

NUMERICAL INVESTIGATION OF THE COOLING ZONE OF A CONTAINER GLASS ANNEALING FURNACE

A Thesis

by

Oğuzhan Aşık

Submitted to the
Graduate School of Sciences and Engineering
in Partial Fulfillment of the Requirements for
the Degree of

Master of Science

in the
Department of Mechanical Engineering

Özyeğin University
February 2024

Copyright © 2024 by Oğuzhan Aşık

NUMERICAL INVESTIGATION OF THE COOLING ZONE OF A CONTAINER GLASS ANNEALING FURNACE

Approved by:

Asst. Prof. Altuğ Melik Başol, Advisor
Dept. of Mechanical Eng.
Özyeğin University

Prof. Dr. M. Pınar Mengüç
Dept. of Mechanical
Özyeğin University

Prof. Dr. Nimeti Döner
Dept. Mechanical Eng.
Gazi University

Date Approved: January 5, 2024



To science and education

ABSTRACT

Annealing, a crucial heat treatment process in the glass manufacturing industry, aims to alleviate the residual thermal stresses induced during the rapid cooling of glass following the forming stage. In the production of container glass, continuous annealing furnaces are employed for this process, consisting of distinct zones tailored to meet specific process requirements. The initial segment of the furnace, the upstream heating zone, is responsible for raising the glassware's temperature to the annealing point. After the heating zone, multiple cooling zones gradually lower the temperature of the glass objects under precise control. To enhance convective cooling rates and ensure uniform air temperatures within the furnace, circulation fans are typically installed in the cooling zones. This study focuses on numerically investigating the influence of circulation airflow rates, namely $0.6 \text{ m}^3/\text{s}$, $0.8 \text{ m}^3/\text{s}$, and $1 \text{ m}^3/\text{s}$, on the cooling performance within an industrial-scale continuous container glass annealing furnace. Two separate solvers are employed in an iterative approach. One solver addresses transient heat conduction within the glass bottles, utilizing radiative and convective heat fluxes as boundary conditions. The other solver is focused on solving the steady convection pattern inside the furnace. Results showed that increasing the circulation fan flow rate enhances the convective heat transfer inside the zone and increases the cooling rate of moving glass bottles. Moreover, increasing the fan flow rate also resulted in a more uniform cooling behavior in glass indicating a reduction in the residual thermal stress levels.

ÖZETÇE

Cam üretim endüstrisinde önemli bir ısı işlem süreci olan tavlama, camın şekillendirme aşamasını takiben hızlı soğutma sürecinden kaynaklanan kalıntı termal gerilmeleri hafifletmeyi amaçlar. Konteyner cam üretiminde, bu süreç için sürekli tavlama fırınları kullanılır ve belirli işlem gereksinimlerini karşılamak üzere uyarlanmış farklı bölgelerden oluşur. Fırının ilk bölümü olan ısıtma bölgesi, cam eşyaların sıcaklığını tavlama noktasına çıkarmaktan sorumludur. Isıtma bölgesinden sonra, birden fazla soğutma bölgesi, cam nesnelerinin sıcaklığını hassas kontrol altında aşamalı olarak düşürür. Fırın içinde konvektif soğutma hızlarını artırmak ve fırın içinde homojen hava sıcaklıklarını sağlamak için genellikle soğutma bölgelerine dolaşım fanları yerleştirilir. Bu çalışma, endüstriyel ölçekte bir sürekli konteyner cam tavlama fırını içinde dolaşım fanlarının hava akış hızlarının, bu projede kullanılan akış hızları $0.6 \text{ m}^3/\text{s}$, $0.8 \text{ m}^3/\text{s}$, ve $1 \text{ m}^3/\text{s}$, soğutma performansına olan etkisini sayısal olarak incelemektedir. İki ayrı çözücü, iteratif bir yaklaşım içinde kullanılmaktadır. Bir çözücü, cam şişeler içinde zamana bağlı ısı iletimini ele alır ve sınırlama koşulları olarak ışıma ve konvektif ısı akılarını kullanır. Diğer çözücü ise fırın içindeki kararlı haldeki konveksiyon dağılımını çözmeye odaklanmıştır. Sonuçlar, fanın akış hızının artırılmasının bölge içinde konvektif ısı transferini artırdığını ve hareket eden cam şişelerinin soğuma hızını artırdığını göstermiştir. Ayrıca, fan akış hızını artırmanın camda daha homojen bir soğuma davranışına yol açtığı göstermiştir.

ACKNOWLEDGEMENTS

I am deeply thankful to my esteemed committee members, whose scholarly expertise and insightful feedback have been invaluable throughout the thesis development. Asst. Prof. Altuğ Melik Başol, Prof. Dr. M. Pınar Mengüç, and Prof. Dr. Nimeti Döner, your commitment to academic excellence has profoundly influenced the depth of this research. A special acknowledgment goes to my supervisor, Asst. Prof. Altuğ Melik Başol at Özyeğin University. Your mentorship, continuous support, and scholarly guidance have been instrumental in shaping the trajectory of this thesis. Your commitment to excellence has been a constant source of inspiration.

To my wife, your unwavering support, patience, and understanding have been my anchor throughout this demanding academic journey. To my mother, my father, and my brother, your enduring belief in my capabilities has fueled my determination to pursue academic excellence. I express my gratitude to my project partners, Gönenc Can Altun and Hazar Şişik. The collaborative efforts and shared insights have significantly enriched the research process at Özyeğin University, and I appreciate the dedication each of you brought to the project.

I extend my thanks to all other friends who have provided emotional support and encouragement. Your friendship has been a source of strength during challenging times. I am grateful for the resources and facilities provided by Özyeğin University, which have been crucial in conducting the necessary research and analysis for this thesis.

TABLE OF CONTENTS

DEDICATION	iii
ABSTRACT	iv
ÖZETÇE	v
ACKNOWLEDGEMENTS	vi
LIST OF TABLES	ix
LIST OF FIGURES	x
I INTRODUCTION	1
1.1 Heat Transfer in Cooling Zones of Continuous Annealing Furnaces .	2
1.1.1 Computational Modeling Approaches and Computational Cost	5
II LITERATURE SURVEY	7
III NUMERICAL METHODOLOGY	12
3.1 Numerical Solvers and Computational Models	13
3.1.1 Convection-Radiation Solver for Air Domain (Steady Domain of the Furnace)	15
3.1.2 Conduction-Radiation Solver for the Transient Solver (Glass Domain)	21
3.2 Coupling of the Solvers	22
IV COMPUTATIONAL STUDY	25
4.1 Definition of the Problem	25
4.2 Computational Furnace Domain	25
V RESULTS	31
5.1 Convergence Monitoring	31
5.2 Furnace	32
5.2.1 Furnace Temperature Distribution	32
5.2.2 Air Temperature in the Furnace Domain	33
5.2.3 Energy Transfer Ratio in Furnace	34

5.3	Bottle	35
5.3.1	Radiative vs. Convective Heat Transfer Rate on the Bottles .	35
5.3.2	Averaged Decrease in Bottle Temperatures and Standard De- viation	37
5.3.3	Bottle Surface Temperatures	38
VI	CONCLUSIONS	41
	REFERENCES	44
	VITA	46



LIST OF TABLES

1	Physics and Models	
	Solid Domain: Glassware inside the furnace (Transient)	
	Air Domain: Atmosphere inside the furnace (Steady)	14
2	scalarSemiImplicitSource in fvOptions	20
3	Fan Model in fvOptions	21
4	Operation conditions of the cooling zone	26
5	Convective and Radiative cooling contribution	35

LIST OF FIGURES

1	General sketch of continuous annealing furnace	1
2	General sketch of air inside the furnace	2
3	Schematic diagram of the continuous annealing process[1]	5
4	Cooling Zone	12
5	Solid Glass (left) and Furnace Air Domains (right)	13
6	Convection-Radiation Solver for Steady Domain (Air Domain of the Furnace)	15
7	One Bottle Simulation	17
8	Pressure vs Velocity Graph	18
9	Forcheimer Coefficient vs Velocity Graph	19
10	Heat Transfer Coefficient For Case 1, Case 2, Case 3	20
11	Schematic of Conduction-Radiation Solver for the Transient Solver (Glass Domain)	22
12	Schematic of coupling solvers	24
13	Size of the cooling zone of annealing furnace	26
14	Demonstration of ventilation fan and air intake locations of the furnace	27
15	Demonstration of ventilation fan and air intake locations of the furnace	27
16	Bottle Mesh	28
17	Air Domain Mesh	29
18	Inner and Outer Walls Mesh	29
19	Porous zone of the bottles	30
20	Air Domain Convergence Monitoring	31
21	Solid Domain Convergence Monitoring	32
22	Outer wall temperature of the furnace	33
23	Inner wall temperature of the furnace	34
24	Axial Cuts Location in the Furnace: 0.6m, 1.5m, 2.5m	34
25	Air temperature Distributions at Axial Cuts of the Furnace: 0.6m, 1.5m, 2.5m	35

26	Bottles Radiative Cooling: Case 1, Case 2, Case 3	36
27	Bottles Convective Cooling: Case 1, Case 2, Case 3	36
28	Mid Bottle Average Decrease in Temperature and Standard Deviation	37
29	Side Bottle Average Decrease in Temperature and Standard Deviation	38
30	Surface temperature field of bottles for the case with the high fan flow rate	39
31	Surface temperature field of bottles for Case 1, Case 2, Case 3 at the last row	40



CHAPTER I

INTRODUCTION

Continuous heat treatment furnaces are a type of thermal processing equipment utilized in industrial processes. In these processes, materials entering from the heating zone are raised to a specific temperature and cooled back down at a precisely controlled rate. The cooling rate of the materials has a very substantial influence on the mechanical properties of the objects. In this sense, the designs of the cooling zones of these furnaces have a major influence on the quality of the heat treatment process.

Annealing is a heat treatment method used in the glass industry, specifically designed to alleviate residual thermal stresses that arise during the uneven heating/-cooling of glass as it solidifies. In container glass manufacturing, annealing takes place in continuous annealing furnaces, divided into three zones. These zones are shown in Figure 1. The first zone, the heating zone, warms the glassware to the required annealing temperature. After heating, the glassware moves to the soaking zone to even out temperature irregularities. Subsequently, the glass enters the cooling zone, where the objective is to gradually cool it to prevent the resurgence of thermal stresses.

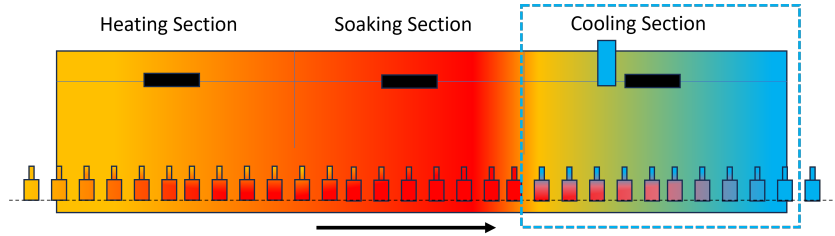


Figure 1: General sketch of continuous annealing furnace

Effective control of the cooling zone's operational parameters significantly impacts the residual thermal stress in the final glass product. Maintaining a uniform temperature distribution inside the cooling zone is important. To ensure temperature

uniformity and augment the cooling rate of the objects circulation fans are usually used in the cooling zones of the furnaces. The main duty of this fan is to drive the air circulation in the furnace and ensure temperature uniformity in the zone. In this regard, the flow rate of the circulation fan has a role in the cooling behavior of the glass in the zone. Numerical models of the furnace can provide insight into the influence of the operational setting of the furnace.

1.1 *Heat Transfer in Cooling Zones of Continuous Annealing Furnaces*

Furnaces are equipped with fans based on their design, and they come in diverse configurations. Within the scope of this thesis, a double-layered furnace design is employed. In double-layered furnaces, radial fans circulate air between these layers, creating a more homogeneous air mixture within the furnace. Finally, a moving conveyor belt is incorporated as a fundamental component to facilitate the movement of materials into the furnace.

