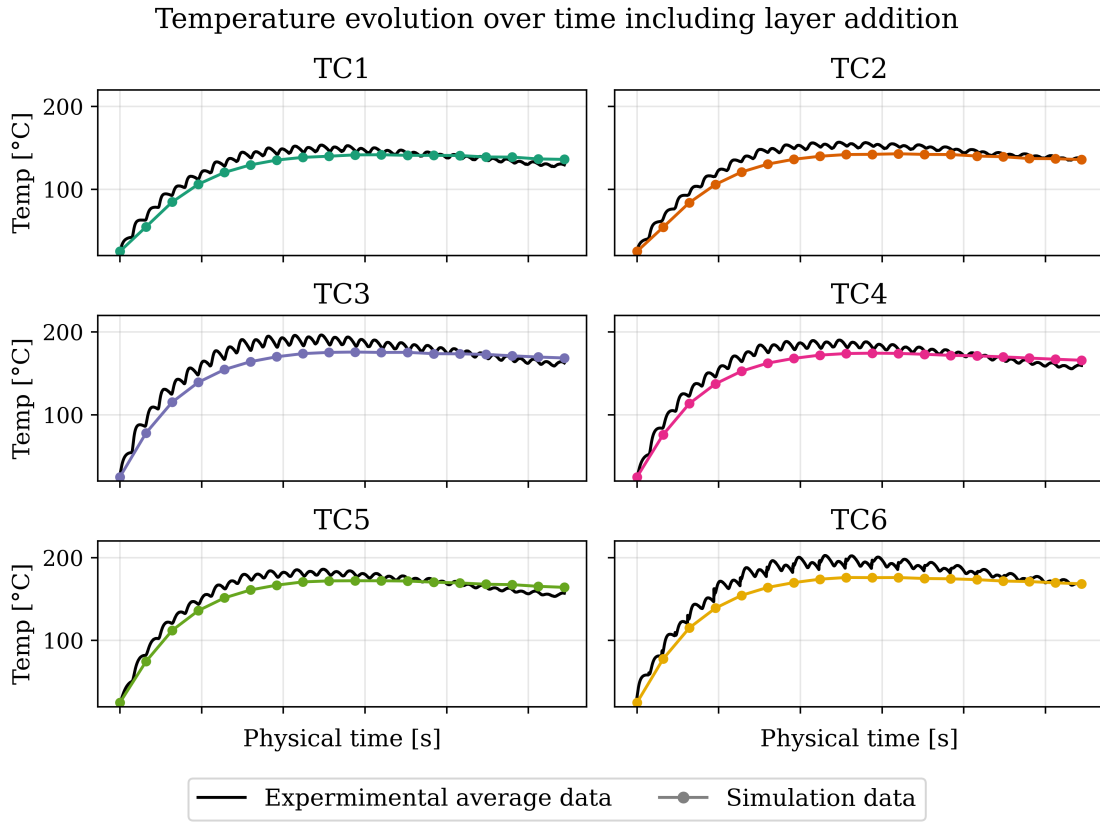


Figure 4.13: Comparison of simulated and experimental temperature evolution over time at all thermocouple locations for the 17 layer configurations.

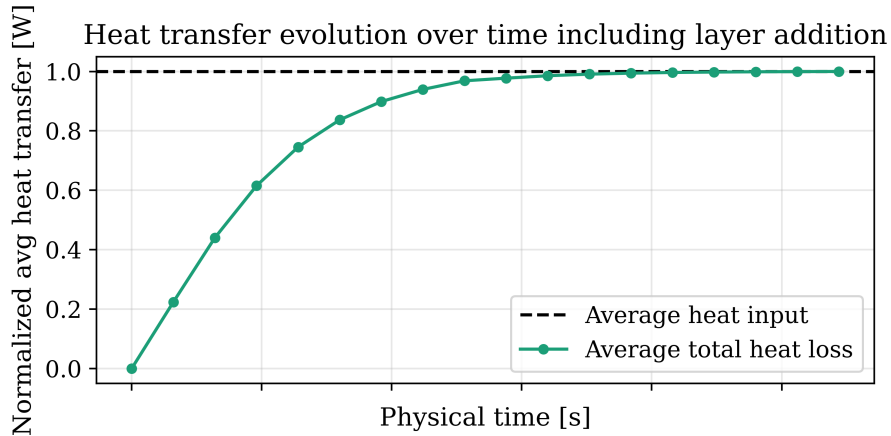


Figure 4.14: Comparison of average heat transfer during one cycle of both heat addition through the heat source and heat losses through convection, radiation and conduction, for the 17 layer configurations.

In contrast, during the later stages of the build, approximately from 10 layers, the system approaches a quasi-steady thermal state, where the heat input and losses are in balance. For a given geometry, this corresponds to a more stable temperature

level, which is primarily governed by the total amount of material rather than the transient heat accumulation. As a result, if the simulation is run for a sufficiently long time, the solution will approach the correct temperature level for that geometry, making it less sensitive to the earlier thermal history. This indicates that the assumption of constant geometry, as used in the model calibration, is valid for the later stages of the process, since the system has effectively reached thermal equilibrium.

A more physically accurate approach would involve a fully coupled layer-by-layer simulation, where material is progressively added and the solution is transferred between simulation steps. However, such an approach is computationally demanding and was therefore not considered feasible within the scope of this work.

Despite this simplification, the steady-state simulations are considered to provide a reasonable approximation of the thermal behavior, particularly in the later stages of the process, which are of primary interest in this study. The model is therefore considered sufficiently calibrated and suitable for analyzing the LMD-w process.

5

Assessment of the Validity of Steady-State Modeling for an Industrial Drum

This chapter focuses on the real drum case, which represents the primary geometry investigated in this work and corresponds to the actual component currently being developed at GKN using the LMD-w process. The component consists of a cylindrical drum (annular cylinder) with an outer diameter of 568.5 mm, an inner diameter of 518.5 mm, a wall thickness of 25 mm, and a total height of 120 mm.

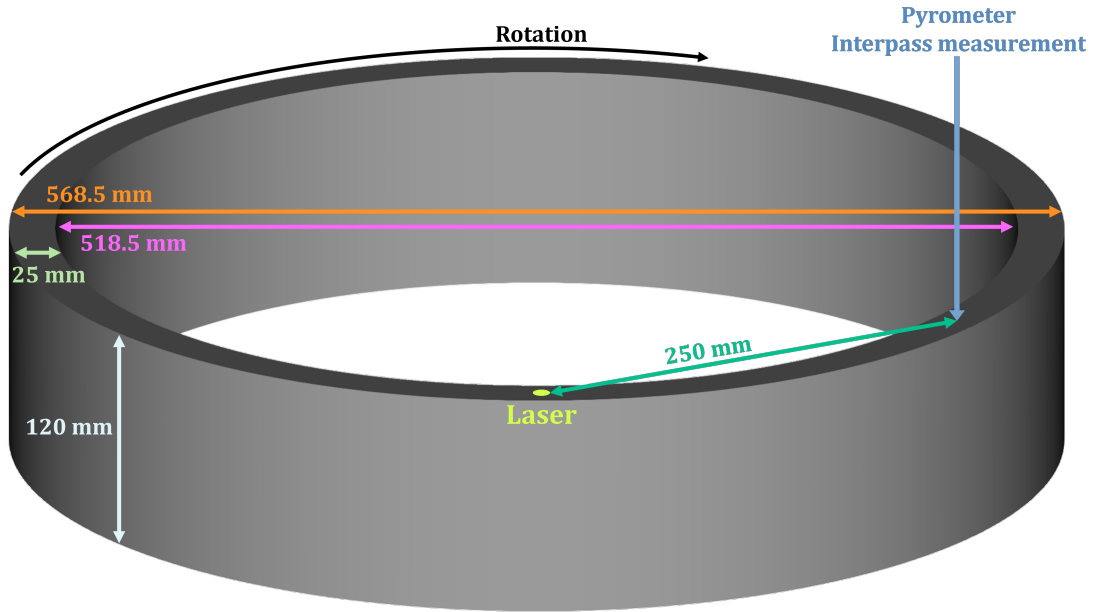


Figure 5.1: Geometry for the drum with dimensions given in mm. The rotational direction, laser location and interpass measurement location are also shown.

The laser is positioned at the center of the surface where material deposition occurs. At a distance of 250 mm from the laser center, a pyrometer is used to measure the temperature at the interpass location. The interpass temperature refers to the surface temperature measured a short distance upstream of the laser, prior to laser interaction. This measurement point provides the experimental temperature data used for comparison with the numerical results. In this case, the laser, and thus the

entire printing machine, is stationary in the room, while the cylindrical drum rotates at a constant angular velocity in the clockwise direction. A geometry overview is illustrated in Figure 5.1.

The available experimental data for the real domain consists of temperature measurements at a single spatial location, recorded by the pyrometer at three discrete time instances during continuous operation, shown in Table 5.1. In the following chapter, all measured temperatures are normalized relative to the temperature measured after 39 minutes. While these measurements indicate a temperature increase over time, they do not provide sufficient information to determine the underlying thermal evolution of the system, as illustrated in Figure 5.2.

In addition, measuring high temperatures using a pyrometer is associated with significant measurement uncertainty. Factors such as surface emissivity, reflections, and environmental conditions can strongly influence the measured values. As a result, the experimental temperature data can be assumed to vary by up to approximately 18% lower relative to the temperature after 39 minutes. This substantial measurement uncertainty directly affects how simulation results can be validated against experimental data.

Table 5.1: Experimentally determined interpass temperatures from pyrometer, normalized with the temperature after 39 minutes.

Time [min]	Time [s]	Normal- ized Temp [-]
0	0	0.54
20	1200	0.91
39	2340	1

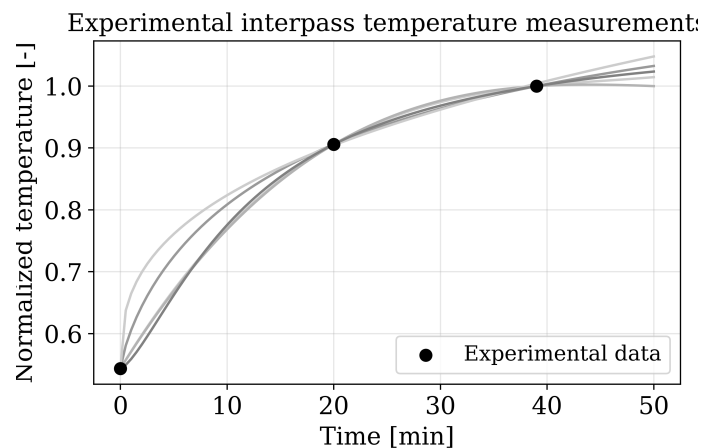


Figure 5.2: Normalized experimental data at the interpass location and multiple possible temperature evolutions representing different heat buildup scenarios.

The limited temporal resolution, however, makes it difficult to determine whether the observed temperature increase represents a continuous heat buildup or whether the system approaches a thermally stabilized condition after a certain operating time. This uncertainty is central when deciding whether a steady-state thermal model can provide a meaningful representation of the system. From a modeling perspective, transient simulations offer insight into the time-dependent temperature evolution. However, resolving more than 30 minutes of physical time leads to a significant computational cost, a limitation that becomes even more pronounced when considering future coupled CFD-CHT simulations. At the same time, the primary

objective of the present work is to assess how the surrounding flow conditions affect the thermal response in the solid. From this perspective, the asymptotic temperature distribution is of particular interest. If the system can be reasonably assumed to approach a thermally stabilized state, steady-state simulations provide a direct and computationally efficient way to compare different flow conditions.

The following key questions are therefore addressed in this chapter:

- Can the available measurements be used to determine whether the temperature has stabilized?
- Is it reasonable to assume that a converged steady-state temperature distribution can represent the long-term thermal behavior of the domain?
- Given the computational cost of transient simulations, is steady-state modeling sufficient to capture the relevant thermal behavior?

The remainder of this chapter therefore focuses on evaluating whether a steady-state thermal representation can be considered a meaningful and informative approximation for the real domain, given the available experimental data and the objectives of the study.

For this domain, new data and a new set of process parameters was provided, but due to confidentiality, the exact values cannot be disclosed. However, all parameters used in the simulations are representative of the actual process conditions, and an overview of the variables used in the simulations are shown in Table 5.2.

Table 5.2: Process parameters for the drum.

Parameter	Value	Parameter	Value
Printing velocity, v_{traverse}	- mm/s	Wire diameter, d_{wire}	- mm
Laser Power, P_{laser}	- kW	Wire feeding velocity, v_{wirefeed}	- m/min
Laser Diameter, d_{laser}	- mm	Wire feed rate, $\dot{m}_{\text{wire}}$	- kg/s
Laser Absorption, η_{laser}	-	Wire heating power, P_{wire}	- W
		Wire heating efficiency, η_{wire}	-

5.1 Solver Settings

The simulations were performed using the segregated solid energy model. The solver settings were largely identical to those used in the transient calibration case, ensuring consistency between the calibration case and thermal modeling of the drum.

The under-relaxation factor (URF) for the energy equation was increased to 0.9999 [7]. This choice was based on the observed stable and smooth convergence of the energy residuals. Increasing the URF reduced the number of iterations required to reach convergence by approximately a factor of two, by allowing larger updates per iteration without introducing instability.

Furthermore, a coupled solid energy solver was also tested. The results were found to be identical to those obtained using the segregated solver, indicating that the thermal solution in the solid domain is independent of the solver formulation for the present case. However, since the segregated approach provided faster and more robust convergence, it was used for subsequent simulations.

5.1.1 Transient Simulation

The rotational motion is defined by selecting a rotation axis passing through the centre of the drum, coinciding with the geometrical symmetry axis of the cylinder. A constant angular velocity is then prescribed using rigid-body motion, with the specific rotation rate.

Convergence was assessed using the same stopping criteria as in the calibration case. An asymptotic stopping criterion based on the energy residual was applied, with a normalized tolerance of 0.05 over 10 successive iterations. A maximum of 100 inner iterations per time step was also used as a secondary stopping criterion, to ensure that the solver advances in time even in cases where the residual convergence is slow. Similarly to the calibration case, the simulations converged rapidly, with the asymptotic stopping criterion being the primary condition satisfied.

A constant time step was used in the transient simulations, as the laser remained continuously active. Different time step sizes were initially evaluated, and a value of 0.05 s was selected based on the observed convergence behavior. A second-order temporal discretization scheme was employed to improve the accuracy of the transient solution.

5.1.2 Steady-State Simulation

The *Convective Velocity option* in STAR-CCM+ was used to account for energy transport in solid regions undergoing rigid-body motion, such as rotation or translation [32]. As shown in the solid energy equation, see Equation (2.30), heat conduction is a diffusive process and the default formulation does not include a convective (advection) term. Consequently, if a solid is moving and the simulation is performed as steady-state, the effect of this motion on the temperature field would be neglected. By enabling the *Convective Velocity option*, an additional advection term based on a prescribed translational and/or rotational velocity is introduced into the solid energy equation, allowing the transport of thermal energy due to solid motion to be captured consistently. This approach can be regarded as the solid-domain analogue of a Moving Reference Frame (MRF) formulation. However, the classical MRF approach cannot be applied in the present case, since it is defined for fluid regions and is based on the governing momentum and energy equations of a flowing fluid.

