

Reduction of convective heat loss from an external solar receiver using an air curtain

Razon Kumar Mondal

A thesis submitted for the degree of
Doctor of Philosophy
The Australian National University

Primary supervisor: A/Prof. John Pye

June 2023

Supervisory panel

A/Prof. John Pye	Primary supervisor and Chair of panel
S/Lec. Juan Felipe Torres	Associate supervisor
Prof. Graham Hughes	Associate supervisor

I declare that this PhD thesis is an original work, representing the culmination of my research efforts and scholarly pursuits. No part of this thesis has been previously submitted or accepted for any other degree or diploma. I affirm that all materials presented herein are my own unless appropriately acknowledged and referenced in the text. Furthermore, this thesis does not contain any previously published or written work authored by another person, except where explicit reference is provided. I take full responsibility for the authenticity, accuracy, and originality of the contents presented in this thesis, upholding the principles of academic integrity and ethical conduct.

Razon Kumar Mondal

6 June 2023

Dedications

I want to dedicate this thesis to the incredible support and love of my family. To my parents, brothers, and sisters, thank you for always believing in me and being there for me. Your encouragement and sacrifices have been my motivation throughout this journey. I am grateful for your unwavering support and the joy you bring to my life.

I also want to dedicate this thesis to my advisors and mentors. Their guidance and expertise have been invaluable in shaping my research. Their belief in me has given me the confidence to push my limits and achieve more than I thought possible.

Lastly, I want to dedicate this thesis to all future scientists and researchers. May this work inspire you to explore new ideas and contribute to the growth of knowledge. Let's continue to learn, discover, and make a difference together.

Acknowledgments

I wish to express my heartfelt gratitude to my supervisors, **Dr. John Pye**, **Dr. Juan Felipe Torres**, and **Prof. Graham Hughes**. Their unwavering guidance and support throughout the project were invaluable. Their insights, inspiration, and advice were essential to the exciting research in this thesis. Their expertise greatly benefited the work, and their interactions inspired me to become a better researcher.

I would like to extend my thanks to all members of the Solar Thermal Group (STG). They were not only colleagues but also good friends. Working with them was always enjoyable, and their ideas and stories helped me grow in developing the project.

I would also like to thank my friends, both in and outside of Australia, for their understanding and patience while I was engrossed in my work.

My gratitude goes out to my family, including my parents, siblings, and relatives, for their unwavering support. I missed their company immensely during these years away from home.

Lastly, I am forever indebted to my wife, **Chanda Rani Das**, for her constant inspiration and support, both during good and bad times. Her sacrifices were immeasurable, and I am grateful for her unwavering presence. No matter what the future holds, I will always stand by her side.

Publications

The following journal and conference papers accompany this research:

- **R. Mondal**, J. F. Torres, G. Hughes, and J. Pye. Analysis of air curtains for natural convection heat-loss mitigation. In Proceedings of the 22nd Australasian Fluid Mechanics Conference, Queensland, Australia, December 2020.
doi.org/10.14264/3ad7af9
- **R. Mondal**, J. F. Torres, G. Hughes, and J. Pye. Air curtains for reduction of natural convection heat loss from a heated plate: a numerical investigation. *Int. J. Heat Mass Transfer*, 189:122709, 2022.
doi.org/10.1016/j.ijheatmasstransfer.2022.122709
- **R. Mondal**, J. F. Torres, G. Hughes, and J. Pye. Optimal configuration of oblique air curtains for heat loss suppression from vertical solar-thermal receiver surfaces. *Applied Thermal Engineering* (Accepted subject to necessary revision), 2023.
- **R. Mondal**, J. F. Torres, G. Hughes, and J. Pye. Effect of sidewalls on the performance of air curtain to minimise convection losses from a heated wall. In preparation, 2023.

Abstract

Concentrated Solar Power (CSP) plants encounter inefficiencies at all stages of electricity generation. Convection from the solar thermal receiver is a significant mode of heat loss in CSP systems and is challenging to mitigate. This study investigates the mitigation of convective heat loss from an external heated surface by introducing an air curtain that disrupts the buoyant flow rising along the heated surface. The external CSP receiver surface is an important setting where this mode of heat transfer frequently occurs. An isothermal flat plate with a height of 1.8 m is used to model the receiver, and an obliquely introduced planar jet is used to form the air curtain. This thesis aims to understand the potential benefits of air curtains for CSP receiver heat loss reduction via a series of numerical case studies at the smaller 1.8 m scale.

The computational fluid dynamics (CFD) model is first validated for heated wall and air curtain cases. The subsequently validated models are then implemented to conduct a parametric study on the heat transfer from the isothermal plate with an air curtain, varying different parameters. The results showed that the air curtain successfully generates a stagnation zone adjacent to the wall and does reduce convective heat losses locally. The air curtain effectiveness, defined as the relative reduction in local heat loss when the air curtain is added, reaches a local value of 31.2% in the stagnation zone for a vertical wall when the air curtain had a jet speed and angle of 2.5 m s^{-1} and 45° , respectively. Smaller air curtain angles relative to the wall resulted in lower effectiveness. A 45° air curtain on a vertical wall can offer performance benefits similar in magnitude to inclining a wall from the vertical. A higher wall temperature was accompanied by better effectiveness near the jet outlet, particularly in the stagnation region, while lower wall temperatures produced higher effectiveness further from the jet. Following that, different air curtain parameters, including jet speed, angle, thickness, and gap, are optimised to increase the mitigation of convection loss from a vertical exter-

nal heated surface, focusing on global air curtain effectiveness. More than 18% global effectiveness was achieved for an optimum jet with a speed of $U_{ac} = 5.5 \text{ m s}^{-1}$, an angle of $\alpha = 45^\circ$, a thickness of $b = 30 \text{ mm}$, and a gap of $G = 30 \text{ mm}$. The jet thickness is found to be the key parameter for significant improvement of air curtain effectiveness. As an option for using waste heat streams in a CSP plant, the potential benefit of heated jets was considered; the effectiveness is gradually decreased with increasing the temperature of the air curtain. Finally, the comparison between 2D and 3D simulations was performed, where no significant changes were revealed. A validation has also been conducted between the 3D CFD modelling result against the experimental data, conducted by Abbasi-Shavazi (2019). Both results show a similar trend and confirm that an overall beneficial air curtain configuration was found to reduce convective heat losses from the heated surface.

Overall, this thesis thoroughly investigates the performance of air curtains to minimise convection loss from an external heated surface. A significant improvement of more than 18% global effectiveness was achieved by using an air curtain. This thesis also contributes to a rigorous validation of numerical modelling with experimental data. Therefore, using this concept in the CSP plant may provide significant economic value to mitigate convection losses from the receiver surface.

Contents

Acknowledgments	vii
Publications	ix
Abstract	xi
1 Introduction and Background Literature	1
1.1 Motivation	1
1.2 Research scope	7
1.3 Background Literature	10
1.3.1 Natural convection along a heated flat plate	10
1.3.2 Planar jet (Air curtain)	14
1.3.3 Use of air curtains in the solar-thermal systems	17
1.4 Outline of the thesis	25
2 Methodology	27
2.1 Pre-processing	27
2.1.1 Geometrical definition of the computational domain	28
2.1.2 Strategy of meshing the domain	29
2.1.3 Mesh refinement study	30
2.1.4 Boundary conditions	32
2.1.5 Fluid properties	34
2.2 Solving with OpenFOAM	35
2.2.1 Overview of OpenFOAM software	35
2.2.2 OpenFOAM solver	37
2.2.3 Turbulence modelling	39
2.3 Post-processing	42
2.3.1 ParaView	42
2.3.2 Effectiveness calculation	43

3	Air curtains for natural convection heat loss reduction: a parametric study	45
3.1	Validation	45
3.1.1	Natural convection from an isothermal vertical plate . .	46
3.1.2	Isothermal planar jet	49
3.2	Results and discussion	52
3.2.1	Transient analysis	52
3.2.2	Effect of jet speed on wall heat flux	54
3.2.3	Effect of jet angle on effectiveness	58
3.2.4	Effect of wall inclination on effectiveness	60
3.2.5	Effect of wall temperature on effectiveness	63
3.3	Conclusions	64
4	Optimization of air curtain parameters to reduce heat loss from heated surface	67
4.1	Grid dependency test	68
4.2	Domain dependency test	71
4.3	Optimization technique	71
4.4	Analysis of results and discussion	74
4.4.1	Loop 1	74
4.4.1.1	Jet speed	74
4.4.1.2	Jet angle	77
4.4.1.3	Jet thickness	80
4.4.1.4	Jet position	82
4.4.2	Additional loops	85
4.4.3	Power consumption of the air curtain	87
4.4.4	Effect of jet outlet temperatures	89
4.5	Non-dimensional Analysis	91
4.6	Conclusion	98
5	Validation of CFD modelling against experimental data	99
5.1	Geometrical set up	99
5.2	Validation of natural convection simulations	104
5.3	Air curtain effectiveness	108
5.3.1	Jet angle, $\alpha = 45^\circ$	108

5.3.2	Jet angle, $\alpha = 25^\circ$	112
5.4	Conclusion	114
6	Conclusions and Recommendations	117
6.1	Conclusions	117
6.2	Recommendations	119

List of Figures

1.1	Black summer bushfire 2019-2020 (Bradstock et al.).	2
1.2	Concentrated solar power (CSP) tower and heliostat field (Ahmadi et al., 2018).	3
1.3	Four main CSP technologies with basic components (Bank, 2021).	5
1.4	Key subsystems of a molten salt solar power tower systems (Lovegrove et al., 2018).	6
1.5	Schematic of external (left) and cavity (right) receivers in solar power tower system (Ho and Iverson, 2014).	7
1.6	Conceptual sketch showing the application of air curtains to reduce convective losses in a cavity receiver.	8
1.7	Conceptual sketch showing the application of air curtains to reduce convective losses from an external tubular receiver.	9
1.8	Natural convection boundary layer along a heated vertical flat plate.	12
1.9	Schematic of flow regions for planar jet (Guyonnaud et al., 2000).	15
1.10	Convective heat losses against wind speeds for $i = 0^\circ$ (Taumoe-folau, 2004).	18
1.11	McIntosh et al. (2014) geometrical setup (top) and air curtain effectiveness for different jet speeds (bottom).	19
1.12	Simulation results based on Zhang et al. (2015).	20
1.13	Simulation results of Flesch et al. (2016).	21
1.14	Saline water experiments of Hughes et al. (2016).	22
1.15	Yang et al. (2018) modified receiver with a special air channel.	23
1.16	Geometrical configuration of Fang et al. (2019).	23
1.17	Experimental set up of Alipourtarzanagh et al. (2020a).	24
1.18	Idea developed by Kelly and Robert (1990) to introduce a protective air-stream for external receiver.	24

1.19	Wang et al. (2021) geometry of the external receiver having an air curtain installed on top.	25
2.1	Two-dimensional geometry of the computational domain subject to θ inclination. Here, the black dash lines are open boundaries, the solid black lines are adiabatic walls, the solid yellow line is the gap between the heated wall and jet, the solid red line is an isothermal wall, and the black dash-dot line is the vertical axis.	29
2.2	2D mesh surrounded the domain for $\theta = 0^\circ$ and $\alpha = 45^\circ$	31
2.3	Basic OpenFOAM case structure for a typical OpenFOAM simulation.	36
2.4	Graphical user interface of ParaView.	43
3.1	Temperature (left) and velocity (right) contour with vector plot for natural convection flow along an isothermal plate when, $T_w = 110^\circ\text{C}$, $\theta = 0^\circ$, $\alpha = 45^\circ$, $b = 3\text{ mm}$, and $G = 0\text{ mm}$	46
3.2	Dimensionless laminar velocity and temperature profile for an isothermal vertical wall.	47
3.3	Local Nusselt number as a function of local Rayleigh number for isothermal vertical wall; the inset shows the difference between modelling results and the experimental results of Tsuji and Nagano (1988).	49
3.4	Velocity contour of turbulent planar jet and associated coordinates system for $\text{Re} = 7000$	50
3.5	Centre line velocity decay of turbulent planar jet for $\text{Re} = 7000$ when positioned at $\alpha = 45^\circ$	51
3.6	Lateral velocity distribution of jet for $\text{Re} = 4000$ when positioned at $\alpha = 45^\circ$	52
3.7	Transition to steady-state heat transfer rate for natural convection case when $T_w = 110^\circ\text{C}$ and $\theta = 0^\circ$	53
3.8	Transition to steady-state heat transfer rate for natural convection with air curtain when $T_w = 110^\circ\text{C}$, $\theta = 0^\circ$, and $\alpha = 45^\circ$	53
3.9	Local heat flux of a vertically oriented heated wall ($\theta = 0^\circ$) for different jet speeds when $\alpha = 45^\circ$	54

3.10	Global effectiveness for different jet speeds when $\theta = 0^\circ$ and $\alpha = 45^\circ$	55
3.11	Temperature profile with velocity vector for $\alpha = 45^\circ$, $U_{ac} = 2.5 \text{ m s}^{-1}$	56
3.12	Local effectiveness ε_y of a vertical heated wall for different jet speeds when $\alpha = 45^\circ$	57
3.13	Local effectiveness at $y = 0.75 \text{ m}$ and 1.75 m for different jet speeds when $\alpha = 45^\circ$, indicated by grey dashed line in Figure 3.12.	57
3.14	Temperature profile of the top 0.1 m of the heated wall, for different jet speeds when $\alpha = 45^\circ$	58
3.15	Local effectiveness for different jet angles when $U_{ac} = 5.1 \text{ m s}^{-1}$	59
3.16	Temperature contour with velocity vector for a 30° tilted air curtain with jet speed of 5.1 m s^{-1}	60
3.17	Local effectiveness for different wall inclinations when $\alpha = 45^\circ$ and $U_{ac} = 5.1 \text{ m s}^{-1}$	61
3.18	Local effectiveness (relative to a vertical heated wall) for different wall inclinations when $U_{ac} = 0 \text{ m s}^{-1}$	62
3.19	Local effectiveness (ε) for different wall temperatures when $\alpha = 45^\circ$ and $U_{ac} = 5.1 \text{ m s}^{-1}$	64
4.1	Two-dimensional computational domain and associated boundary conditions.	68
4.2	Two-dimensional mesh around the heated plate and jet, focusing on different jet widths (b) inclined at $\alpha = 45^\circ$	69
4.3	Grid dependency test was performed when $T_w = 110^\circ\text{C}$, $b = 3 \text{ mm}$, $G = 0 \text{ mm}$, and $\alpha = 45^\circ$. (i) natural convection case. (ii) natural convection with jet of speed $U_{ac} = 6 \text{ m s}^{-1}$	70
4.4	Global effectiveness against different jet speeds for two separate domains when $T_w = 110^\circ\text{C}$, $\alpha = 45^\circ$, $b = 3 \text{ mm}$, and $G = 0 \text{ mm}$	71
4.5	Schematic of the univariate search method.	72
4.6	Global effectiveness (ε) of air curtain for different jet speeds when $\alpha = 45^\circ$, $b = 3 \text{ mm}$, and $G = 0 \text{ mm}$	75

4.7	Comparison of local Nusselt number along the heated surface for cases without and with a jet when $\alpha = 45^\circ$, $b = 3$ mm, and $G = 0$ mm.	75
4.8	Visualization of temperature field along heated surface for $U_{ac} = 5 \text{ m s}^{-1}$, $\alpha = 45^\circ$, $b = 3$ mm, and $G = 0$ mm.	76
4.9	Transverse velocity profile at $y = 1.75$ m for $U_{ac} = 5 \text{ m s}^{-1}$, $\alpha = 45^\circ$, $b = 3$ mm, and $G = 0$ mm.	77
4.10	Global effectiveness as a function of jet angle when $U_{ac} = 5 \text{ m s}^{-1}$, $b = 3$ mm, and $G = 0$ mm; inset shows only those jet angles that produced positive effectiveness.	78
4.11	Comparison of temperature field when $U_{ac} = 5 \text{ m s}^{-1}$, $b = 3$ mm, and $G = 0$ mm; inset 1 describes the upper section of the wall for a wall attached jet ($\alpha = 30^\circ$) and inset 2 depicts the upper section of the wall for optimal jet angle in this iteration ($\alpha = 50^\circ$).	79
4.12	Comparison of wall heat flux for $\alpha = 50^\circ$ and 55° when $T_w = 110^\circ\text{C}$, $b = 3$ mm, $G = 0$ mm, and $U_{ac} = 5 \text{ m s}^{-1}$	80
4.13	Global effectiveness for different jet thicknesses when $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $G = 0$ mm.	80
4.14	Local effectiveness for three different jet thicknesses when $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $G = 0$ mm.	81
4.15	Global effectiveness (ϵ) of air curtain for different jet gaps when $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $b = 60$ mm.	83
4.16	Local effectiveness of different jet gaps for $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $b = 60$ mm.	83
4.17	Temperature contour for jet gap, $G = 10$ mm and 30 mm when $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $b = 60$ mm.	84
4.18	Effectiveness as a function of jet thickness in Loop 2 when $U_{ac} = 5.5 \text{ m s}^{-1}$, $\alpha = 45^\circ$, and $G = 30$ mm.	85

4.19	Global effectiveness obtained in Loop 3 as a function of the jet parameter: (a) Jet speed when $\alpha = 45^\circ$, $b = 30$ mm and $G = 30$ mm. (b) Jet angle when $U_{ac} = 5.5 \text{ m s}^{-1}$, $b = 30$ mm and $G = 30$ mm. (c) Jet thickness when $U_{ac} = 5.5 \text{ m s}^{-1}$, $\alpha = 45^\circ$ and $G = 30$ mm. (d) Jet gap when $U_{ac} = 5.5 \text{ m s}^{-1}$, $\alpha = 45^\circ$ and $b = 30$ mm.	86
4.20	Global effectiveness for different jet outlet temperatures when $T_w = 110^\circ\text{C}$, $b = 30$ mm, $G = 30$ mm.	90
4.21	Wall heat flux for different jet outlet temperatures when $T_w = 110^\circ\text{C}$, $b = 30$ mm, $G = 30$ mm.	90
5.1	Experimental configuration of heated plate and air curtain when $T_w = 110^\circ\text{C}$ and $\alpha = 45^\circ$	100
5.2	3D modelling for heated plate and air curtain when $T_w = 110^\circ\text{C}$ and $\alpha = 45^\circ$	101
5.3	3D mesh surrounded by the domain for $\alpha = 45^\circ$	102
5.4	Mesh refinement test for four different mesh sizes when $T_w = 110^\circ\text{C}$, $\alpha = 45^\circ$. (a) natural convection. (b) natural convection with air curtain of speed $U_{ac} = 5.75 \text{ m s}^{-1}$	103
5.5	Temperature contour and velocity vector of the natural convection flow (2D view) developed along an isothermal wall when $T_w = 110^\circ\text{C}$ and $\alpha = 45^\circ$. Note the domain has been sliced along z -axis to show the velocity vector.	105
5.6	Nusselt number as a function of Rayleigh number for natural convection flow when $T_w = 110^\circ\text{C}$	106
5.7	Heat losses through the edge of the plate unit, reported in the experimental setup. Image was taken from (Abbasi-Shavazi, 2019).	106
5.8	Deviation of simulated data from the experimental data produced by Abbasi-Shavazi (2019).	108
5.9	Comparison of effectiveness for air curtain speeds of $U_{ac} = 3.45 \text{ m s}^{-1}$ and 5.75 m s^{-1}	109

5.10	Temperature contour (left) and velocity vector (right) when an air curtain of speed $U_{ac} = 3.45 \text{ m s}^{-1}$ interacts with the natural convection of a heated wall. The temperature of the wall is scaled at 350 K for better viewing of the contour.	110
5.11	Heat transfer rate from different units of the plate in transition region for $U_{ac} = 3.45 \text{ m s}^{-1}$	111
5.12	Heat flux distribution of the plate for the case of natural convection and natural convection with an air curtain of speed $U_{ac} = 3.45 \text{ m s}^{-1}$	112
5.13	Effectiveness of air curtain for the combination of $\alpha = 25^\circ$ and $U_{ac} = 5.75 \text{ m s}^{-1}$	113
5.14	Temperature contour (left) and velocity vector (right) of the flow attachment case when $\alpha = 25^\circ$ and $U_{ac} = 5.75 \text{ m s}^{-1}$. Note the maximum temperature scaled at 350 K for better visualisation.	114