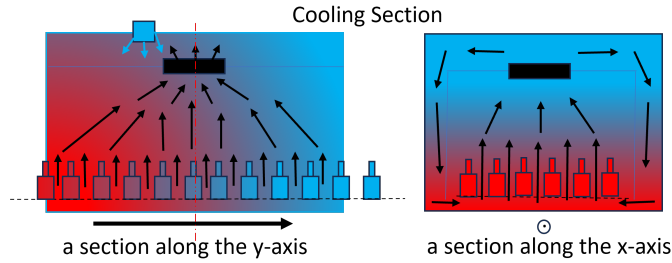


Figure 2: General sketch of air inside the furnace

A radial fan is a mechanical device with an impeller powered by an electrical motor. The impeller gives the air direction while rotating on its own. Usually, the air it operates on is drawn in through the fan's inlet and sucked from the fan's outlet. The fan in the furnace sucks the air from the inner side of the furnace and blows air into the channel which is defined between the inner and outer walls. The used fan in the furnace circulates the air as shown in Figure 2. It is placed at the top of the inner

furnace wall. It creates a pressure difference and forces the air to move inside the layer between the inner and outer furnace walls. Pressure differences are created with the help of the fan to circulate the air. The air circulated by the fan passes through the glassware area. The rotational speed of the fan affects the air velocity. It directly affects the heat transfer coefficient over the glassware surface. At the cooling zone of the annealing furnaces, circulated air has a lower temperature than the glassware. Therefore, the thermal energy of the hot glassware can be extracted with the help of cold air. This information is valuable for optimizing furnace conditions to enhance treatment outcomes. The effect of the circulation fans on the heat transfer inside the furnace is examined with different volumetric fan air flow rates. The thermal energy inside the bottle is extracted with the help of the circulated air in two different ways. Directly convection over the glassware surface and radiation between the glassware surface and furnace walls. It is important to note that, this extracted energy from glassware is always taken cold air. Absorbed energy by the furnace walls is transferred to the cold air with the convection.

When hot glass enters the cooling section, it is faced with a cold furnace surface and furnace air. The energy can be dissipated in two different ways: convection and radiation. Bottles entering the cooling section have hot surfaces. Due to the temperature difference between the furnace and glass bottles, it becomes important to solve the radiation heat transfer between the furnace surfaces and the bottle surfaces.

There are three types of heat transfer modes in the continuous annealing furnace: conduction, convection, and radiation. The energy source within the different zones of the furnace undergoes continuous variations. In the heating zone, glass materials acquire energy through electric heaters or heating chambers used as an energy source. However, in the cooling zone, the hot glass material becomes a source from which energy is dissipated. Incoming glassware has a high temperature at the inlet of the cooling zones. To determine the energy distribution within the furnace, radiation

and convection must be solved simultaneously. Radiative heat fluxes upon the solid surfaces are calculated by

$$q_r = \epsilon \sigma F_{i \rightarrow j} (T_j^4 - T_i^4) \quad (1)$$

where the emissivity according to black surface assumption is 1, view factor $F_{i \rightarrow j}$, source temperature T_j , object temperature T_i , Boltzmann constant σ . The fraction of radiant energy that leaves a surface when it is incident upon another surface is known as the view factor. It is possible to define the view factor relation between surfaces of finite size as:

$$F_{i \rightarrow j} = \int_{A_i} \int_{A_j} \frac{\cos(\theta_1) \cos(\theta_2)}{\pi S^2} dA_j dA_i \quad (2)$$

where S is the distance between the center of the finite areas, θ is the angle between the centers to each surface's normal, and dA is the differential area of surfaces. Convective heat fluxes between air and solid surface are calculated:

$$q_c = h(T_s - T_{inf})dA \quad (3)$$

where h is the heat transfer coefficient, T_s solid surface temperature, T_{inf} air temperature and dA is the differential area of the solid surface.

$$q_{bottle} = q_c + q_r \quad (4)$$

The impact of glassware on heat transfer within the furnace is easily discernible through the equation 4. While the equation may appear straightforward, the calculation of heat transfer through radiation on the right side of the equation becomes significantly more complex due to the influence of the view factor. Moreover, the convective heat transfer coefficient used in the convection calculation varies with different velocities, directly affecting the speed of the flow within the furnace. The view factor, being easily detectable through ray tracing technology commonly used in graphics cards, facilitates the determination of the radiation effect more conveniently.

Associating the heat transfer coefficient with fan speed and determining it through a separate simulation provides more accurate results compared to assuming a fixed heat transfer coefficient.

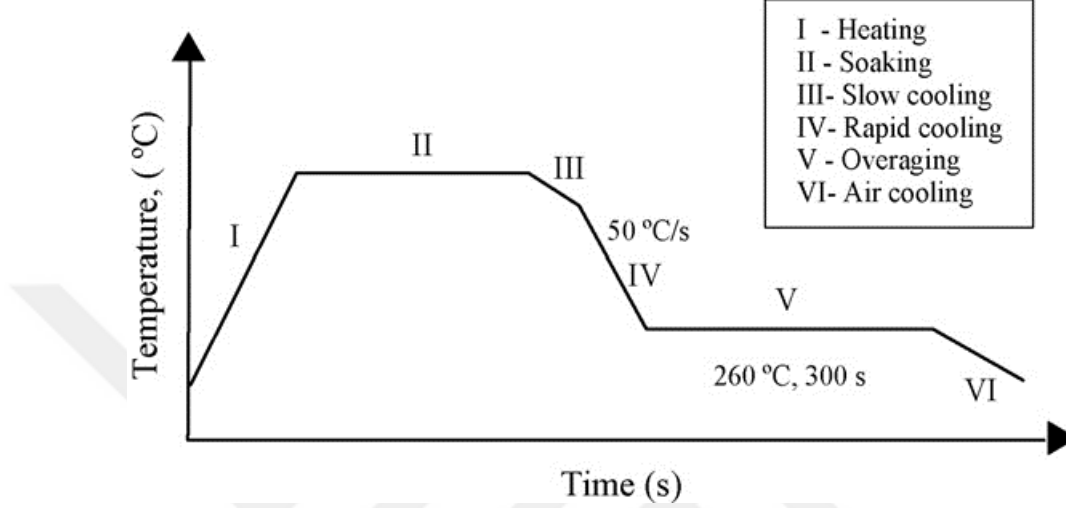


Figure 3: Schematic diagram of the continuous annealing process[1]

Figure 3 illustrates the overall heating and cooling profile of the material within the furnace. The temperature inside the furnace undergoes continuous changes throughout the thermal process. Particularly in the region identified as the slow cooling zone (Zone 3), the change of temperature of the material is controlled. The heat transfer mechanisms responsible for this variation can be accurately computed using CFD programs, allowing for a more detailed analysis of the material.

1.1.1 Computational Modeling Approaches and Computational Cost

Radiation has a high computational cost. For that reason, the computational cost of numerical techniques may be a limiting factor, particularly in large-scale industrial furnaces. Continuous annealing furnaces used in industries are typically longer than 10 meters, wider, and higher than 2 meters. In the realm of physics, various factors come into play when describing these furnaces, such as the three modes of heat transfer, interactions between fluids and the moving solid product, and temperature-dependent thermal properties like thermal conductivity and specific heat capacity.

Representing the intricate details of these physical entities and furnaces in numerical investigations can be computationally challenging, considering the large dimensions involved. To conduct effective numerical investigations on annealing furnaces, it is crucial to employ high-fidelity models. These models should strike a balance, minimizing computational power requirements, ensuring numerical stability, and accurately capturing the continuous and differentiable aspects of the annealing furnace. Cooling processes in these furnaces, which can be time-consuming and material-dependent, add another layer of complexity to numerical simulations. To address these challenges and enhance both computational time and accuracy, GPU-accelerated computing emerges as a promising solution. GPUs, designed specifically for efficient rendering applications, consist of multiple computing units that operate in parallel, reducing the computational cost of processing tasks. With advancements in computational resources and optimized algorithms in code libraries, GPUs have gained popularity in scientific research, particularly in solving problems related to radiative heat transfer. A significant challenge in radiative heat transfer calculations lies in the spontaneous emission of photons in various directions and random emission points. Each photon possesses a unique propagation direction and energy, and as they traverse through a medium, events like absorption, emission, and scattering can alter their direction and energy. GPUs have proven effective in overcoming these challenges, demonstrating considerable speed-up compared to traditional CPUs.

Convection also has a high computational cost. However, the computational cost of the convection over the glassware can be reduced by using porosity-based modeling. Thanks to the porous structure, the amount of mesh used has been reduced. In the mesh of bottles for one row used in this thesis, there are 1.1 million cells of sufficiently good quality. With the addition of the porous structure, this number has been reduced to 5,000 for one row. Additionally, the required computation time has also decreased.

CHAPTER II

LITERATURE SURVEY

Numerous studies in the literature focus on modeling heat treatment processes using Computational Fluid Dynamics (CFD), primarily concentrating on metal and steel annealing. These studies align with the present investigation of the glass annealing process in the sense of furnace operation and heat transfer characteristics. Chapman [2] studied a mathematical model for a batch reheating furnace used in heat treatment. The model considers heat conduction in walls and load, convective heat transfer, and radiative heat transfer using Hottel's zone method. Marlow [3] presents a mathematical model for a direct-fired annealing furnace that helps operators adjust parameters like line speed and gas flow rates when changing the coil type. Gao et al. [4] presented a method to calculate the heating time for metalwork pieces of arbitrary shape. However, in their study, the furnace temperature and heat transfer coefficients were assumed as constant. Tang et al. [5] created an extensive 3D CFD model for a reheating furnace with a walking beam mechanism. They utilized the outcomes of this 3D model to establish the boundary conditions for a 2D Finite Difference model, resulting in a significant reduction in computational time while still maintaining reasonable accuracy. Similarly, Han et al. [6] partitioned the computational domain into multiple sub-regions and determined temperature distributions within each sub-region by applying overall heat balances. This method streamlines the modeling process but assumes a uniform temperature distribution within each sub-area. Another approach developed to deal with computational complexity was undertaken by Prieler et al. [7]. They employed an iterative solution approach, treating gas phase combustion as steady-state and conduction within the billets as transient. The heat fluxes derived

from steady-state simulations were employed as boundary conditions for transient simulations. Depree et al. [8] formulated a comprehensive three-dimensional model to examine the temperature distribution of steel strips within a continuous annealing furnace. The furnace spans 160 meters and is segmented into 20 zones, comprising 12 heating zones and 8 cooling zones, each of which is further subdivided into two zones. The intricate 3D model incorporates the complex geometries of furnace components, including heart rolls, cooling tubes, and steel strips, with a few simplifications. The model addresses the heat conduction equation throughout the entire furnace, encompassing furnace walls, and factors in convective heat loss from the furnace surfaces. Radiative heat transfer calculations employ the hemicube method and encompass all surfaces within the furnace. The temperatures predicted by the 3D model were compared with those measured by thermocouples, revealing good agreement. However, the 3D model demands substantial computational power, requiring 9 hours for a complete solution. To address this computational demand, a simplified 1D/2D model is developed. The insights gained from the 3D model are utilized to enhance the simple models, enabling rapid solutions in approximately 10 seconds on a standard desktop computer, while maintaining high accuracy. In a study cited as [[9]], By building several mathematical models to predict furnace and strip temperature as well as strip recrystallization under both steady and transient conditions, the optimization and enhanced control of the annealing furnace on a continuous galvanizing line were studied. Due to the high level of reflected radiation inside the furnace and its extremely poor measurability of temperatures, a modeling approach is used. The temperature distribution of the steel strip and the furnace thermocouple probes are examined using a 3D model, and the data from the 3D model allows for the creation of a greatly simplified 1D/2D model. The system can simulate furnace operating conditions quickly by combining a steel recrystallization calculation with a 1D/2D model using COMSOL.

Numerical studies focusing on the cooling process of metals also exist in the literature. Hu [10] proposed a model to analyze transient temperature profiles in a row of circular wafers during cooling, revealing significant radial temperature variations and the potential for thermal stresses exceeding the yield stress in silicon wafers, causing plastic deformation. Chen et al. [11] modeled a strip steel cooling process. For the heat transfer coefficient in the cooling zone, there is a strong relationship between jet speed and cooling time. They solve 1-D heat conduction to establish a feasible model. The average error of the model is 4%. The heat transfer coefficient becomes important when the inlet temperature of the strip steel changes. The cooling of strip steel can be adjusted by adjusting the flow of the cooling medium or the operating speed of the strip steel. However, in their experiment, the steel strip was stationary.

There are numerical studies focusing on the fan effect in the furnace. In a laboratory drying oven, Reference [12] conducted experimental research to enhance temperature uniformity by modifying the ventilation system. Utilizing Computational Fluid Dynamics (CFD) simulations, the study examined how adjusting the position of heaters and deflectors could optimize air-flow redirection. The heated air was directed precisely onto the storage space of the equipment, and the authors investigated the impact of varying fan speeds on temperature distribution. Four different speeds were experimented with 1500 rpm, 2000 rpm (the existing fan), 2500 rpm, and 3000 rpm. The findings indicated that higher fan rotation speeds correlated with reduced temperature differences. Nevertheless, the improvement observed was marginal, approximately 0.1 K, suggesting a minimal impact. The study recommended employing CFD to design geometric prototypes ensuring temperature uniformity in convection ovens. However, the size of laboratory drying furnace is significantly smaller than industrial furnaces. Ivan et al. [13] investigated the impact of fans on electric steel tempering furnaces. They conducted simulations at four different speeds: 720, 990,

1350, and 1800 rpm. Elevated rotation speeds led to enhanced thermal uniformity, attributed to the fan-induced re-circulation zone improving flow mixing and heat transfer. Nevertheless, a noteworthy finding was the reduction in the thermal efficiency of the tempering furnace with increasing rotation speeds. Therefore, it is essential to balance these features for optimal results when managing fan speed. For instance, despite a 20% decrease in thermal efficiency when doubling the rotation speed, there was simultaneously up to a 50% increase in heat transfer to the load. This trade-off can be advantageous for reducing the tempering process duration. However, the computational time for each run was extended to 100 hours due to the utilization of a 3M cell in the mesh. Lee et al. [14] addressed the simultaneous effects of conduction, convection, and radiation by employing a discrete-ordered method for radiation calculations. The fan's role in facilitating flow mixing and convective heat transfer within the furnace was identified as crucial. They modeled the fan region using MRF (moving reference frame). Elevating the fan speed proved instrumental in enhancing convective heat transfer, consequently improving temperature uniformity. Additionally, introducing swirl flow to the inlet gas resulted in enhanced temperature uniformity in the lower furnace zone.