Convergence for the steady-state simulations was also assessed the asymptotic stopping criterion based on the energy residual. However due to fast simulation times, the normalized tolerance was decreased to 0.01 over 10 successive iterations. In addition, a second asymptotic stopping criterion was defined based on the monitored

interpass temperature, using a normalized tolerance of 1×10^{-6} over 5 samples. Both criteria were required to be satisfied before the simulation was terminated. This ensures not only that the residuals have stabilized, but also that the physical temperature response has reached a steady-state. The use of both residual- and solution-based criteria provides a more robust measure of convergence. A maximum of 200 iterations was imposed to prevent excessive computation in the event of non-convergence. However, this limit was not reached, as all simulations converged within approximately 50-100 iterations.

5.2 Boundary Condition Treatment

Given the limited availability of experimental data for the current case, a new calibration would be highly uncertain. Therefore, the boundary conditions in this study are based on results from the simple calibration domain, where key parameters such as the heat transfer coefficient and emissivity were established. Rather than recalibrating, these results are applied together with the given process parameters to define physically reasonable boundary conditions. All parameters are kept fixed throughout the study to ensure consistency. A summary of the applied boundary conditions is provided in Table 5.3.

Table 5.3: Summary of the prescribed thermal boundary conditions applied to the drum in the solid-only simulation.

Heat transfer mechanism	Variable	Value
Convection	Temperature Argon, T_{Argon}	60 °C
	Convective heat transfer coefficient, $h_{\text{Convection}}$	3.41 W/m ² K
Radiation	Temperature Argon, T_{Argon}	60 °C
	Emissivity ε	0.3
Conduction	Normalized temperature drum bottom, T_{bottom}	0.68
	Contact conductance, h_{Fixture}	10 000 W/m ² K

5.2.1 Treatment of Wire Melting and Energy Balance

In contrast to the validation case, this model represents continuous deposition without cooling pauses, resulting in a continuously maintained melt pool during the entire operation. Consequently, an appropriate approach in this case is to subtract the energy required to heat and melt the incoming wire from the total heat input. The energy necessary to raise the wire temperature from ambient conditions to the melting point, as well as to provide the latent heat of fusion, is therefore treated as an effective energy sink. The required power for wire heating and melting is calculated based on the wire mass rate and the properties of the material, given by

$$\dot{m}_{\text{wire}} = \rho_{\text{wire}}(T = T_0) \cdot A \cdot v_{\text{wirefeed}} \quad (5.1)$$

$$P_{\text{melt wire}} = \dot{m}_{\text{wire}}[c_p(T_{\text{liquidus}} - T_0) + H_m] = \dot{m}_{\text{wire}}\Delta h \quad (5.2)$$

where Δh is the corresponding enthalpy difference between initial temperature and liquidus temperature. This expression represents the energy required to heat and melt the wire, from which the effective power is defined.

$$P_{\text{eff}} = \eta_{\text{laser}} P_{\text{laser}} + \eta_{\text{wire heating}} P_{\text{wire heating}} - P_{\text{melt wire}}. \quad (5.3)$$

P_{eff} therefore represents the net energy available for heating the previously deposited material and maintaining the melt pool. By applying this reduced laser power, the thermal effect of continuous wire feeding is implicitly included without explicitly modeling the wire.

5.2.2 Conduction to Bottom Fixture

In the calibration case, heat conduction through the screws was modeled using the fixture temperature. However, the drum is mounted on a fixture whose geometry, thermal mass, and temperature distribution are unknown. To simplify the computational domain, the drum was truncated at the bottom, and only the upper part was included in the simulation. Consequently, a suitable thermal boundary condition had to be defined at the truncated surface, since the conduction boundary condition used in the calibration model could not be directly reused.

In analogy with how conduction through the screws was treated in the calibration model, the interaction with the fixture is represented by a convective boundary condition at the bottom surface of the truncated domain. A high heat transfer coefficient, corresponding to a high contact conductance, is applied in order to mimic conductive heat transfer into the fixture. Although a high contact conductance is a reasonable assumption, the corresponding boundary temperature is unknown. Therefore, a parameter sweep was performed in which the imposed bottom temperature was varied. The resulting interpass temperatures for different prescribed bottom temperatures are summarized in Table 5.4, while a vertical cross-section showing the temperature distribution and temperature gradient from the interpass location down toward the bottom of the cylinder is shown in Figure 5.3.

The simulations in this section were performed using steady-state and the mesh configuration denoted as ANSA map 2400, 30, 62, which is further evaluated later in this chapter.

From a physical perspective, it is reasonable to assume that heat is transferred from the drum into the fixture. The heat source applied on the drum raises its temperature during operation, and the fixture is therefore expected to be progressively heated by the drum. Consequently, the heat flux at the bottom boundary must be positive, corresponding to heat leaving the domain. It can therefore be directly observed from Table 5.4 that a prescribed bottom temperature resulting in negative values for the conductive heat transfer is not physically plausible. In these cases, negative heat fluxes are obtained, which is also evident from Figure 5.3 where the temperature increases toward the bottom boundary, resulting in a non-physical temperature gradient.

While this reasoning establishes the direction of heat transfer, it does not provide

Table 5.4: Interpass temperatures obtained for different bottom temperatures used in the conductive boundary condition are presented together with the corresponding heat transfer through conduction. The bottom temperatures and the interpass temperature is normalized with the reference temperature, and the conduction is normalized with the value corresponding to the lowest tested bottom temperature.

Normalized Bottom Temperature [-]	Contact Conductance [$\text{W}/\text{m}^2\text{K}$]	Normalized Interpass Temperature [-]	Conduction [W]
0.45	10 000	0.86	1.00
0.50	10 000	0.87	0.83
0.54	10 000	0.88	0.64
0.59	10 000	0.89	0.45
0.63	10 000	0.90	0.24
0.68	10 000	0.91	0.02
0.72	10 000	0.93	-0.21
0.77	10 000	0.94	-0.45
0.82	10 000	0.95	-0.71

Calibration of the bottom temperature of the cylinder vertical profiles at interpass location

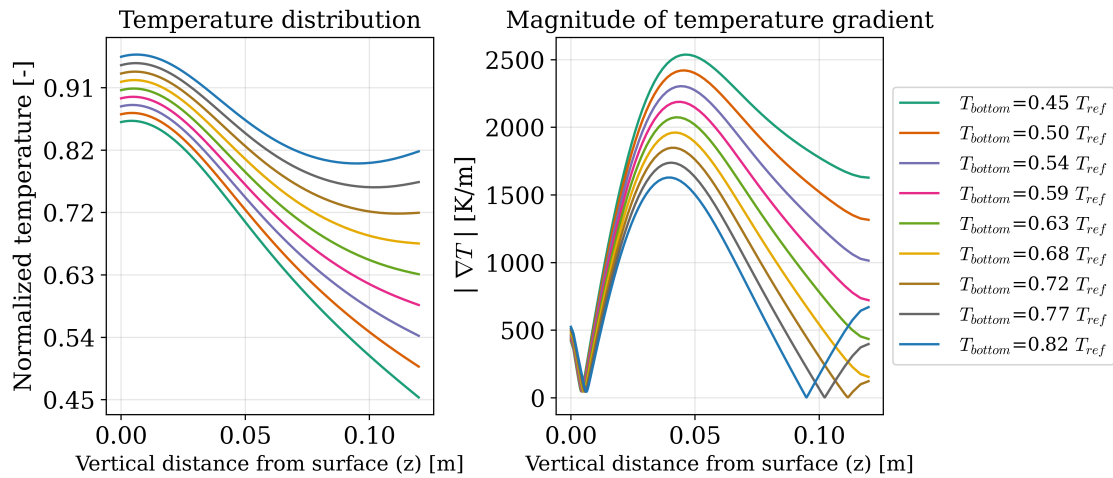


Figure 5.3: Thermal response in a vertical cross-section of the drum from the interpass location down to the bottom, along with the corresponding temperature gradient.

information about either the magnitude of the heat flux or the prescribed bottom temperature. However, it can be observed that all resulting interpass temperatures are lower than the measured temperature of at 39 minutes. Based on this observation, the normalized bottom temperature of 0.68 is considered the most reasonable approximation of the asymptotic temperature. This choice results in an interpass temperature that is in close agreement with the experimental data, particularly when

considering that the measurements may be underestimated by up to 18%.

5.2.3 Radiation

When only the solid domain is simulated, radiative heat transfer can be incorporated into an effective heat transfer coefficient, defined in Equation 4.3, as in the calibration case. However, when fluid flow is included, as addressed in the next chapter, such an effective heat transfer coefficient can no longer be prescribed, and radiation must instead be modeled explicitly using the solver’s radiation functionality.

To assess the suitability of this approach for future CFD-CHT simulations, a comparison was performed between the effective heat transfer coefficient formulation and the explicit radiation model. The results show a temperature difference of -0.01°C , indicating very good agreement. Although the explicit radiation model is slightly more computationally expensive, it yields equivalent results. For this reason, it is considered appropriate for future coupled simulations, while the effective heat transfer coefficient formulation is used for the analyses presented in this chapter.

5.3 Geometry and Mesh

Initial steady-state simulations revealed numerical issues, indicating a strong sensitivity of the solution to the chosen volume mesh. The *Convective Velocity option* in the solid energy equation was found to significantly influence the solution depending on mesh topology. As a result, the selection of an appropriate volume mesh was identified as a key aspect of the simulation setup. The mesh needed to provide numerically stable and physically consistent solutions for the rigid-body rotation of the solid, while also remaining suitable for future extension involving solid-fluid heat transfer. For such CHT simulations, a conformal mesh is generally preferred, as it improves consistent heat flux transfer between regions [7]. However, not all mesh types are compatible with both conformal mesh and the *Convective Velocity option*. Consequently, several meshing approaches were evaluated to assess their robustness and compatibility with the physical model. Volume meshes were generated using both the internal meshing in STAR-CCM+ and the external meshing tool ANSA.

A limitation of *Convective Velocity option* is that it should only be used when the solid region can be treated as a pure body of rotation or translation [32]. This requirement implies that the geometry and motion of the solid must be time-independent and rigid. Although the studied LMD-w process is based on layer-wise material addition, the resulting solid geometry is time-dependent and locally non-uniform. Each newly deposited layer introduces a small geometric step, resulting in a surface that is neither perfectly smooth nor axisymmetric. To enable the use of a steady-state model, the drum is therefore assumed to be perfectly symmetric and geometrically smooth, and the discrete height increment associated with newly deposited layers is neglected. This simplification allows the solid to be treated as a rigid body with a well-defined motion, which is a necessary condition for the validity

of the steady-state energy equation and the associated convective velocity formulation. A more detailed representation of the layer-wise build-up would instead require a fully transient simulation with time-varying geometry.

Furthermore, for steady-state simulations, it would be beneficial to locally refine the mesh in regions with large temperature gradients, such as near the melt zone and close to the surface. While such refinements can be applied in the axial direction without major complications, introducing localized refinement in the radial-circumferential plane is less suitable in the present study. This is because a consistent reference solution requires the process to also be analysed in a transient framework, where the rotation of the component is explicitly accounted for through rigid body motion (RBM). In the transient framework, the entire domain rotates, meaning that the mesh, and any local refinement, also rotates with it. As a result, localized refinement cannot remain fixed to regions of interest, such as the melt zone, without remeshing at each time step. To avoid this, a uniform and axisymmetric mesh distribution was adopted in the radial and circumferential directions. This ensures consistent resolution during rotation and allows for a fair comparison between different mesh topologies.

5.3.1 Mesh Generation

Initial mesh generation was performed using STAR-CCM+. This approach enables the mesh to be an integrated part of the simulation workflow, allowing meshing, refinements, and subsequent simulations to be carried out directly in the same software. As a result, different mesh topologies and resolutions could be tested efficiently without the need to generate meshes in external software and repeatedly import updated meshes into the solver.

A polyhedral mesher, which is an unstructured mesh, was initially considered, as it is the only meshing approach in STAR-CCM+ that allows a fully conformal mesh to be generated between solid and fluid regions [32]. This makes the polyhedral mesh particularly attractive for CHT simulations.

Following the polyhedral approach, a tetrahedral mesh was evaluated as an alternative unstructured meshing strategy. Despite the lack of support for conformal mesh generation, the tetrahedral mesher was included to assess its performance with the *Convective Velocity option*.

A trimmed-cell mesher was subsequently evaluated as a structured mesh type, offering a more ordered cell topology compared to unstructured meshes. Compared to unstructured meshes, trimmed-cells generally provide improved numerical accuracy due to reduced numerical diffusion and dissipation. However, since the domain geometry is cylindrical while the trimmed-cell mesher is based on a Cartesian grid that is aligned with the global coordinate system, a large number of cut, skewed, and distorted cells were generated near the curved boundaries. Furthermore, similar to the tetrahedral mesh, the trimmed-cell mesher does not support conformal mesh generation.

For the polyhedral, tetrahedral, and trimmed-cell meshes, consistent baseline set-

tings were applied, with only minor method-specific adjustments. All meshes were generated using standard parameters, where a global base size controlled both surface and volume resolution, ensuring comparable discretizations across mesh types. Two resolutions were considered in each case, a coarse mesh with base size of 5 mm and a fine mesh with base size 0.5 mm, to assess mesh sensitivity and verify that the results were not dependent on mesh size. To increase resolution in regions with high thermal gradients, local mesh refinement was applied on the top surface of the drum, where deposition occurs, by reducing the surface base size by approximately 20% relative to the global value. To ensure smooth transitions between refined and coarser regions, both surface and volumetric growth rates were kept low, resulting in a gradual transition from the refined region to the coarser parts of the domain. The polyhedral, tetrahedral, and trimmed-cell meshes are shown in Figure 5.4, Figure 5.5 and Figure 5.6, respectively.

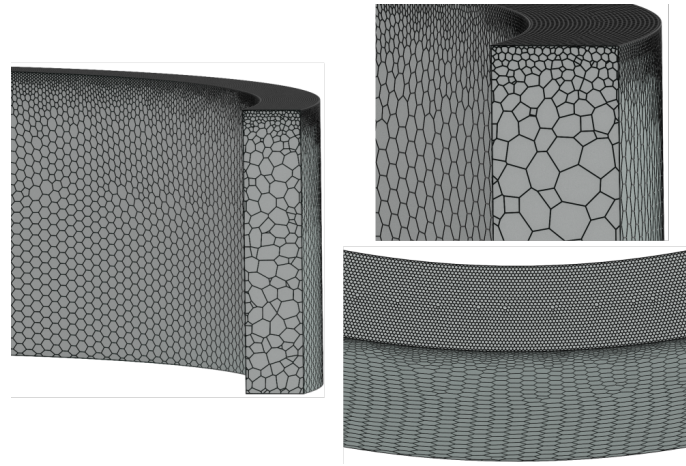


Figure 5.4: Mesh of the unstructured mesh generated by the polyhedral function in STAR-CCM+.

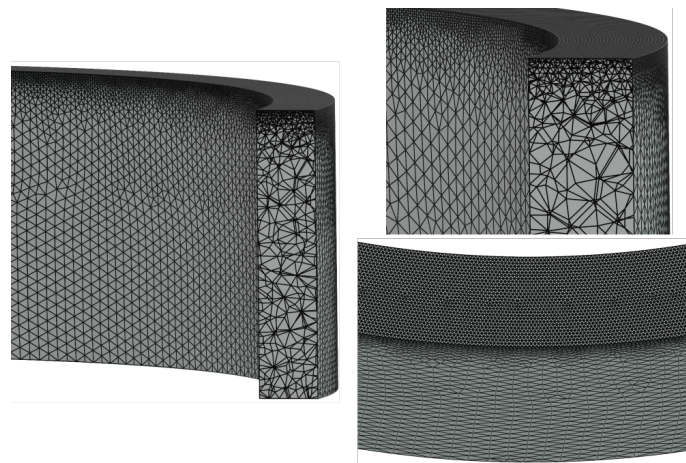


Figure 5.5: Mesh of the unstructured mesh generated by the tetrahedral function in STAR-CCM+.