List of Tables

2.1	Mesh refinement study for the case of $\theta = 0^\circ$ and $\alpha = 45^\circ$	32
2.2	Calculated fluid properties at $T_f = 338.5$ K.	35
4.1	The optimal parameters obtained in Loop 2.	85

Nomenclature

Symbols

$\dot{Q}$	Heat transfer rate (W)
$\dot{Q}''$	Heat flux (Wm^{-2})
$\dot{Q}_{AC}$	Convective heat loss with air curtain (W)
$\dot{Q}_{NC}$	Convective heat loss for natural convection (W)
Ri^*	Modified Richardson number
V	Variable in optimisation problem
ΔT	Temperature difference (K)
$\dot{m}$	mass flow rate (Kgs^{-1})
$\dot{Q}_{th}$	Thermal energy required to produce electricity (J)
g	Acceleration due to gravity (ms^{-2})
CFD	Computational fluid dynamics
Gr	Grashof number
Pr	Prandtl number
Re	Reynold number
ρ_a	density of air (Kgm^{-3})
b	Thickness of the jet outlet (m)
G	Gap between heated surface and jet (m)
T_∞	Ambient temperature (K)
T_w	Surface temperature (K)
A_{ac}	area of jet outlet (m^2)
b	Thickness of the jet outlet (m)
C_μ	Turbulent coefficient
I	Turbulent Intensity
k	Turbulent kinetic energy (m^2s^{-2})
l	Reference length scale of jet outlet (m)
P_{elec}	Required electricity power to operate the fan (W)
P_{fan}	Power used by the fan (W)

P_{jet}	Power consumption at the jet outlet (W)
P_s	Power savings by air curtain (W)
T_{ac}	Jet outlet temperature (K)
U_{ac}	Jet speed (m s^{-1})
Nu_y	Local Nusselt number
Ra_y	Local Rayleigh number
h	Heat transfer coefficient ($\text{W m}^{-2}\text{K}$)

Greek

α	Air curtain angle ($^\circ$)
β	Thermal expansion coefficient (K^{-1})
η_{fan}	Efficiency of the fan
η_{motor}	Efficiency of the motor
η_{PB}	Efficiency of the power block
κ	Thermal conductivity of the fluid ($\text{W m}^{-1}\text{K}^{-1}$)
ν	Kinematic viscosity (m^2s^{-1})
ω	Turbulent dissipation rate (m^2s^{-3})
σ	Thermal diffusivity (m^2s^{-1})
θ	Heated wall inclination ($^\circ$)
ε	Global effectiveness
ε^*	Actual air curtain effectiveness
ε_y	Local effectiveness

Subscripts

AC or ac	Air curtain
j	jet
NC	Natural convection
s	Savings
w	Wall
t	turbulent
x	Horizontal axis (m)
y	Vertical axis (m)

Introduction and Background Literature¹

1.1 Motivation

Combating global warming is one of the most challenging issues facing the planet today. The average surface temperature of the Earth's atmosphere is rapidly increasing. According to the Intergovernmental Panel on Climate Change's sixth assessment report (Hennessy et al., 2022), the earth is projected to approach the critical 1.5 °C warming limit in the early 2030s. As the temperature rises, various impacts are changing aspects of our climate, such as hotter summers, rising ocean temperatures, melting polar ice, and increased storm activities. If these impacts grow more frequent and severe, the Paris Agreement (Rogelj et al., 2016) signed by 196 countries may not be enough to prevent the hothouse climate. Extreme weather conditions are already posing severe problems in many parts of the world. One recent example is Australia, which had record-breaking temperatures and an intense drought season in the country's east during the black summer bushfires of 2019-2020 (Figure 1.1). The fires damaged approximately 3113 dwellings, killed 33 people and numerous wildlife and had an estimated economic impact of up to 20 billion Australian dollars (Filkov et al., 2020).

Over 97% of scientists agree that humans are actively causing climate change (Hennessy et al., 2022). The increased use of fossil fuels such as coal, oil,

¹Material in sections 1.2 and 1.3.3 of this chapter has been published as: R. Mondal, J. F. Torres, G. Hughes, and J. Pye. Air curtains for reduction of natural convection heat loss from a heated plate: a numerical investigation. *Int. J. Heat Mass Transfer*, 189:122709, 2022.



Figure 1.1: Black summer bushfire 2019-2020 (Bradstock et al.).

and gas has resulted in dramatic increases in carbon dioxide (CO₂) in the atmosphere. Carbon dioxide is the most damaging of the greenhouse gas emissions. It traps heat in the Earth's atmosphere, resulting in increasing the Earth's temperature significantly. The growing production of carbon dioxide is a major contributor to climate change. Hence, steps must be taken to limit greenhouse gas emissions from the Earth's atmosphere. One strategy for mitigating the long-term impacts of climate change is to increase our reliance on renewable energy sources that replenish quicker than they can be used. Solar energy, or the power of the Sun, is one of the widely available renewable energy sources. The amount of sunlight intercepted by the earth continuously is 1.75×10^{14} kW and 1.05×10^{14} kW reaches the earth's surface continually by considering a 60% transmittance through the atmospheric cloud cover (Kalogirou, 2013). If 0.1% of the total solar energy could be converted to electricity with an efficiency of 10%, it would provide a resource base of 105 TW which is approximately 3 times the world's total generating capacity of about 3.8 TW (Motyka and Jones, 2022; Thirugnanasambandam et al., 2010). Moreover, unlike fossil fuel, solar energy is clean and can be supplied without polluting the environment (Kalogirou, 2004).

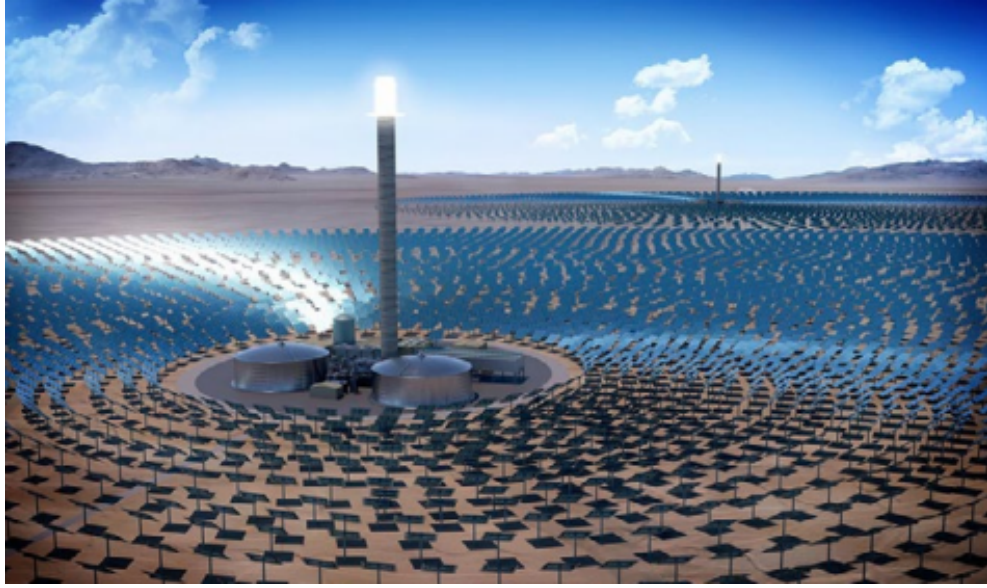


Figure 1.2: Concentrated solar power (CSP) tower and heliostat field (Ahmadi et al., 2018).

Two primary types of solar power production technologies are photovoltaic (PV) and concentrated solar power (CSP) (Figure 1.2) (Barlev et al., 2011). PV technology generates electricity directly by utilizing the photoelectric effect of semiconductor materials. Because of the lowering cost of PV systems, the PV industry has seen a successful exponential development era during the last decade. Overall capacity has grown by more than 22 times, from 23 GW in 2010 to 512 GW at the end of 2019 (Song et al., 2021). Nonetheless, this method lacks a natural heat storage mechanism. As a result, to compensate for the intermittent nature of their power source, supplemental storage solutions such as batteries are necessary. Batteries are not renewable, are expensive, and important components such as lithium and cobalt are expected to run out by 2050 (Vaalma et al., 2018).

Unlike flat plate photovoltaic (PV) or non-concentrating thermal systems, CSP technologies concentrate solar energy into a small area using a combination of mirrors or lenses (Lovegrove and Stein, 2021). The heat generated by solar irradiation is then transferred to a working fluid, such as liquid, molten salt, or air, and used in thermodynamic cycles to produce electricity. These systems can generate power even when the Sun is not shining, which gives them a substantial advantage over solar-generating technology. This is achievable by

including energy storage technology, such as thermal storage tanks, to store extra thermal energy in the daylight for use when sunlight is absent. CSP is a commercially mature technology and has the potential to make significant contributions to global energy demands. Because of the highly volatile surge in natural gas prices and environmental hazards, this technology is gaining significant attention around the globe. Besides, the price of CSP modules has fallen dramatically in the last decade, providing a lower levelised cost of electricity (Lilliestam et al., 2017; Kurup and Turchi, 2015). The overall amount of installed capacity is promisingly increasing day by day. By the end of 2021, the cumulative installed global solar thermal power capacity has increased to 6800 MW and a large number of CSP projects are still under development (Mehos et al., 2017; Wang, 2021). Currently, the market offers four of the most common CSP technologies. They are the parabolic trough collector (PTC), the linear Fresnel collector (LFC), the parabolic dish collector (PDC), and the solar power tower (SPT), also known as central receiver technology (Krothapalli and Greska, 2011). They differ depending on whether they are line or point-focusing. There is also a distinction between either the distribution of an optical concentrator in the form of small planar fragments and continuous curved concentrators. Figure 1.3 depicts the four principal types of CSP technologies (Bank, 2021).

As the name implies, line-focus technologies, such as the parabolic trough collector and linear Fresnel collector, concentrate sunlight along the focal length of the collector. Whereas the point focus technologies, such as the Stirling and solar tower, concentrate sunlight at a single point at the top of the tower or in the middle of the parabolic dish. Line focus methods concentrate solar energy around 100 times, reaching temperatures in the heat transfer fluid (HTF) of 400-550 °C (Zhang et al., 2013). The steam produced at this temperature range is of moderate quality. Point focus technologies concentrate solar energy roughly 1000 times, heating the HTF to 600-1000 °C, which is double the temperature obtained by line focus (Richter et al., 2009). This distinction makes the point-focus approach more efficient since it uses less land and has the lowest cost per KWh.

The parabolic trough is the most economically and technically proven of all solar technologies (95.7% of all operating CSP projects), providing for the low-

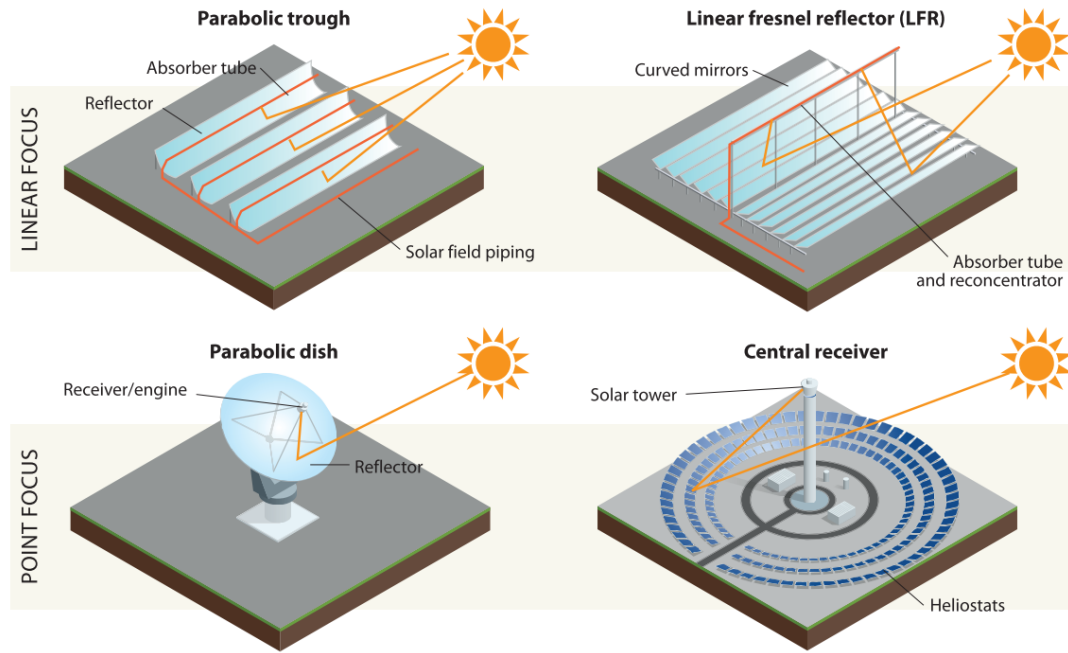


Figure 1.3: Four main CSP technologies with basic components (Bank, 2021).

est costs and economic risks (Sánchez, 2015). However, there is a tendency toward larger-scale CSP technologies; for example, solar power tower technology now accounts for 71.43% of projects under construction, compared to 28.57% for parabolic trough technology, with capacities of 110 MWe and 17.5 hours of thermal energy storage (Baharoon et al., 2015). Given the growing interest in large-scale applications, this PhD thesis focuses on solar power tower technology.

Solar power towers are complicated integrated systems comprised of several subsystems (see Figure 1.4). The important subsystems are:

- The mirror field that gathers solar radiation and directs it to a focal point by tracking the Sun during the day.
- The heat transfer fluid system extracts heat from the receiver and distributes it to the storage and/or power block.
- The receiver captures the radiation and transforms it to high temperatures.

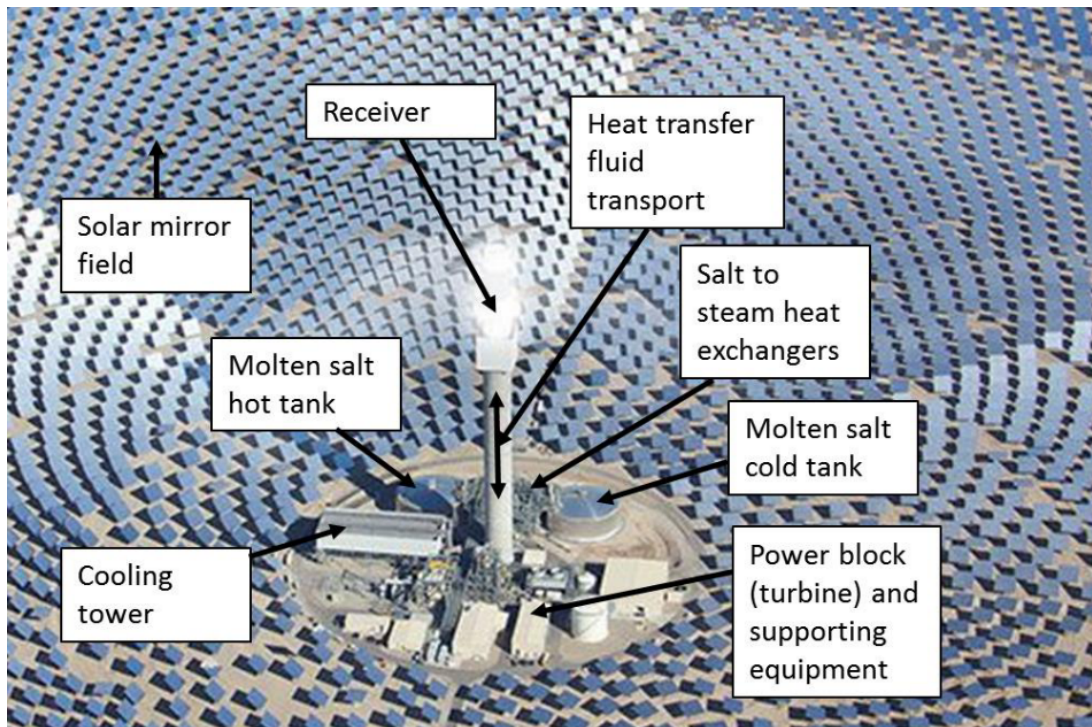


Figure 1.4: Key subsystems of a molten salt solar power tower systems (Lovegrove et al., 2018).

- The power plant and its related equipment, which is normally based on a steam turbine and an electrical generator.
- The thermal storage subsystem is commonly comprised of two tanks of heated liquid salt, although alternative techniques may also be used.

The receiver is one of the most significant components of a solar power tower system since it absorbs incoming solar energy on its surface. Two conventional types of receivers are widely employed in SPT systems: cubical cavity and external receivers (Figure 1.5). The external receivers are relatively inexpensive, durable, and proven technology based on concepts from heat exchanger design (Boerema et al., 2013). However, this design incurs more considerable reflection and thermal emission losses than cavity receivers due to the hot surfaces being completely exposed to the environment (IRENA, 2018). According to Ho and Iverson (2014), the receiver efficiency drastically reduce from 94% to 85%, as the temperature of the receiver increases from

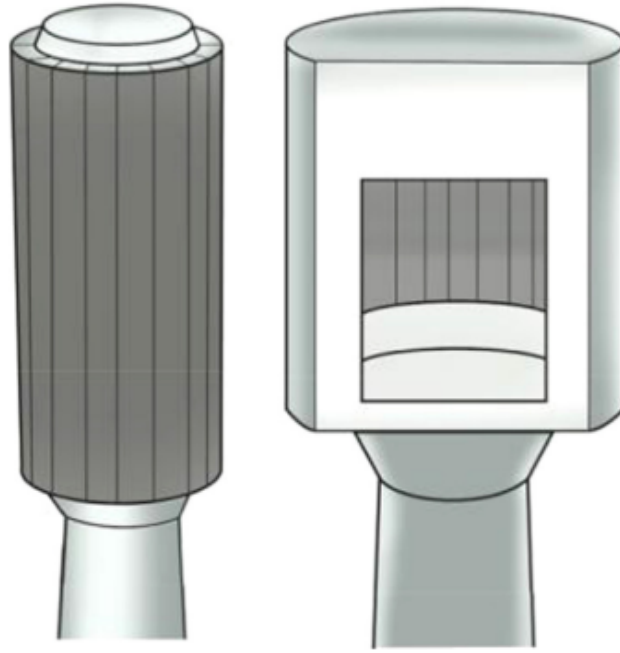


Figure 1.5: Schematic of external (left) and cavity (right) receivers in solar power tower system (Ho and Iverson, 2014).

200 °C to 800 °C. Therefore, attention must be paid to minimizing these losses from the external receiver surface to ensure maximum efficiency. This is why this thesis has attempted to propose a technological advancement in the external receiver design to improve its thermal performance.

1.2 Research scope

Solar receiver encounters significant energy losses from their surface, reducing overall efficiency. Mitigating these losses is a significant challenge for the research community. Energy losses from the solar receiver in CSP systems primarily occur through conduction, radiation, and convection. Although analytical approaches may be used to reasonably estimate conduction and radiation terms, convective heat losses are the most challenging to predict due to their complicated dependency on various geometrical parameters and operating conditions. Convection in the receiver is caused by fluid density gradients between ambient and receiver temperatures. External receivers are the dominant design and experience more considerable convection losses than

cavity receivers because the receiver surfaces are completely exposed to the environment. Hence, reducing the convective heat loss from the external receiver would be a significant value to limit convective heat loss and improve thermal performance efficiently. Thus, this study motivates us to develop a new, cost-effective strategy to minimize convective heat losses from the external receiver surface. Reducing this loss from the receiver ensures higher operating temperatures, making the receiver more efficient.