In the cooling zone of heat treatment processes for both metals and glass, thermal radiation plays a significant role in heat transfer primarily due to the elevated temperatures of the materials involved. Consequently, the accuracy of the results is influenced by the reliability of the radiation model. One approach used to solve radiative heat transfer is the Monte Carlo (MC) method, which has gained widespread use in multi-core and many-core computer architectures. Silvestri and Pecnik [15] applied the MC method to a non-grey participating medium, achieving three orders of magnitude increase in speed on a single graphic card (GPU) compared to the central processing unit (CPU). Additionally, Martinsen et al. [16] significantly accelerated their Monte Carlo solver, designed for calculating photon transport in turbid

media, by utilizing a high-end graphics card, resulting in seventy times compared to single-threaded desktop computer implementation.



CHAPTER III

NUMERICAL METHODOLOGY

This study focuses on the thermal analysis of the cooling zone of a continuous glass annealing furnace. In these continuous furnaces glass objects are transferred inside the furnace on a moving conveyor belt. The process is transient from the moving glass object's point of view. There is a substantial change in the glass object temperature between the inlet and outlet of the zone. Moreover, the thermal conditions inside the furnace also vary depending on the position inside the furnace. So, while the glass objects move through the furnace they are faced with varying thermal conditions inside the furnace, which needs to be considered as an additional transient effect in the modeling approach. On the other side, the furnace itself has a rather steady temperature distribution inside the furnace. The zone temperatures of the furnace are kept at a constant level in the normal operation of these furnaces. So, this differencing in the thermal conditions between the moving glass objects and the furnace necessitates using different approaches for the modeling of these two domains. In this regard, the entire computational domain shown in Figure 4 is split into two separate domains as shown in Figure 5.

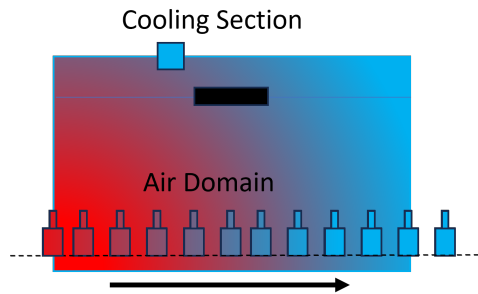


Figure 4: Cooling Zone

The solid glass region shown on the left-hand side in Figure 5 is solved in a moving

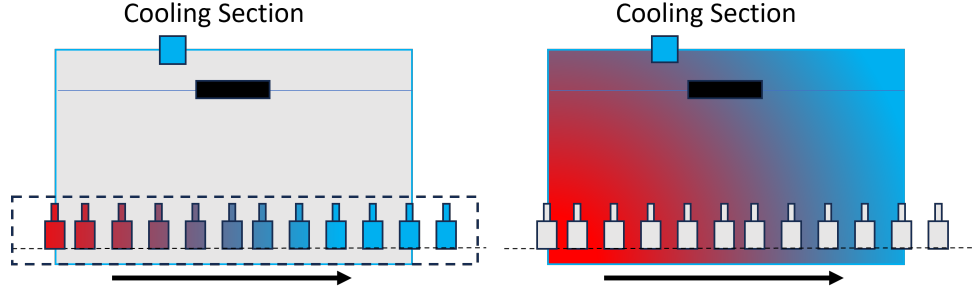


Figure 5: Solid Glass (left) and Furnace Air Domains (right)

frame of reference using a transient conduction solver. At the glass boundary, the convective and radiation heat transfer is imposed as heat fluxes. For the modeling of the radiation heat transfer, a surface-to-surface radiation model is used. For the thermal analysis of the furnace air domain, a steady convective flow solver is utilized. Here, the radiation heat transfer between the furnace walls and between the glass objects and the furnace walls is calculated using a surface-to-surface radiation model. The air domain region is shown in Figure 5. This region includes the furnace atmosphere, furnace walls, and bottle surfaces.

3.1 Numerical Solvers and Computational Models

In the present investigation, the computational domain is discretized into distinct solid and air domains. Using physics and models in the solvers shown in Table 1. Two distinct solvers are employed to address the solution within each of these domains. The solid domain is characterized by the application of a transient heat conduction solver in the moving frame of reference. Specifically, boundary conditions representing radiation and convection are imposed at the interface to account for heat transfer between the glass and furnace. Concurrently, the air domain is subjected to the solution of compressible Reynolds-Averaged Navier-Stokes (RANS) equations using a steady solver within the stationary frame of reference. To enhance computational efficiency in simulating the air domain and mitigate the influence of glass rows on the convective patterns within the furnace, a porosity-based approach is adopted.

Table 1: Physics and Models

Solid Domain: Glassware inside the furnace (Transient)

Air Domain: Atmosphere inside the furnace (Steady)

	Solid Domain	Air Domain
Heat Transfer Physics	Unsteady Conduction	Steady Convection
Governing Equations	Transient Heat Conduction	Compressible RANS with $k-\omega$ turb. Model with wall functions
Frame of reference	Moving	Stationary
Thermal BC's	Radiation and Convection	Radiation and Convection
Velocity BC's	No slip	No slip
Additional Models	-	Porosity based modeling for glass rows
Solver	OpenFoam with in-house Radiation Model	OpenFoam with in-house Radiation Model

Both solvers are implemented using the OpenFOAM platform, incorporating diverse radiation solvers for comprehensive analysis.

Within the furnace, separate regions exist that enable separate analyses. These regions can be classified as in Figure 5 the regions of the air, defined as the furnace domain, and solids, specifically defined as the areas occupied by glass bottles. Hot glassware entering the furnace can be subjected to heat transfer through two modes, particularly relevant in the cooling zone. These modes are facilitated by the cold furnace walls and the relatively cool air within the furnace. The cold furnace walls enable heat transfer from the glass surfaces through radiation. Simultaneously, the cool air inside the furnace engages in convective heat removal from the glass by coming into contact with its surface. Therefore, the hot glass surfaces interact with the boundary regions within the furnace, comprising the cold walls and the cool furnace air. It is essential to jointly analyze these two regions to determine the combined

heat flux effects on the glass. To achieve this, we employed a convection-radiation solver developed for this purpose. On the other hand, the impact of heat transmitted through contacts within the glass needs separate consideration. Heat removal from the bottle surface reduces its temperature, creating temperature differences within the solid bottle. Over time, heat transfer occurs through contact from these hot regions towards the surfaces. Hence, a solid region solver was developed to address temperatures within the solid region. Detecting the energy removed from the surface is essential for determining this heat transfer. The convection and radiation solver mentioned above provides the necessary information for the solid glass domain solver to facilitate heat transfer through contact within the glass. Further details on each solver are provided in the respective solver descriptions below.

3.1.1 Convection-Radiation Solver for Air Domain (Steady Domain of the Furnace)

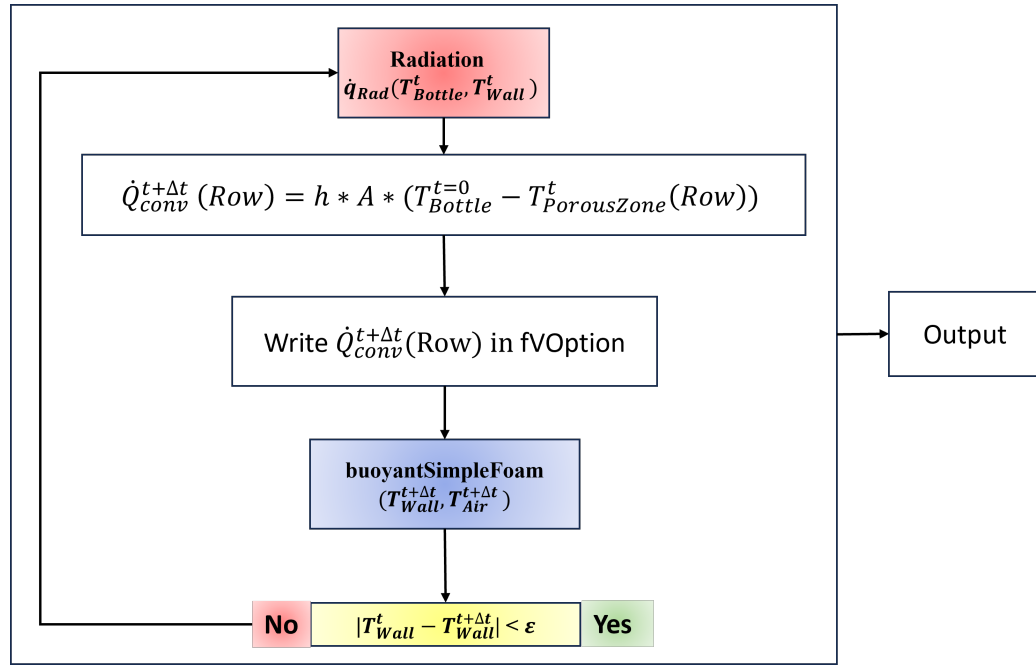


Figure 6: Convection-Radiation Solver for Steady Domain (Air Domain of the Furnace)

The developed convection-radiation solver is employed to determine furnace temperatures using assumed steady-state bottle temperatures. To provide a comprehensive overview, the solver is initially defined using assumed bottle temperatures, leading to the determination of both furnace and internal air temperatures. Subsequently, these temperatures are utilized to ascertain bottle temperatures. The interrelationships between the solvers will be elucidated in greater detail following their descriptions. To solve convective heat transfer inside the furnace, the flow behavior needs to be solved. For that reason, `buoyantSimpleFoam` is used. This solver can be used for steady-state, compressible, turbulence, and heat transfer[17]. To account for turbulence in the flow, the $k - \omega$ turbulence model was employed.

Examining the convection-radiation solver more closely, this solver is designed to determine wall temperatures using the Darcy-Forchheimer model, aiming to reduce computation time by avoiding the direct utilization of bottle geometry within the furnace. Assuming the furnace is filled, each row of bottles is individually modeled as a porous region, thereby mitigating challenges associated with the large dimensions of industrial furnaces and numerous bottle geometries. The Darcy-Forchheimer model employed is explained in more detail below.

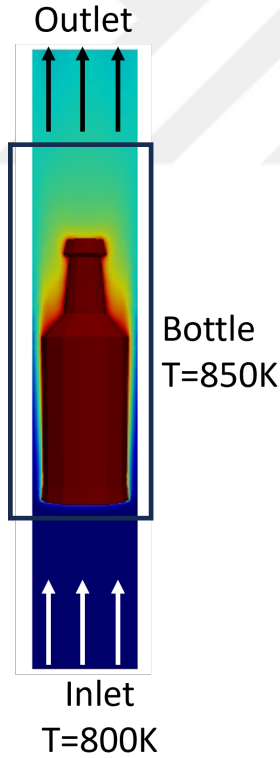
The rows of bottles, defined as porous structures, also facilitate the interaction of the two separate solvers embedded within the convection-radiation solver. These porous structures are simultaneously utilized as heat sources. Each defined row serves as an energy source, enabling the transfer of convected heat from the region with defined bottle temperatures to the air region. More detailed information on this process is provided in the relevant section.

Furthermore, a momentum source is employed to capture the fan's impact rather than directly modeling the fan region. In the steady-state simulation, the direct unstable effects of the fan are disregarded using this approach. However, it is noteworthy

that neglecting the swirl effect due to this modeling approach may influence the results. Detailed information regarding the data utilized for fan regions is explained in greater detail in the relevant section below.

3.1.1.1 Porosity-Based Blockage and Convective Heat Transfer Model

In the furnace domain, the bottle geometry repeats itself 435 times. Using all of the bottles to create a mesh of the furnace environment is computationally impossible because of the number of cells in the one-bottle mesh. By using 1 bottle geometry, a simulation was created. The created simulation geometry is shown with inlet, outlet, and bottle locations in Figure 7



The inlet velocity is defined as 0.35m/s and the temperature is set at 800K. Bottle surface is defined as an isothermal wall that has 850K temperature. The created mesh has 1.7M cells to find the heat transfer coefficient and pressure drop that is created because of the bottle. With changing the inlet velocity, heat transfer coefficients are found for different fan speeds.

$$\Delta p = (\mu * D + 0.5 * \rho * F * U) * U \quad (5)$$

In the equation 5, the first term on the right-hand side represents the Darcy contribution, considered zero. The second term represents the

Figure 7: One Bottle Simulation Forchheimer contribution, which accounts for the nonlinear resistance to flow. The parameter F was derived from external CFD simulations' single bottle simulations.