To address the cell quality issues observed for the trimmed-cell mesh, a mesh generated using a directed mesh approach was employed. This meshing approach enables

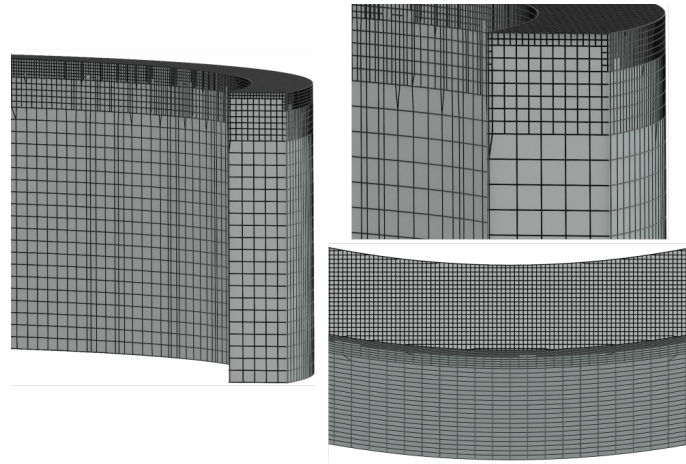


Figure 5.6: Mesh of the structured mesh generated by the trimmed-cell function in STAR-CCM+.

the generation of structured meshes while allowing the cells to be aligned with the underlying geometry. Unlike the cartesian-based trimmed-cell mesh, the directed mesher allows the grid to follow curved surfaces, making it particularly suitable for cylindrical domains. By aligning the cells with the radial and axial directions of the geometry, the formation of distorted cells near curved boundaries could be avoided. As a result, the directed mesh retained the advantages of a structured topology while providing significantly improved cell quality and geometric conformity compared to the trimmed-cell approach.

In contrast to the other mesh types, the mesh resolution for the directed mesher was controlled by specifying the number of cells in the radial, circumferential, and axial directions, as can be seen in Figure 5.7. Two levels of refinement were investigated. The resulting characteristic element lengths were approximately 2.7 mm and 0.7 mm for the coarse and fine meshes, respectively. As in the other mesh configurations, increased resolution was applied near the top surface, where the highest thermal gradients were expected. Consequently, smaller cells were used near the top of the cylinder, while progressively larger cells were allowed toward the bottom. A stretching factor of approximately 1.05 was therefore applied in the axial direction, resulting in a gradual increase in cell height away from the deposition region.

Due to limitations in STAR-CCM+ regarding conformal solid-fluid interfaces, mesh generation was also performed in ANSA to enable structured hexahedral meshes with conformal interfaces. A surface mesh, identical to the Directed Mesher in STAR-CCM+, was generated in ANSA. Based on this surface mesh, volume meshes were then created using two different ANSA meshing techniques.

The first method was the HexaInterior mesh, which can be seen in Figure 5.8. Another meshing technique available in ANSA is generating the volume mesh using the mapping function. This method gives the same result as the directed mesher in STAR-CCM+ and is shown in Figure 5.7. The resulting meshes were exported from ANSA and imported into STAR-CCM+, where they were used within the same simulation framework as the internally generated meshes.

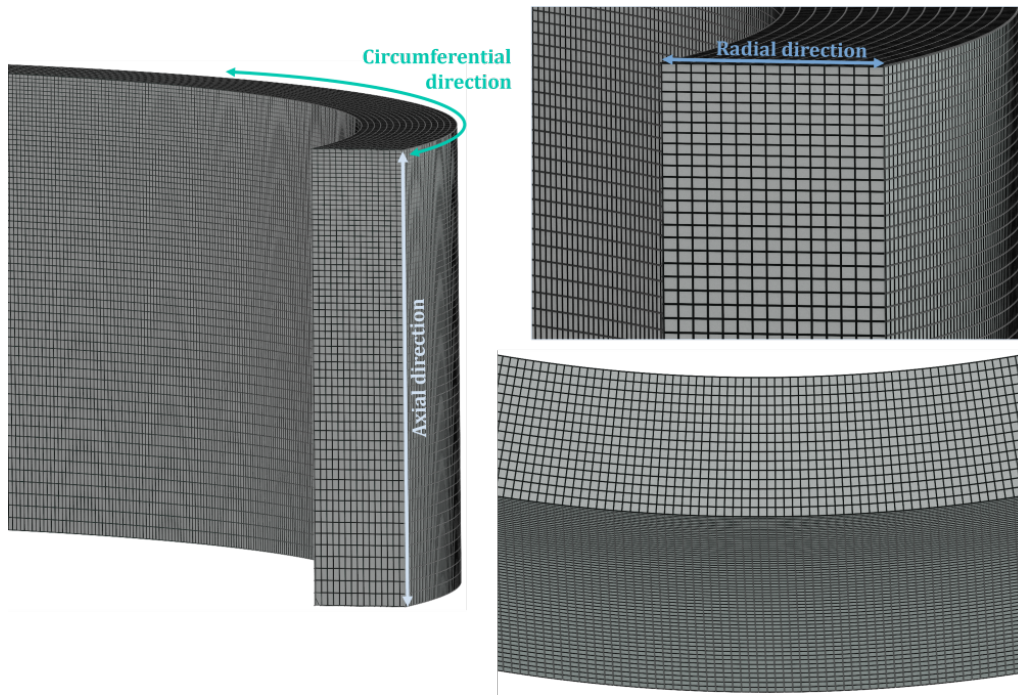


Figure 5.7: Structured mesh topology used for both the STAR-CCM+ directed mesher and the ANSA mapped mesh. The figure also illustrates the cell divisions in the radial, circumferential, and axial directions, which were varied during the mesh sensitivity study.

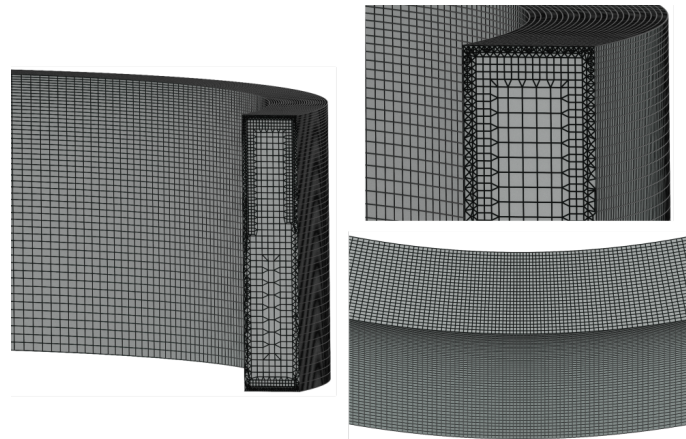


Figure 5.8: Mesh of the structured mesh generated by the HexaInterior function in ANSA.

5.3.2 Mesh Topology Evaluation

To ensure that differences in solver behavior could be attributed to mesh topology and resolution rather than other modeling choices, all simulations were performed using identical physical models, material properties, boundary conditions, and solver settings.

The mesh evaluation was carried out in two stages. First, a limited transient study

was performed to identify a suitable mesh topology for the subsequent analyses. Since transient simulations are computationally demanding, only two representative mesh configurations were investigated, a highly resolved polyhedral mesh and a structured mapped mesh generated in ANSA with a substantially lower cell count. The objective of this comparison was not to determine the optimal mesh in terms of accuracy and computational cost, but rather to evaluate the predicted long-term thermal response and assess whether steady-state simulations could provide a meaningful approximation of the system behavior.

In the second stage, steady-state simulations were used to evaluate all mesh topologies described in Section 5.3.1. For each mesh topology, both a coarse and a fine mesh were generated and analyzed. The evaluated mesh topologies and resolutions are presented in Section 5.4.2 and in Table 5.6.

5.3.3 Mesh Sensitivity Analysis

Based on the evaluated mesh configurations, a mesh sensitivity study was subsequently performed using the mapped ANSA mesh, as it provided the most reliable results (details on this selection are presented in Section 5.4.2). The objective of the study was to further assess the influence of mesh resolution on the predicted temperature field and associated thermal gradients. Mesh refinement was therefore applied in the radial, circumferential, and axial directions of the cylindrical solid.

The mesh resolution in the radial and circumferential directions was varied in a combined manner to maintain a uniform cell aspect ratio in the cross-sectional plane. By jointly adjusting the number of divisions in these directions, different surface mesh configurations were generated. Additionally, mesh sensitivity was also investigated in the axial direction. The same axial stretching factor of 1.05 was retained while the total number of cells was varied, resulting in uniformly larger or smaller elements throughout the domain.

5.4 Results and discussion

This chapter presents and discusses the simulation results. Initially, the influence of mesh topology is examined in the context of transient simulations, followed by an analysis of the transient behavior. Finally, the steady-state assumption is evaluated in comparison with the transient results, with particular focus on identifying the most suitable mesh.

5.4.1 Transient behavior

Two different mesh configurations were evaluated for the transient analysis. The first consisted of a polyhedral mesh with very fine resolution, while the second was a structured mapped mesh generated in ANSA containing approximately six times fewer cells. The primary objective of this study was not to identify the optimal mesh in terms of accuracy versus computational cost, but rather to investigate the transient thermal evolution and assess whether steady-state simulations could

provide a meaningful approximation of the thermal behavior. Since the transient simulations were computationally expensive, with simulation times exceeding one day, only these two mesh configurations were investigated. Table 5.5 provides an overview of the investigated mesh configurations and the corresponding simulation results. The reported temperatures were calculated as the mean value over the final 10 minutes of the simulation, i.e., between 50 and 60 minutes.

Table 5.5: Table containing mesh type results for transient simulations for the polyhedral mesh and ANSA-generated mapped mesh.

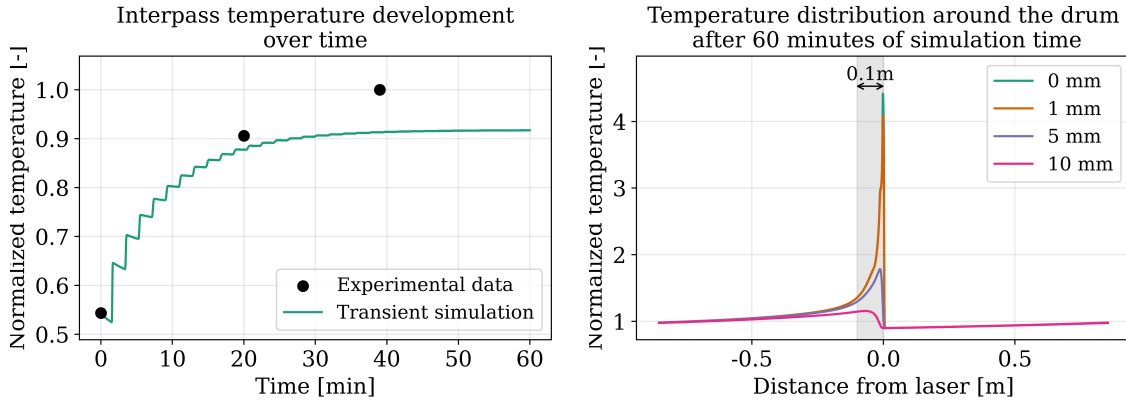
Mesh type	Base size / Approximate Element Length [mm]	Cells in circumf-, radial-, axial direction	Number of cells	Normalized Interpass Temperature [-]
STAR-CCM+ Polyhedral mesh	0.5	-	24 282 636	0.916
ANSA Map	0.7	2400, 30, 62	4 464 000	0.917

Based on the comparison, the ANSA mapped mesh was selected due to its substantially lower computational cost while still providing sufficiently resolved thermal behavior. This mesh further used as the reference mesh in the subsequent section with steady-state modeling.

Using the selected mesh, the transient thermal response of the system was then investigated. The transient temperature evolution exhibits a characteristic sawtooth-like behavior during the initial phase of the simulation. After this initial transient phase, the temperature gradually approaches a more stable and asymptotic level, indicating that a periodic steady-state has been reached. In this regime, the overall temperature increase is no longer significant.

The sawtooth-like pattern is characterized by a rapid increase followed by a gradual decrease in temperature. When examining the first rotation in detail, this behavior can initially be understood from the imposed initial conditions. The entire drum is uniformly preheated, corresponding to the desired temperature for deposition, while the surrounding environment is significantly cooler. As a result, the cylinder undergoes continuous cooling due to convection and radiation, leading to a gradual decrease in temperature during the first rotation. As the simulation progresses, material that has passed under the laser heat source is transported to the measurement location. When this heated region reaches the monitoring point, a rapid increase in temperature is observed followed by a slow decrease. This sequence is repeated during subsequent rotations, producing the characteristic sawtooth pattern.

At first glance, this behavior may appear counterintuitive, particularly since the heat source operates continuously. However, the observed response can be explained by considering the combined effects of thermal history and heat input. As the drum cools down during the first revolution, the incoming temperature to the heat source varies over time, and the resulting temperature increase depends on this initial condition. A cooler material element receiving the same heat input as a warmer one therefore reaches a lower peak temperature. This leads to the observed variation in peak temperatures and the sawtooth-like behavior.



(a) Transient temperature evolution at the interpass location. The simulation matches the first two experimental data points but then converges toward a steady-state, deviating from the experimental trend. A periodic sawtooth-like variation is observed instead of a continuous increase.

(b) Temperature distribution around the drum at depths of 0 mm, 1 mm, 5 mm and 10 mm below the surface. The highlighted region indicates the characteristic length scale, defined by the distance over which the temperature decays significantly from the melt zone.

Figure 5.9: Simulation results for the transient simulation of the drum.

One might expect thermal conduction within the solid to rapidly smooth out these variations and eliminate the sawtooth behavior. However, this is not observed. The characteristic length scale of the temperature field, obtained from the simulated surface distribution (Figure 5.9b), is approximately 0.1 m. This length scale reflects the combined effects of localized heating, diffusion, and rotational transport.

To assess the governing heat transfer mechanism, the Péclet number was evaluated as

$$Pe = \frac{vL}{\alpha} \approx 160 \gg 1, \quad (5.4)$$

indicating that heat transport due to rotation dominates over thermal diffusion. As a result, temperature gradients are not smoothed out by conduction but instead persist and are transported with the rotating solid, giving rise to the observed sawtooth pattern.

This behavior is further clarified by considering the Fourier number based on the rotational timescale,

$$Fo = \frac{\alpha t}{L^2} \approx 0.01 \ll 1, \quad (5.5)$$

which shows that thermal diffusion is very limited during a single rotation. Consequently, one revolution does not provide sufficient time for the temperature field to equilibrate, and the spatial variations remain largely intact.

To estimate the timescale required for the disappearance of the sawtooth temperature pattern, the Fourier number can be considered of order unity, $Fo \sim 1$, corresponding to the onset of effective thermal diffusion. This yields a characteristic

diffusion timescale of the form

$$t \sim \frac{L^2}{\alpha} \approx 1100 \text{ s} = 18 \text{ min}, \quad (5.6)$$

which represents the time required for thermal conduction to substantially reduce spatial temperature gradients.

This estimate can be directly related to the transient behavior observed in the simulation. After approximately this time, the sawtooth pattern is largely diminished, indicating that thermal diffusion has become sufficiently effective to smooth out the temperature variations. The agreement between the estimated diffusion timescale and the simulated evolution demonstrates that the disappearance of the sawtooth pattern is governed by thermal diffusion over the characteristic length scale.