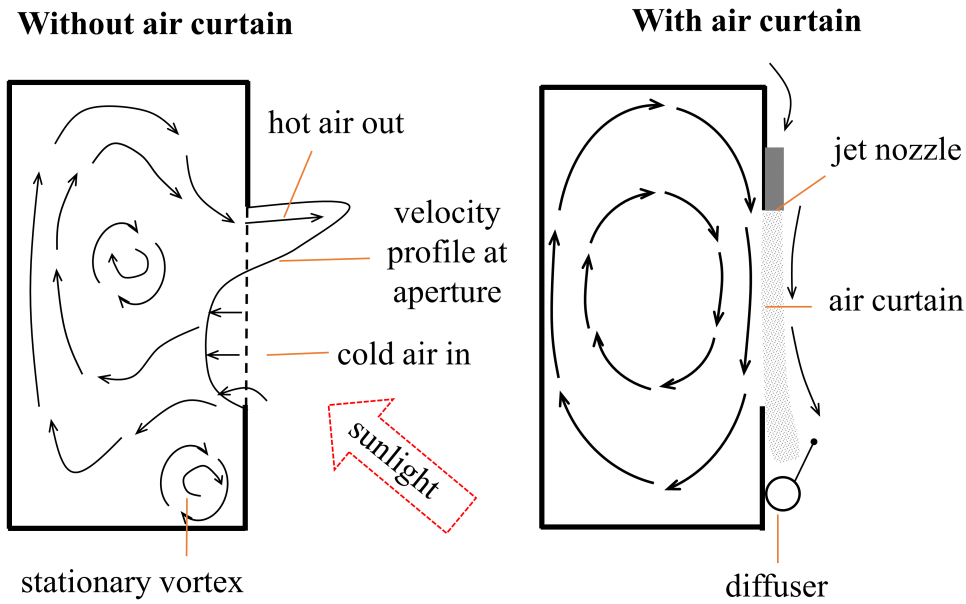


Figure 1.6: Conceptual sketch showing the application of air curtains to reduce convective losses in a cavity receiver.

Active airflow control in the form of an air curtain could be a potential method to mitigate the convection losses from the external receiver surface. A substantial amount of work was dedicated to improving the thermal performance of cavity receivers using air curtain devices (Section 1.3.3). Figure 1.6 demonstrates a conceptual sketch showing the application of air curtains to reduce convective loss in a cavity receiver. Nevertheless, relatively little research has been undertaken for the external receivers in central tower systems in the presence of the air curtain (Kelly and Robert, 1990; Wang et al., 2021), despite the latter being increasingly dominant in commercial applications. This leaves the potential space to implement this method further for the

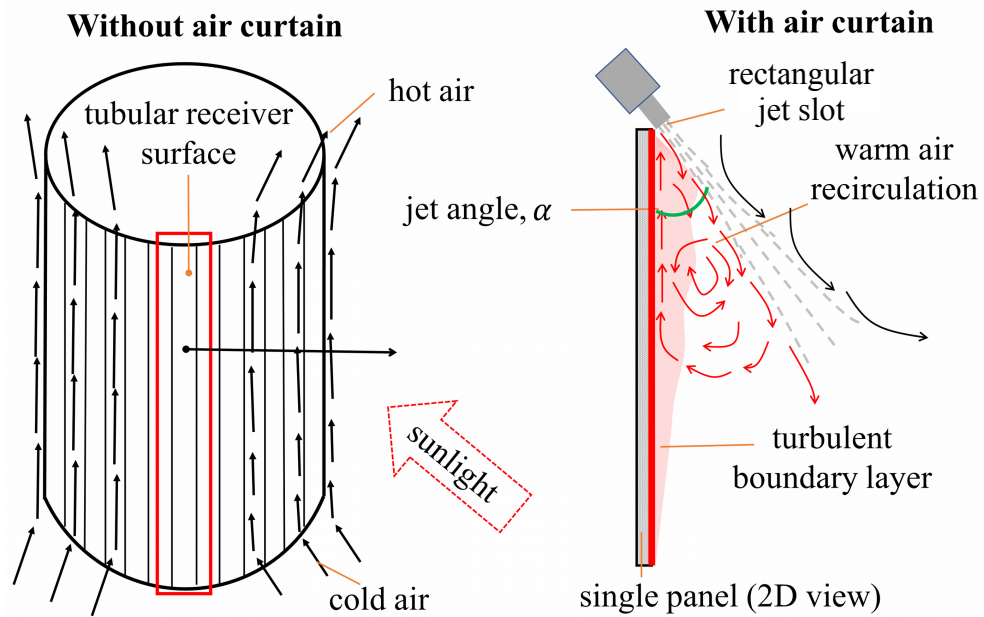


Figure 1.7: Conceptual sketch showing the application of air curtains to reduce convective losses from an external tubular receiver.

investigation of improving the thermal performance of the external receiver.

Given the aforementioned research gaps identified above, this study aims to propose an obliquely introduced air curtain to mitigate convective heat loss from an external heated surface. The external receiver is typically cylindrical or flat in shape and comprises a series of distinct vertical panels. Each panel is formed by a bank of tubes that transport the heat transfer fluid. If the surface roughness of each panel (due to tube curvature) is ignored, each panel can be characterized as a vertical flat plate. Therefore, the proposed method in this thesis can later be engineered on an external receiver surface. To the best of the author's knowledge, this flow configuration has not been studied in a detailed and isolated way in any prior works. Figure 1.7 presents a conceptual sketch showing how an air curtain can be implemented on an external receiver panel to suppress the natural convection heat loss from its surface.

1.3 Background Literature

This research combines two important key studies which have been broadly investigated by many researchers in the last few decades. One is the natural convection flow over an isothermal heated surface, and another is the planar jet air curtain. This section has made an effort to provide a theoretical background on these topics.

1.3.1 Natural convection along a heated flat plate

Natural convection is an omnipresent thermal flow phenomenon induced by buoyant force (Gebhart, 1962, 1973; Bodenschatz et al., 2000), which may emerge in various geometries, for instance, cylinder (Ganesan and Rani, 1998; Chen and Yuh, 1980; Lee et al., 1988), sphere (Bishop et al., 1966; Jia and Gogos, 1996), flat plate (Zhao et al., 2013; Tsuji and Nagano, 1988; Shavazi et al., 2018; Cheesewright, 1968; Churchill and Chu, 1975), and channels under different boundary and flow conditions (Desrayaud et al., 2013; Marcondes and Maliska, 1999). Natural convection from flat plate geometry is a fundamental case and has received more attention from the research community because of its wide range of applications in various engineering and natural systems (Fan et al., 2021; Kalendar and Oosthuizen, 2011). An important setting of this configuration is found in concentrating solar power (CSP) receivers (Pye et al., 2017; Torres et al., 2018; Dennis and Kraabel, 1984). As mentioned above, each vertical panel of an external receiver surface is characterized as a flat plate if the surface roughness is ignored. The vertical length of this panel can be up to 15 m to 25 m or more (Siebers and Kraabel, 1984). Given the large dimensions and high temperatures at which these receivers operate, turbulent natural convection is a frequently-occurring flow regime, and if relatively low wind speeds are present in the vicinity of an external receiver, natural convection becomes the dominant mode of heat transfer (Nock et al., 2016).

Natural convection corresponds to situations where the fluid is set in motion by buoyancy forces due to density variations. These density variations may result from heat transfer inside or between the fluid and heated or cooled solid walls. Figure 1.8 describes how the natural convection flow is developed around a heated vertical plate. The fluid temperature surrounding the

heated plate increases, making it less dense and starting to rise by thermal expansion. This establishes a natural convection boundary layer along the plate (similar to Figure 1.8). The velocity distribution inside the boundary layer changes along the vertical direction. Due to the no-slip boundary conditions, the velocity at the wall is zero; it rises to a certain peak value and then reduces to zero at the edge of the boundary layer as the free stream is at rest. The growth of boundary layers is laminar. However, turbulent eddies are formed at some distance from the leading edges, and the transition to turbulent boundary layers starts depending on the fluid characteristics and the temperature difference between the wall and the atmosphere. The boundary layer further up the plate is entirely turbulent.

Convective heat transfer is the transfer of heat from one place to another by the motion of fluids. It constitutes any heat transfer between a solid surface and an adjacent body of fluid, such as air. According to Newton's law of cooling, convective heat transfer from the surface can be written as,

$$\dot{Q} = hA(T_w - T_\infty) \quad (1.1)$$

Here, h is the heat transfer coefficient, and A is the area of the surface.

As mentioned above, when a cold fluid passes through a heated plate, the fluid is exposed to the no-slip condition at the wall. Therefore, the fluid movement near the wall is very low. Heat transfer in this area occurs through conduction. There is a region above the sub-layer where viscous forces delay fluid motion; convection may happen in this region, but conduction may play a major role. Convective heat transfer rate can be demonstrated at the wall as,

$$\dot{Q} = -\kappa \frac{\partial T}{\partial y} \quad (1.2)$$

Heat transfer between the solid surface and fluid is a notable feature of convection through conduction. By equating convective and conductive heat transfer and integrating along the surface, the convective heat transfer co-

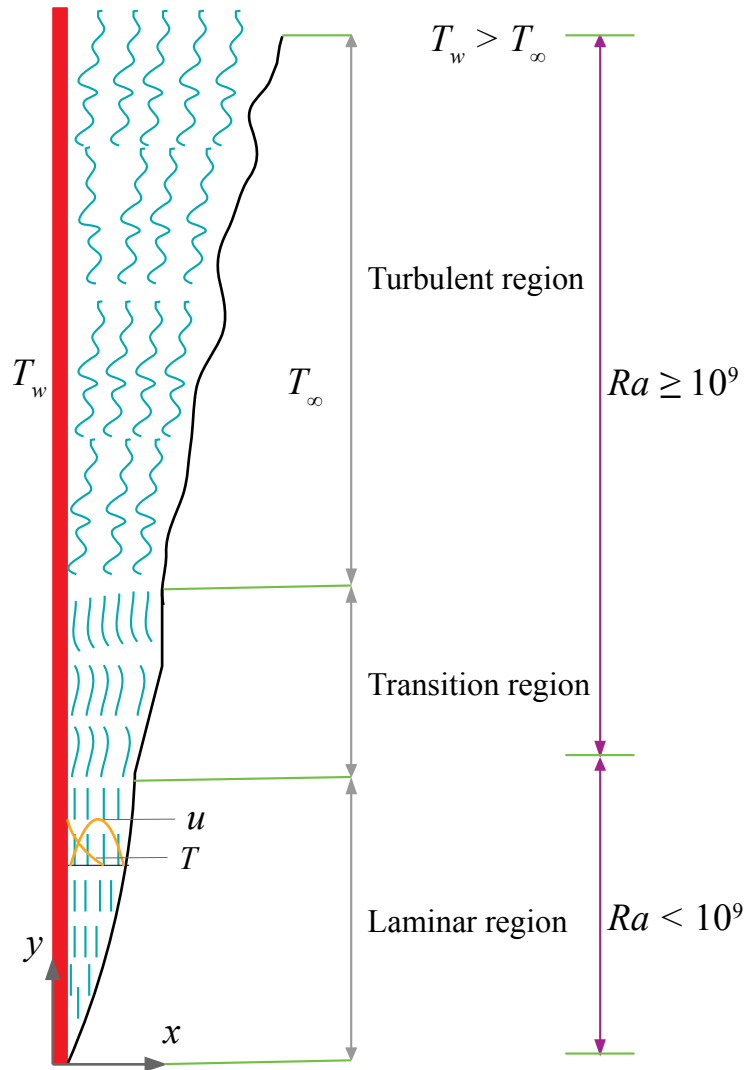


Figure 1.8: Natural convection boundary layer along a heated vertical flat plate.

efficient can be calculated using the following equations,

$$h = \frac{-\kappa \frac{\partial T}{\partial y}}{A(T_w - T_\infty)} \quad (1.3)$$

It is worth recalling some important control parameters, such as the Rayleigh number, Grashof number, Prandtl number, and Nusselt number, that are frequently adopted to characterize the natural convection behaviour. Rayleigh

number is an important dimensionless number for characterizing natural convection adjacent to a vertical heated surface. It is described as the product of the Grashof number and Prandtl number, where the Grashof number shows the relationship between buoyancy and viscosity forces within a fluid, and the Prandtl number depicts the relationship between momentum diffusivity and thermal diffusivity,

$$\text{Ra}_y = \text{Gr}_y \cdot \text{Pr} \quad (1.4)$$

$$= \frac{g\beta\Delta T y^3}{\nu^2} \times \frac{\nu}{\sigma} \quad (1.5)$$

$$= \frac{g\beta\Delta T y^3}{\nu\sigma}, \quad (1.6)$$

Characterizing the laminar to turbulence transition of a free convection boundary layer flow is the most significant use of the Rayleigh number. The boundary layer acting as a thermal resistance to the leading edge of the surface will be comparatively thin, resulting in a relatively large convection coefficient. At a Rayleigh number of about 10^9 the flow over a flat plate will become transitional and eventually become turbulent. The excessive turbulence within the boundary layer will increase heat transfer resulting in relatively strong convection coefficients. For vertical flat plate, $\text{Ra} < 10^9$ represents laminar flow and $\text{Ra} > 10^9$ depicts turbulent flow.

Nusselt number is another essential parameter used for characterizing the natural convection phenomenon. It represents the ratio of convective heat transfer to the conductive heat transfer at a boundary in a fluid. Convective heat transfer occurs through advection (fluid motion) and diffusion (conduction). Between these two, the conductive part is measured under the same principle of convective but hypothetically for a motionless fluid. Nusselt number is defined as,

$$\text{Nu}_y = \frac{hy}{\kappa} \quad (1.7)$$

The selection of the characteristic length of a flat plate is in the direction of

growth or thickness of the boundary layer. Nusselt number value usually characterizes whether the flow is laminar or turbulent. A larger value of the Nusselt number indicates more active convection, with turbulent flow typically occurring in the 100-1000 K range. Nusselt number values between 1-10 represent slug flow or laminar flow.

There are some empirical correlations of Nusselt number that have been developed for the vertical plate. Churchill and Chu (1975) have suggested a correlation that can be employed over the entire range of Ra_y which is of the form,

$$Nu_y = \left[0.825 + \frac{0.387Ra_y^{1/6}}{[1 + (0.492/Pr)^{9/16}]^{8/27}} \right] \quad \text{for } 10^{-1} < Ra_y < 10^{12} \quad (1.8)$$

Although the correlation stated in equation 1.8 is ideal for all engineering problems, the following equation 1.9 (Churchill and Chu, 1975) can make a slightly better approximation for the laminar regime.

$$Nu_y = \left[0.68 + \frac{0.670Ra_y^{1/4}}{[1 + (0.492/Pr)^{9/16}]^{4/9}} \right] \quad \text{for } Ra_y \leq 10^9 \quad (1.9)$$

By convention, it is common to assess the fluid properties for these dimensionless parameters and convective heat transfer problems at the film temperature. The film temperature is defined as the average wall temperature and ambient temperature.

1.3.2 Planar jet (Air curtain)

Air curtains are turbulent plane jets that create a transparent air barrier across the entrance to separate two distinct areas with specific conditions. It is commonly utilized across a wide variety of practical applications, including industrial furnaces (Oliveira et al., 1991), open refrigerated display cabinets (Sun et al., 2017), buildings (Goubran et al., 2016), and energy savings (Van Belleghem et al., 2012; Costa et al., 2006). With wide-ranging appli-

cations, significant effort has gone into researching the fluid mechanics of planar jets. Guyonnaud et al. (2000) presented it well in a study examining the design of air curtains.

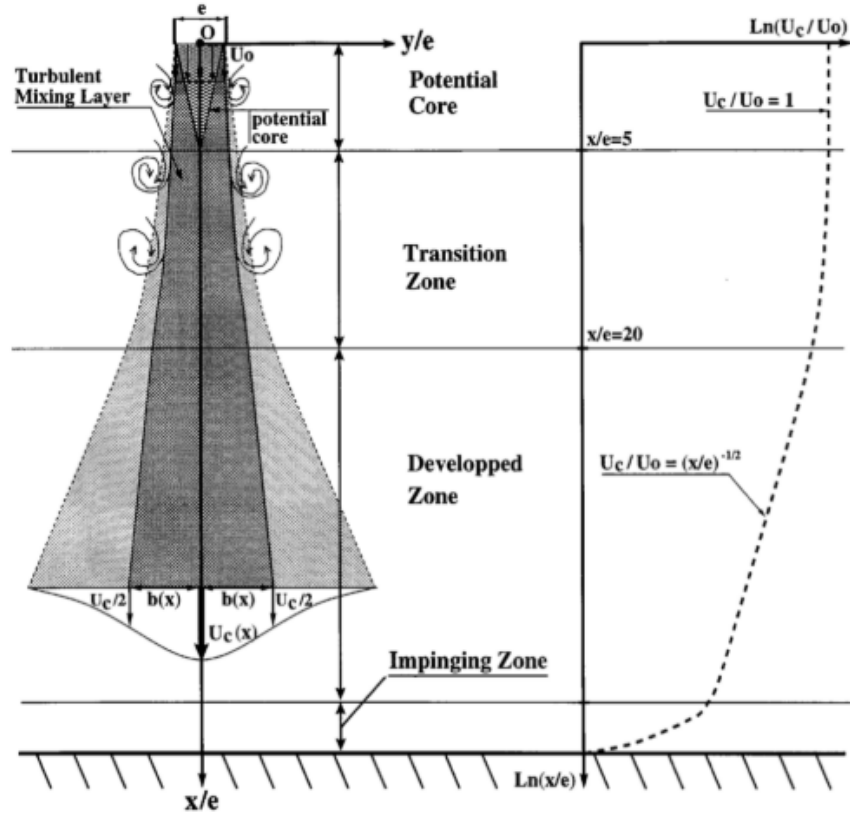


Figure 1.9: Schematic of flow regions for planar jet (Guyonnaud et al., 2000).

There are four distinct zones in a planar jet flow field: potential core, transition, developed and impinging. A constant centre-line velocity characterizes the potential core region. That is, the central velocity of the jet does not diminish. The spread of the jet is also limited, as its thickness does not increase significantly. Depending on the nozzle shape and initial flow conditions, the length of this zone is typically five to eight times the nozzle thickness (Van and Howell, 1976). Beyond this point lies the jet's transition zone, which extends for about 20 times the nozzle thickness. The centre line velocity starts to exhibit some decay while the jet thickness spreads at a greater rate. Eventually, the developed zone occurs, with a larger deceleration to the jet's centre line and significant spreading. The impinging zone, which happens when the

flow touches the floor, is irrelevant to this project.

When the planar jets are formed into an air curtain, some essential parameters need to be considered in terms of implementation. Costa et al. (2006) and Van Belleghem et al. (2012) characterized three distinct flow settings based on jet constraints. A low-speed air jet has insufficient momentum to tackle pressure variations across the doorway. These pressure differences primarily comprise two components. The first one is a consequence of the air curtains operation. Air is drawn from one place but thrown into both places, resulting in pressure build-up. This pressure, known as the auxiliary pressure, allows the jet to flow in the direction of the air being taken. The second component of the difference in pressure happens owing to the difference in temperature across the gate, creating a difference in density. This type of pressure is called stack pressure and varies linearly from top to bottom of the aperture.

On the other hand, a powerful jet creates a super-stable flow regime that enables the opening to be completely sealed. This flow pattern is dominated by forced convection at a high-speed jet. There is another jet flow configuration between the sub-stable and super-stable flow regime, called the breakthrough system, where the jet has enough velocity to initiate a full seal. The efficiency of the air curtain is greatest in the breakthrough regime as the jet is stable enough to reach the opposite side and has minimal air entrainment through the opening.