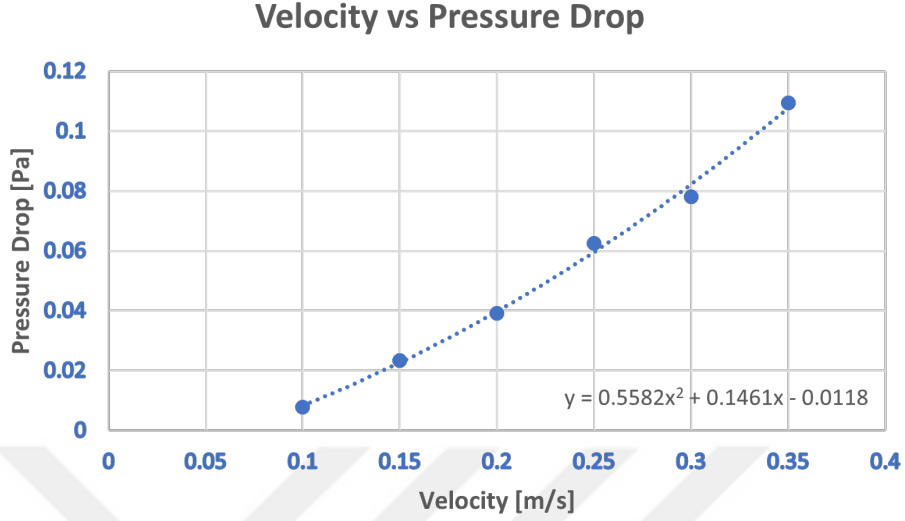


Figure 8: Pressure vs Velocity Graph

Upon examination of Figure 8, it becomes evident that a second-degree polynomial equation can represent the variation in pressure. This observation suggests the appropriateness of utilizing the Darcy-Forchheimer model, as the relationship between pressure and the square of velocity is apparent. This alignment supports the use of the Darcy-Forchheimer model, given the proportional nature between pressure and the square of velocity. In OpenFOAM, solvers have been prepared using the Darcy-Forchheimer model[18]. Utilizing this model, each row of bottles in Figure 19 is defined as a porous region. For each bottle geometry, the Forchheimer coefficient is calculated based on the pressure loss created by six different velocities, employing the above mentioned equation 5.

In Figure 9 the Forchheimer coefficient was found as 16. Utilizing this value, porous structures in the glass regions have been characterized. Figure also exhibits a distribution along a linear trend, given the proportional relationship between the Forchheimer coefficient and the square of velocity.

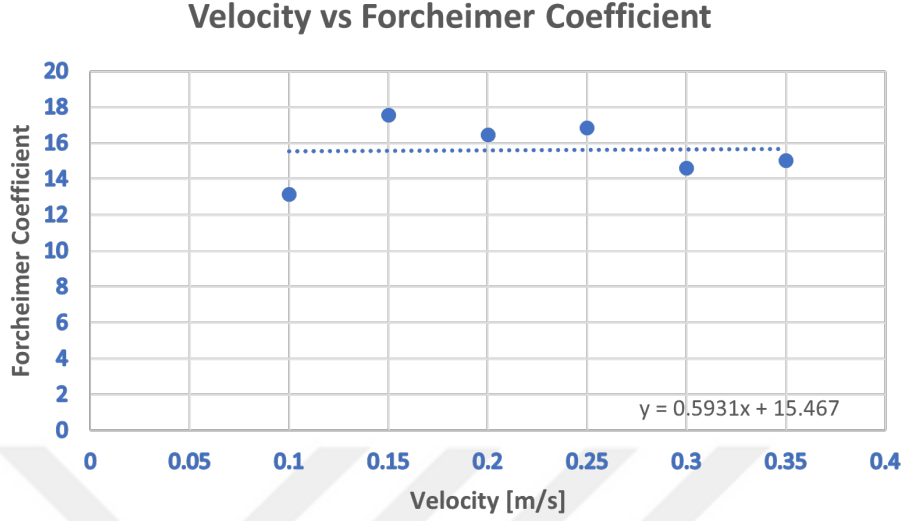


Figure 9: Forcheimer Coefficient vs Velocity Graph

3.1.1.2 Heat Transfer Model in Porous Zones

$$\dot{Q}_{bottle} = h * A * (T_{bottle} - \bar{T}_{PorousZone}(row)) \quad (6)$$

The convective heat transfer on the surface of glass bottles was calculated using Newton's law of cooling. In the equation 6, the non-uniform convective heat transfer h was determined for a single glass bottle CFD studies conducted in three cases. Convective heat transfer coefficients derived from this simulation were imposed on all the bottles inside the furnace. The simulations' results are depicted in Figure 10 below. According to this figure, as expected, the highest heat transfer coefficient is observed at the highest fan speed. In addition, T_{bottle} in Equation 6 represents the local surface temperature of the glass bottles and $\bar{T}_{porousZone}(row)$ represents the mean temperature of the corresponding porous zone for the glass bottle rows. In OpenFOAM, the fvOptions file is used to define the heat source/sink term. scalarSemiImplicitSource [19] utility is used. Cell zone is defined in the topoSet dictionary to create the heat source/sink region. The created file is shown in Table 2. The selection mode is defined as cellZone. With using related row cell zone (in Table 2 just shown for row 1), each rows are defined separately. in the fvOptions file, there are 29 different heat

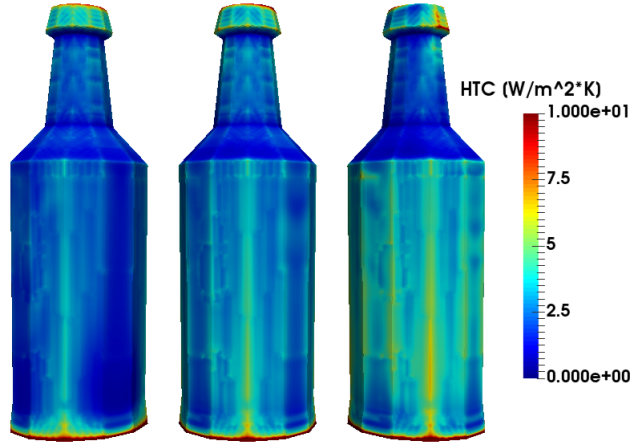


Figure 10: Heat Transfer Coefficient For Case 1, Case 2, Case 3

source/sink terms are used. `volumeMode` allows the user to set whether the source is indicated as an absolute quantity (`absolute`) or a quantity per unit of volume (`specific`). Finally within `injectionRateSuSp` the `Su` and `Sp` coefficients are indicated after the name of the field of the corresponding equation (`h`). In this case, since the specific mode is used.

Table 2: `scalarSemiImplicitSource` in `fvOptions`

```
rowH1
{
    type                scalarSemiImplicitSource;
    active              yes;
    scalarSemiImplicitSourceCoeffs
    {
        timeStart      5000;
        duration        50000.0;
        selectionMode   cellZone;
        cellZone        rown1;
        volumeMode      specific;
        injectionRateSuSp
        {
            h            (9149 0);
        }
    }
}
```

3.1.1.3 Fan Model in fvOptions

The meanVelocityForce model was used to define the fan region. The model used is shown in the table below for one fan zone only. The region to be selected for selectionMode is defined as cellZone, and this region was created geometrically in topoSetDic. The speed of the air passing through the region is defined differently in the z direction (the notation for the fan speed of $0.6 \text{ m}^3/s$ is used in the table).

Table 3: Fan Model in fvOptions

momentumSource1	
{	
type	meanVelocityForce;
active	yes;
meanVelocityForceCoeffs	
{	
selectionMode	cellZone;
cellZone	fanCellZone1;
fields	(U);
Ubar	(0 0 15);
relaxation	1;
}	
}	

3.1.2 Conduction-Radiation Solver for the Transient Solver (Glass Domain)

For the solution of the transient heat conduction solver, the laplacianFoam solver [20] in OpenFoam was used. The solver uses the backward Euler method for the time discretization and the central differencing scheme for the spatial discretization. An in-house developed surface-to-surface radiation model solved the radiation heat transfer to the glass walls [21]. The created approach partitions the computational field into row sets, in contrast to the commercial solvers that handle the computational field as a whole. Since there are seven rows of bottles in the computational domain, the division in this investigation is set to seven. In doing so, rather than calculating the radiation and conduction calculations all at once, they are done row by row. From

the first bottle row (first row) to the last bottle row (seventh row), the bottle rows are calculated in a sequential order. First, the radiation solver is programmed with the surface temperatures of each bottle and the furnace wall. Second, a conduction solver is calibrated using the first bottle's internal temperature. Thirdly, using the surface temperatures of every participant in the domain at time t , the radiative heat fluxes that happened on the first bottle row's surface are computed. The temperature distribution in the time $t + \Delta t$ is computed by solving the transient heat conduction equation while maintaining the computed radiative fluxes constant during the Δt time step. The first bottle row is reserved for storage immediately following the new temperature field. The conduction solver is tasked with calculating the internal temperature of the second bottle row once the first bottle row has been solved. Until every bottle row was resolved, this process was repeated. Following the resolution of each bottle row, the radiation solver maps the new surface temperature field onto the bottles, and the bottles are pushed forward through the furnace.

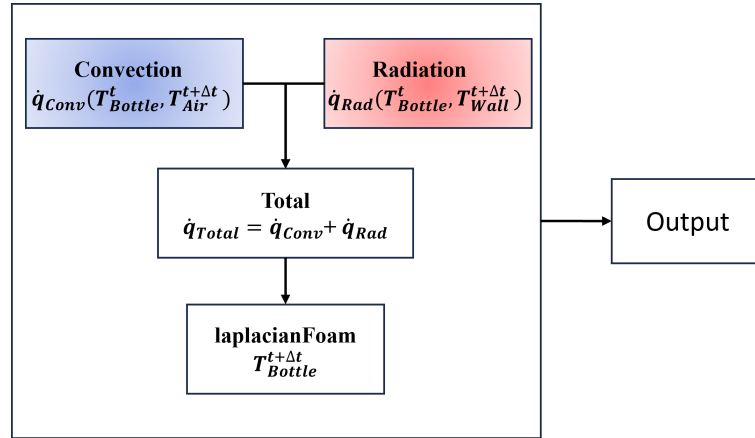


Figure 11: Schematic of Conduction-Radiation Solver for the Transient Solver (Glass Domain)

3.2 Coupling of the Solvers

Thermal analysis of the cooling zone requires a conjugate heat transfer solver. The solid glass domain requires the solution of the transient conduction equation with

the convective and radiative boundary conditions imposed on the glass walls. On the other hand, the furnace domain requires a convection solver with radiation boundary conditions on the walls. However, due to the movement of the glass bottles inside the zone, these two separate domains cannot be solved in a fully coupled way without modeling the movement of the bottles with remeshing. Instead, a segregated modeling approach was chosen in which the solid and air domains were solved sequentially in an iterative approach by transferring the required boundary conditions between the domains. The flowchart of the iterative numerical approach is provided in Figure 12. The information sharing between these two solvers is made possible by their constant solution of the glass and wall temperatures. First, the initial conditions that are given keep the glass temperature constant, and wall temperatures are calculated using the steady solver. Until the temperature difference between successive iterations drops below a predetermined threshold, it iteratively looks for results. If the condition is not satisfied after 200 iterations, the temperatures of the walls from the 200th iteration are regarded as the final result. The temperatures of the bottles are then determined using a transient solver employing these wall temperatures. The operation of the two solvers is completed when the temperature difference between successive iterations of the glass falls below a specific threshold, resulting in the achievement of the ultimate result.