However, the system has not yet reached thermal equilibrium. Although local temperature gradients are smoothed out, the overall temperature continues to increase as heat is still being accumulated. Over additional time, the system gradually approaches a quasi-steady-state, where the heat input is balanced by thermal losses.

This convergence is observed after approximately 30-40 minutes, where the temperature evolution becomes significantly slower. The final stage is characterized by small temperature fluctuations, with a standard deviation of 0.130°C (0.026 %) over the last 10 minutes. Due to these residual variations, the temperature does not reach a perfectly constant value. The representative temperature that was used for comparison with the steady-state solver was therefore defined as the mean over the final 10 minutes of the simulation, yielding a normalized temperature of 0.917.

The transient temperature evolution agrees well with the first two experimental data points, but a clear deviation is observed at the last point, where the experimental temperatures continue to increase while the simulation approaches a steady-state condition. Within the present modeling framework, this continued increase cannot be reproduced, as all simulations converge toward a steady-state. The discrepancy may be related to uncertainties in the boundary conditions, but it may also arise from measurement effects associated with the long operating time. Extended operation could potentially lead to thermal accumulation in surrounding components or the measurement system, potentially introducing drift in the recorded temperatures.

To determine whether the deviation originates from model limitations or experimental uncertainty, additional measurements would be required. In particular, repeated experiments with improved temporal resolution and extended datasets would allow for a more reliable assessment. In the absence of such data, the deviation is treated as a limitation of the current model.

5.4.2 Mesh Topology and Sensitivity Evaluation

This section presents the mesh evaluation study for steady-state simulations using the *Convective Velocity option*. Since the transient analysis showed that the thermal

response approaches an asymptotic state over time, steady-state modeling was investigated as a computationally efficient method for predicting the long-term thermal behavior.

All investigated mesh configurations converged numerically, exhibiting low residual levels and stable monitored variables such as temperature and heat flux. However, despite numerical convergence, significant differences were observed in the resulting temperature fields, depending on the mesh topology.

Several mesh types were found to converge to stable but physically incorrect temperature fields, characterized by unrealistically low temperature levels. The evaluated mesh topologies, resolutions, and corresponding results are summarized in Table 5.6. Additionally, the temperature distributions in a vertical cross-section at the interpass location for the different cases are shown in Figure 5.10. Furthermore, the temperature fields on the cylindrical surface for a selection of simulations are presented in Figure 5.11. All results are evaluated relative to the transient solution, presented and discussed in the previous section, Section 5.4.1.

Table 5.6: Table containing mesh type results categorised by mesh type for steady-state compared to the corresponding transient results. The temperature difference is normalized with the transient reference solution.

	Mesh type	Base size / Approximate Element Length [mm]	Cells in circumf-, radial-, axial direction	Number of cells	Interpass ΔT [%]
Trans- sient	STAR-CCM+ Polyhedral	0.5	-	24 282 636	-0.13
	ANSA Map	0.7	2400, 30, 62	4 464 000	-
Steady state	STAR-CCM+ Polyhedral	0.5	-	24 282 636	-7.87
	STAR-CCM+ Polyhedral	5	-	195 715	-19.24
	STAR-CCM+ Tetrahedral	0.5	-	125 057 329	-3.45
	STAR-CCM+ Tetrahedral	5	-	906 875	-12.10
	STAR-CCM+ Trimmed-cell	0.5	-	18 551 112	-0.13
	STAR-CCM+ Trimmed-cell	5	-	167 821	-0.49
	STAR-CCM+ Directed mesher	0.7	2400, 30, 62	4 464 000	-0.00
	STAR-CCM+ Directed mesher	2.7	600, 10, 62	372 000	-0.66
	ANSA HexaInterior	0.7	2400, 30, 62	7 683 021	-1.10
	ANSA HexaInterior	2.7	600, 10, 62	1 370 683	-3.24
	ANSA Map	0.7	2400, 30, 62	4 464 000	-0.00
	ANSA Map	1.5	1200, 15, 62	1 116 000	-0.23
	ANSA Map	2.7	600, 10, 62	372 000	-0.49
	ANSA Map	1.5	1200, 15, 31	558 000	-0.25
	ANSA Map	1.5	1200, 15, 128	2 232 000	-0.17

The polyhedral mesh resulted in temperature fields that, although numerically converged, were physically incorrect. The predicted temperature levels were significantly lower than expected, and the temperature distribution deviates substantially from the transient reference solution. A clear dependence on mesh resolution was observed, where the fine mesh yielded results closer to the reference solution than the coarse mesh. However, even with the fine resolution, large discrepancies remained,

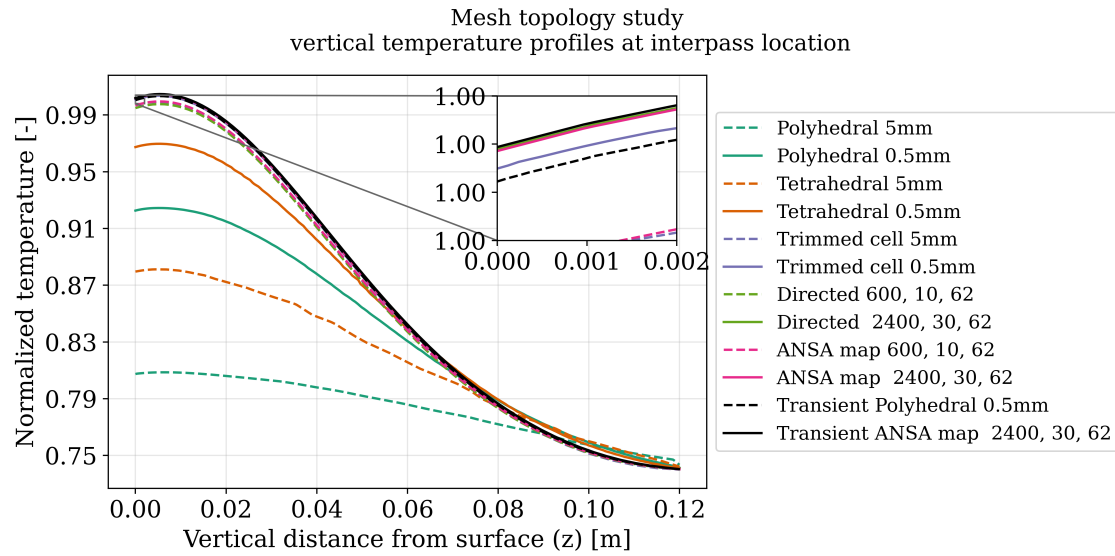


Figure 5.10: Temperature distributions for the different cases in a vertical cross-section from the interpass location down to the bottom. All temperatures are normalized using the transient reference simulation.

and the coarse polyhedral mesh showed the poorest performance.

A similar trend was observed for the tetrahedral mesh. The predicted temperature fields deviated considerably from the reference solution, although the results were somewhat improved compared to the polyhedral mesh. As for the polyhedral case, the fine mesh provided better agreement than the coarse mesh, but the overall accuracy remained insufficient, indicating that mesh refinement alone was not sufficient to recover physically correct results.

The trimmed-cell mesh produced more reasonable temperatures than the unstructured meshes, likely due to reduced numerical diffusion. While the fine mesh showed improved agreement with the reference solution, the difference between the coarse and fine meshes was relatively small, suggesting a weaker sensitivity to mesh resolution.

The directed mesh yielded physically consistent temperature fields across both mesh resolutions. The predicted temperature levels and gradients were in close agreement with the transient reference solution, and only minor differences were observed between the coarse and fine meshes. This indicates both strong numerical robustness and low sensitivity to mesh resolution.

The hexa-interior mesh generated in ANSA produced results that can be regarded as intermediate between the fully structured meshes and the unstructured meshes. While the majority of the cells are structured, the presence of a tetrahedral transition region near the surface appears to introduce some additional numerical diffusion. As a result, the solution accuracy was slightly reduced compared to fully structured meshes, although still significantly better than for fully unstructured meshes.

When the domain was discretized using a fully structured mapped mesh in ANSA,

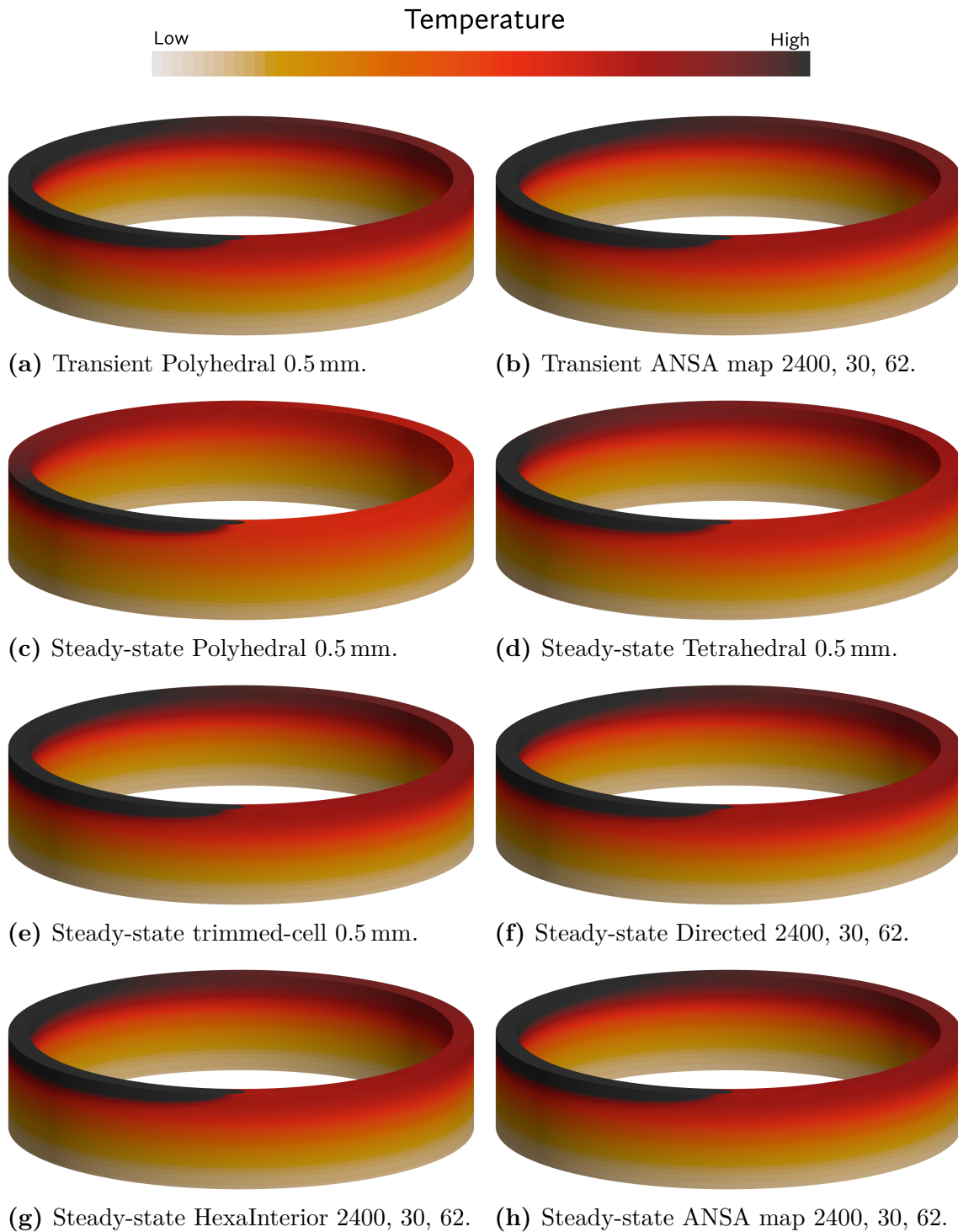


Figure 5.11: Temperature fields of the drum for different mesh setups, and a transient reference simulation.

the resulting temperature fields were physically consistent and closely matched the transient reference solution. The results were also consistent with those obtained using the directed mesh in STAR-CCM+, indicating that mesh topology is the dom-

inant factor governing solution accuracy, rather than the specific meshing software used. However, in the present study, fully conformal meshes could only be generated in ANSA, which makes it the preferred choice in this work.

The observed behavior is primarily attributed to the discretization of the convective term in the solid energy equation when the *Convective Velocity option* is used in a steady-state formulation. In combination with upwind schemes, unstructured meshes introduce significant numerical diffusion due to irregular cell shapes and misalignment with the transport direction. This leads to smeared temperature fields, reduced peak temperatures, and inaccurate gradients. In contrast, structured meshes provide better alignment with the convective transport, thereby reducing numerical diffusion and enabling physically consistent solutions.

Overall, this study clearly demonstrates that the *Convective Velocity option* exhibits significant limitations and strong sensitivity to mesh topology. All simulations converged numerically without apparent issues, yet several cases resulted in physically incorrect temperature fields, highlighting that convergence alone is not a sufficient indicator of solution quality. Careful validation of temperature distributions is therefore essential when applying this approach. Furthermore, the results indicate that the limitations are primarily associated with the *Convective Velocity* formulation rather than the mesh types themselves. This is supported by the fact that even the polyhedral mesh, which performed poorly in the present steady-state simulations, was able to produce accurate results in the transient simulations with rigid body motion. Consequently, while different mesh types have inherent advantages and disadvantages, the main source of inaccuracy in this study is the *Convective Velocity option* itself, rather than the mesh discretization.

Since mapped mesh generated in ANSA provided the most accurate results, a more detailed mesh sensitivity study was performed. When varying the resolution in the axial direction, only negligible differences were observed between the coarsest, intermediate, and finest meshes, 0.25%, 0.23%, and 0.17%, respectively, indicating a very weak sensitivity to the number of cells in this direction. In contrast, variations in the radial and circumferential directions resulted in slightly larger differences, 0.49%, 0.23%, and 0.0%, respectively, suggesting a stronger influence on the solution, although the overall sensitivity remained low. Given that the steady-state simulations converged within ≈ 100 iterations, the computational cost remained modest even for relatively fine meshes. This makes it feasible to employ a higher resolution while still maintaining low computational effort. Nevertheless, to limit the total cell count, a mesh with 1200, 15, and 62 cells in the circumferential, axial and radial directions, respectively, was selected as a suitable compromise for subsequent CFD simulations. This mesh contains 1 116 000 cells and showed a deviation of only 0.23% compared to the transient reference simulation.

5.4.3 Assessment of Steady-State Assumption

The transient analysis shows that the system does not reach a perfectly constant temperature, but instead approaches a quasi-steady-state after approximately 30-40 minutes. Based on this behavior, the system can be considered to have reached a

sufficiently stable state for the purpose of the present work, even though a strict steady-state is not reached. This supports the use of a steady-state formulation as an approximation of the long-term thermal behavior.

A comparison between the transient and steady-state solutions shows that the steady-state model accurately reproduces the temperature field while significantly reducing computational cost compared to a 60 min transient simulation. However, the steady-state solution using the *Convective Velocity* formulation is highly sensitive to mesh topology. Although the ANSA map mesh yielded physically reasonable results, careful validation of the temperature field is essential when applying this approach in similar simulations.

Overall, the results indicate that steady-state modeling represents a computationally efficient and reasonable approach for capturing the relevant thermal behavior in the present problem. The steady-state model based on the ANSA map mesh further provides a suitable foundation for extension to a fully coupled CFD-CHT model.