Dimensioning of the air jet is necessary to achieve a breakthrough regime. The dimensioning can be performed based on various principles, namely the superposition of flows, momentum balance, the balance of kinetic energy, and empirical methods (Sirén, 2003). According to his study, momentum balance is the most significant parameter for jet dimensioning. Some models have already been developed based on numerical correlations incorporating dimensional parameters. The essential parameter that characterizes the stability of the air curtain is that the deflection modulus (D_m), which is the proportion of air curtain momentum to transverse forces, is triggered by either side of the curtain's temperature difference. It was first analytically described by Hayes and Stoecker (1969) under the assumption that temperature is inversely proportional to density,

$$D_m = \frac{b\rho_0 U_{ac}^2}{gH^2(\rho_w - \rho_\infty)} \quad (1.10)$$

$$= \frac{bU_{ac}^2}{gH^2(\frac{\rho_w}{\rho_0} - \frac{\rho_\infty}{\rho_0})} \quad (1.11)$$

$$= \frac{bT_w T_\infty U_{ac}^2}{gH^2 T_0 (T_w - T_\infty)} \quad (1.12)$$

Some authors also used the Richardson number to characterize the stability of the air curtain instead of the deflection modulus (Chen and Yuan, 2005). Richardson number is the proportion of the buoyancy force to the momentum force. It represents the significance of natural convection relative to forced convection.

$$Ri = \frac{Gr}{Re^2} \quad (1.13)$$

$$= \frac{g\beta H^3(T_w - T_\infty)}{\nu^2} \times \frac{\nu^2}{U_{ac}^2 H^2} \quad (1.14)$$

$$= \frac{g\beta H(T_w - T_\infty)}{U_{ac}^2} \quad (1.15)$$

1.3.3 Use of air curtains in the solar-thermal systems

Air curtain applications in the solar-thermal context are limited. Taussig (1984) first developed the concept in 1984, proposing an aerodynamic method for a central receiver. Despite having limited computational resources compared to today, he conducted extensive calculations showing that the idea was feasible and showed energy benefit within the range of cavity temperature 1000 K-1600 K. However, further development of this concept stopped until the works of Taumoefolau (2004) and Paitoonsurikarn et al. (2011) concerning the convective heat losses on a solar cavity receiver. The outcome of those studies revealed that certain crosswind velocities and orientations past the cavity aperture could mitigate convective heat losses (Figure 1.10), suggesting that a similar result could be achieved by using an air curtain. Based

on this concept, the following results were reproduced by researchers, both experimentally and computationally.

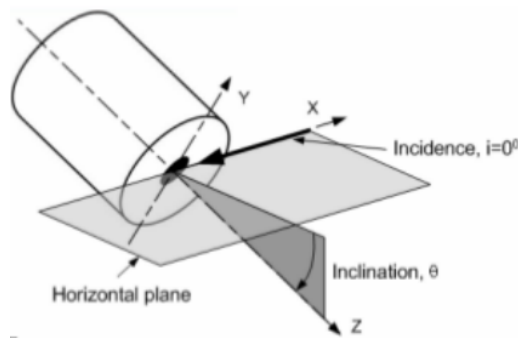
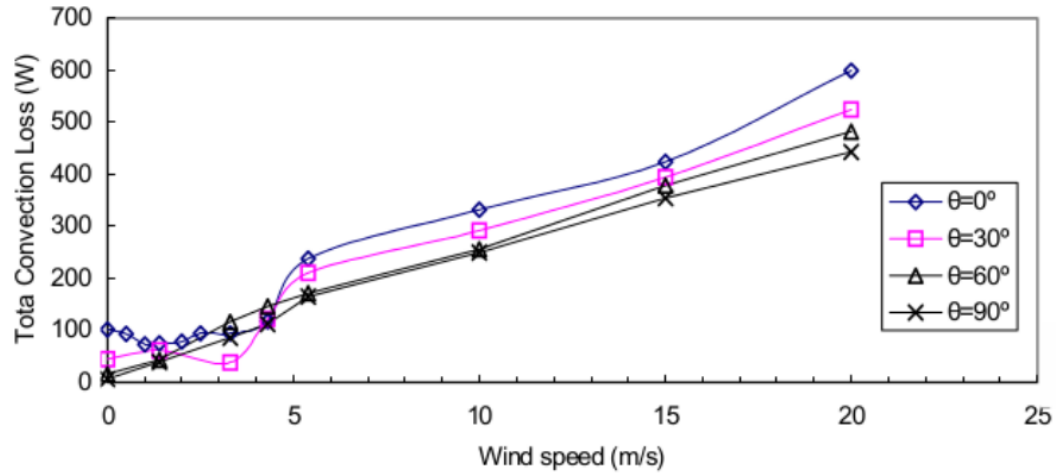


Figure 1.10: Convective heat losses against wind speeds for $i = 0^\circ$ (Taumoe-
folau, 2004).

McIntosh et al. (2014) used a two-dimensional computational fluid dynamics (CFD) model to understand heat transfer mechanisms when an air curtain is directed perpendicularly across the aperture of an open-ended cavity (Figure 1.11). A range of cavity inclinations and air curtain velocities was studied in their study. However, they did not vary the air curtain inclination with respect to the aperture plane. They reported a maximum reduction in 54% convective heat loss with an air curtain.

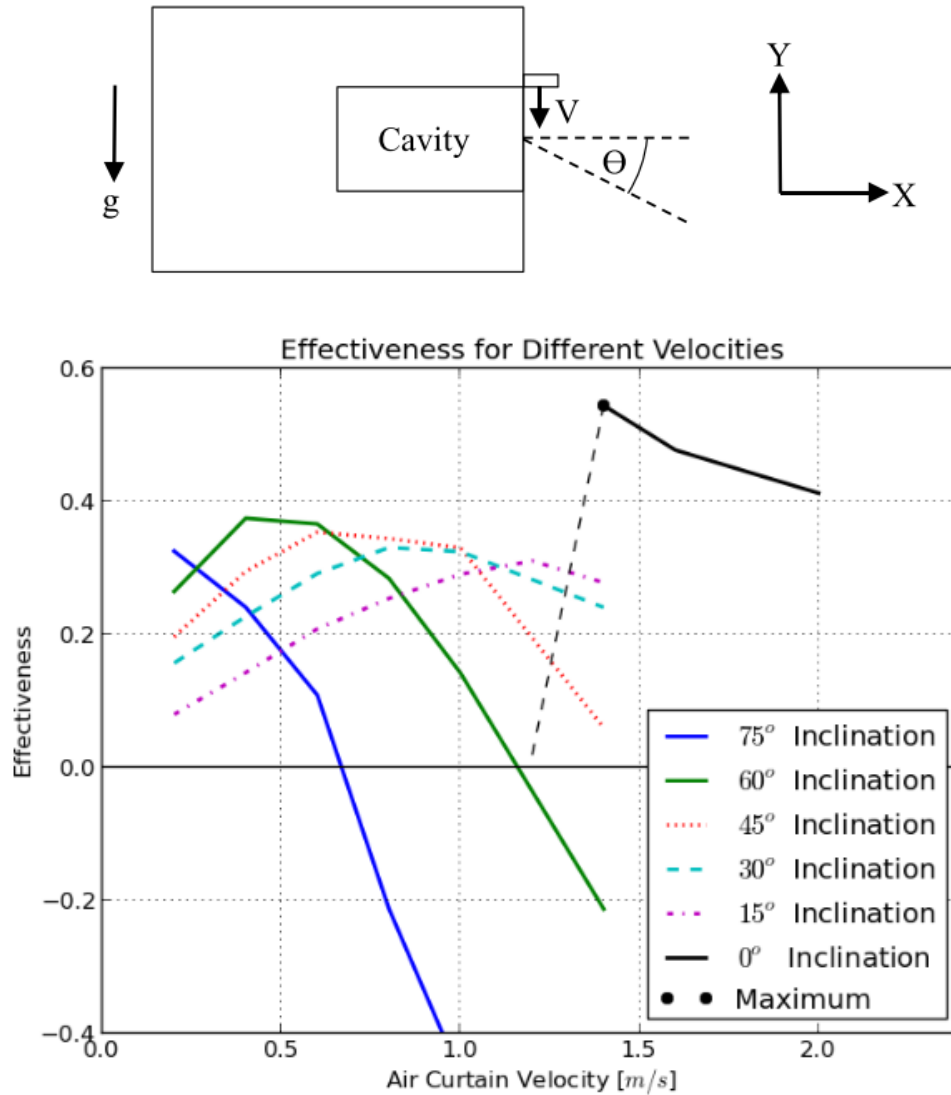


Figure 1.11: McIntosh et al. (2014) geometrical setup (top) and air curtain effectiveness for different jet speeds (bottom).

Zhang et al. (2015) extended this work via CFD simulations to examine air curtain inclination (Figure 1.12). Their findings unveiled two distinct mechanisms for the reduction of heat losses. Firstly, a partially sealed mode could be achieved by applying a low-speed jet that does not isolate the cavity aperture completely; here, the air curtain increases the stagnation zone inside the cavity and could retain more heat. Secondly, a relatively high-speed turbu-

lent jet could achieve a fully sealed mode. The jet had sufficient momentum to reach the far side of the aperture. The results suggested that using an air curtain could reduce convective heat loss by up to 70% for a cavity receiver. However, these losses can also be increased if the planar jet speed is too high or misdirected. Therefore, it is necessary to control jet speed and direction to achieve optimum effectiveness.

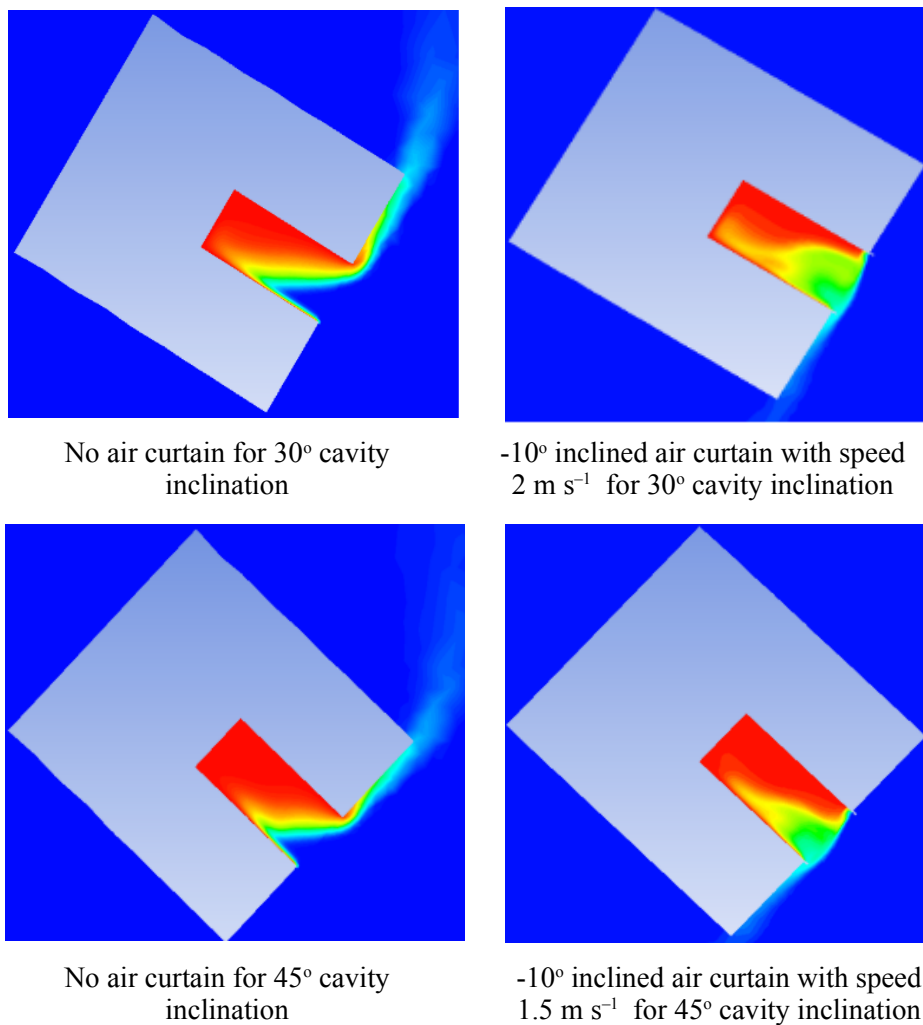


Figure 1.12: Simulation results based on Zhang et al. (2015).

In a related study, Flesch et al. (2016) used computational simulations to investigate the effect of wind on the air curtain operation (Figure 1.13). They found that an air curtain can mitigate the heat losses from the receiver in windy conditions if the air curtain speed is sufficiently large. Though a high-

speed curtain could disturb the temperature field inside the cavity, this was acceptable because it had less impact than the wind without the air curtain. Their results showed that convective heat losses could be reduced to a maximum of 50%.

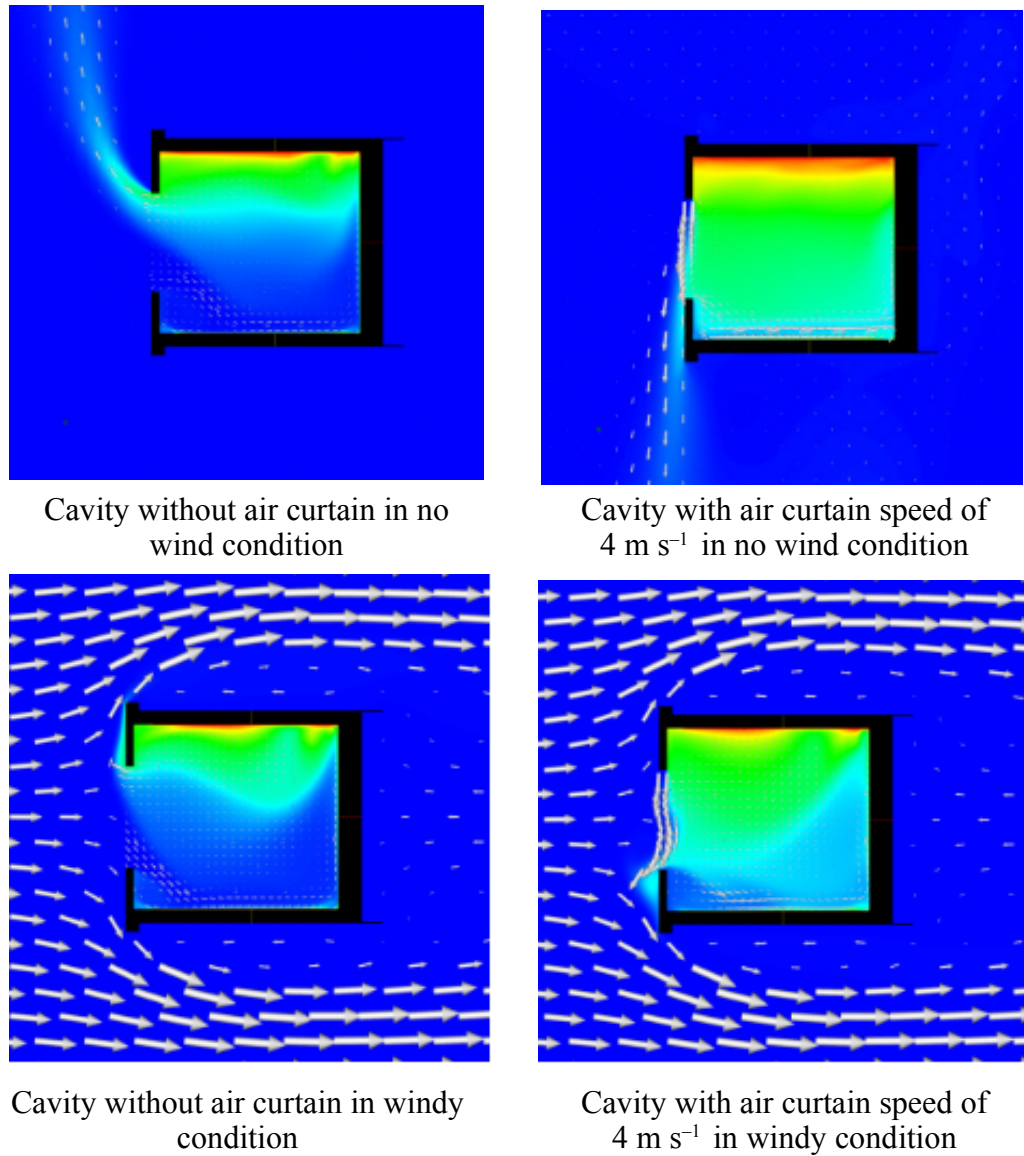


Figure 1.13: Simulation results of Flesch et al. (2016).

Hughes et al. (2016) showed how active flow control works qualitatively through an experiment where buoyancy was adjusted by concentration dif-

ferences in a saline solution rather than heat (Figure 1.14). The buoyant fluid replicated a stagnation zone inside the cavity. When a water jet was used as a shield (i.e. a water curtain), the stagnation region expanded in size because it reduced buoyant fluid loss from the cavity.

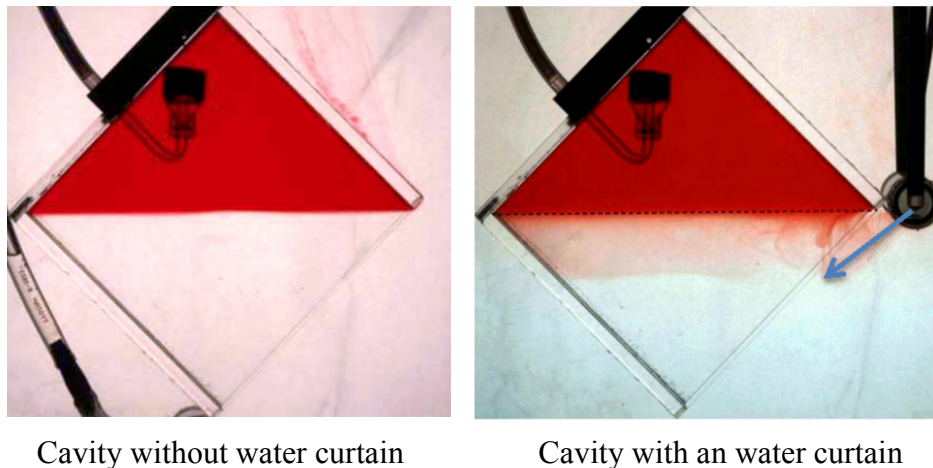


Figure 1.14: Saline water experiments of Hughes et al. (2016).

Yang et al. (2018) then conducted a numerical simulation of convective heat loss reduction from a solar cavity receiver using a U-shaped air curtain (Figure 1.15). This form of air channel helps create a confined airflow system and reduces convective heat loss, which leads to an impressive increase in the performance of a cavity receiver. Two types of airflow modes were investigated; the clockwise and anti-clockwise directions, and the air flowing in the anti-clockwise direction showed better results up to 58% of convective heat losses.

Fang et al. (2019) numerically evaluated the influence of an air curtain on the natural convective heat loss from a solar cavity receiver (Figure 1.16). In their geometrical design, a plane air nozzle was mounted at the top of the cavity opening. Four air nozzle parameters were considered such as inclination, width, outlet temperature, and outlet velocity. Their results confirmed that convective heat loss could be reduced using an air curtain. A maximum of 28.6% reduction was achieved under the optimum condition.

Finally, Alipourtarzanagh et al. (2020a,b,c) conducted some experimental and numerical investigations on reducing convective heat losses from cavity shaped

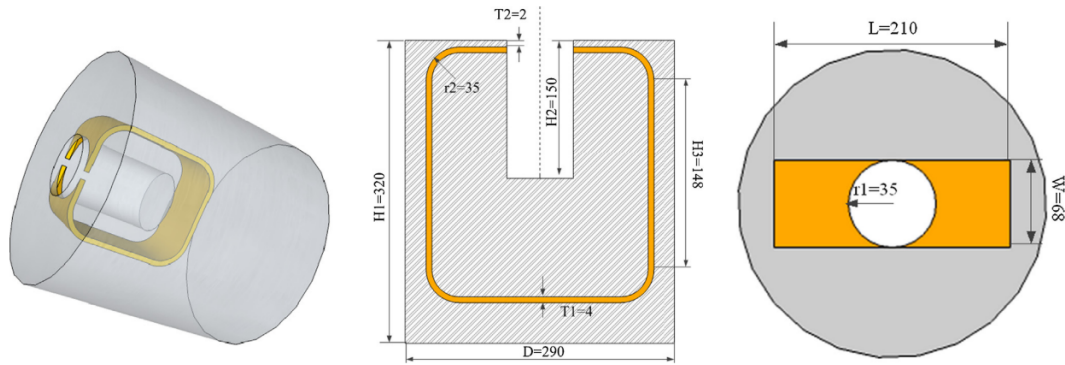


Figure 1.15: Yang et al. (2018) modified receiver with a special air channel.