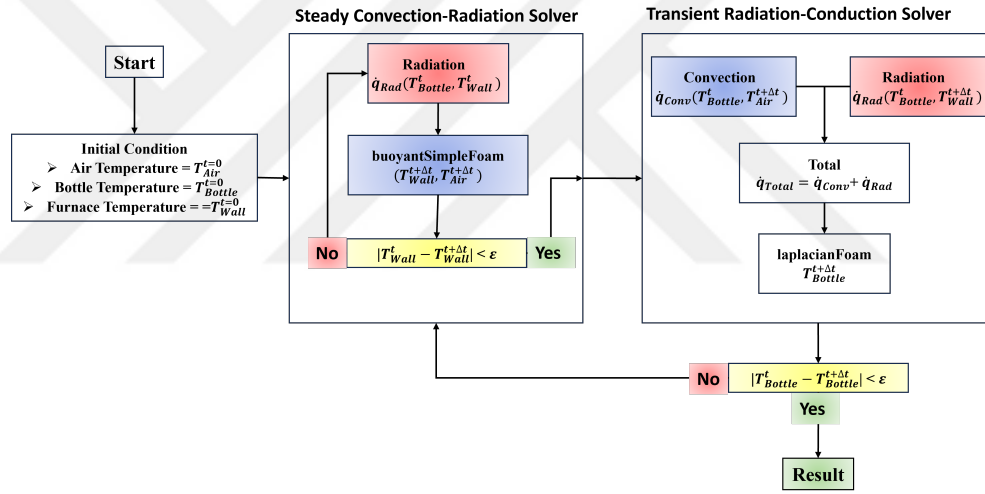


Figure 12: Schematic of coupling solvers

CHAPTER IV

COMPUTATIONAL STUDY

4.1 Definition of the Problem

Temperature data for materials entering the furnace offers important information about the level of quality of the final product. It is not feasible to measure glass bottle temperatures in real-world scenarios by experimentation for each zone. Consequently, glass temperatures were measured, considering each of the three heat transfer modes. This information makes it easier to comprehend the bottle's temperature distribution, which allows furnace operations to be optimized. Building on this idea, the study's main goal is to find out how the furnace's fan speed affects objects.

4.2 Computational Furnace Domain

The cooling zone of the annealing furnace was constructed with air intakes, inlet, and outlet as depicted in Figure 13. The furnace walls have a double-layer structure. This double-wall layer structure is used to circulate the furnace air with the use of a circulation fan mounted on top of the furnace. Inside the zone, there are 29 rows of glass bottles, and each row consists of fifteen glass bottles. The furnace has 3.10-meter length, 1.00-meter height, and 2.16-meter width. The operational parameters of the cooling zone for cases are provided in Table 4. Cases work under the same operational conditions except for the flow rate of the circulation fan. The thermal equilibrium condition reached at the end of the soaking zone between the glass and air is imposed at the inlet of the cooling zone. In this regard, glass bottles and air are set to the same temperature at the cooling zone inlet. To lower the furnace temperature cooling air is injected from the two intakes placed at the top of the furnace. Circulation fans

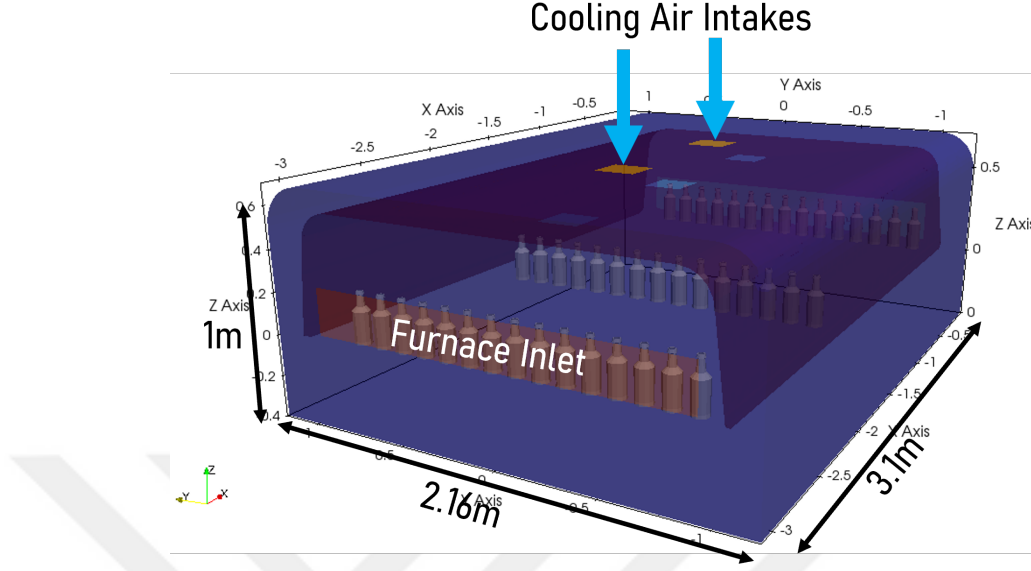


Figure 13: Size of the cooling zone of annealing furnace

Table 4: Operation conditions of the cooling zone

	Case 1	Case 2	Case 3
Bottle Temperatures at the Inlet	850 K	850 K	850 K
Air Temperature at Zone Inlet	850 K	850 K	850 K
Air Flow Rate at the Zone Inlet	$1 \text{ m}^3/\text{s}$	$1 \text{ m}^3/\text{s}$	$1 \text{ m}^3/\text{s}$
Flow rate of circulation fans	$0.6 \text{ m}^3/\text{s}$	$0.8 \text{ m}^3/\text{s}$	$1 \text{ m}^3/\text{s}$
Total Cooling Air Flow Rate	$0.8 \text{ m}^3/\text{s}$	$0.8 \text{ m}^3/\text{s}$	$0.8 \text{ m}^3/\text{s}$
Cooling Air Temperature	300K	300K	300K
Conveyor belt speed	0.011 m/s	0.011 m/s	0.011 m/s

are mounted at the top of the inner furnace. The positioning of the air intake and circulation fans can be seen in Figure 14. Circulation fans were modeled using source terms for the momentum equations in the solver. Circulation fans direct hot air ingested from the inner furnace region into the channel between the inner and outer furnace walls where it mixes with the cooling air injected from the air intakes. Finally, the mixed air is directed to the inner furnace.

The air intake locations, fan positions, glass movement directions, and furnace inlet and outlet are all depicted in figure 14. The cold air coming from the air intakes lowers the furnace's air temperature. High-temperature glasses enter the furnace and

lose energy during their trajectories due to radiation and convection, which heats the air inside the furnace as well as the walls.

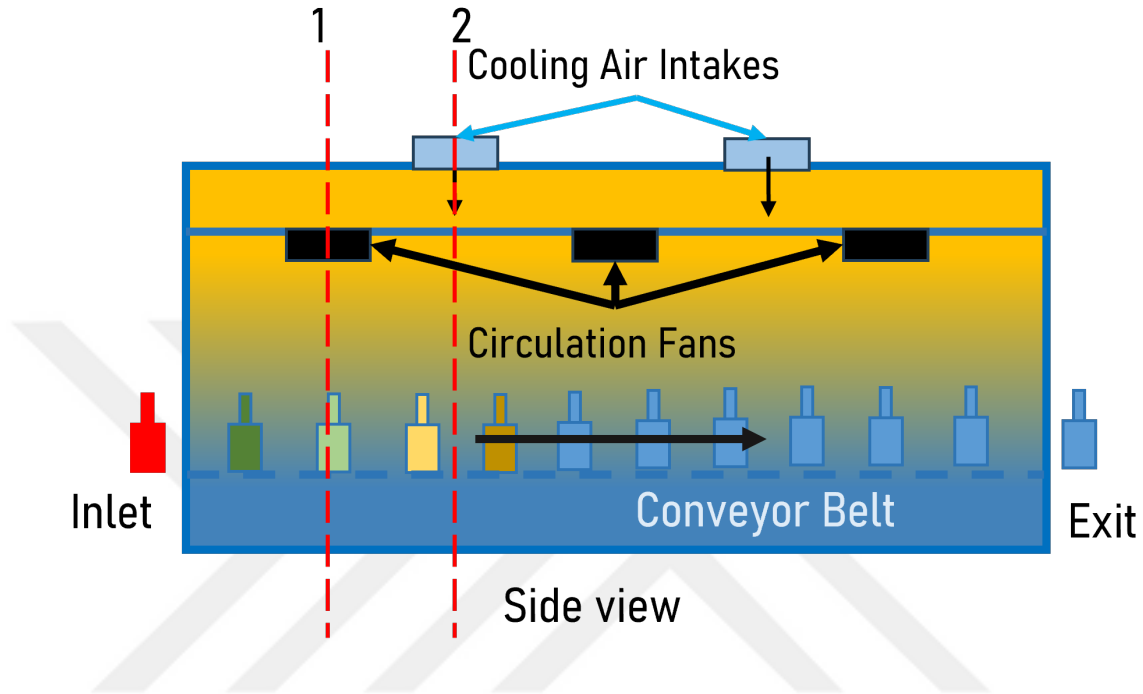


Figure 14: Demonstration of ventilation fan and air intake locations of the furnace

The front views depicted in Figure 15 illustrate the behavior of air in the fan section and air intakes section. The fan facilitates the mixing of cold air coming from the air intakes with the air inside the furnace.

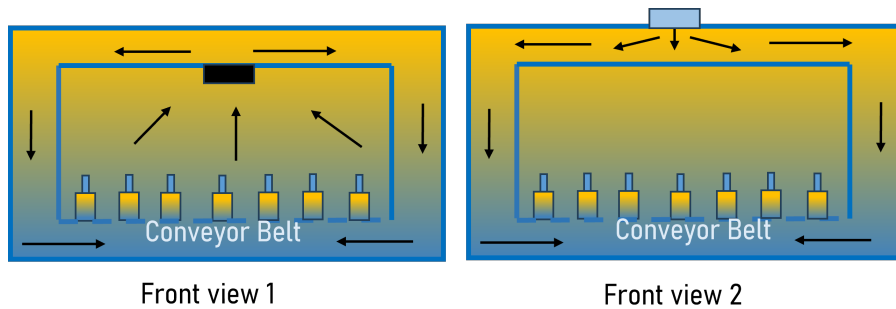


Figure 15: Demonstration of ventilation fan and air intake locations of the furnace

The mesh for the glass bottles was created using the commercial mesh generator Ansys IcemCFD (Figure 16). A multi-block structured mesh topology was employed.

Ten cells have been used across the thickness of the bottle. The same bottle mesh was used for all the bottles in the furnace.

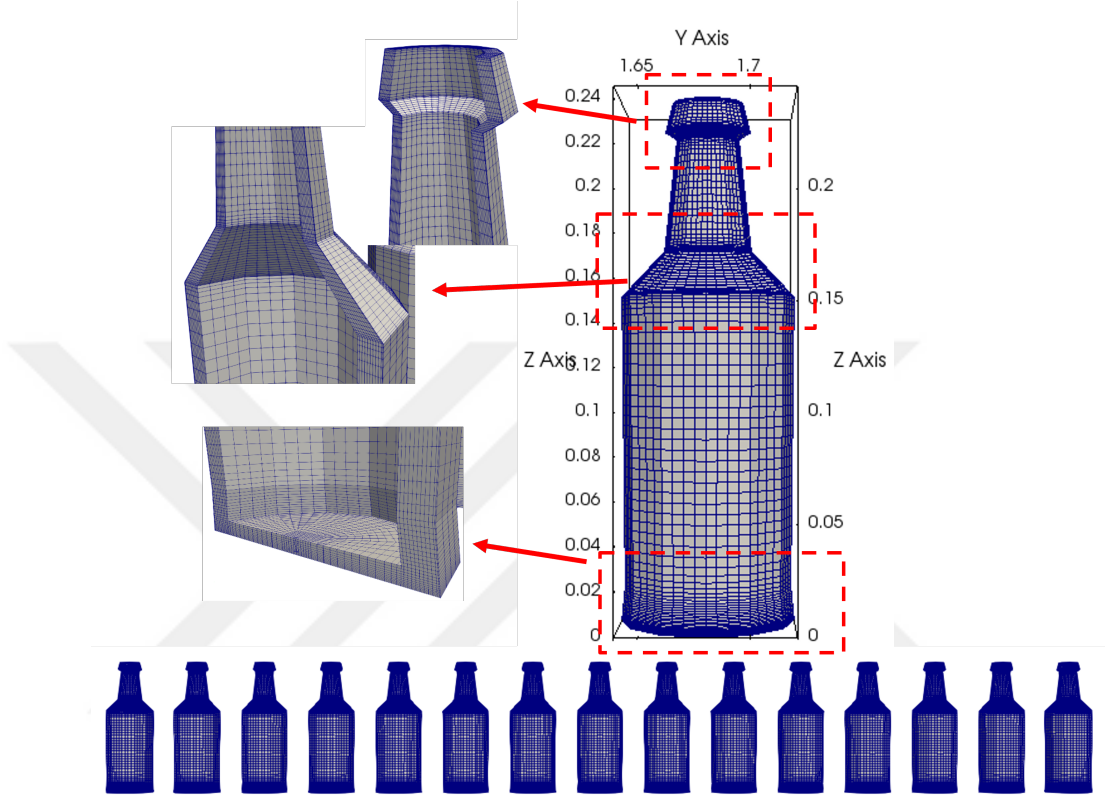


Figure 16: Bottle Mesh

The furnace mesh shown in 17 was generated using the snappyHexMesh utility of OpenFOAM. Six prism layers were used at the inner and outer walls of the furnace. The average y^+ value is 45. At the center of the furnace where the glass rows are located a relatively coarse mesh was used. The reason for that is the modeling of the bottle rows as porous zones instead of including the bottles directly in the domain using a body-fitted mesh. This modeling approach decreases the total number of cells in the furnace domain. The positioning of the porous zones is shown in Figure 17.