5. Assessment of the Validity of Steady-State Modeling for an Industrial Drum

6

Coupled CFD Analysis and Forced Cooling Evaluation

To improve the thermal prediction of the LMD-w process, a conjugate heat transfer (CHT) model was developed. Previous chapters were limited to solid-only thermal modeling, where the effect of the surrounding fluid was represented through simplified boundary conditions. In the present approach, both the solid and fluid domains are solved simultaneously, with the fluid flow explicitly resolved using computational fluid dynamics (CFD). This enables a more physically realistic representation of the thermal interaction between the rotating drum and the surrounding gas.

The primary objective of this chapter is to investigate how the inclusion of a resolved flow field influences the thermal response of the rotating drum, with particular emphasis on the interpass temperature. Additionally, the impact of forced cooling is evaluated.

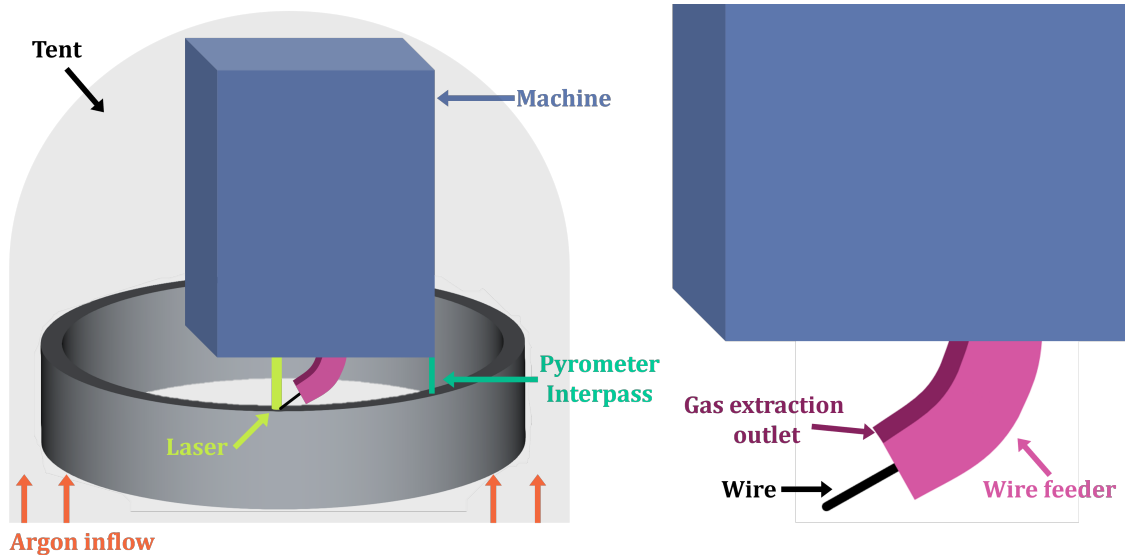


Figure 6.1: Representative schematic of the computational domain, showing the drum, machine, wire feeder, wire, laser, pyrometer, gas extraction outlet, and enclosing tent. The geometry is simplified due to confidentiality.

6.1 Geometry and Computational Domain

The computational domain consists of both solid and fluid regions, representing the drum, the machine, and the surrounding gas within the tent. The solid domain includes the rotating drum and the relevant machine components, while the fluid domain captures the argon inside the tent, enclosing the entire system. A representative schematic of the full geometry is shown in Figure 6.1. Moreover, the planes used for the subsequent flow field analysis, including two cross-sectional planes and one cylindrical surface, are shown in Figure 6.2.

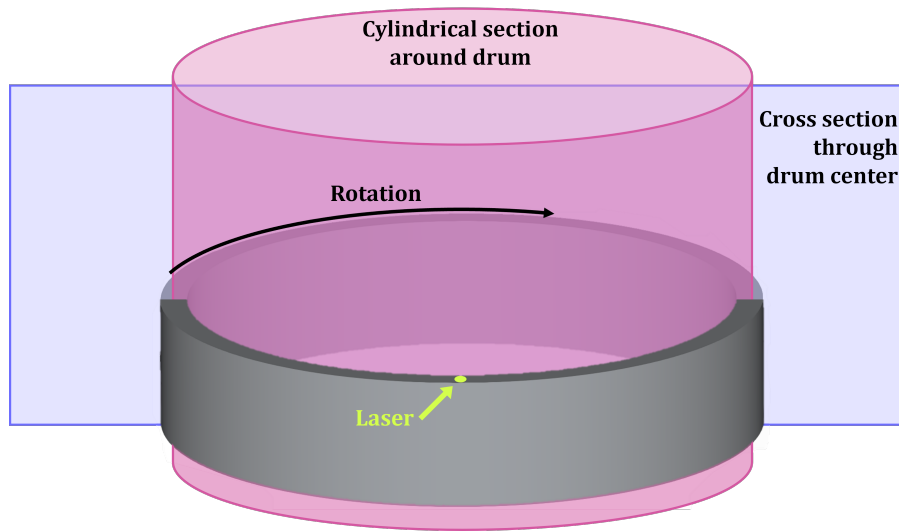


Figure 6.2: Drum geometry and selected sectional planes used for flow field analysis.

6.1.1 Geometry Pre-Processing

The original CAD model of the deposition system was highly detailed, including the machine, laser, and surrounding components. For CFD purposes, such detail is not necessary, as the objective is to capture the overall flow behavior rather than small-scale flow features. The focus of the study is the flow and heat transfer near the deposition region. Therefore, a geometry cleanup for the machine was performed, where small features were removed or simplified. However, key elements affecting the local flow were retained. In particular, the wirefeeder and gas extraction outlet were kept in detail due to their influence on the flow near the deposition region.

The computational domain was constructed to represent the full physical setup, including the rotating drum, the deposition system, and the surrounding argon within the tent. The computational domain was constructed to match the full scale of the tent, ensuring that the overall flow behavior is realistically represented.

The gasflow within the domain is primarily driven by a pressure inlet located at the bottom of the enclosure. This boundary represents the incoming gas to the system. Additionally, an outlet is defined within the deposition head, corresponding to the air extraction system present in the physical setup. This creates a global flow

pattern through the tent and across the drum surface. However, the bottom surface at the center of the drum was defined as a wall boundary. This simplification is based on the fixture design, where no significant airflow is expected within the inner region of the drum.

Overall, the geometry was designed to balance physical realism with computational efficiency, ensuring that the dominant flow and heat transfer mechanisms, particularly near the deposition zone, are accurately captured.

6.1.2 Forced Cooling Setup

To investigate potential methods for reducing thermal accumulation, a forced cooling strategy was evaluated in which argon was injected through small tubes directed toward the deposition surface. The objective was to assess how localized cooling influences the flow field and whether it can effectively reduce the thermal load on the component. Compared with the base case with natural convection, the forced cooling configuration includes additional localized cooling jets. These are intended to enhance heat removal from the newly deposited material by directing cold gas toward the surface shortly after the melt zone.



Figure 6.3: Representative illustration of three cooling jets directing cold argon gas onto the newly deposited, hot surface downstream of the laser.

In the present study, three cooling jets were introduced and oriented toward the surface downstream of the melt pool. The configuration is based on the physical setup, from which the geometry was approximated. As the exact dimensions and positioning of the cooling tubes are not available, their placement in the model is therefore approximate and does not reflect the actual industrial configuration. A representative setup of the cooling jets is shown in Figure 6.3.

Furthermore, due to their small diameter, the tubes themselves were not explicitly modeled. Instead, each jet was represented by a boundary surface with a prescribed mass flow, effectively simplifying the geometry while retaining the dominant flow effects. This simplification reduces computational cost and complexity, while still capturing the primary influence of the forced cooling on the flow and thermal fields.

6.2 Mesh Generation

The computational mesh was generated in ANSA, consistent with the previous findings on solid mesh quality in Chapter 5, and to enable the creation of conformal meshes suitable for CHT simulations.

The meshing strategy for the solid domain follows the approach established in the previous chapter, where a structured mapped mesh was shown to provide improved numerical stability and accuracy. As a result, the same mesh settings were retained for the solid, and no modifications were introduced.

Consequently, the focus of this section is on the fluid mesh generation, which introduces additional complexity due to the need to accurately resolve flow features and thermal boundary layers.

6.2.1 Surface Mesh

The surface mesh was generated predominantly using ANSA's mapped meshing approach, enabling the use of structured quadrilateral elements. This approach was applied wherever the geometry allowed, particularly for the wire feeder and machine surfaces. The use of quadrilateral elements improves mesh quality by reducing skewness and providing better alignment of cells, which is beneficial for numerical accuracy and convergence.

For more complex surfaces where mapped meshing was not feasible, an unstructured CFD surface mesh was applied. To reduce the overall cell count while maintaining accuracy near the region of interest, a non-uniform mesh resolution was applied to the enclosure walls. Finer elements were used in the lower part of the domain near the drum, while coarser elements were applied further away. These surfaces were meshed using ANSA's CFD meshing approach to allow for a smooth transition in element size.

The approximate element size for different components is summarized in Table 6.1.

Table 6.1: Surface mesh element sizes for the different parts.

Component		Characteristic Element Length [mm]
Drum surface		1.5
Machine		8-9
Wire feeder	Overall	0.5-0.7
	Gas outlet	0.3-0.5
Tent walls	(lower region)	4
	(upper region)	25
Bottom center		4

6.2.2 Boundary Layer Treatment

To accurately resolve both the velocity and thermal boundary layers, prism layers were applied on all relevant solid surfaces in contact with the fluid. Prism layers were generated on the entire surface of the rotating drum, and on all external surfaces of the machine.

The prism layers on the drum are of particular importance, as they ensure proper resolution of the near-wall gradients governing convective heat transfer. Resolving both the hydrodynamic and thermal boundary layers is essential for accurately predicting the heat flux from the drum surface, which directly influences the interpass temperature. Additional prism layer refinement was introduced in regions of high flow gradients, specifically near the extraction outlet. Due to the elevated velocities in this region, a finer near-wall resolution is required to capture high velocities and steep gradients.

The prism layer setup was defined using first prism layer height, number of layers and growth rate. These parameters were evaluated by monitoring the resulting y^+ values, ensuring that the target condition of $y^+ < 1$ was satisfied across all relevant surfaces. In addition to the y^+ criterion, the quality of the transition between the prism layer region and the core volume mesh was assessed. This was assessed by examining the continuity of both the temperature field and its gradients across the interface. A smooth and physically consistent transition was verified by ensuring that no artificial discontinuities or numerical artifacts were present in either the temperature or velocity fields. These checks were essential to confirm that the prism layer setup provides an accurate representation of the near-wall behavior, without introducing numerical errors due to poor mesh transitions.

No prism layers were applied to the tent walls. Since these surfaces are located far from the region of interest, resolving the boundary layer in these regions was deemed unnecessary. This significantly reduces the total cell count without affecting the accuracy of the solution near the drum and deposition region. Similarly, no prism layers were applied to the bottom surface at the center of the drum. Although this surface is geometrically close to the drum, the flow in this region is assumed to be relatively slow due to the surrounding fixture design, which limits fluid motion. As a result, large velocity and thermal gradients are not expected to develop near this surface. Based on this assumption, resolving the boundary layer in this region was considered of secondary importance compared to the drum surface and the melt pool region. Omitting prism layers in this area therefore simplifies the mesh while maintaining sufficient accuracy for the quantities of interest.

An overview of the prism layer settings for different regions is provided in Table 6.2.

6.2.3 Volume Mesh

For the volume mesh, the *HexaInterior* function in ANSA was employed. A low growth rate of 1.005 was used to ensure that the cell size increases gradually away from the refined regions. This allows for a smooth spatial transition and helps

Table 6.2: Prism layer settings for different regions.

Region		First Layer Height [mm]	Number of Layers	Growth Rate
Drum	Top surface	0.1	12	1.3
	Sides	0.1	12	1.4
Machine		0.1	8	1.4
Wirefeeder	Overall	0.05	8	1.3
	Gas outlet	0.005	12	1.4

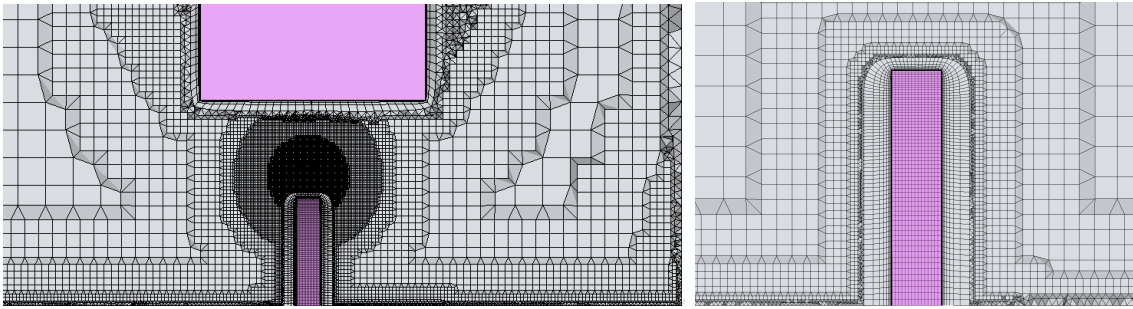
maintain high mesh quality by avoiding abrupt changes in cell size.

In addition, the *smooth transition* option was enabled. This further improves mesh quality by ensuring a gradual variation in cell size between refined and coarse regions, reducing the risk of numerical errors associated with strong mesh gradients.

The volume mesh is largely governed by the underlying surface mesh resolution. Since different surface regions were assigned different characteristic element sizes, this naturally results in finer volumetric cells in areas of higher interest, such as near the drum and the melt pool region. Conversely, larger cells are generated in regions outside of the area of interest, improving computational efficiency without compromising solution accuracy where it matters most.

To further control the mesh distribution, local refinements were introduced using ANSA's *Size Field* functionality. A spherical refinement region was defined around the melt zone and close to the gas extraction outlet. These regions are characterized by strong gradients in both the velocity and temperature fields. Near the melt pool, high temperatures significantly influence the surrounding fluid, leading to pronounced thermal gradients and buoyancy-driven flow effects. Additionally, the extraction outlet is associated with the highest local velocities in the domain. For the forced cooling configuration, additional local refinements were introduced along the jet paths between the inlet surfaces and the impingement region on the drum. A total of three refinement regions were applied to resolve the high-velocity jets. These refinements are necessary to accurately capture the jet structure and the resulting local heat transfer, as coarse mesh resolution would otherwise lead to excessive numerical diffusion and underprediction of the jet impact. The combination of high velocity gradients and strong thermal coupling in these regions makes it critical to resolve the flow accurately. Therefore, a finer mesh was applied to ensure sufficient resolution of both the aerodynamic and thermal phenomena, which directly affect the predicted interpass temperature.

The cell count is approximately 13 000 000 in both cases, with the similarity primarily due to the majority of cells, especially in the prism layers, being unchanged. Due to confidentiality reasons, the exact geometry of the machine cannot be disclosed. Therefore, Figure 6.4 are limited to selected sections of the mesh where no sensitive details are visible, and more detailed images cannot be presented.



(a) Mesh close to meltzone with refinements in high gradient regions. (b) Mesh around the drum with prism layers and volume mesh.

Figure 6.4: Mesh of a selected region of the geometry, illustrating the solid domain, prism layers, and volume mesh. The solid region is represented in pink, while the fluid region is shown in light grey.

6.3 Physical and Numerical Models

This section outlines the numerical setup used in STAR-CCM+, including the solver strategy, turbulence modeling, and convergence criteria applied in the simulations.