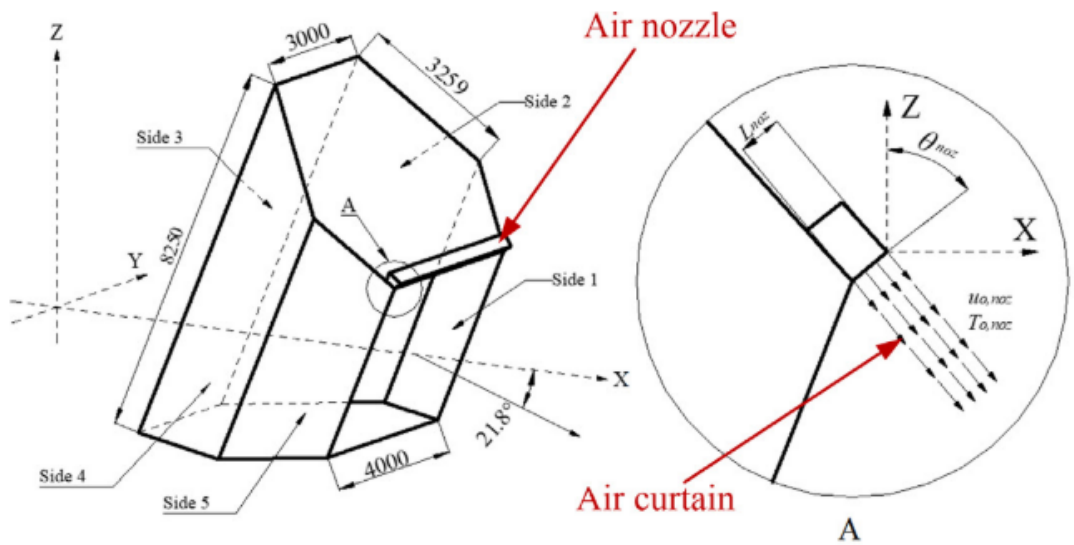


Figure 1.16: Geometrical configuration of Fang et al. (2019).

receivers using air curtain configurations (Figure 1.17). One study focused on the effectiveness of an air curtain for a tilted cavity receiver in windy conditions, varying the air curtain speed and angles. Their findings suggested that the air curtain orientation should be 30° inclined towards the wind direction instead of parallel to the aperture plane for inclined cavity receivers and reported a maximum of 60% convective heat loss reduction. In a separate study, they introduced two aerodynamic approaches to the cavity aperture plane: suction and blowing to mitigate heat losses from the receiver; suction

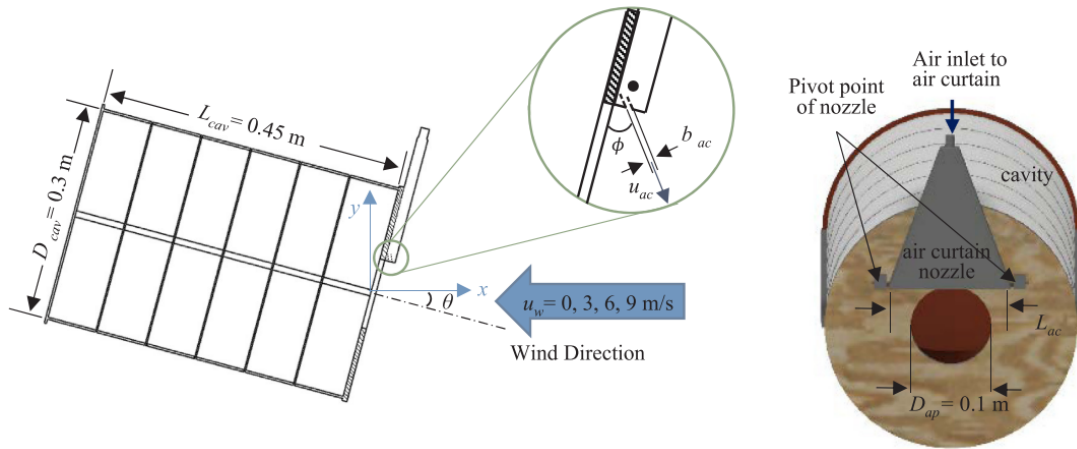


Figure 1.17: Experimental set up of Alipourtarzanagh et al. (2020a).

performed better than flow blowing at high wind speed. Furthermore, they found that a downward-facing air curtain was less effective than an upward-facing air curtain when the wind speed was low.

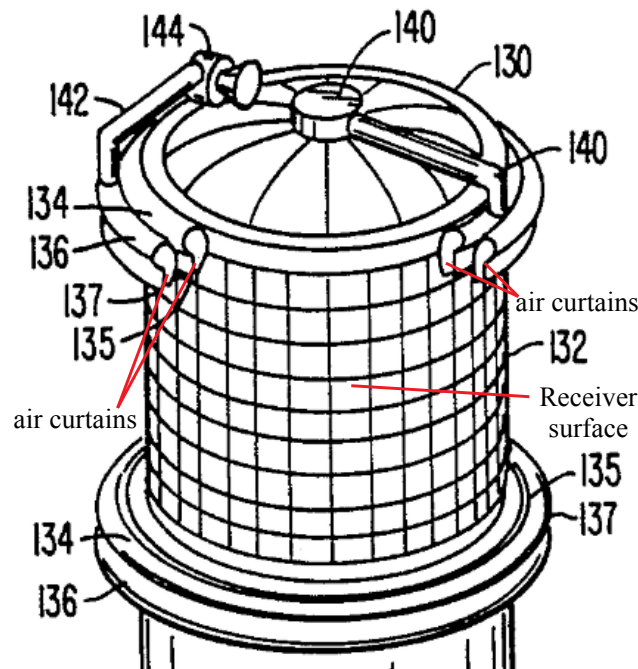


Figure 1.18: Idea developed by Kelly and Robert (1990) to introduce a protective air-stream for external receiver.

While most of the studies mentioned above were focused on cavity receivers in the presence of an air curtain, attention is being paid to improving the ther-

mal performance of external receivers using air curtains. Kelly and Robert (1990) first developed the idea of introducing a protective air stream that could minimize convective heat losses from the external solar central receiver (Figure 1.18). However, this idea was unexplored until 2021. A recent numerical study performed by Wang et al. (2021) confirmed that an air curtain could effectively reduce convective heat loss from the external solar receiver and enhance receiver performance (Figure 1.19). They investigated different air curtain configurations on the thermal performance of the solar receiver and reported a maximum of 9.60% reduction in convective heat loss, which led to an increase in the electricity power production of approximately 0.49%, clarifying the way that jet should be introduced in the thermal boundary layer to reduce convective heat loss from the receiver surface. It is essential as flow attachment and flow-resonance effects could increase heat loss Ogasawara et al. (2021), which is detrimental to solar-thermal energy conversion.

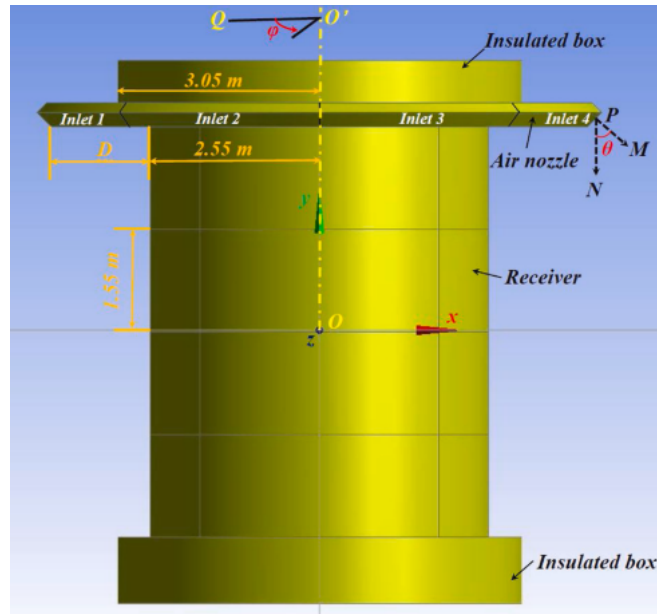


Figure 1.19: Wang et al. (2021) geometry of the external receiver having an air curtain installed on top.

1.4 Outline of the thesis

This thesis aims to address the scope outlined in Section 1.2 by investigating the minimization of convective heat loss from an external heated surface

using an air curtain. The thesis is structured into six chapters, which are organized to address the research gaps. A brief overview of each chapter is provided below.

Chapter 2 introduces the numerical methodology employed to examine the reduction of convective heat loss from an isothermal wall utilizing an air curtain. The methodology encompasses the design of the computational domain, meshing strategy, mesh refinement test, specification of boundary conditions, fluid properties, solver description, solving procedures, turbulence modeling, and post-processing utilities.

Chapter 3 presents the validation of base models for an isothermally heated wall and a turbulent planar jet. These validated models are then combined to conduct a parametric study on heat transfer from the isothermal plate with an air curtain. Four key parameters of the heated wall and air curtains are varied, including jet speed, angle, plate temperature, and inclination.

Chapter 4 focuses on optimizing the parameters of the air curtain to minimize convective heat loss from the heated surface. Four air curtain parameters, namely jet speed, angle, thickness, and the gap between the plate and jet, are considered in this investigation. The influence of jet outlet temperature is also examined under optimal jet conditions.

Chapter 5 presents a qualitative comparison between the CFD model and experimental setup. Additionally, the accompanying flow behaviors of the interaction between the buoyant plume emerging along the heated wall and the air curtain are included to enhance the modeling findings, which were absent in the experimental research.

Lastly, Chapter 6 provides a summary of the research findings and offers recommendations for potential future work in this project that should be undertaken.

Methodology¹

The current project numerically investigated the reduction of convective heat loss from an external heated surface using an air curtain. Some specific processes were followed to ensure the validation and verification of the CFD model with available literature. Versteeg and Malalasekera (2007) provided a well-documented summary of these processes in three different steps that include,

- Pre-processing
- Solving
- Post-processing

Following that, the methodology of this thesis work is also illustrated in three subsections. These processes are described in the following sections.

2.1 Pre-processing

There are some necessary steps in numerical simulations before solving any problem. These steps need to be followed in a pre-processing stage, including the design of the model, selection of size and dimension of the domain, discretization of the domain through a numerical mesh, choosing specific fluid properties, and boundary conditions. All these steps are discussed in this section.

¹Material in sections 2.1 and 2.3.2 of this chapter has been published as: R. Mondal, J. F. Torres, G. Hughes, and J. Pye. Air curtains for reduction of natural convection heat loss from a heated plate: a numerical investigation. *Int. J. Heat Mass Transfer*, 189:122709, 2022.

2.1.1 Geometrical definition of the computational domain

The physical problem is depicted in Figure 2.1, together with the dimension of the computational domain and associated boundary conditions. This proposed problem modelled a 1.8 m high wall as an external receiver surface. The surface is subject to isothermally heated at $T_w = 110^\circ\text{C}$. Surface temperatures are generally affected by the flowing fluid as well as the incident flux on different parts of the receiver. Uniform surface temperature is an assumption that simplifies and generalises the present work, but it is also an approximation to the real case (Asselineau et al., 2022). The height and temperature of the plate were chosen to effectively generate turbulent natural convection on its surface, such that it is expected around an external receiver. The Rayleigh number in terms of the characteristic length of the heated surface and temperature difference is 3.17×10^{10} , which indicates a fully turbulent boundary layer flow on top of the surface (expected around 1.0 m to 1.8 m) with a possible laminar to turbulent transition flow at the lower part (around 0.6 m to 0.9 m) of the heated surface (Shavazi et al., 2018; Cheesewright, 1968).

To suppress the natural convection flow on the receiver surface, a plane air nozzle of thickness b is positioned at the top of the receiver with an inclined (α) opening that can create an effective air curtain. A gap (G) was maintained between the surface and air curtain (similar to that in Figure 2.1). The temperature of the air curtain opening is considered as same as ambient fluid temperature $T_\infty = 21^\circ\text{C}$. To create different air curtain characteristics, the jet speed (U_{ac}), thickness (b), angle (α), gap (G), and outlet temperature (T_{ac}) were changed. It is worth mentioning that the boundary of the air curtain is parametrically controlled, allowing the angle, thickness, and gap of the jet to be varied without affecting the mesh significantly (see Figure 2.2).

There was an active choice to model the simulation in 2D. The boundary layer flow on a flat, wide plate is predominantly two-dimensional. Besides, the preferred geometry of some power towers is a cylinder. An infinitesimal vertical section of a cylinder approaches a 2D flat plate; as such the physics of the two are closely related. Therefore, the present study feels justified in modelling the domain in two dimensions. Also, the assumptions of the 2D models reduce the computational cost significantly (McIntosh et al., 2014).

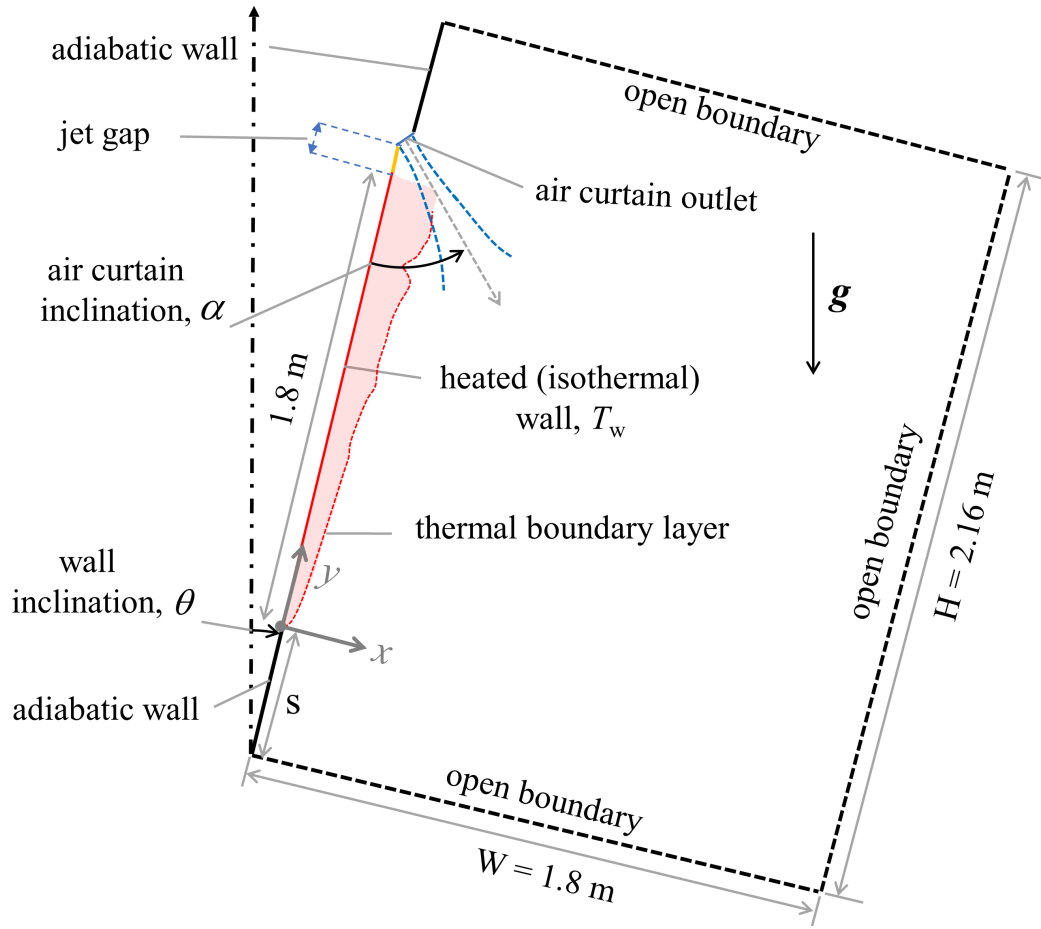


Figure 2.1: Two-dimensional geometry of the computational domain subject to θ inclination. Here, the black dash lines are open boundaries, the solid black lines are adiabatic walls, the solid yellow line is the gap between the heated wall and jet, the solid red line is an isothermal wall, and the black dash-dot line is the vertical axis.

2.1.2 Strategy of meshing the domain

The computational domain is discretized by fully structured mesh elements. An automated mesh generation process was undertaken using OpenFOAM built-in meshing tools blockMesh and snappyHexMesh, similar to that used by Torres et al. (2020). To create the stereolithography (STL) surface of the jet boundary, Gmsh (Geuzaine and Remacle, 2009), a finite element mesh generator was used. The two-dimensional mesh around the heated wall and

jet boundary is presented in Figure 2.2. A considerable effort was put into meshing the jet boundary as the jet outlet was always inclined relative to the heated wall. As a result, the cells generated along the jet outlet were not initially perpendicular to it. A layered mesh was created on the jet boundary so that the jet flow emerges via the cells that are orthogonal to the boundary, as shown in the inset of Figure 2.2.

To resolve the boundary layer on the heated wall, a high-resolution mesh was created using blockMesh, which applies geometric progression to decrease the cell size with distance from the wall (OpenFOAM, 2021). The initial mesh gradient in the vicinity of the wall was chosen to reach the y^+ value less than 5 and further refined to keep $y^+ \leq 1$. Here, y^+ is the dimensionless distance from the wall to the first mesh node and defines the viscous sub-layer where fluid is dominated by viscous effects. Studies (Versteeg and Malalasekera, 2007) have shown that more accurate results are produced for $y^+ \leq 1$.

2.1.3 Mesh refinement study

The right choice of mesh size is important to obtain a credible resolution of grid numbers inside the domain. Therefore, a mesh refinement study is necessary. Table 2.1 summarises the mesh refinement study for five different mesh sizes while $\theta = 0^\circ$, $\alpha = 45^\circ$. This refinement test was carried out for both natural convection when $U_{ac} = 0 \text{ m s}^{-1}$ and the natural convection equipped with an air curtain when $U_{ac} = 5.1 \text{ m s}^{-1}$. The mesh elements increased continuously to the whole domain and adjusted the mesh gradient to vary first cell thickness from 0.89 mm to 0.25 mm normal to the hot wall. It is clearly visible in Table 2.1 that transitioning from Mesh 1 to Mesh 3, the percentage of heat transfer difference is noticeable with a maximum of 12.70% in natural convection case and 18.33% in convection with an air curtain case. However, this difference is below 1% when transiting from Mesh 4 to Mesh 5 in both cases. Hence, Mesh 4 was deemed accurate enough for producing satisfactory results.