The oven is designed with a double-layer configuration. The meshes of the inner and outer walls of the furnace are illustrated in Figure 18.

There are 29 rows of glasses inside the furnace, and there are 15 glasses in each

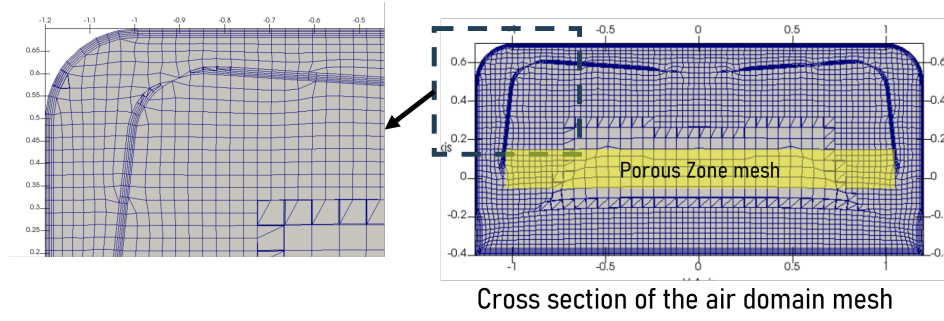


Figure 17: Air Domain Mesh

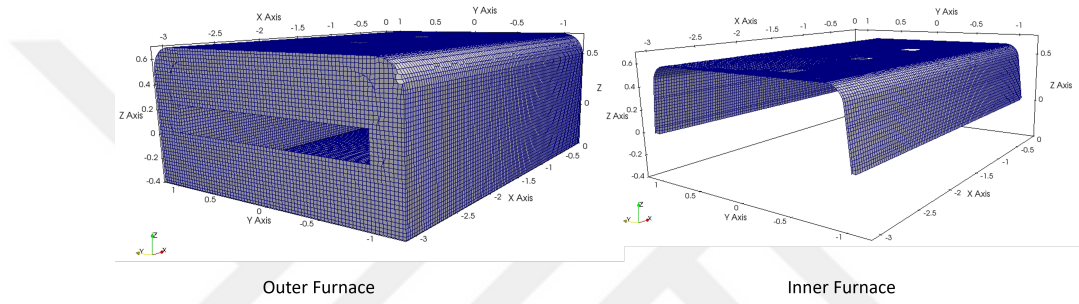


Figure 18: Inner and Outer Walls Mesh

row. Each row of glasses in the furnace has a specific set of porous regions defined by the Convection-Radiation solver for the steady Domain because these 29 rows are arranged evenly throughout the furnace. These regions are used to model the thermal contribution of the glasses to the furnace as well as the pressure drop caused by the porous regions. These zones' locations are shown in Figure 19, as are the locations of the fan and air intake zones. For the bottles in the relevant region of Figure, a more detailed representation is supplied to improve understanding of the regions.

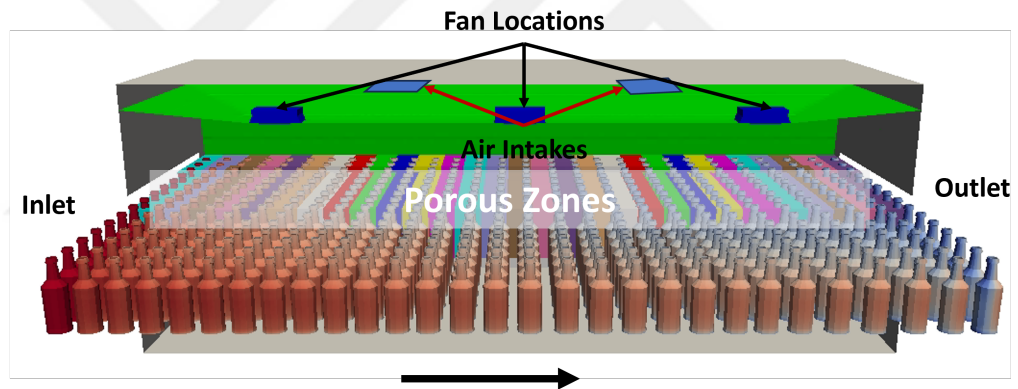


Figure 19: Porous zone of the bottles

CHAPTER V

RESULTS

5.1 Convergence Monitoring

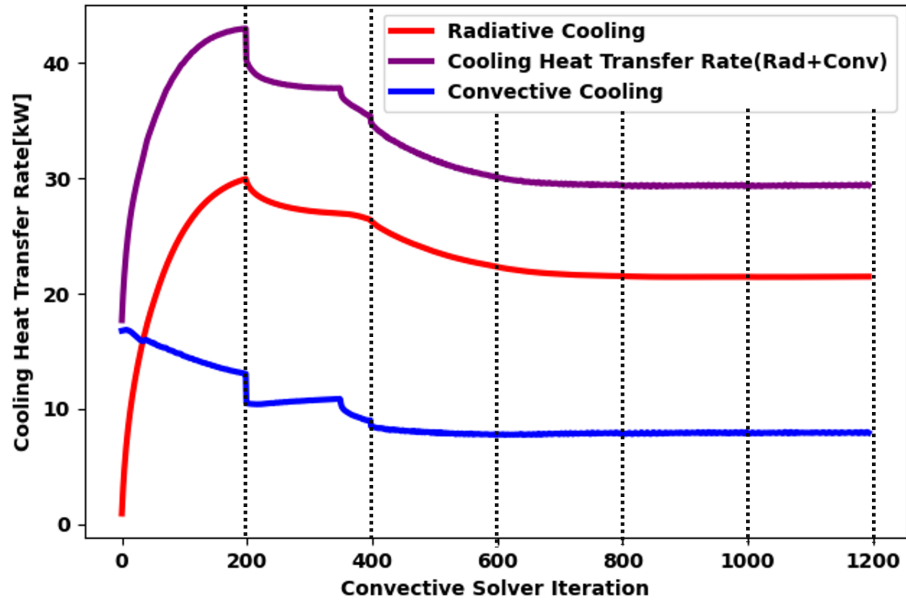


Figure 20: Air Domain Convergence Monitoring

In Figure 20, the convergence monitoring for the air domain simulation of Case 3 is shown. For convergence, the overall cooling heat transfer rate to the bottles was monitored. Every two hundred convective solver iterations the glass bottle temperatures were updated running the transient conduction solver on the moving solid glass domain. The row-averaged bottle temperatures calculated with the transient conduction solver are shown in Figure 21. Results show that after the fifth iteration, the row averaged temperatures reached a steady state.

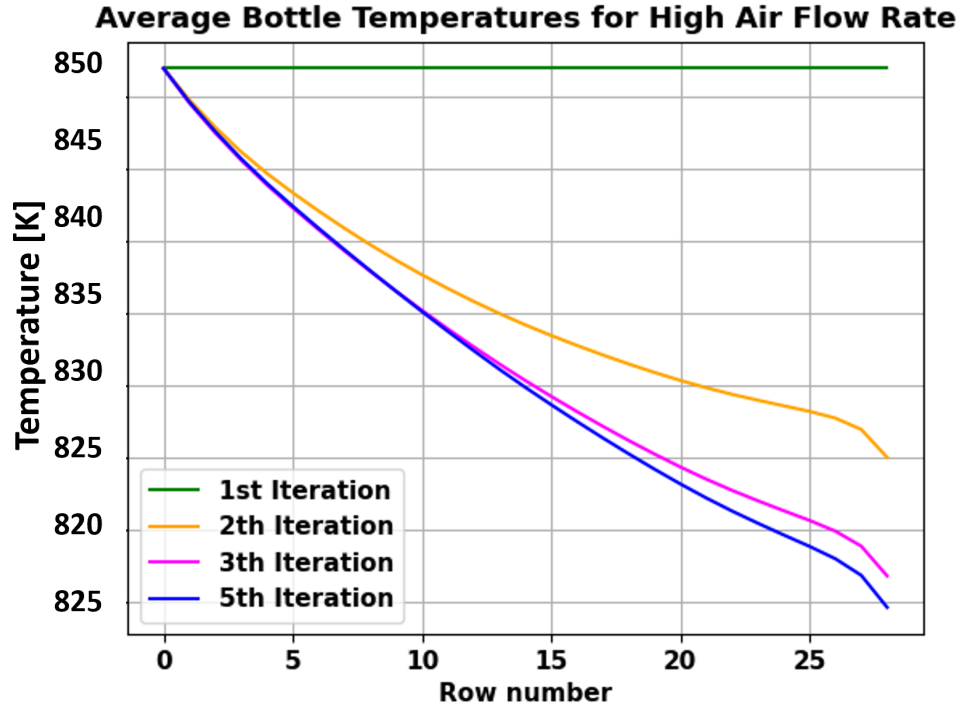


Figure 21: Solid Domain Convergence Monitoring

5.2 *Furnace*

5.2.1 Furnace Temperature Distribution

Figure 22 and 23 illustrate the temperature profiles of both the inner and outer walls of the furnace for Case 3, characterized by the higher fan flow rate. Noteworthy is the absence of significant differences in the temperature distributions of the walls between the two cases with varying circulation fan flow rates. Consequently, the discussion on the temperature distribution of the furnace walls applies uniformly to both cases.

As depicted in Figure 23, distinct cold spots appear on the walls near the cooling air intake. This observation underscores the influence of the cooling air intake on the temperature patterns of the furnace walls.

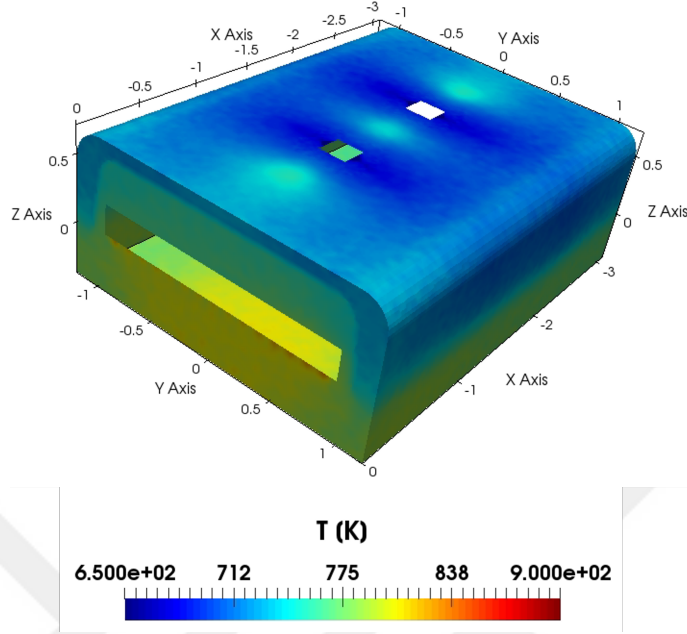


Figure 22: Outer wall temperature of the furnace

5.2.2 Air Temperature in the Furnace Domain

Figure 25 depicts the temperature distribution at constant axial cross-sections spanning the three circulation fans along the furnace's axis. The position of the axial cross-sections spanning is shown in Figure 24. In scenarios, the circulation fans induce a noticeable flow of hot air from the inner furnace region, indicated by the vertical arrows. In Case 1, characterized by the lower fan flow rate, a significant reduction in air temperature is observed at the outer layer of the furnace. This implies a diminished mixing of the cooling air with the heated furnace air at the lower fan flow rate.

Moreover, distinct circulation zones at the corners of the inner furnace are identified, generated by the action of the circulation fans. These zones exhibit relatively elevated temperatures compared to their surroundings. Notably, the temperature within these circulation regions gradually increases towards the outlet of the furnace. This temperature elevation within the circulation zones is attributed to the heat release generated by the solid glass bottles. The findings suggest that variations in fan

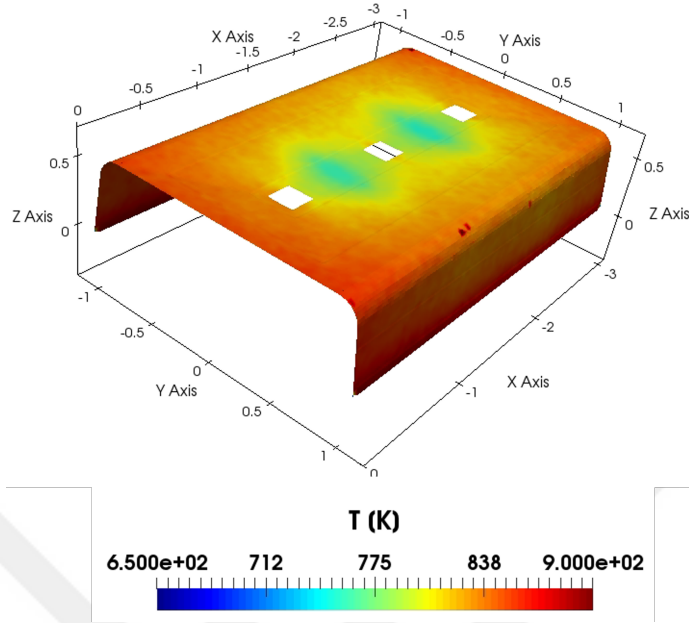


Figure 23: Inner wall temperature of the furnace

flow rates have a discernible impact on the distribution and mixing of air within the furnace, influencing temperature profiles and circulation patterns.