6.3.1 Solver Settings

When extending the model to include fluid flow and CHT, the problem becomes significantly more complex due to the strong coupling between momentum, pressure, and energy transport in the fluid. In such cases, a coupled solver approach is generally preferred over the segregated, as it solves the governing equations simultaneously, leading to improved robustness and convergence behavior for strongly coupled multiphysics problems. Since it is not possible to combine a coupled solver in the fluid domain with a segregated solver in the solid domain, the coupled approach was applied to the entire model. This choice is justified because the solid solution was previously verified to be independent of the solver formulation, as described in the Section 5.1.

CFL and convergence behavior

Simulations were performed using constant CFL numbers of 1 and 5, allowing for a comparison between a more conservative and a more aggressive pseudo-time-stepping approach. Here, the CFL number refers to the pseudo-time-step CFL number used by the coupled steady-state solver to control the magnitude of the iterative solution updates.

Although a steady-state formulation was applied, initial simulations exhibited oscillations in both velocity and temperature fields, particularly in regions influenced by buoyancy, indicating that the physical system is naturally unsteady. Due to the significant computational cost associated with fully transient simulations, a steady-state approach was kept. Instead, the influence of the CFL number was investigated. A lower CFL number results in a more gradual, pseudo-transient-like evolution of

the solution, improving stability at the expense of slower convergence. In contrast, higher CFL values improve convergence speed but tend to damp unsteady flow structures and may lead to difficulties in obtaining a stable solution. Based on this, two simulation cases were defined to assess these effects, as described in Section 6.5 below.

Continuity convergence accelerator

In addition, the *continuity convergence accelerator* (CCA) was enabled in the coupled solver. This method introduces an additional pressure-correction equation to reduce mass imbalances at each iteration. Initial simulations showed significant difficulties in achieving convergence of the mass flow balance, which motivated the use of the CCA. A relaxation factor of 0.05 was selected, as higher values resulted in oscillatory behavior, while the lower value provided a more stable solution and significantly improved the mass flow balance compared to simulations without CCA. It should be noted that the CCA does not affect the final solution, but improves the convergence behavior. [32]

Discretization

Second-order discretization schemes were applied for both the momentum and energy equations. This reduces numerical diffusion and improves the accuracy of predicted velocity and temperature gradients, which is essential for resolving both boundary layer behavior and convective heat transfer.

Conjugate Heat Transfer Coupling

The coupling between the fluid and solid domains was handled using a conformal interface. This allows for direct node-to-node connectivity between the two domains, enabling the use of a conjugate heat transfer (CHT) formulation without interpolation errors.

At the interface, both temperature continuity and heat flux conservation are enforced. This ensures that the heat transfer between the solid and fluid is governed by the local flow conditions, rather than prescribed boundary conditions, providing a physically consistent representation of convective cooling.

6.3.2 Turbulence Modeling

Turbulence was modeled using the SST k - ω model. This model was selected due to its suitability for near-wall modeling, which is important for predicting convective heat transfer [25]. The mesh was designed to achieve $y^+ \lesssim 1$ in near-wall regions, consistent with a low-Reynolds-number approach. This allows the viscous sub-layer to be resolved using a sufficiently fine mesh, avoiding the use of standard wall functions [32], and improving the prediction of wall heat transfer [25].

An under-relaxation factor of 0.7 for the turbulence model was applied to improve convergence stability. This value was selected based on observed convergence behavior, as it reduced oscillations and resulted in lower residual levels compared to higher URF values.

6.3.3 Convergence Criteria

Convergence was assessed using both residual monitoring and evaluation of physically relevant quantities. Absolute residual values for continuity, momentum, and energy were monitored to ensure an adequate reduction. However, due to the inherently unsteady nature of the solution, strict residual convergence to a fixed threshold was not achievable.

Instead, convergence was evaluated based on the behavior of key physical quantities, such as interpass temperature, velocity, temperature gradients, and heat transfer coefficient. The simulations were run for 50 000 iterations, and a solution was considered sufficiently converged when the majority of these quantities exhibited stable behavior over a large number of iterations.

It is noted that certain variables still display significant oscillations, as discussed in Section 6.6. These are interpreted as physical transient effects rather than a lack of numerical convergence. This approach ensures that the thermal response of the system, which is the primary quantity of interest, is captured in a consistent and physically meaningful manner despite the absence of a strictly steady solution.

6.4 Boundary Conditions

The solid domain was modeled as in the previous chapter. The main difference is that the thermal boundary conditions on the drum are no longer prescribed, but instead computed through conjugate heat transfer with the surrounding fluid. Heat conduction to the bottom and the applied laser heat input were however retained.

6.4.1 Momentum Boundary Conditions

The airflow within the enclosure is driven by an gas extraction system, with an estimated flow rate on the order of 400 L/min to 600 L/min. Due to the truncated domain, the inflow was modeled using a pressure inlet of 50 Pa at the bottom surface, while the extraction was represented by a pressure outlet of -3300 Pa. This outlet value was calibrated through preliminary simulations to obtain a representative volumetric flow rate. To improve numerical stability during the initial iterations, the pressure boundary conditions were ramped from 0 Pa to the target pressure over the first 100 iterations.

The tent walls and the bottom surface at the center of the drum were modeled as slip walls. This assumption eliminates the formation of boundary layers at these surfaces, which is justified since they are either located far from the region of interest or expected to have negligible influence on the local flow due to the fixture design.

Machine and wire feeder surfaces were, however, treated as no-slip walls. This allows boundary layers to develop where the flow directly interacts with solid surfaces that influence the convective heat transfer near the melt pool.

6.4.2 Thermal Boundary Conditions

The inlet temperature was specified based on measurements and set to 20 °C. For the enclosure walls, convective boundary conditions were applied using a heat transfer coefficient h_{conv} obtained from the previous chapter together with an assumed ambient temperature of 25 °C.

The thermal condition of the bottom surface at the drum center is uncertain because of the complex fixture geometry. Therefore, an adiabatic boundary condition was applied as an assumption.

For the machine and wire feeder surfaces, no experimental temperature data was available. These components were therefore assigned a constant temperature boundary condition of 100 °C.

6.4.3 Forced Cooling Configuration

For the forced cooling configuration, additional inlet boundaries were introduced near the deposition region. These inlets were defined with prescribed mass flow rates and directed toward the newly deposited surface.

In this case, the total mass flow from these inlets corresponded to a volume flow of 400 L/min, which is comparable to the inflow previously entering through the bottom boundary. Since the extraction system remains of similar strength and the bottom inlet is pressure-driven, this results in a reduced inflow from the bottom surface. As a result, the bottom inlet contributes only a small fraction of the total inflow, and the flow field becomes dominated by the injected cooling jets.

6.4.4 Initial Conditions

The first simulations were initialized with stationary fluid with a temperature of 20 °C. The solid domain was initialized with a uniform temperature corresponding to a normalized temperature of 0.54, consistent with the previous case. For subsequent simulations, the solution fields from previous runs were used as initial conditions. This provided a more developed initial flow, pressure, and temperature field, which significantly improved convergence behavior.

6.5 Simulation Cases and Experimental Data

A set of four simulation cases was considered to investigate the influence of both the CFL number and the cooling configuration. The cases include two CFL numbers, 1 and 5, and two cooling conditions, natural convection and forced cooling. An overview of the simulation cases is given in Table 6.3.

In addition to the numerical results, experimental measurements were used for comparison and validation purposes. The corresponding measurements used for comparison are summarized in Table 6.4.

Table 6.3: Overview of simulated cases with varying CFL numbers and cooling type.

Case	Cooling Type	CFL Number	Description
1	Natural	5	Natural convection, high CFL
2	Natural	1	Natural convection, low CFL
3	Forced	5	Forced cooling, high CFL
4	Forced	1	Forced cooling, low CFL

Table 6.4: Normalized experimental data for natural and forced cooling configurations at different times.

No forced cooling			Forced cooling		
Time [min]	Time [s]	Normalized Temp [-]	Time [min]	Time [s]	Normalized Temp [-]
0	0	0.54	0	0	0.56
30	1200	0.91			
39	2340	1.00	38	2277	0.81

6.6 Results and Discussion

The results are shown for all four simulation cases, enabling comparison between different CFL numbers as well as between natural and forced cooling configurations. Data were extracted using multiple probes. In the solid domain, the primary probe was located at the interpass position, with an additional probe placed approximately 10 cm downstream of the laser and at a depth of about 1 cm. In the fluid domain, velocities, temperatures, and temperature gradients were evaluated at positions 1 cm and 5 cm above the melt pool. A comparison between the two CFL cases was obtained by plotting 25 000 iterations for $CFL = 1$ and 5000 iterations for $CFL = 5$. The data were further normalized between 0 and 1 to allow for a clearer comparison of the evolution over iterations. The simulation results are presented in Figure 6.5 and Figure 6.6, while iteration-averaged values over the same interval are reported in Table 6.5.

6.6.1 CFL Sensitivity and Transient Flow Behavior

A comparison of $CFL = 1$ and $CFL = 5$ shows that both cases exhibit very similar behavior when scaled. The oscillations display comparable amplitudes and share a similarly irregular character, with no clear periodicity. This indicates that, although the $CFL = 5$ case initially appeared more oscillatory, the difference is largely related to how quickly the solution evolves between iterations rather than a fundamentally different flow behavior. Specifically, while the $CFL = 5$ case still exhibits faster variations per iteration, the normalized representation reveals that the effective oscillation frequency is in fact quite similar for both cases. Thus, the lower CFL

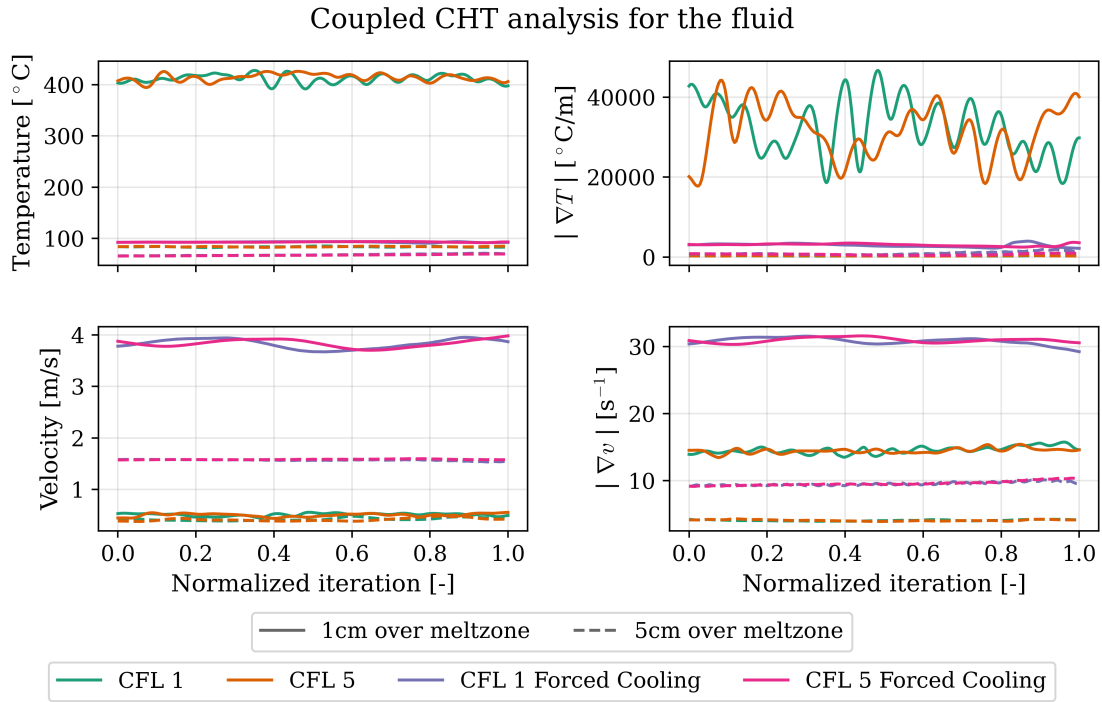


Figure 6.5: Temperature, temperature gradient, velocity, and velocity gradient for all four simulation cases, evaluated at two probe locations positioned 1 cm and 5 cm above the melt pool.

Table 6.5: Average values of the parameters over the selected normalized iteration interval.

		CFL 1 No forced cooling	CFL 5 No forced cooling	CFL 1 Forced cooling	CFL 5 Forced cooling
Massflow [L/min]	Gas extraction outlet	432.59	432.51	430.69	430.81
Temp fluid [°C]	1 cm over meltzone	408.4	413.8	92.2	92.9
	5 cm over meltzone	83.0	83.8	69.7	67.2
∇T [°C/m]	1 cm over meltzone	27156	31521	2737	3074
	5 cm over meltzone	351	331	1461	684
Velocity [°C]	1 cm over meltzone	0.50	0.50	3.89	3.84
	5 cm over meltzone	0.43	0.41	1.55	1.58
∇v [s ⁻¹]	1 cm over meltzone	15.1	14.5	30.1	30.9
	5 cm over meltzone	4.19	4.10	9.78	9.61
Normalized Temp solid [-]	Interpass location	0.82	0.82	0.65	0.65
	[-0.1, 0.02, -0.01] m	1.03	1.03	0.87	0.87
Average heat transfer coefficient [W/m ² K]	All surfaces	10.42	10.35	16.97	17.06
	Side surfaces	11.00	10.93	13.87	13.92
	Top surface	4.80	4.75	46.76	47.21
	Top rear-side (90°-270°)	4.13	4.14	12.99	11.74

number mainly slows down the temporal progression of the solution rather than significantly altering its physical characteristics.

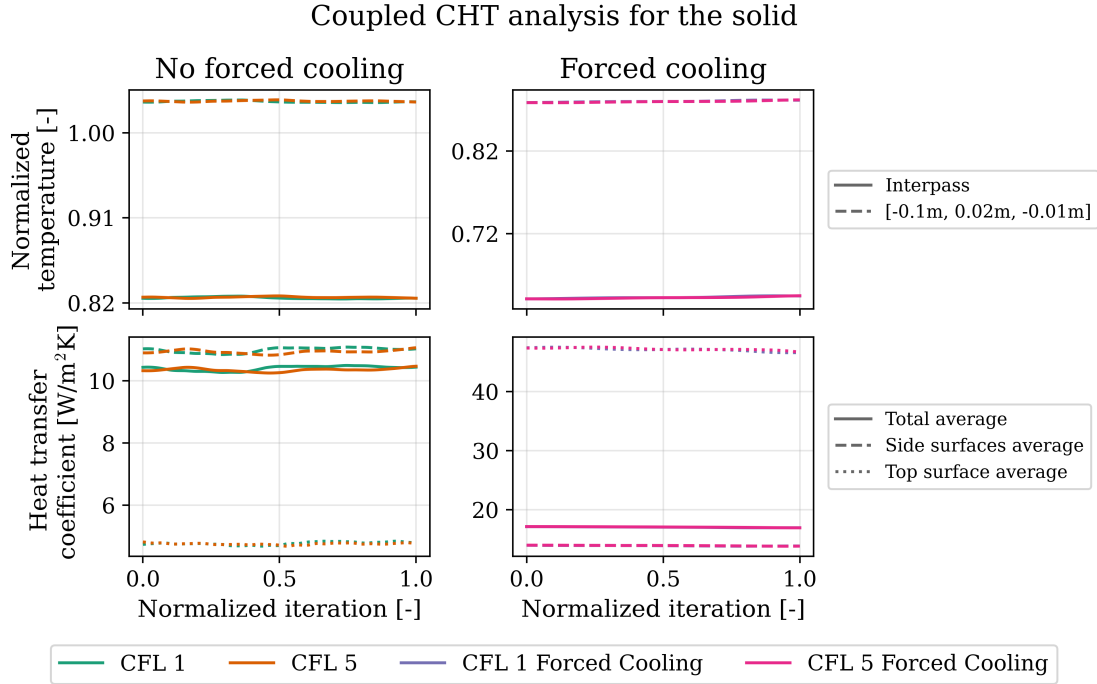


Figure 6.6: Comparison between natural and forced cooling cases. Temperatures at the two solid probe locations (interpass and downstream of the melt zone) are shown together with the heat transfer coefficient, including both values averaged over the entire drum as well as averaged values for the top and side surfaces.