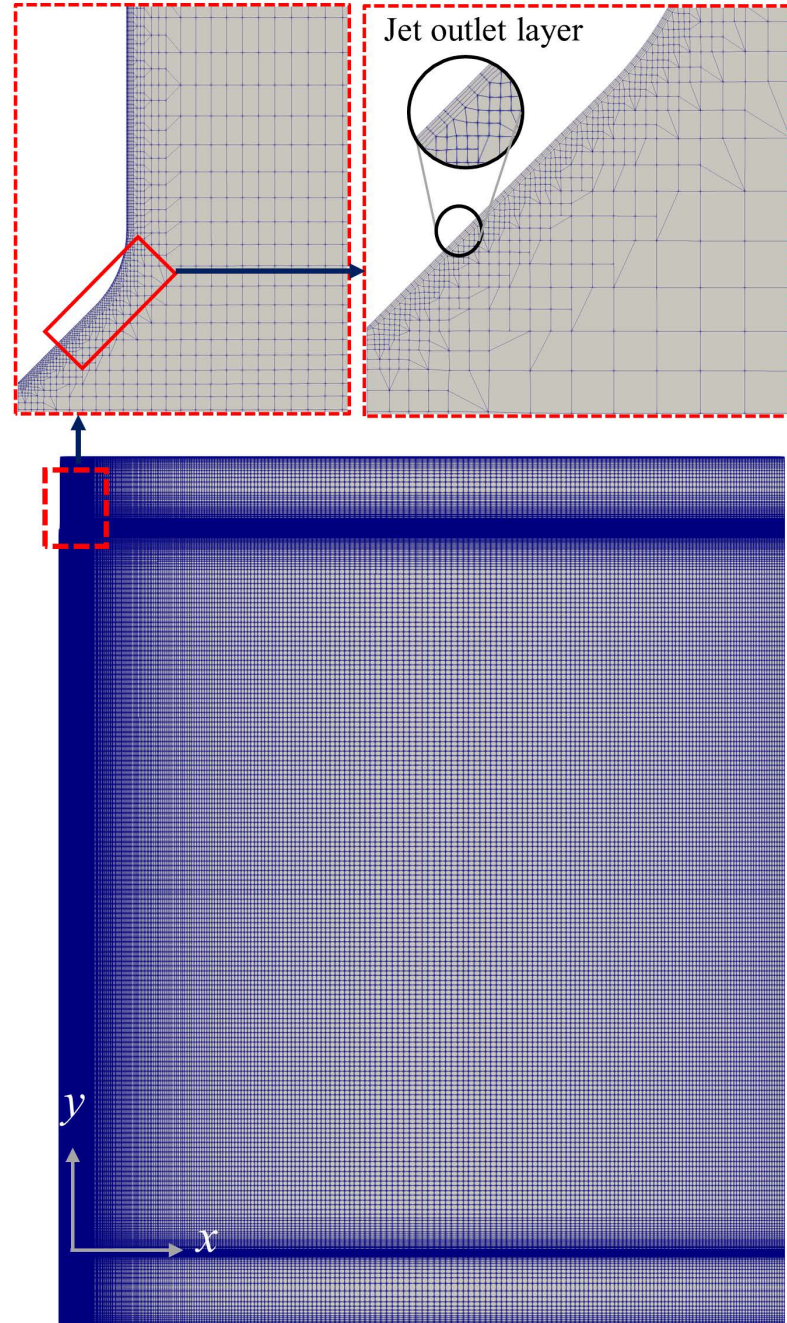


Figure 2.2: 2D mesh surrounded the domain for $\theta = 0^\circ$ and $\alpha = 45^\circ$.

Table 2.1: Mesh refinement study for the case of $\theta = 0^\circ$ and $\alpha = 45^\circ$. Here shown, for each numbered mesh, is the total number of cells N_e , first layer thickness normal to the wall δ , heat transfer rate $\dot{Q}$ for natural convection (NC, $U_{ac} = 0 \text{ m s}^{-1}$) and the wall with air curtain (AC, $U_{ac} = 5.1 \text{ m s}^{-1}$), the percentage of difference from the finest mesh Δ_c , and the non-dimensional wall distance y^+ . Conditions here correspond to $Gr = 4.55 \times 10^{10}$ and $Re = 784.62$.

Mesh	N_e	δ (mm)	$\dot{Q}$ (W)		Δ_c (%)		y^+
			NC	AC	NC	AC	
1	18443	0.89	209.13	216.09	12.70	18.33	2.09
2	37960	0.59	192.70	193.81	3.85	6.15	1.33
3	65075	0.45	188.06	186.78	1.35	2.30	1.02
4	99001	0.33	185.90	183.02	0.19	0.24	0.77
5	140199	0.25	185.55	182.58	–	–	0.65

2.1.4 Boundary conditions

Boundary conditions are different for each specific flow problem. In our case, two different flow phenomena exists: a pure natural convection along an isothermal wall and a forced convection for planar jet flow. This is why defining the appropriate boundary conditions for the domain was challenging as we had to implement the same conditions for validating both the heated wall and jet. This was necessary due to avoid the effect of boundary conditions while combining wall and jet. The separate validations of the isothermal heated wall and planar jet are shown in Chapter 3.

The boundary conditions were chosen such that it satisfies the validations of both isothermal heated wall and planar jet with available literature. The following conditions are imposed on the domain boundaries with reference to Figure 2.1, with B_a , B_h , B_j , and B_o referring to the adiabatic wall, heated wall, jet outlet and open boundaries respectively:

- Inflow conditions for velocity, temperature, turbulent kinetic energy and turbulent dissipation energy,

$$\mathbf{u} = k = \omega = 0, \quad T = T_\infty \quad \forall \mathbf{x} \in B_o \quad (2.1)$$

- Outflow conditions for velocity, temperature, turbulent kinetic energy

and turbulent dissipation energy,

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial y} = 0, \quad \frac{\partial T}{\partial x} = \frac{\partial T}{\partial y} = 0, \quad \frac{\partial k}{\partial x} = \frac{\partial k}{\partial y} = 0, \quad \frac{\partial \omega}{\partial x} = \frac{\partial \omega}{\partial y} = 0 \quad \forall \mathbf{x} \in B_o \quad (2.2)$$

- Pressure conditions on the open boundary,

$$\begin{cases} p = p_0 - \frac{1}{2}\rho|\mathbf{u}|^2 & \forall x = W \text{ or } y = -s \\ \frac{\partial p}{\partial y} = 0 & \forall y = H - s \end{cases} \quad (2.3)$$

- Adiabatic wall conditions,

$$\mathbf{u} = 0, \quad \frac{\partial T}{\partial x} = 0 \quad \forall \mathbf{x} \in B_a \quad (2.4)$$

- Heated wall condition,

$$\mathbf{u} = 0, \quad T = T_w \quad \forall \mathbf{x} \in B_h \quad (2.5)$$

where $\mathbf{u}$ is the velocity vector, u and v are the velocity components in x and y directions, k is the turbulent kinetic energy, ω is the turbulent dissipation energy, $\mathbf{x}$ is the position vector, p and p_0 are the calculated and fixed total pressure, T is the air temperature, and ρ is the density of the fluid.

Defining boundary conditions at the jet outlet B_j is also necessary as they affect the turbulent entrainment in the flow (Navaz et al., 2005). Uniform profiles for velocity, turbulent kinetic energy, and turbulent dissipation energy were used at the nozzle exit. The temperature was set to the ambient. For pressure boundary conditions, a fixedFluxPressure was employed, where the pressure is adjusted by the velocity conditions. Since the air curtain is positioned at a certain angle relative to the wall, to apply the desired momentum perpendicular to the jet inlet, the horizontal (u) and vertical (v) velocity components were calculated according to the air curtain inclination α to obtain the velocity magnitude (U_m) of the jet outlet. The magnitude of the velocity

(U_{ac}) is calculated as follows,

$$U_m = \sqrt{u^2 + v^2} \quad (2.6)$$

$$= \sqrt{(U_{ac} \sin \alpha)^2 + (U_{ac} \cos \alpha)^2} \quad (2.7)$$

$$= \sqrt{U_{ac}^2 (\sin^2 \alpha + \cos^2 \alpha)} \quad (2.8)$$

$$= U_{ac} \quad (2.9)$$

where, $u = U_{ac} \sin \alpha$ is the jet horizontal velocity component, $v = U_{ac} \cos \alpha$ is the jet vertical velocity component.

Turbulent kinetic energy k and turbulent dissipation energy ω also needed to be defined at the jet slot to introduce turbulence in the jet flow. For a fully developed jet flow, the k and ω are obtained as follows (Chen and Yuan, 2005),

$$k = \frac{3}{2}(IU_{ac})^2, \quad \omega = \frac{k^{0.5}}{C_\mu^{0.25}l}, \quad l = 0.07b \quad (2.10)$$

where U_{ac} is the speed of the jet, C_μ is the turbulent coefficient of value 0.09, l is the reference length scale of jet outlet, b is the jet thickness, and I is the turbulence intensity which was selected 1% for all these simulations. Flow in not-so-complex devices like pipes, ventilation flows; the turbulence intensity typically considers 1% to 5% (TI).¹

2.1.5 Fluid properties

Determining the appropriate fluid properties is necessary for any CFD investigations. These investigations used atmospheric air as the working fluid,

¹Note that a trial investigation was conducted for the case of a heated plate with an air curtain case (not included in the thesis) to examine the impact of turbulence intensity (at levels of 1%, 3%, and 5%) on heat transfer, considering a plate temperature of $T_w = 110^\circ\text{C}$, an ambient temperature of $T_\infty = 21^\circ\text{C}$, a jet speed of $U_{ac} = 5.1 \text{ m s}^{-1}$, a jet thickness of $b = 3 \text{ mm}$, no gap between the jet and the plate ($G = 0 \text{ mm}$), a jet inclination of $\alpha = 45^\circ$, and a plate inclination of $\theta = 0^\circ$. However, the results did not show any significant differences, which led the author to choose a turbulence intensity of 1% to avoid introducing any non-physical effects from the jet outlet.

where the fluid properties were approximated at film temperature T_f . The film temperature is calculated as the average of wall (T_w) and ambient (T_∞) temperatures. The necessary fluid properties were estimated through basic linear interpolation to derive the values at T_f , using the properties of air at atmospheric pressure, provided by Holman (2010). The fluid properties used in these simulations are summarised in Table 2.2.

Table 2.2: Calculated fluid properties at $T_f = 338.5$ K.

Fluid properties	Value
Specific heat (C_p)	$1008.241 \text{ Jkg}^{-1}\text{K}^{-1}$
Dynamic viscosity (μ)	$2.02 \times 10^{-5} \text{ kgm}^{-1}\text{s}^{-1}$
Prandtl number (Pr)	0.70
Kinematic viscosity (ν)	$1.95 \times 10^{-5} \text{ m}^2\text{s}^{-1}$
Thermal conductivity (κ)	$0.0291 \text{ Wm}^{-1}\text{K}^{-1}$
Thermal diffusivity (σ)	$2.80 \times 10^{-5} \text{ m}^2\text{s}^{-1}$

2.2 Solving with OpenFOAM

This section illustrates the solving procedures of this problem, including the overview of the solving tool, solver descriptions, and turbulence modelling.

2.2.1 Overview of OpenFOAM software

OpenFOAM is an open-source computational fluid dynamics (CFD) software developed by OpenCFD and the OpenFOAM Foundation. OpenFOAM stands for *Open Source Field Operation and Manipulation* and is a library of object-oriented software written in the C++ programming language (Weller et al., 1998). The software was initially developed in 1989 as a commercially licensed product called FOAM, but afterwards, it was released as open-source software in 2004.

OpenFOAM has a number of benefits for CFD simulations; because it is open source, and the user has access to the source code and can customize it for their own objectives. Moreover, the ability to run in parallel over large processor arrays makes it attractive for simulations on large-scale meshes. In

addition to this, it also has a large number of utilities for pre-processing and post-processing of results. In contrast, this software has some drawbacks, including limited documentation except for accessing the code, a steep learning curve for the user without prior knowledge of C++, and training costs.

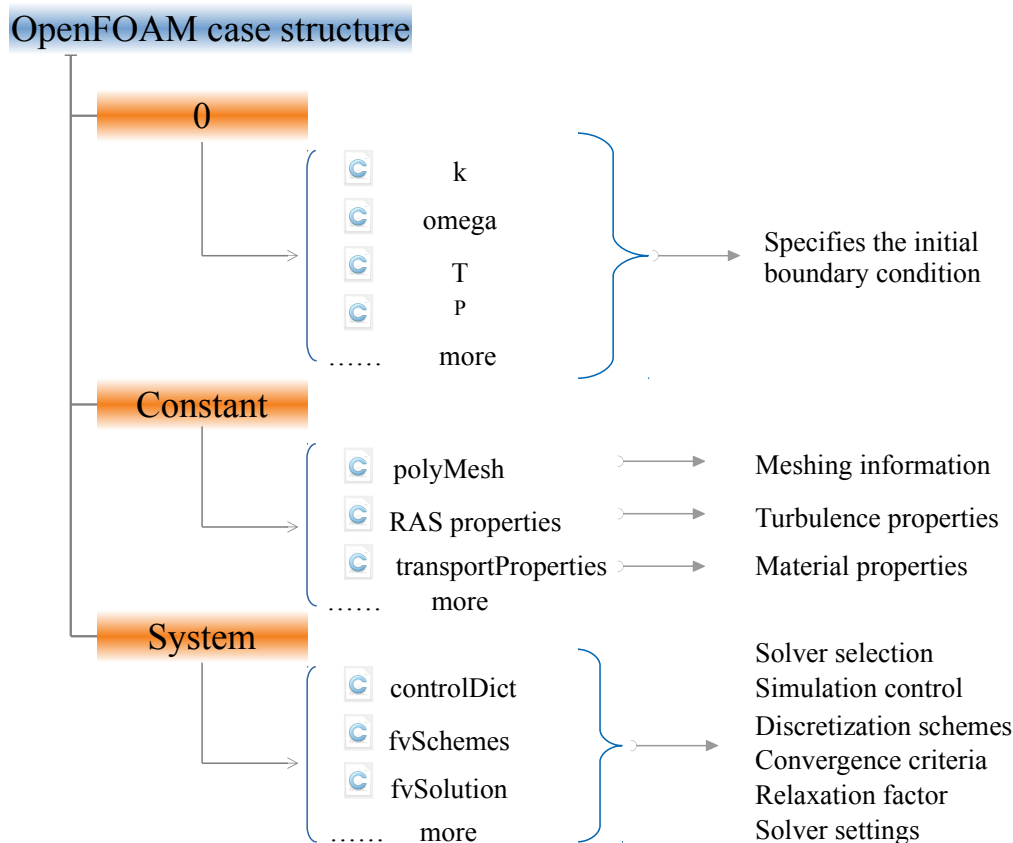


Figure 2.3: Basic OpenFOAM case structure for a typical OpenFOAM simulation.

OpenFOAM is quite different in terms of running CFD simulations than other commercial packages such as Fluent, CFX etc. Most Commercial software packages use a graphical user interface (GUI) to set up the case and data entry, whereas, in OpenFOAM, case set-up is done by storing text files in some specific folders. The basic case structure for any OpenFOAM simulations is shown in Figure 2.3.

Similar to Figure 2.3, to run an OpenFOAM simulation, there must be three different existing directories exist in the case file, namely zero, constant, and

system. The zero folders specify the initial boundary conditions for each boundary field. The constant folder accommodates the information on physical properties and the turbulence model of the system. Depending on the user requirements, different source terms can also be implemented in the governing equations by adding files inside the constant folder. Additionally, the constant directory contains a subdirectory named polyMesh, which houses all the files explaining the geometry and mesh. A mesh can be generated in a variety of methods for an OpenFOAM simulation. OpenFOAM includes two mesh-generating utilities: blockMesh and snappyHexMesh, which can create a pure structured mesh. OpenFOAM also has the ability to import mesh from other third-party software such as Fluent, Salome, Gmsh, and so on.

In the system directory, three files must be present to run any OpenFOAM simulation: the controlDict, fvSchemes, and fvSolution. Depending on the size of the simulation or the type of post-processing to be performed the system folder may also contain additional files. The controlDict file controls the overall running of the simulation such as the start and end time, time step, and information which specifies when data is to be written to an output file and the format of that data. Different post-processing statements may also be stored in the controlDict to extract data from the flow field. All the discretization schemes used for the time dependence, divergence, laplacian and gradient terms are specified in the fvSchemes file. In fvSolution, the information of the solver algorithm, convergence criteria, and relaxation factor is specified.

2.2.2 OpenFOAM solver

In OpenFOAM, the governing partial differential equations of the problem are solved using a cell-centred finite volume method (FVM). In FVM, the equations are integrated over each control volume on the mesh and the divergence term of the volume integrals is converted into surface integrals using Gauss's theorem. After that, the surface integrals are evaluated by summing the contributions from each of the cell faces. This approach requires a technique to extrapolate the stored velocity at the cell centroid to the velocity at the cell face. Multiple methods are available to perform this extrapolation in Open-

FOAM and these are described in the OpenFOAM Users Guide (OpenFOAM, 2021).

OpenFOAM package contains a large number of solvers for solving CFD simulations. BuoyantPimpleFoam solver is used for these simulations, which is described as a transient solver for buoyant, turbulent flow of compressible fluids for ventilation and heat-transfer (OpenFOAM, 2021). The algorithm of buoyantPimpleFoam solver solves the governing equations of mass continuity, conservation of momentum, and conservation of energy. Since the OpenFOAM is built based on the finite volume method, the appropriate discretization schemes of the variables need to be selected for each governing equation. The unsteady terms are discretized using the Euler method, which is a first-order, bounded implicit scheme. The gradient terms of the governing equations were discretized using the Gauss linear interpolation scheme, while the divergence terms were discretized using the Gauss upwind scheme. To discretize the Laplacian terms, the Gauss linear corrected, an unbounded, second-order was applied. The Gauss entry, which defines the standard finite volume discretization of Gaussian integration, requires interpolating values from cell centres to face centres.

The equations for mass continuity,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \quad (2.11)$$

where ρ and $\mathbf{u}$ are the density and velocity field respectively.

The equation of momentum conservation,

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = \rho \mathbf{g} - \nabla p + \nabla \cdot (\mu_{\text{eff}} D(\mathbf{u})) - \nabla \left(\frac{2}{3} \mu_{\text{eff}} D(\mathbf{u}) \right), \quad (2.12)$$

where $\mu_{\text{eff}} = \mu + \mu_t$, defines as the effective viscosity due to both molecular and turbulent viscosity, p is the static pressure, $\mathbf{g}$ is the acceleration due to gravity that is in the negative y -direction, and the rate of strain tensor $D(\mathbf{u})$ is defined as $D(\mathbf{u}) = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$.

During the implementation in OpenFOAM, the gravity force term and pres-

sure gradient are rearranged in the following form,

$$\nabla p + \rho \mathbf{g} = -\nabla(p_{rgh} + \rho \mathbf{g} \cdot \mathbf{r}) + \rho \mathbf{g} \quad (2.13)$$

$$= -\nabla p_{rgh} - \rho \mathbf{g} - (\mathbf{g} \cdot \mathbf{r}) \nabla \rho + \rho \mathbf{g} \quad (2.14)$$

$$= -\nabla p_{rgh} - (\mathbf{g} \cdot \mathbf{r}) \nabla \rho \quad (2.15)$$

Here, $\nabla \rho \mathbf{g} \cdot \mathbf{r} = -\rho \mathbf{g} - (\mathbf{g} \cdot \mathbf{r}) \nabla \rho$

And the conservation of energy,

$$\frac{\partial(\rho h)}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho \mathbf{u} h) + \nabla \cdot (\rho \mathbf{u} k) = \rho \mathbf{u} \mathbf{g} - \nabla \cdot (\mu_{\text{eff}} \nabla h), \quad (2.16)$$

where h is the enthalpy per unit mass defined as $h = e + \frac{p}{\rho}$ in terms of internal energy per unit mass e and the kinematic pressure. k is the kinetic energy per unit mass defined as $k = \frac{1}{2} \mathbf{u}^2$, and α_{eff} is the effective thermal diffusivity defined as the sum of laminar and thermal diffusivities.

$$\alpha_{\text{eff}} = \alpha + \alpha_t \quad (2.17)$$

$$= \frac{\nu_t}{Pr_t} + \frac{\kappa}{\rho C_p} \quad (2.18)$$

where κ is the thermal conductivity, C_p is the specific heat at constant pressure, μ is the dynamic viscosity, ν_t is the turbulent (kinematic) viscosity, Pr is the Prandtl number and Pr_t is the turbulent Prandtl number.

2.2.3 Turbulence modelling

To introduce the turbulence in the model, Menter's Shear Stress Transport turbulence model $k-\omega$ SST was applied. It is a two-equation eddy viscosity model that combines the widely used $k-\epsilon$ and $k-\omega$ turbulence models (Menter et al., 2003). Studies have shown that the $k-\epsilon$ and $k-\omega$ turbulence models are most accurate far and near, respectively, from the boundary (Ferziger and

Peric, 2002). Also, the k - ϵ model is not reliable for estimating the mean Nusselt number along a hot wall (Barakos et al., 1994). Choi and Kim (2012) demonstrated that k - ω SST model agrees better with the hot-wall experimental data reported by Cheesewright (1968). Thus, the overall opinion in the CFD community appears k - ω SST provides more accurate results than the k - ϵ model close to the boundaries (Menter et al., 2003).

The k - ω SST model combines the conventional k - ϵ and k - ω turbulence model with the help of a blending function. This blending function is defined in such a way that the k - ω model is applied in the near-wall region and the k - ϵ model has activated far away from the wall. In the k - ω SST model, the derivative of a dumped cross-diffusion term is added in the ω equation. There are some modifications in the turbulent shear stress and model constants are also been made in the k - ω SST model. The k - ω SST turbulence model solves two extra equations for turbulent kinetic energy k and specific turbulence dissipation ω .