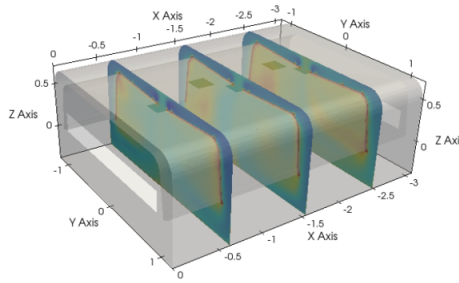


Figure 24: Axial Cuts Location in the Furnace: 0.6m, 1.5m, 2.5m

5.2.3 Energy Transfer Ratio in Furnace

The rates of heat loss from the glasses through convection and radiation, two significant modes of heat transfer, are outlined in Table 5. This table shows that the heat transfer through convection stays at about 27% at low fan speeds and rises to about 41% at the highest fan speed. The fan speed has a direct correlation with the amount of energy extracted from the glasses.

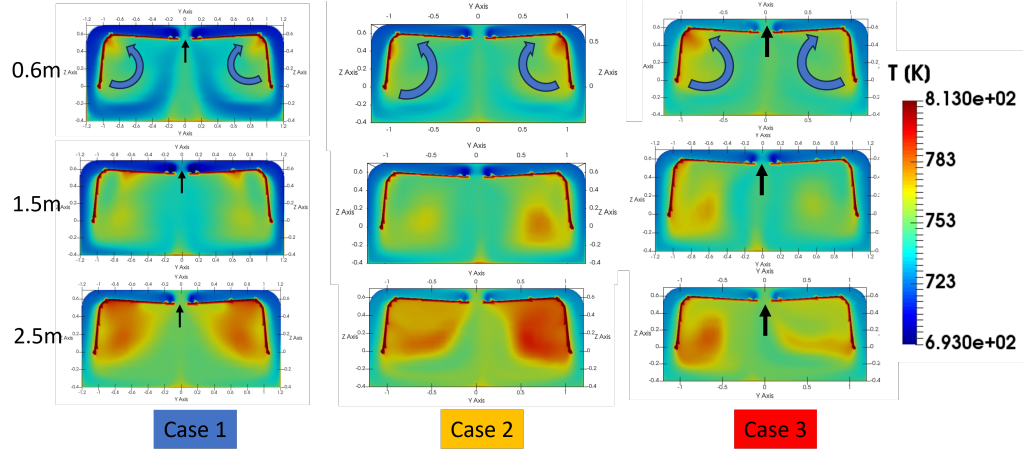


Figure 25: Air temperature Distributions at Axial Cuts of the Furnace: 0.6m, 1.5m, 2.5m

Table 5: Convective and Radiative cooling contribution

	Case 1	Case 2	Case 3
Convective Cooling Ratio	27%	33%	41%
Radiative Cooling Ratio	73%	68%	59%

5.3 Bottle

5.3.1 Radiative vs. Convective Heat Transfer Rate on the Bottles

Figure 26 shows the distribution of heat loss from radiation from the glasses. The color blue denotes no heat loss, while the color red represents the greatest heat loss. In this instance, one could argue that hot glasses lose more energy at the furnace inlet. It has been noted that reduced heat transfer through radiation occurs when fan speeds are high due to the entrance region's lower air circulation over the furnace wall. At high fan speeds, the walls of the furnace cool more quickly and come to balance with the furnace air. Towards the outlet of the furnace, heat transfer through radiation begins to decrease. However, upon reaching the furnace outlet, the glassware, now exposed to the atmosphere, undergoes rapid cooling. This rapid cooling was not considered in the simulations. To address this, boundary conditions at the furnace outlet can be adjusted as if there were a secondary cooling zone, effectively mitigating the impact of rapid cooling in that region.

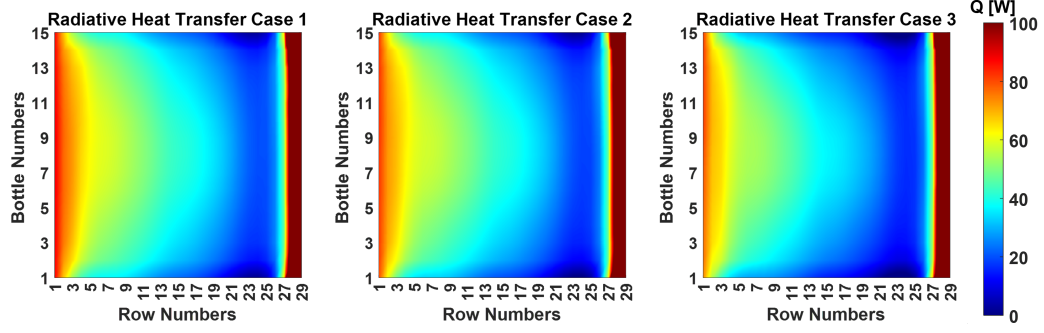


Figure 26: Bottles Radiative Cooling: Case 1, Case 2, Case 3

The energy transferred from the glasses through convection is shown in Figure 27, with red denoting the greatest heat loss and blue denoting the lowest heat loss. In this case, greater convective heat transfer is seen at the furnace inlet, much like radiation heat transfer in the preceding Figure 26. The convective heat transfer diminishes toward the furnace outlet. When evaluating fan speeds, the areas farther from the furnace outlet have higher convective heat transfer at high fan speeds. This is a result of the circulation fan speed. Towards the furnace outlet, a decrease in heat transfer through convection is observed. This is primarily due to the sudden cooling of the glass surfaces as they start to encounter the atmosphere. Since cold air entry into that region from the outside is restricted, the air in that zone has remained close to the surface temperature of the glass bottles.

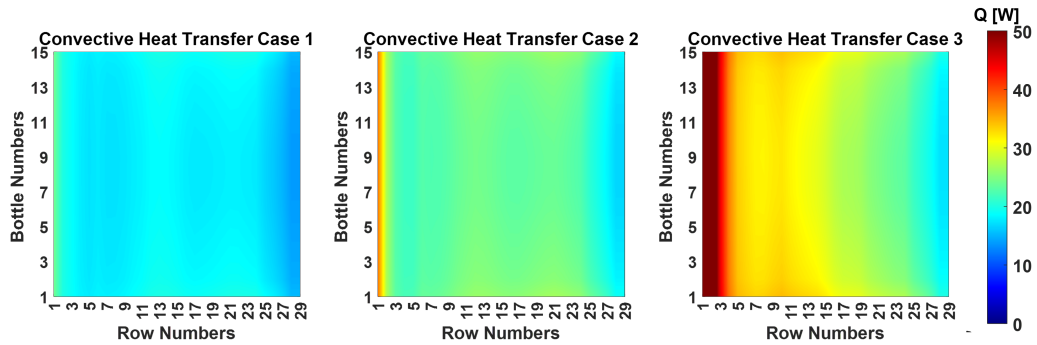


Figure 27: Bottles Convective Cooling: Case 1, Case 2, Case 3

5.3.2 Averaged Decrease in Bottle Temperatures and Standard Deviation

The average temperature drops of middle bottles at various fan speeds are shown in Figure 28. Compared to bottles with other fan speeds, glasses at higher fan speeds have continuously seen larger temperature drops and their standard deviations have stayed lower. There has been a roughly 40 K decrease across all fan speeds. It is found that there is a 6 K temperature differential between the highest (Case 3) and lowest (Case 1) fan speeds. Taking into account that the most cooled glass has dropped by 46 K, the 6 K drop corresponds to almost a 13% increase in cooling. Figure 29 below

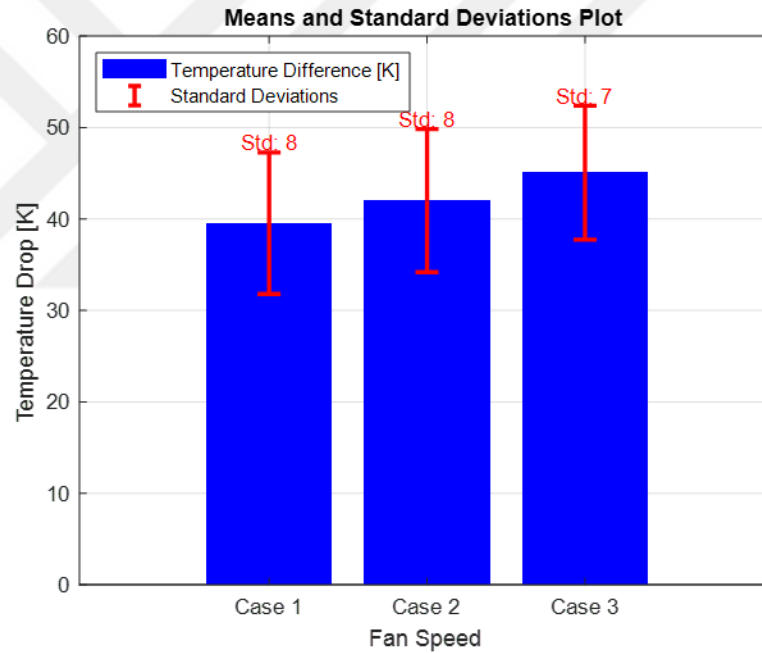


Figure 28: Mid Bottle Average Decrease in Temperature and Standard Deviation

depicts the average temperature loss and standard deviation of bottles at the side based on different fan speeds. Similar to bottles in the middle, bottles at the side with the highest fan speed have experienced more cooling. However, the amount of cooling for glasses at the side is nearly 7 K lower on average compared to glasses in the center.

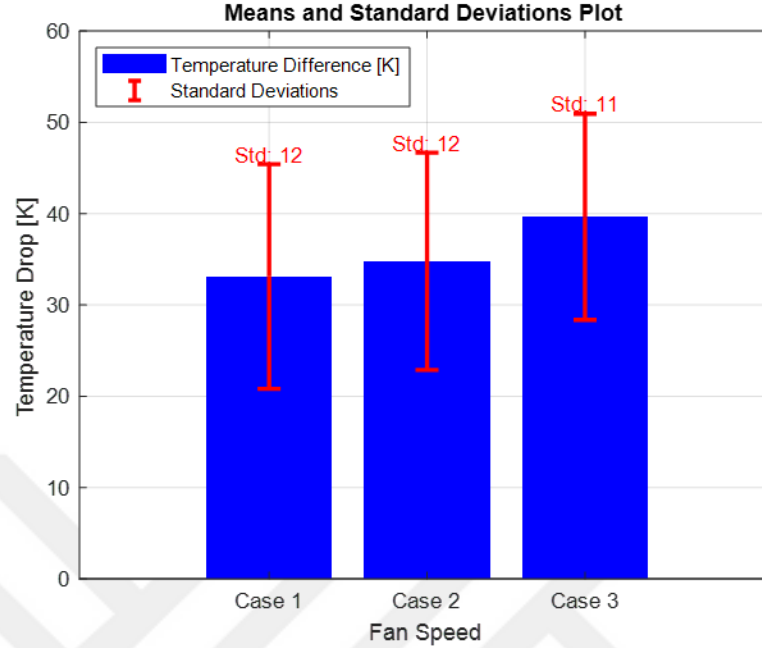


Figure 29: Side Bottle Average Decrease in Temperature and Standard Deviation

5.3.3 Bottle Surface Temperatures

Figure 30 illustrates the surface temperature distribution within the specified bottle rows in the zone, focusing on the scenario with a high fan flow rate. The outcomes are exclusively presented for this particular fan flow rate case. The findings highlight an accelerated cooling trend at the neck of the bottles, primarily attributed to radiative heat transfer between the glass and the furnace walls. The relatively significant spacing between adjacent bottles at this height facilitates more effective radiative heat transfer.

Conversely, the effect of radiative cooling diminishes at lower heights due to the shadowing effect induced by neighboring bottles. This effect inhibits the cooling rate of the bottles, resulting in hot spots in the lower body of the bottles. Beyond the temperature non-uniformity within individual bottles, there is also observable temperature non-uniformity between bottles within the same row. Particularly noticeable towards the zone outlet, temperature disparities become more apparent. Bottles along

the furnace's center line exhibit relatively lower temperatures than those positioned at the side walls.

The temperature non-uniformity is attributed to the accumulation of hot air in the circulation zones, as discussed in Section 5.2.2. This accumulation likely diminishes the cooling rate of the bottles positioned at the side walls, maintaining them at a higher temperature than bottles along the centerline.