This supports the conclusion that the observed oscillations are primarily physical rather than numerical in origin. Reducing the CFL number does not remove the unsteady behavior, but only changes its rate of development. The flow is therefore inherently unstable, and adjusting numerical parameters such as the CFL number does not lead to a fully steady or converged solution in the classical sense.

It is, however, important to note that different variables exhibit varying sensitivity to these oscillations. For instance, the temperature in the solid region remains relatively stable compared to quantities such as velocity or temperature gradients in the fluid region. This suggests that while the flow field is strongly affected by the instability, some derived or coupled quantities are less sensitive.

Overall, since both CFL numbers lead to similar physical results when properly scaled, the choice of CFL should be guided primarily by computational cost. Although $CFL = 1$ provides a smoother and more gradual evolution, it does not fundamentally change the solution. The $CFL = 5$ case, on the other hand, achieves the same behavior in significantly fewer iterations. Therefore, $CFL = 5$ is the preferable choice, as it provides faster convergence while still capturing the same underlying physical behavior.

6.6.2 Comparison of Solid-Only and Coupled CHT Models

The first aspect to consider is the magnitude and spatial variation of the heat transfer coefficient obtained from the CFD-CHT simulations, and how these compare to previous assumptions. As shown in Figure 6.9 and summarized in Table 6.5, the heat transfer coefficient is significantly higher along the vertical side surfaces compared to the top surface. This behavior can be explained by differences in the flow field. Higher flow velocities are observed along the vertical side walls, driven by both natural convection and the imposed bottom inflow, which promote upward motion. In contrast, the flow near the top surface remains weak and, in some regions, nearly stagnant. As a result, convective heat transfer is enhanced along the sides, while it is reduced on the top surface. In addition to this overall trend, a pronounced increase in heat transfer coefficient is observed close to the melt zone. This is attributed to the very high local temperatures, which induce strong buoyancy-driven flow and create a more complex and dynamic flow field. Furthermore, the presence of the gas extraction outlet modifies the local flow structure, contributing to additional spatial variations in the heat transfer coefficient.

Comparing these results with the assumptions used in the calibration case and the steady-state drum simulations, it is evident that the heat transfer coefficient obtained from the CFD-CHT analysis is significantly higher. This difference is also reflected in the thermal response of the solid. In the CFD-CHT simulation, the interpass temperature is approximately 18 percentage points lower compared to the corresponding solid-only simulation. Since a higher heat transfer coefficient implies more efficient heat transfer from the solid to the surrounding fluid, this naturally leads to reduced solid temperatures.

These results highlight a fundamental difference between the solid-only and CFD-CHT approaches. In the solid-only model, the heat transfer coefficient is prescribed based on simplified assumptions, including a calibrated heat transfer coefficient based on natural convection and a quiescent surrounding gas. In contrast, the CFD model resolves the flow field explicitly, so convective heat transfer develops according to the local velocity and temperature gradients.

As a result, the heat transfer coefficient in the CFD simulations is not imposed or calibrated for a specific case, but instead emerges naturally from the governing physics. This leads to a more realistic and physically consistent representation of the heat transfer, particularly since the CFD model accounts for the actual flow conditions, geometry, and temperature levels present in the system.

6.6.3 Effect of Forced Cooling

The introduction of forced cooling significantly alters the flow field. By injecting argon directly toward the newly deposited material, the velocity field is fundamentally changed, leading to different flow structures and temperature distributions compared to the natural convection case, as can be seen in Figure 6.7. As a result, direct comparison of flow quantities such as velocity magnitude between cases with

and without forced cooling becomes less meaningful as the underlying flow physics differ substantially.

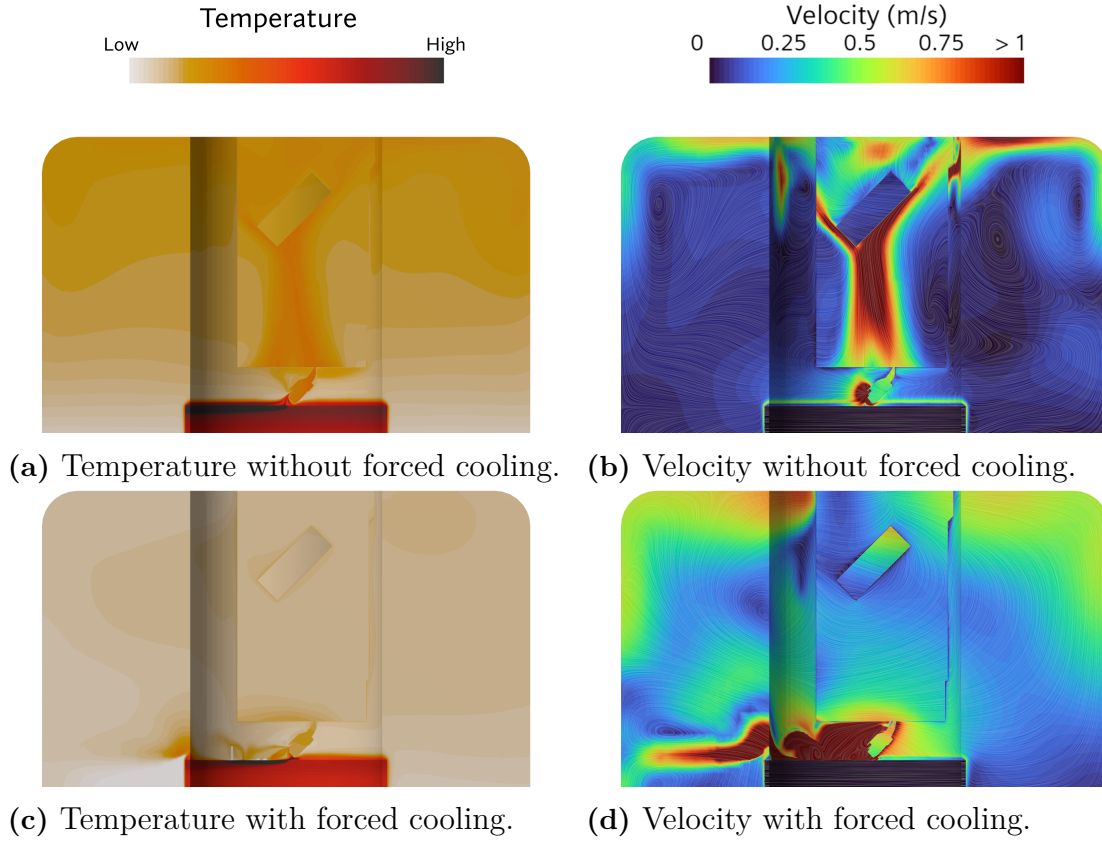


Figure 6.7: Temperature and velocity distribution for CFL 5 with and without forced cooling, shown in a combined cylindrical surface view and cross-sectional view through the center of the drum.

Looking at the discrepancies between experimental data and simulations, it is again observed that the simulations tend to predict lower temperatures.

$$\begin{aligned}
 T^{exp}(\text{without forced cooling}) - T^{sim}(\text{without forced cooling}) &= 18 \text{ percentage points} \\
 T^{exp}(\text{with forced cooling}) - T^{sim}(\text{with forced cooling}) &= 16 \text{ percentage points}
 \end{aligned}
 \tag{6.1}$$

However, this discrepancy is considered to be in reasonable agreement when taking into account the uncertainty associated with the pyrometer measurements. The measurement uncertainty was estimated to be on the order of up to 18% lower, indicating that the experimental temperatures may be overestimated. Within this margin, the difference between simulations and experiments is therefore considered acceptable. Nevertheless, it would be beneficial to perform additional experimental measurements in order to reduce uncertainty and better establish the actual temperature levels.

One of the primary variables of interest is the thermal response of the solid when comparing cases with and without forced cooling, as illustrated in Figure 6.8. In

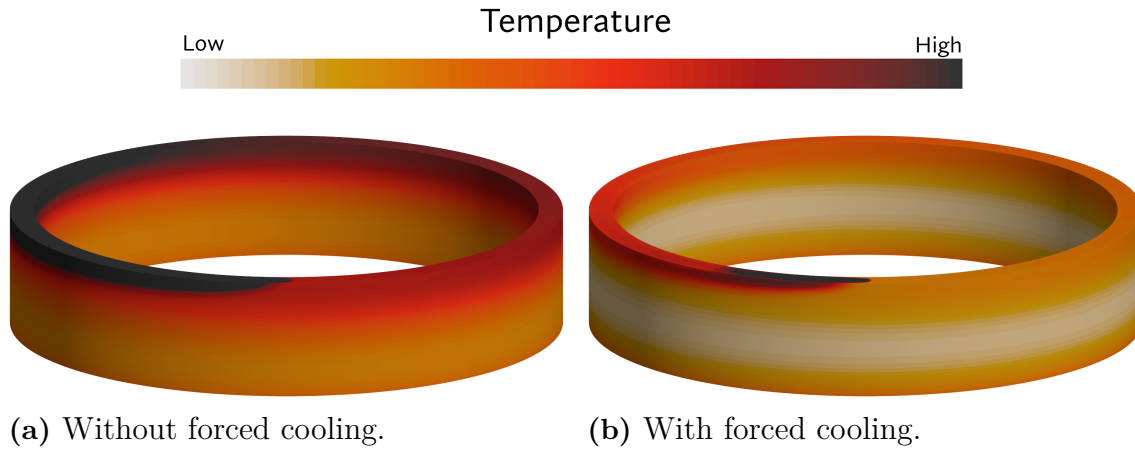


Figure 6.8: Temperature distribution of the drum for CFL 5.

particular, the interpass temperature shows a notable difference of approximately 16 percentage points between with and without forced cooling, indicating that forced cooling has a significant impact on the temperature evolution.

However, when accounting for the uncertainty associated with the pyrometer measurements, and assuming that this measurement error is consistent across both the forced and non-forced cooling cases, a more meaningful comparison can be made.

$$\begin{aligned} T^{exp}(\text{without forced cooling}) - T^{exp}(\text{with forced cooling}) &= 19 \text{ percentage points} \\ T^{sim}(\text{without forced cooling}) - T^{sim}(\text{with forced cooling}) &= 17 \text{ percentage points} \end{aligned} \quad (6.2)$$

This suggests that while there may be inaccuracies in the absolute temperature levels, the relative thermal response is captured more reliably.

An important aspect to highlight is the temperature distribution observed in Figure 6.8b. As can be seen, the temperature is highest near the surface, decreases towards the mid-section, and then increases again near the bottom of the domain. This behavior is a direct consequence of the conduction boundary condition applied at the bottom boundary. This boundary condition was calibrated in Section 5.2.2 to represent the heat transfer to the bottom fixture. However, its validity depends on the thermal state of the system. When forced cooling is introduced, the temperature of the drum is reduced. As a result, the prescribed boundary condition becomes relatively too high compared to the drum temperature. This creates an unphysical situation where heat is effectively introduced into the domain from the bottom boundary, acting as a heat source rather than allowing heat to leave the domain. This behavior was not accounted for in the present model and represents a limitation of the current setup. To obtain a more physically consistent simulation, this aspect should be addressed in future work, for example by redefining the boundary condition to ensure that it remains representative under varying thermal conditions.

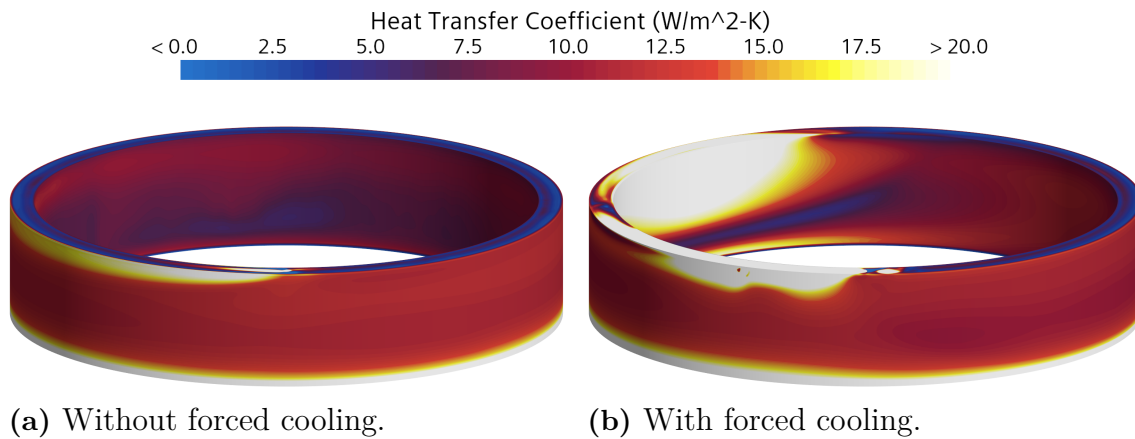


Figure 6.9: Heat transfer coefficient for CFL 5.

The spatial distribution of the heat transfer coefficient for the different cooling configurations is shown in Figure 6.9. As expected, the presence of impinging cooling jets has a pronounced effect on the heat transfer coefficient field, leading to significantly higher local values compared to the case without forced cooling. This is also reflected in the surface-averaged values, where the average increases from $10.35 \text{ W/m}^2\cdot\text{K}$ in the case without forced cooling to $17.06 \text{ W/m}^2\cdot\text{K}$ when forced cooling is applied. The increase is primarily localized to the regions directly impacted by the cooling jets, where the flow impingement strongly enhances convective heat transfer. The increased heat transfer coefficient leads to enhanced convection, which is clearly reflected in the temperature field, where the drum is cooled more efficiently. This demonstrates that directing airflow toward the surface is considerably more effective than supplying a similar amount of air through the bottom.

Overall, the results clearly demonstrate that forced cooling is highly effective in reducing the thermal load on the drum, and that actively directing cooling gas toward the surface is significantly more efficient than relying on bottom inflow alone. This highlights forced cooling as a key strategy for improved thermal control and an important area for further development and optimization.

6.7 Future Work

Despite the insights gained from the CFD analysis, significant further work is required to improve both the accuracy and the overall understanding of the process.

A key area of improvement is model validation. More comprehensive experimental data are needed to ensure the reliability of the simulations, including measurements of inlet and outlet flow rates as well as temperatures of the machine and wire feeder. Additional measurement points on the drum are required, and the measurements must be associated with lower uncertainty. The use of a single interpass temperature is therefore insufficient to fully validate the CFD model, and a more extensive and reliable dataset is required.

In addition, further studies are required to better understand the governing physical processes. This includes investigating the influence of gas temperature and gas flow rate, as well as identifying the optimal location for forced cooling to maximize its effectiveness. A sensitivity analysis or Design of Experiments (DoE) approach could be applied to systematically evaluate the impact of different parameters. This would enable identification of the most influential variables and improve the overall understanding of the process and its controlling mechanisms.

Another important aspect that requires attention is the behavior of smoke generated from the melt pool. Future work should include studying the formation of smoke and its transport within the system, particularly the impact of gas injection on smoke movement. This is closely linked to the need for optimizing the gas extraction system to ensure efficient removal of fumes.