The governing equation for turbulent kinetic energy,

$$\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{u} k) = \tilde{P}_k - \beta^* \rho k \omega + \nabla \cdot \left[(\mu + \mu_t \sigma_k) \nabla k \right] \quad (2.19)$$

The governing equation for specific turbulent dissipation energy,

$$\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho \mathbf{u} \omega) = \alpha \rho S^2 - \beta \rho \omega^2 + \nabla \cdot \left[(\mu + \mu_t \sigma_k) \nabla \omega \right] + 2 (1 - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \nabla k : \nabla \omega \quad (2.20)$$

Here, F_1 is the blending function and defined as,

$$F_1 = \tanh \left\{ \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500 \nu}{y^2 \omega} \right), \frac{4 \rho \sigma_{\omega 2} k}{CD_{k\omega} y^2} \right] \right\}^4 \right\} \quad (2.21)$$

where $CD_{k\omega} = \max \left(2 \rho \sigma_{\omega 2} \frac{1}{\omega} \nabla k : \nabla \omega, 10^{-10} \right)$ and y is the first cell thickness normal to the wall.

The turbulent eddy viscosity is defined as,

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, S F_2)} \quad (2.22)$$

Where S is the invariant measure of the strain rate and F_2 is a second blending function defined by:

$$F_2 = \tanh \left[\left[\max \left(\frac{2\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right) \right]^2 \right] \quad (2.23)$$

One of the drawbacks of k - ω SST model is the generation of excessive turbulence near stagnation regions. To avoid building up the excessive turbulence in this region, a production limiter is applied. The production limiter is defined as,

$$\tilde{P}_k = \min(P_k, 10 \cdot \rho \beta^* k \omega) \quad (2.24)$$

The closure coefficients of k - ω SST model are computed through a linear combination by blending the coefficients of corresponding k - ϵ and k - ω model, and is expressed as,

$$\phi = \phi_1 F_1 + \phi_2 (1 - F_2) \quad (2.25)$$

Here ϕ , ϕ_1 , and ϕ_2 are the closure coefficients of k - ω SST, k - ϵ , and k - ω models.

The computed coefficients of k - ω SST are: $\alpha_1 = \frac{5}{9}$, $\beta^* = 0.09$, $\beta_1 = \frac{3}{40}$, $\alpha_{k1} = 0.85$, $\alpha_{\omega 1} = 0.5$, $\alpha_2 = 0.44$, $\beta_2 = 0.0828$, $\alpha_{k2} = 1$, $\alpha_{\omega 2} = 0.856$.

There also exists some rigorous turbulence models that can capture a more accurate flow behaviour, for example, direct numerical simulations (DNS) and large eddy simulation (LES) (Ogasawara et al., 2021); nonetheless, the need for significant computational resources (Torres et al., 2020) led to the use of the Reynolds-Averaged Navier Stokes (RANS) approach here. The Boussinesq approximation was not adopted, as differences in density triggered by temperature differences were significant, and the air was treated as a compressible ideal gas.

2.3 Post-processing

2.3.1 ParaView

The post-processing of the simulation was performed using ParaView, an open-source, multi-platform scientific data processing and visualisation software that is capable of handling enormous data sets (Ahrens et al., 2005). ParaView is a general-purpose end-user application with a distributed design that allows for easy manipulation from any desktop or other remote parallel computing resource. Moreover, it has extensive scripting and batch-processing capabilities. The standard scripting interface uses the widely used Python programming language for scripted control. As with the graphical user interface (GUI) (Figure 2.4), the Python scripted control is easy to learn, notably including the ability to record actions in the GUI and save them out as concise human-readable Python programs and powerful, including the ability to write scripted filters that run on the server that has access to every bit of your data on a large parallel machine.

ParaView enables the interactive display of enormous data sets by utilizing parallel data processing and rendering. Additionally, it supports large screens, such as tiled displays and immersive 3D displays equipped with head tracking and pointer control. The design of the ParaView includes many conceptual features that make it stand apart from other scientific visualization solutions.

- An open-source, scalable, multi-platform visualization software.
- An user interface that is open, flexible, and easy to use.
- An extensible, modular architecture based on open standards.
- Support for distributed computation models in order to process large data sets.
- Availability of commercial support and maintenance.

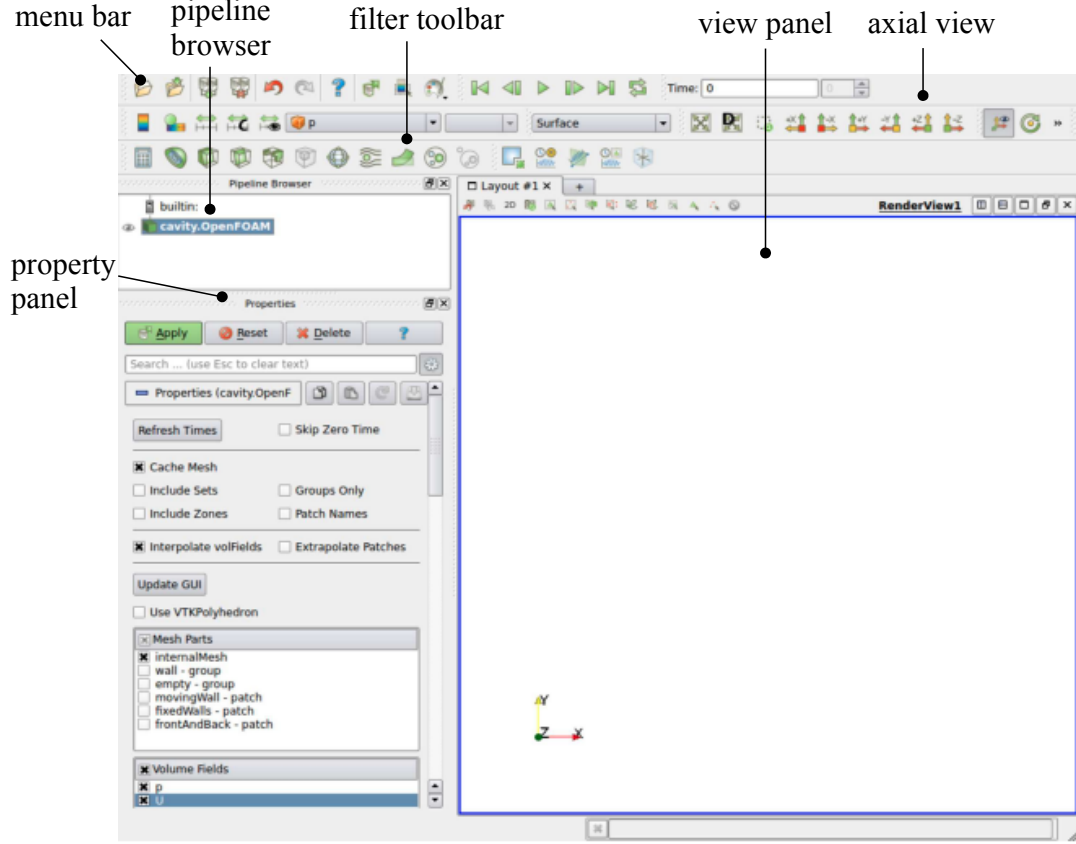


Figure 2.4: Graphical user interface of ParaView.

2.3.2 Effectiveness calculation

An objective function is defined to calculate the global (ε) and local (ε_y) effectiveness of an air curtain, i.e. its ability to minimize heat loss from an external heated surface. Both global and local effectiveness is defined as the ratio (as a percentage) of heat loss reduction (globally and locally) due to the air curtain, i.e. the difference between heat loss with and without an air curtain, relative to the benchmark case of heat loss due to natural convection only (Frank and Linden, 2014). The global effectiveness is determined using the convective heat transfer rate $\dot{Q}$ throughout the entire plate, whereas local effectiveness is determined using the convective heat flux $\dot{Q}''$ at given plate locations. Radiative heat transfer is excluded from this analysis as the receiver design necessarily requires an iterative solution to the heat transfer problem

in order to determine the surface temperatures and heat balance (Abbasi-Shavazi et al., 2020). It is suggested that the radiation part can be decoupled from the convection part, using the surface temperatures as the common link. The following mathematical equations are used to compute the effectiveness.

For global effectiveness,

$$\varepsilon = \frac{\dot{Q}_{NC} - \dot{Q}_{AC}}{\dot{Q}_{NC}} \times 100\%. \quad (2.26)$$

For local effectiveness,

$$\varepsilon_y = \frac{\dot{Q}_{NC}'' - \dot{Q}_{AC}''}{\dot{Q}_{NC}''} \times 100\%. \quad (2.27)$$

Here, the $\dot{Q}_{NC}$ and $\dot{Q}_{AC}$ are convective heat transfer rates from the entire plate for natural convection and convection with an air curtain respectively. The $\dot{Q}_{NC}''$ and $\dot{Q}_{AC}''$ are convective heat flux at given plate locations for natural convection and convection with an air curtain respectively.

This chapter presented a comprehensive analysis of the methodology used in this thesis. The analysis is performed in three stages: pre-processing, solving, and post-processing. Following this methodology, an extensive parametric study has been performed in Chapter 3 to investigate the effect of different air curtain and heated wall parameters to reduce the convective heat loss from the heated surface.

Air curtains for natural convection heat loss reduction: a parametric study¹

The aim of the present chapter is to demonstrate the effect of different air curtain and heated wall parameters to reduce convection heat losses from an external heated plate. First of all, the base models, i.e., an isothermal heated wall and a planar jet air curtain, are validated. The validated models are then implemented to conduct a parametric study to mitigate convective heat losses from the heated wall.

3.1 Validation

Validation of base case studies is necessary for any numerical simulations. In this study, two different essential components are combined: an isothermally heated wall that can successfully generate turbulent natural convection flow on its surface and a planar jet air curtain. We validated both parts separately with experimental data to be confident in our numerical model.

¹Material in this chapter has been published as: R. Mondal, J. F. Torres, G. Hughes, and J. Pye. Air curtains for reduction of natural convection heat loss from a heated plate: a numerical investigation. *Int. J. Heat Mass Transfer*, 189:122709, 2022.

3.1.1 Natural convection from an isothermal vertical plate

In the first stage, the turbulent natural convection flow on the heated surface needs to be modelled accurately so that it satisfies the physics of similar flow conditions generated around an external solar receiver surface. The simulated steady-state temperature and velocity field for turbulent natural convection flow along the heated surface has been presented in Figure 3.1. It is worth to mention that the jet was present at top of the plate during validation; however, turned off. This approach was trialed to see if the jet boundary creates any nonphysical effect on natural convection flow. As can be seen in Figure 3.1 that the jet boundary did not show any noticeable effect qualitatively or quantitatively.

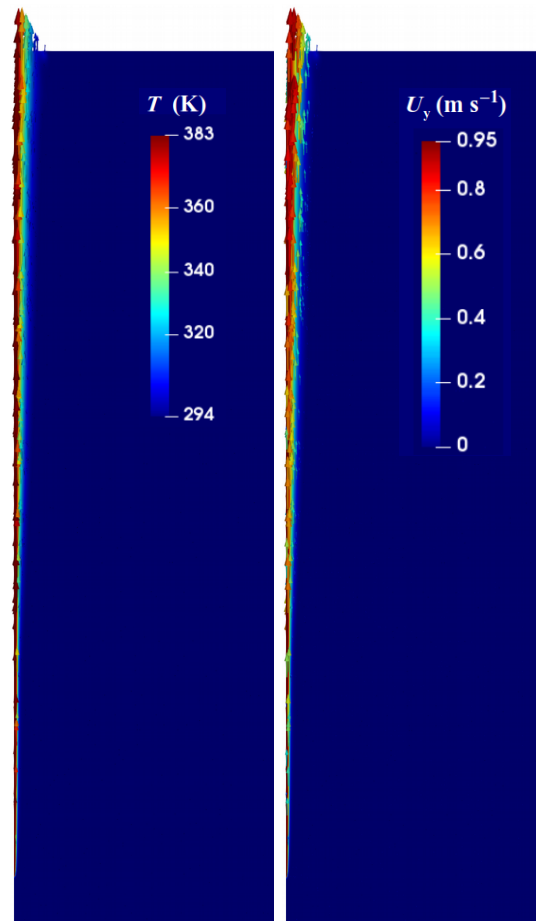


Figure 3.1: Temperature (left) and velocity (right) contour with vector plot for natural convection flow along an isothermal plate when, $T_w = 110^\circ\text{C}$, $\theta = 0^\circ$, $\alpha = 45^\circ$, $b = 3\text{ mm}$, and $G = 0\text{ mm}$.

A quantitative comparison of simulated dimensionless laminar velocity u^* , temperature T^* , and local Nusselt number Nu_y profile is also made with experimental data obtained by Tsuji and Nagano (1988). The u^* and T^* are computed as similar to that defined by Tsuji and Nagano (1988) (see in nomenclature). It should be noted that the isothermal flat plate was oriented vertically, and the jet outlet was turned off while validating the natural convection case. Figure 3.2 compares the non-dimensional laminar velocity and temperature profile for Grashof number, $Gr = 1.95 \times 10^8$, which matches closely with the experimental data, except for a slight difference in velocity profile away from the boundary layer. This discrepancy is likely to be because of the differences in the boundary conditions between the experimental setup and the numerical model. The heated wall of the experimental setup of Tsuji and Nagano (1988) was in a closed box. However, in our simulation, the boundaries are kept open, which allows the stream-wise velocity to develop. Therefore, a non-zero velocity is observed away from the heated wall. Apart from that, the temperature and velocity profiles matched well for $x^* \leq 5$; thus, the heat transfer obtained in our modelling agrees well with the experimental observations.

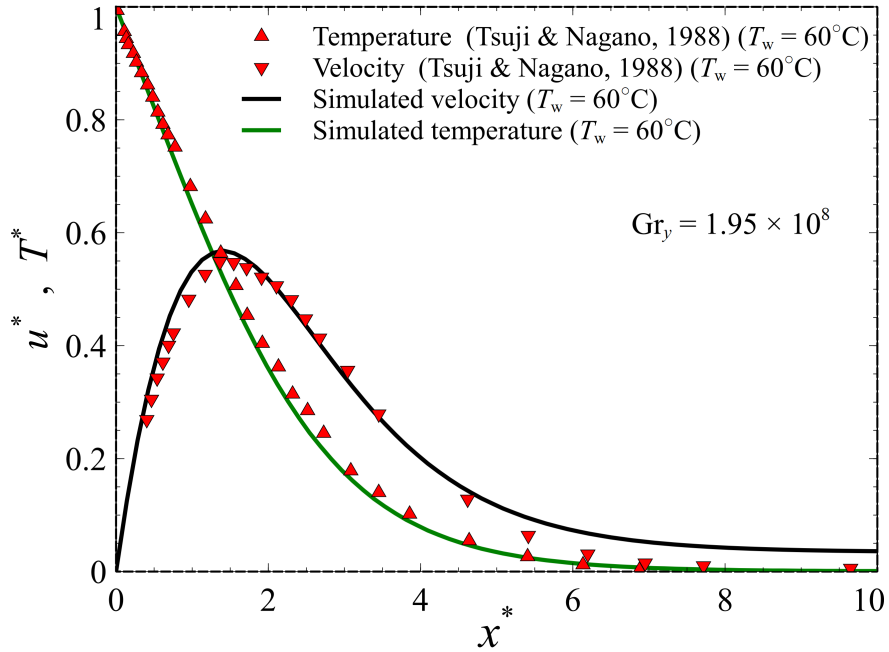


Figure 3.2: Dimensionless laminar velocity and temperature profile for an isothermal vertical wall.

Figure 3.3 shows the local Nusselt number Nu_y profile as a function of local Rayleigh number Ra_y together with an inset plot which shows the difference in percentage between simulated and experimental data. The comparison was made for the local Nusselt number, but average Nusselt numbers were also calculated for both wall temperatures. The average Nusselt numbers for wall temperatures of 100°C and 110°C are 126.02 and 129.20, respectively. The comparison between simulated and experimental results showed a very good agreement over most of the plates, particularly in laminar and fully turbulent regions, with an error of within 3%. Nevertheless, the transition region experienced a maximum difference of 16.60%. This difference might have occurred because various experimental aspects affect the onset of turbulence, such as small irregularities in the surface or ambient flows, which may have existed in the experiment, but certainly, no perturbations were introduced in the simulations (Zhao et al., 2019). Therefore, we would expect that the transition to turbulence is not expected to be the same in simulations and experiments. As noted in Section 2.2.3, higher-accuracy LES or DNS simulations could have been used to achieve improved resolution of the transition flow, but these methods are highly computationally expensive and would have reduced our ability to show the effects of varying air curtain and heated wall parameters for minimizing convective heat loss. Thus, RANS was considered to be the ideal choice for this study. We also examined the temporal variations in heat transfer from the heated plate, since transient simulations had been performed. The heat transfer rate reached a quasi-steady state (see Figure 3.7 and 3.8), indicating that the transient variations of average and local Nusselt number are very small, and would unlikely affect the conclusions.

Here, the Rayleigh number along the heated plate is defined as,

$$Ra_y = GrPr = \frac{(g \cos \theta) \beta \Delta T y^3}{\nu^2} \times \frac{\nu}{\sigma} = \frac{(g \cos \theta) \beta \Delta T y^3}{\nu \sigma}, \quad (3.1)$$

where Gr is the Grashof number, Pr is the Prandtl number, g is the gravitational acceleration, β is the thermal expansion coefficient, ν is the kinematic viscosity, σ is the thermal diffusivity, ΔT is the temperature difference between heated wall and ambient, y is the distance from leading edge of heated plate.

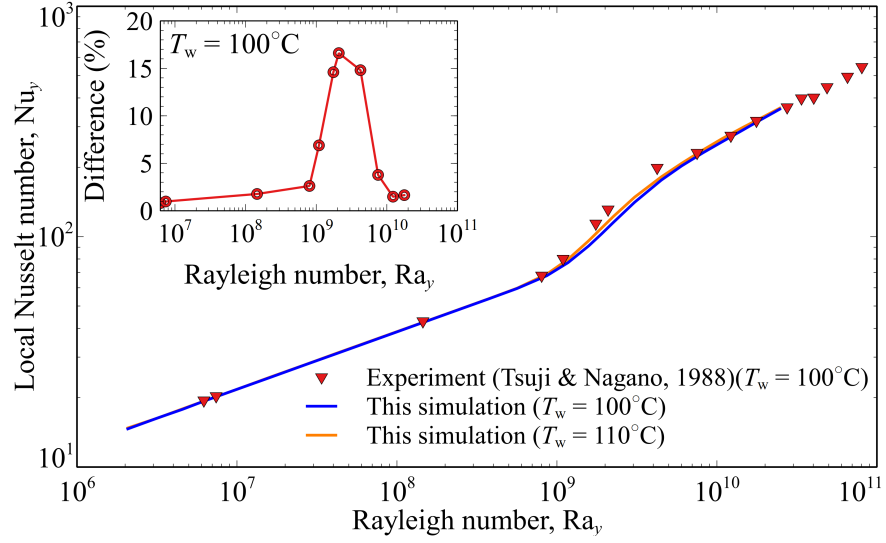


Figure 3.3: Local Nusselt number as a function of local Rayleigh number for isothermal vertical wall; the inset shows the difference between modelling results and the experimental results of Tsuji and Nagano (1988).

The local Nusselt number is defined as follows,

$$\text{Nu}_y = \frac{y\dot{Q}''}{\kappa\Delta T}. \quad (3.2)$$

Here, $\dot{Q}''$ is the heat flux per unit area of the wall, and κ is the thermal conductivity of air.

The thermophysical properties in equations 3.1 and 3.2 were determined in a way consistent with how Tsuji and Nagano (1988) reported their results, where all the fluid properties were calculated at film temperature T_f . Table 2.2 summarises the fluid properties used for this study. The film temperature is defined as the average of the wall T_w and ambient T_∞ temperatures. Thermal expansion coefficient β was measured at the ambient temperature T_∞ .