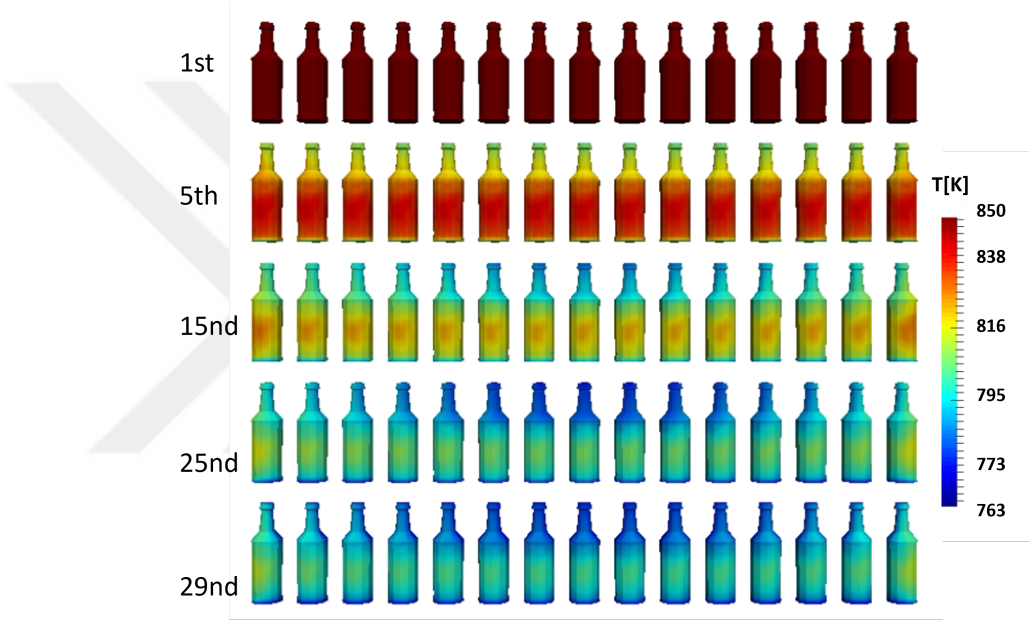


Figure 30: Surface temperature field of bottles for the case with the high fan flow rate

The temperature distributions on the surfaces of bottles in the last row at different fan speeds are shown in Figure 31. The most noticeable situation is that at low fan speeds, the surface temperatures of the bottles at the edges remain very high. This can be correlated with Figure 25 explaining the flow inside the furnace. Circulation zones formed near the inner wall region inside the furnace have prevented the wall temperatures from decreasing. Therefore, the walls remain at high temperatures at low fan speeds.

Additionally, the surfaces of bottles at high fan speeds have cooled more than the surfaces of bottles in the furnace with low fan speeds. The reason for this can

be attributed to increased convection. This is because, due to the shading effect of the bottles on each other, the body regions of the bottles lose more heat through convection.

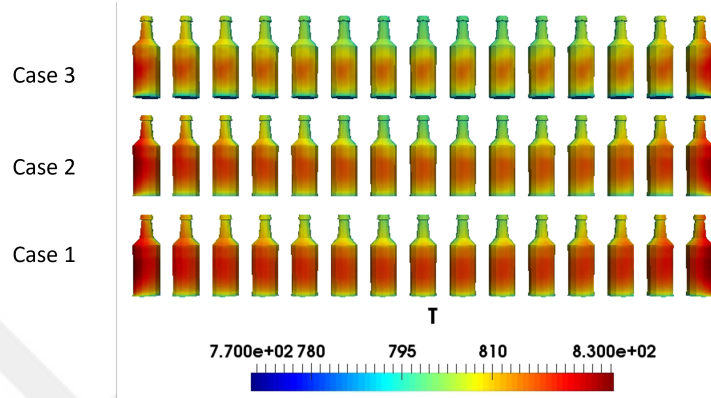


Figure 31: Surface temperature field of bottles for Case 1, Case 2, Case 3 at the last row

CHAPTER VI

CONCLUSIONS

Transient convective and radiative cooling of 3D moving bottles inside the cooling zone of the continuous annealing furnace is investigated. The problem inside the cooling zone of air intakes is considered combined conduction, convection, and radiation. Conduction and radiation calculations are separately conducted by a hybrid in-house developed CPU-GPU solver. To solve radiation, there are two different solvers are applied. One of them is placed in the steady-radiation solver. This solver used the forward Monte Carlo ray tracing method. Radiative heat fluxes are calculated between bottles and furnace walls. Since the air inside the furnace is assumed steady-state, wall temperatures were calculated according to the constant bottle temperatures. the furnace wall's temperatures change due to convection heat transfer until a steady state is reached.

The primary reason for employing an iterative method between these two solvers is rooted in establishing accurate boundary conditions inside furnaces. The initial step of the algorithm involves utilizing initial guesses for the temperature of the bottles, air temperature, and furnace wall temperatures. The temperature of the walls is determined using the steady-convection solver with an initial guess for the bottle temperatures. Even if the results for the furnace wall temperatures are still considerably deviate from the correct values based on the initial estimates, a point of equilibrium will be reached between them due to the utilization of the two solvers. The bottle temperatures are then updated using the results from the steady solver, and similarly, the furnace wall temperatures are updated. While the thermal inertia of the furnace may be higher than that of the glassware, consecutive results from the steady

solver have exhibited differences in wall temperature. Consequently, the updated solution for the glassware must be employed to determine the furnace temperature until both solvers converge to similar results in the next iteration.

The second one is used in transient radiation-conduction solver. This solver used the backward Monte Carlo ray tracing method. The furnace wall's temperatures determined using the steady-state radiation-convection solver were assigned as boundary conditions to find the bottle temperatures. Calculations are made for the radiative heat fluxes between bottles in the same row, bottles in the neighboring row, and bottles in contact with the furnace walls. The boundaries of the surfaces inside the furnace are thought to be black and diffuse emitters. Heat fluxes on the bottle surface are approximately 25% overestimated as a result of the black assumption. Radiative heat fluxes are imposed as a boundary condition to solve the heat conduction equation on the CPU side. The discretization of the conduction equation is done using the first-order accurate Backward Euler method in time and the second-order accurate central method in space. Instead of taking into account the cooling zone being fully loaded with 29 bottle rows in the computational furnace model, 7 bottle rows are proposed to represent the fully loaded furnace condition. Only the fourth bottle row—out of seven—is meant to show the distribution of bottle temperatures inside the furnace. However, since they serve as the boundary condition for the bottles in row 4, the temperature of the 66 bottles inside the other rows is also solved.

To solve convection in the cooling zone of the furnace, a porosity-based model is applied in the steady radiation-convection solver. The number of mesh for one-row bottles (15 bottles in one row) is reduced by 1.1M to 5k cells. Reducing the mesh number also reduced computational time. For comparison, a simulation with all the bottles was not run, but the simulation used to calculate the heat transfer coefficient and pressure loss with only one bottle takes nearly half an hour to reach stability when run on 1 core. This computational cost was reduced with the help of

the Darcy-Forcheimer porosity model.

Three distinct volumetric fan flow rates, namely $0.6 \text{ m}^3/\text{s}$, $0.8 \text{ m}^3/\text{s}$, and $1 \text{ m}^3/\text{s}$ were employed to investigate convective heat transfer. Notably, at a higher volumetric fan flow rate, both the mean and standard deviation temperatures of the bottles exhibit lower values compared to the two other cases. Specifically, the mid bottle in the final row registers a temperature 6 K lower than the other scenarios, while the side bottle in the same row experiences a 7 K reduction. The standard deviation of the bottles serves as an indicator of temperature uniformity, and for the side and mid bottles in the last row, the highest volumetric fan flow rate yields temperatures 1 K lower than the alternatives. This nuanced temperature variation across scenarios offers insights into the impact of different volumetric fan flow rates on convective heat transfer within the system.

REFERENCES

- [1] B. Demir and M. Erdogan, “The hardenability of austenite with different alloy content and dispersion in dual-phase steels,” *Journal of Materials Processing Technology - J MATER PROCESS TECHNOL*, vol. 208, pp. 75–84, 11 2008.
- [2] K. Chapman, S. Ramadhyani, and R. Viskanta, “Modeling and parametric studies of heat transfer in a direct-fired batch reheating furnace,” *Journal of Heat Treating*, vol. 8, no. 2, p. 137 – 146, 1990.
- [3] D. O. Marlow, “Modelling direct-fired annealing furnaces for transient operations,” *Applied Mathematical Modelling*, vol. 20, no. 1, p. 34 – 40, 1996.
- [4] M. Gao, C. Reid, M. Jahedi, and Y. Li, “Estimating equilibration times and heating/cooling rates in heat treatment of workpieces with arbitrary geometry,” *Journal of Materials Engineering and Performance*, vol. 9, no. 1, p. 62 – 71, 2000.
- [5] G. Tang, B. Wu, D. Bai, Y. Wang, R. Bodnar, and C. Q. Zhou, “Modeling of the slab heating process in a walking beam reheating furnace for process optimization,” *International Journal of Heat and Mass Transfer*, vol. 113, p. 1142 – 1151, 2017.
- [6] S. H. Han, D. Chang, and C. Huh, “Efficiency analysis of radiative slab heating in a walking-beam-type reheating furnace,” *Energy*, vol. 36, no. 2, p. 1265 – 1272, 2011.
- [7] R. Prieler, B. Mayr, M. Demuth, B. Holleis, and C. Hochenauer, “Prediction of the heating characteristic of billets in a walking hearth type reheating furnace using cfd,” *International Journal of Heat and Mass Transfer*, vol. 92, p. 675 – 688, 2016.
- [8] N. Depree, J. Sneyd, S. Taylor, M. Taylor, J. Chen, S. Wang, and M. O’Connor, “Development and validation of models for annealing furnace control from heat transfer fundamentals,” *Computers and Chemical Engineering*, vol. 34, no. 11, p. 1849 – 1853, 2010.
- [9] N. Depree, J. Sneyd, S. Taylor, M. Taylor, M. O’Connor, and J. Chen, “Mathematical modelling of an annealing furnace for process control applications,” *TMS Annual Meeting*, vol. 3, p. 267 – 274, 2010. Cited by: 1.
- [10] S. Hu, “Temperature distribution and stresses in circular wafers in a row during radiative cooling,” *Journal of Applied Physics*, vol. 40, no. 11, p. 4413 – 4423, 1969.
- [11] P. Chen, F. Dai, L. Pan, Y. Guo, J. Ke, J. Wu, Y. Lei, and Y. Li, “Numerical simulation and experimental study of strip steel jet cooling,” *Applied Thermal Engineering*, vol. 181, no. 12, 2020.

- [12] J. Smolka, Z. Bulinski, and A. J. Nowak, “The experimental validation of a cfd model for a heating oven with natural air circulation,” *Applied Thermal Engineering*, vol. 54, no. 2, p. 387 – 398, 2013.
- [13] I. D. Palacio-Caro, P. N. Alvarado-Torres, and L. F. Cardona-Sepúlveda, “Numerical simulation of the flow and heat transfer in an electric steel tempering furnace,” *Energies*, vol. 13, no. 14, 2020.
- [14] H. J. Lee, J. Park, and S. H. Lee, “Numerical investigation on influence of fan speed and swirling gas injection on thermal-flow characteristics in nitrocarburizing furnace,” *Materials Transactions*, vol. 58, no. 9, p. 1322 – 1328, 2017.
- [15] S. Silvestri and R. Pecnik, “A fast gpu monte carlo radiative heat transfer implementation for coupling with direct numerical simulation,” *Journal of Computational Physics: X*, vol. 3, 2019.
- [16] P. Martinsen, J. Blaschke, R. Künnemeyer, and R. Jordan, “Accelerating monte carlo simulations with an nvidia® graphics processor,” *Computer Physics Communications*, vol. 180, no. 10, p. 1983 – 1989, 2009.
- [17] <https://www.openfoam.com/documentation/guides/latest/doc/guide-applications-solvers-heat-transfer-buoyantSimpleFoam.html#sec-applications-solvers-compressible-buoyantSimpleFoam-overview>.
- [18] https://www.openfoam.com/documentation/guides/latest/api/classFoam_1_1porosityModels_1_1DarcyForchheimer.html.
- [19] <https://www.openfoam.com/documentation/guides/latest/doc/guide-fvoptions-sources-semi-implicit.html>.
- [20] <https://www.openfoam.com/documentation/guides/latest/doc/guide-applications-solvers-basic-laplacianFoam.html>.
- [21] G. Can Altun and A. Melik Başol, “Numerical analysis of the radiant heating effectiveness of a continuous glass annealing furnace,” *Applied Thermal Engineering*, vol. 204, 2022.

VITA

Oğuzhan Aşık graduated from Nilüfer Bursa Istanbul Science High School in 2015. He received his bachelor's degree in Mechanical Engineering from Özyeğin University in 2021. Following these, he pursued a master's degree in mechanical engineering at Özyeğin University. His field of study and research focus is on numerical modelling for heat transfer and flow in continuous annealing furnaces.