Overall, these areas represent important next steps toward improving model fidelity and enabling process optimization.

7

Conclusion

This work has investigated the thermal behavior of the LMD-w process through a sequence of increasingly complex CFD models, providing both validation and insight into the cooling performance.

A solid-only model of a calibration domain was initially developed to calibrate the boundary conditions used throughout the study. The calibrated model showed close agreement with experimental data, indicating that the selected model setup and calibrated boundary conditions are reliable. Furthermore, the assumption of constant geometry was found to be reasonable for this purpose.

Following the calibration case, the same modelling approach and calibrated boundary conditions were applied to the real drum geometry. This enabled the thermal behaviour of the actual system to be investigated while maintaining the validated setup established during the calibration stage. The transient solid-only model of the real drum demonstrated that the system reaches a quasi-steady state after approximately 30-40 minutes. The steady-state solution was found to closely match the asymptotic temperature reached by the transient simulation, confirming that steady-state assumptions are valid for evaluating the long-term thermal behavior of the system. Furthermore, the steady-state simulations provided comparable results at a significantly lower computational cost, making them a practical alternative for future studies. Although the simulations consistently underpredict the temperatures, the results can still be considered reasonable given the relatively large measurement uncertainty associated with the pyrometer data.

To further increase the physical fidelity of the simulations, the solid-only model was extended to include both fluid and solid regions. This led to a coupled CFD simulation with conjugate heat transfer (CHT), enabling a more detailed assessment of the thermal behavior. The CFD-CHT analysis further confirms the underprediction of absolute temperature levels. Nevertheless, the relative trend between cases without and with forced cooling is well captured. The simulations predict a temperature reduction of 17 percentage points, which is in good agreement with the experimental data showing a reduction of about 19 percentage points. However, further validation is required, particularly through additional experimental measurements.

Overall, the findings confirm the robustness of the modeling approach while also identifying areas for improvement. Future work should focus on enhanced validation and the use of the developed models for process optimization.

Bibliography

- [1] Mohammad Abuabiah et al. “Investigating the Impact of Process Parameters on Bead Geometry in Laser Wire-Feed Metal Additive Manufacturing”. In: *Journal of Manufacturing and Materials Processing* 8.5 (2024). ISSN: 2504-4494. DOI: 10.3390/jmmp8050204. URL: <https://www.mdpi.com/2504-4494/8/5/204>.
- [2] “ASTM52900-15; Standard Terminology for Additive Manufacturing—General Principles—Terminology. ASTM International: West Conshohocken, PA, USA, 2015; Edition 2”. In: (2021).
- [3] T. Balasubramani et al. “Additive Manufacturing of Ti-6 Al-4 V Alloys: Fabrication Techniques and Material Properties-Review”. In: *Transactions of the Indian Institute of Metals* 78.8 (2025), p. 200. ISSN: 0975-1645. DOI: 10.1007/s12666-025-03625-8. URL: <https://doi.org/10.1007/s12666-025-03625-8>.
- [4] Mohamad Bayat et al. “A review of multi-scale and multi-physics simulations of metal additive manufacturing processes with focus on modeling strategies”. In: *Additive Manufacturing* 47 (2021), p. 102278. ISSN: 2214-8604. DOI: <https://doi.org/10.1016/j.addma.2021.102278>. URL: <https://www.sciencedirect.com/science/article/pii/S2214860421004383>.
- [5] T.L. Bergman et al. *Fundamentals of Heat and Mass Transfer*. Wiley, 2020. ISBN: 9781119722489.
- [6] GKN internal documents.
- [7] Steven Dowding. *Best Practices for Modeling Heat Transfer in Simcenter STAR-CCM+*. Accessed: 2026-04-14. 2018. URL: <https://community.sw.siemens.com/s/article/best-practices-for-modeling-heat-transfer-in-simcenter-star-ccm>.
- [8] Panagis Foteinopoulos, Alexios Papacharalampopoulos, and Panagiotis Stavropoulos. “On thermal modeling of Additive Manufacturing processes”. In: *CIRP Journal of Manufacturing Science and Technology* 20 (2018), pp. 66–83. ISSN: 1755-5817. DOI: <https://doi.org/10.1016/j.cirpj.2017.09.007>. URL: <https://www.sciencedirect.com/science/article/pii/S1755581717300494>.
- [9] Samy Hocine, Helena Van Swygenhoven, and Steven Van Petegem. “Verification of selective laser melting heat source models with operando X-ray diffraction data”. In: *Additive Manufacturing* 37 (2021), p. 101747. ISSN: 2214-8604. DOI: <https://doi.org/10.1016/j.addma.2020.101747>. URL: <https://www.sciencedirect.com/science/article/pii/S2214860420311192>.

- [10] Henry Hu and Stavros A Argyropoulos. In: 4.4 (1996), p. 371. DOI: 10.1088/0965-0393/4/4/004. URL: <https://doi.org/10.1088/0965-0393/4/4/004>.
- [11] Bibin John, Senthilkumar Paramasivam, and Sreeja Sadasivan. “Applied and Theoretical Aspects of Conjugate Heat Transfer Analysis: A Review”. In: *Archives of Computational Methods in Engineering* 26 (Jan. 2018). DOI: 10.1007/s11831-018-9252-9.
- [12] Goldak John, Chakravarti Aditya, and Bibby Malcolm. “A new finite element model for welding heat sources”. In: *Metallurgical Transactions B* 15 (1984), pp. 299–305. ISSN: 2379-0229. DOI: <https://doi.org/10.1007/BF02667333>.
- [13] W.P Jones and B.E Launder. “The prediction of laminarization with a two-equation model of turbulence”. In: *International Journal of Heat and Mass Transfer* 15.2 (1972), pp. 301–314. ISSN: 0017-9310. DOI: [https://doi.org/10.1016/0017-9310\(72\)90076-2](https://doi.org/10.1016/0017-9310(72)90076-2). URL: <https://www.sciencedirect.com/science/article/pii/0017931072900762>.
- [14] Jana Kelbassa and Andres Gasser. “Wire Laser Metal Deposition”. In: *Tailored Light 2: Laser Applications*. Ed. by Reinhart Poprawe, Constantin Häfner, and Rolf Wester. Cham: Springer International Publishing, 2024, pp. 437–440. ISBN: 978-3-030-98323-9. DOI: 10.1007/978-3-030-98323-9_12. URL: https://doi.org/10.1007/978-3-030-98323-9_12.
- [15] Navneet Khanna et al. “Cost Modelling for Powder Bed Fusion and Directed Energy Deposition Additive Manufacturing”. In: *Journal of Manufacturing and Materials Processing* 8.4 (2024). ISSN: 2504-4494. DOI: <https://doi.org/10.3390/jmmp8040142>. URL: <https://www.mdpi.com/2504-4494/8/4/142>.
- [16] Hossam A. Kishawy and Ali Hosseini. “Titanium and Titanium Alloys”. In: *Machining Difficult-to-Cut Materials: Basic Principles and Challenges*. Cham: Springer International Publishing, 2019, pp. 55–96. ISBN: 978-3-319-95966-5. DOI: 10.1007/978-3-319-95966-5_3. URL: https://doi.org/10.1007/978-3-319-95966-5_3.
- [17] Philip Kosky et al. “14.3.1.2 Thermal Convection”. In: *Exploring Engineering - An Introduction to Engineering and Design (5th Edition)*. Elsevier, 2021. ISBN: 978-0-12-815073-3. URL: <https://app.knovel.com/hotlink/pdf/id:kt012RML27/exploring-engineering/thermal-convection>.
- [18] E.L. Li et al. “A three-phase model for simulation of heat transfer and melt pool behaviour in laser powder bed fusion process”. In: *Powder Technology* 381 (2021), pp. 298–312. ISSN: 0032-5910. DOI: <https://doi.org/10.1016/j.powtec.2020.11.061>. URL: <https://www.sciencedirect.com/science/article/pii/S0032591020311207>.
- [19] J. R. Lloyd and W. R. Moran. “Natural Convection Adjacent to Horizontal Surface of Various Planforms”. In: *Journal of Heat Transfer* 96.4 (Nov. 1974), pp. 443–447. ISSN: 0022-1481. DOI: 10.1115/1.3450224. eprint: https://asmedigitalcollection.asme.org/heattransfer/article-pdf/96/4/443/5609319/443_1.pdf. URL: <https://doi.org/10.1115/1.3450224>.
- [20] A.P. Mackwood and R.C. Crafer. “Thermal modelling of laser welding and related processes: a literature review”. In: *Optics & Laser Technology* 37.2 (2005), pp. 99–115. ISSN: 0030-3992. DOI: <https://doi.org/10.1016/>

- j.optlastec.2004.02.017. URL: <https://www.sciencedirect.com/science/article/pii/S0030399204000507>.
- [21] Satyapal Mahade et al. "Chapter 7 - Processing of high-performance materials by laser directed energy deposition with wire". In: *Additive Manufacturing of High-Performance Metallic Materials*. Additive Manufacturing Materials and Technologies. Elsevier, 2024, pp. 175–222. ISBN: 978-0-323-91885-5. DOI: <https://doi.org/10.1016/B978-0-323-91885-5.00017-2>. URL: <https://www.sciencedirect.com/science/article/pii/B9780323918855000172>.
 - [22] O. Manca, B. Morrone, and S. Nardini. "Thermal Analysis of Solids at High Peclet Numbers Subjected to Moving Heat Sources". In: *Journal of Heat Transfer* 121.1 (Feb. 1999), pp. 182–186. ISSN: 0022-1481. DOI: 10.1115/1.2825939. URL: <https://doi.org/10.1115/1.2825939>.
 - [23] Ingrid Martorell, Joan Herrero, and Francesc X Grau. "Natural convection from narrow horizontal plates at moderate Rayleigh numbers". In: *International Journal of Heat and Mass Transfer* 46.13 (2003), pp. 2389–2402. ISSN: 0017-9310. DOI: [https://doi.org/10.1016/S0017-9310\(03\)00010-3](https://doi.org/10.1016/S0017-9310(03)00010-3). URL: <https://www.sciencedirect.com/science/article/pii/S0017931003000103>.
 - [24] F. R. Menter. "Two-equation eddy-viscosity turbulence models for engineering applications". In: *AIAA Journal* 32.8 (1994), pp. 1598–1605. DOI: 10.2514/3.12149. URL: <https://doi.org/10.2514/3.12149>.
 - [25] Florian Menter et al. "The SST turbulence model with improved wall treatment for heat transfer predictions in gas turbines". In: *Proceedings of the International Gas Turbine Congress* (Jan. 2003), pp. 2–7.
 - [26] Elham Mirkoochi et al. "Heat Source Modeling in Selective Laser Melting". In: *Materials* 12.13 (2019). ISSN: 1996-1944. DOI: 10.3390/ma12132052. URL: <https://www.mdpi.com/1996-1944/12/13/2052>.
 - [27] Stanley Osher and Ronald P. Fedkiw. "Level Set Methods: An Overview and Some Recent Results". In: *Journal of Computational Physics* 169.2 (2001), pp. 463–502. ISSN: 0021-9991. DOI: <https://doi.org/10.1006/jcph.2000.6636>. URL: <https://www.sciencedirect.com/science/article/pii/S0021999100966361>.
 - [28] Sebastian D. Proell, Wolfgang A. Wall, and Christoph Meier. "On phase change and latent heat models in metal additive manufacturing process simulation". In: *Advanced Modeling and Simulation in Engineering Sciences* 7 (May 2020). ISSN: 2213-7467. DOI: 10.1186/s40323-020-00158-1. URL: <https://doi.org/10.1186/s40323-020-00158-1>.
 - [29] Tahira Raza et al. "Chapter 5 - Processing of high-performance materials by laser beam-powder bed fusion". In: *Additive Manufacturing of High-Performance Metallic Materials*. Additive Manufacturing Materials and Technologies. Elsevier, 2024, pp. 123–154. ISBN: 978-0-323-91885-5. DOI: <https://doi.org/10.1016/B978-0-323-91885-5.00014-7>. URL: <https://www.sciencedirect.com/science/article/pii/B9780323918855000147>.
 - [30] Christoph Reichl et al. "Comparison of Two CFD Approaches Using Constant and Temperature Dependent Heat Capacities during the Phase Transition in PCMs with Experimental and Analytical Results". In: *Processes* 10 (Feb. 2022), p. 302. DOI: 10.3390/pr10020302.

- [31] Martin Schulz et al. “Process Optimization of Wire-Based Laser Metal Deposition of Titanium”. In: *Journal of Engineering for Gas Turbines and Power* 141.5 (Feb. 2019), p. 052102. ISSN: 0742-4795. DOI: 10.1115/1.4041167. URL: <https://doi.org/10.1115/1.4041167>.
- [32] Siemens Digital Industries Software. *Simcenter STAR-CCM+ User Guide, version 2506*. Version 2506. 2025. URL: <https://www.plm.automation.siemens.com/global/en/products/simcenter/STAR-CCM.html>.
- [33] Marián Slodička and Hennie De Schepper. “Determination of the heat-transfer coefficient during solidification of alloys”. In: *Computer Methods in Applied Mechanics and Engineering* 194.2 (2005), pp. 491–498. ISSN: 0045-7825. DOI: <https://doi.org/10.1016/j.cma.2004.04.010>. URL: <https://www.sciencedirect.com/science/article/pii/S0045782504003469>.
- [34] G. Tryggvason et al. “A Front-Tracking Method for the Computations of Multiphase Flow”. In: *Journal of Computational Physics* 169.2 (2001), pp. 708–759. ISSN: 0021-9991. DOI: <https://doi.org/10.1006/jcph.2001.6726>. URL: <https://www.sciencedirect.com/science/article/pii/S0021999101967269>.
- [35] Henk Kaarle Versteeg and Weeratunge Malalasekera. *An introduction to computational fluid dynamics : the finite volume method*. 2. ed. Harlow, England: Pearson Prentice Hall, 2007. ISBN: 9780131274983.
- [36] Joerg Volpp. “Laser light absorption of high-temperature metal surfaces”. In: *Heliyon* 9.10 (2023), e21021. ISSN: 2405-8440. DOI: <https://doi.org/10.1016/j.heliyon.2023.e21021>. URL: <https://www.sciencedirect.com/science/article/pii/S2405844023082294>.
- [37] J. J. Wei, B. Yu, and Y. Kawaguchi. “Simultaneous natural-convection heat transfer above and below an isothermal horizontal thin plate”. In: *Numerical Heat Transfer, Part A: Applications* 44.1 (2003), pp. 39–58. DOI: 10.1080/713838175. URL: <https://doi.org/10.1080/713838175>.
- [38] Jihong Yang et al. “Experimental investigation and 3D finite element prediction of the heat affected zone during laser assisted machining of Ti6Al4V alloy”. In: *Journal of Materials Processing Technology* 210.15 (2010), pp. 2215–2222. ISSN: 0924-0136. DOI: <https://doi.org/10.1016/j.jmatprotec.2010.08.007>. URL: <https://www.sciencedirect.com/science/article/pii/S0924013610002438>.
- [39] Tianlong Zhang and Chain-Tsuan Liu. “Design of titanium alloys by additive manufacturing: A critical review”. In: *Advanced Powder Materials* 1.1 (2022), p. 100014. ISSN: 2772-834X. DOI: <https://doi.org/10.1016/j.apmate.2021.11.001>. URL: <https://www.sciencedirect.com/science/article/pii/S2772834X21000142>.