3.1.2 Isothermal planar jet

The other validation study corresponds to the planar jet. It is required because the jet was always introduced obliquely relative to the wall and the faces of the control volumes inside the domain (far from the layered mesh shown in the inset of Figure 2.2). As a result, the mesh could have an impact on the jet

flow. To validate, a planar jet was set at a 45° inclination with a uniform inlet velocity U_{ac} and mounted at the top of the heated wall. The heated wall was replaced by an open boundary for the purpose of jet validation. Importantly, we used the same mesh for both validation cases, with a heated wall and a planar jet, so that the mesh is likely to be suitable for the combined simulation of the wall and jet.

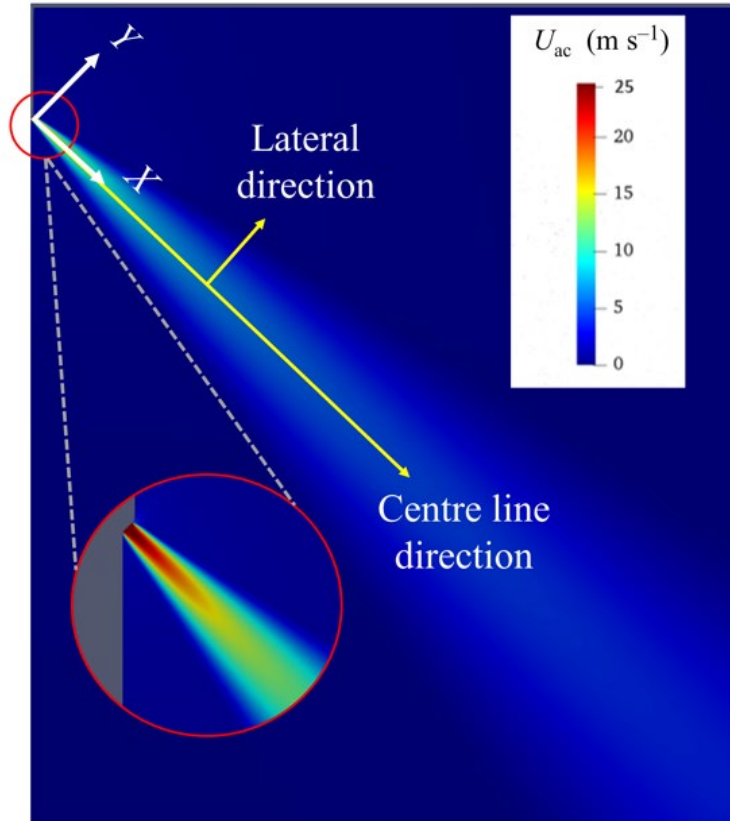


Figure 3.4: Velocity contour of turbulent planar jet and associated coordinates system for $Re = 7000$.

The flow visualisation of the velocity magnitude of an obliquely introduced planar jet is presented in Figure 3.4. The velocity profiles in both centre line and lateral directions were compared with available literature. In Figure 3.5, the simulated jet centre line velocity U_c is plotted against the experimental data produced by Deo et al. (2007). The result showed a good agreement for Reynolds number $Re = 7000$. The lateral velocity distributions of the jet have been plotted in Figure 3.6 for $Re = 4000$ together with an LES simulation

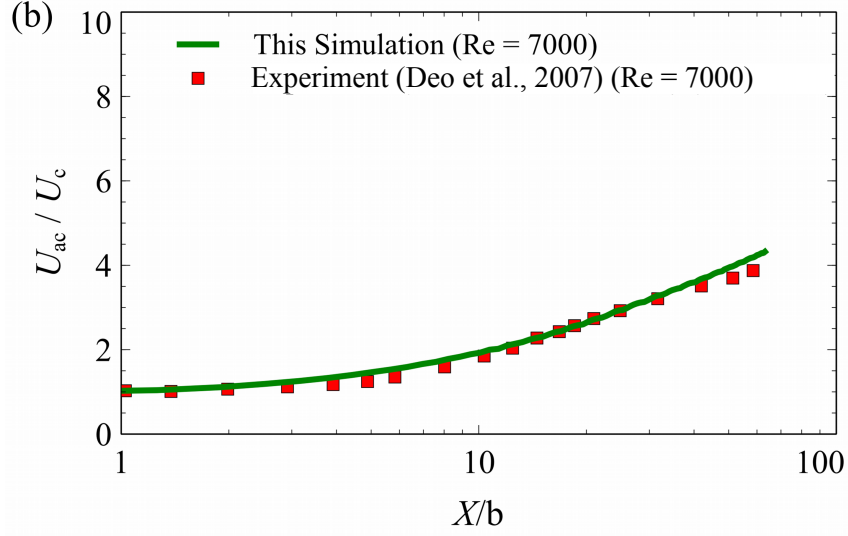


Figure 3.5: Centre line velocity decay of turbulent planar jet for $Re = 7000$ when positioned at $\alpha = 45^\circ$.

result, conducted by Bisoi et al. (2017). It is worth noting that the jet half-width $Y_{1/2}$ is measured along Y coordinate direction to a point at which $U = \frac{1}{2}U_c$ in this figure. It is seen that the lateral velocity profiles also matched well with the literature, except for a slight discrepancy in the far field. This discrepancy has happened because the jet gets close to the open boundaries, and the boundary could have an effect on the lateral velocity distributions of the jet. However, this validation approach is deemed acceptable because we used the same boundary conditions for validating both cases; hence, the boundary condition should not affect the result further while combining wall and jet.

Here, the Reynolds number is defined as,

$$Re = \frac{U_{ac}b}{\nu}, \quad (3.3)$$

where b is the jet outlet thickness.

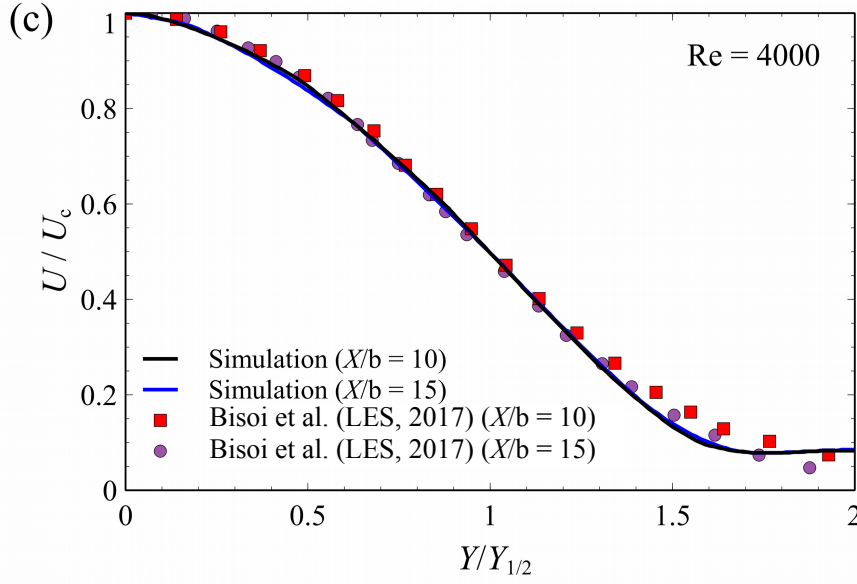


Figure 3.6: Lateral velocity distribution of jet for $Re = 4000$ when positioned at $\alpha = 45^\circ$.

3.2 Results and discussion

3.2.1 Transient analysis

Firstly, the heat transfer rate time series for pure natural convection and convection with air curtain cases were checked, demonstrating that a thermal quasi-steady state is reached (in the transition zone, periodic fluctuations are expected (Zhao et al., 2017), so the plate heat transfer rate is also expected to fluctuate slightly). Since the solver used in this work is transient, we considered that our simulations reached a quasi steady-state condition, i.e. there is a minimal quantitative change in the heat transfer rate with time.

For natural convection in Figure 3.7, the heat transfer rate starts with an extremely high value as the initial temperature difference between the working fluid against the wall was maximum (89°C), i.e. the local temperature gradient at the wall-fluid boundary at $t = 0$ is infinite. The heat transfer rate then remains almost unchanged approximately from 5 s, indicating the boundary layer reached fully quasi steady-state conditions. In Figure 3.8, the time series of the heat transfer rate through the entire plate is shown when the natural convection boundary layer along the plate interacts with a 45° tilted air cur-

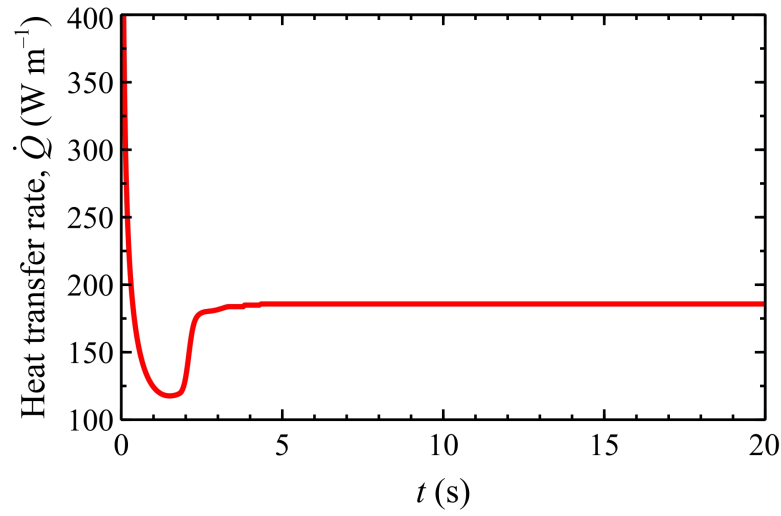


Figure 3.7: Transition to steady-state heat transfer rate for natural convection case when $T_w = 110^\circ\text{C}$ and $\theta = 0^\circ$.

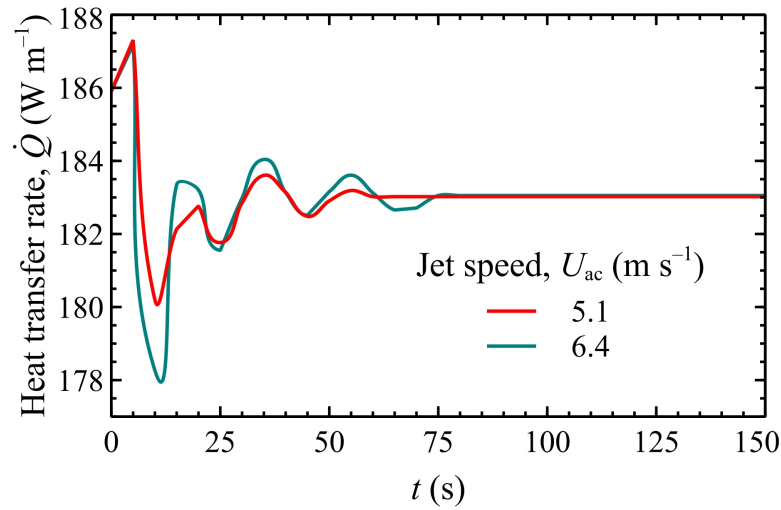


Figure 3.8: Transition to steady-state heat transfer rate for natural convection with air curtain when $T_w = 110^\circ\text{C}$, $\theta = 0^\circ$, and $\alpha = 45^\circ$.

tain with jet speeds 5.1 and 6.4 m s^{-1} respectively. Note that the jet outlet turned on at $t = 20\text{ s}$ when the natural convection simulation finished. As can be seen in this figure, the heat flux starts fluctuating after introducing the jet and reaches fully quasi steady-state conditions after approximately 75 s .

3.2.2 Effect of jet speed on wall heat flux

The local heat flux and global effectiveness of the entire plate for a 45° inclined jet at varying jet speeds are discussed in this section. This jet inclination was chosen consistent with Mondal et al. (2020), because greater heat loss was observed there at $\alpha = 45^\circ$ compared to $\alpha < 45^\circ$. The local heat flux for different jet speeds is presented in Figure 3.9. The jet velocities are varied from 1.2 m s^{-1} to 6.4 m s^{-1} in increments of 1.3 m s^{-1} . The results demonstrate that jet speeds larger or equal to 1.2 m s^{-1} reduced the convective heat losses from the wall in two regions between 0.5 m and 1.8 m . The largest reduction of convective heat loss appears in the fully turbulent boundary layer (between 1.7 m to 1.8 m), and a significant reduction occurs in the transition regions (between 0.6 m to 1.1 m) of the wall.

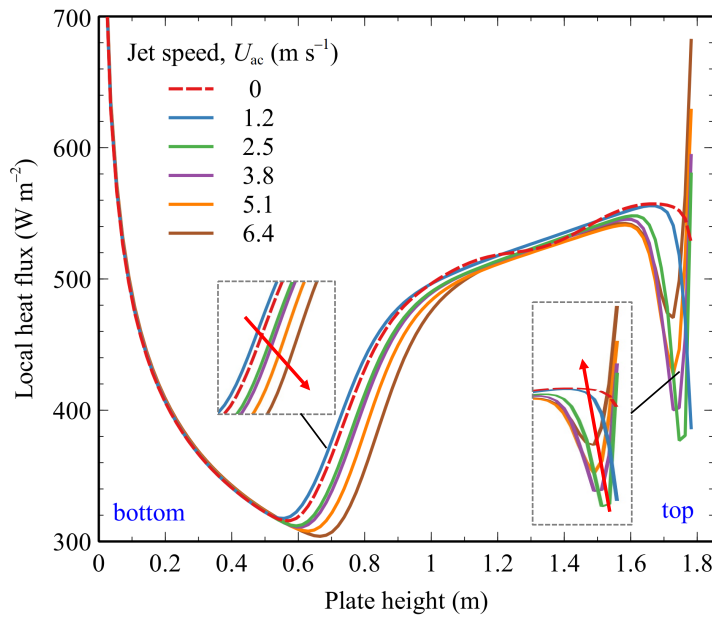


Figure 3.9: Local heat flux of a vertically oriented heated wall ($\theta = 0^\circ$) for different jet speeds when $\alpha = 45^\circ$.

A further attempt was made to find maximum global effectiveness for various jet speeds, with additional data points added in the vicinity of the optimum effectiveness. Figure 3.10 shows the global effectiveness of the air curtain at various jet speeds. The highest effectiveness of $1.79 \pm 0.01\%$ is observed for a jet speed of 4.9 m s^{-1} to 5.1 m s^{-1} . The jet speed of 5.1 m s^{-1} is chosen for

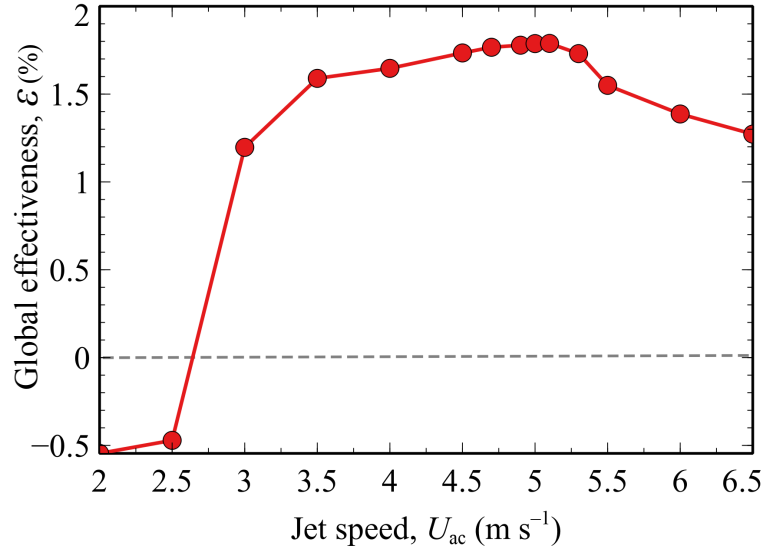


Figure 3.10: Global effectiveness for different jet speeds when $\theta = 0^\circ$ and $\alpha = 45^\circ$.

further investigations.

To understand what is causing variations in global heat loss, we return now to the discussion of local effects. Figure 3.11 to 3.12 discuss the local effectiveness of the air curtain over the entire plate is plotted for different jet speeds. This figure clearly indicates that the air curtain successfully creates a stagnation zone at the top of the wall as expected and produces the largest positive effectiveness at wall heights between 1.6 m to 1.8 m. A temperature contour with the velocity vector in Figure 3.11 demonstrates how the air curtain generates a stagnation zone near the top of the wall. As shown in Figure 3.12, when the air curtain had an exit jet speed of 1.2 m s^{-1} , a reduction of convective heat loss was only noticed in the top section of the wall. In contrast, below 1.5 m, this jet speed showed negative effectiveness, i.e. increased local heat flux. However, when the jet speed ranged from 2.5 m s^{-1} to 6.4 m s^{-1} , the results showed a systematic reduction in convective heat losses between approximately 0.5 m and 1.78 m. A maximum local effectiveness of 31.72% was achieved within this region with a jet speed of 2.5 m s^{-1} . In the uppermost portion of the wall, a very confined region of negative effectiveness was induced for all jet speeds. This is because, for a strong jet, the airflow at the top of the wall was completely reversed, and instead of suppressed natural

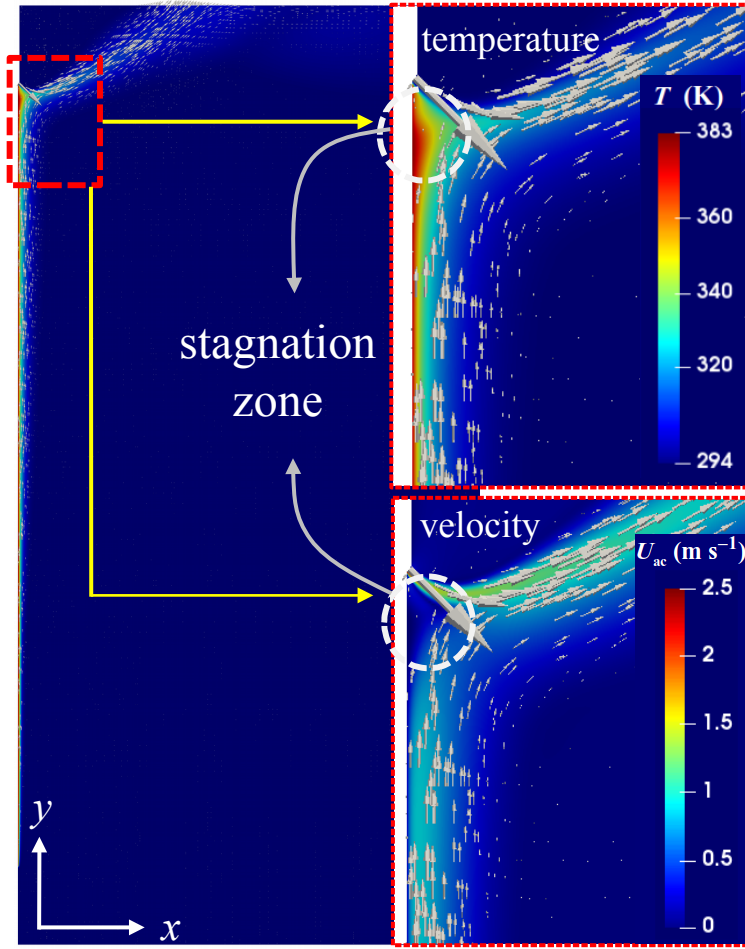


Figure 3.11: Temperature profile with velocity vector for $\alpha = 45^\circ$, $U_{ac} = 2.5 \text{ m s}^{-1}$.

convection, strong forced convection increases the heat transfer locally. A similar negative effect was observed for $y < 0.5 \text{ m}$, where the laminar region is expected to exist. The laminar region, exhibiting relatively low convective heat losses when the air curtain is absent, is probably also disturbed when the air curtain is added. An interesting result is noticed in the transition region where the highest local effectiveness of 18.88% is achieved by the maximum jet speed of 6.4 m s^{-1} . The evident trend in this region is that the effectiveness increases significantly with increased jet speed.

The air curtain showed two peaks in effectiveness in Figure 3.12, which are plotted in Figure 3.13. At wall location 1.75 m where the stagnation zone exists, the local effectiveness significantly increased with jet speed up to

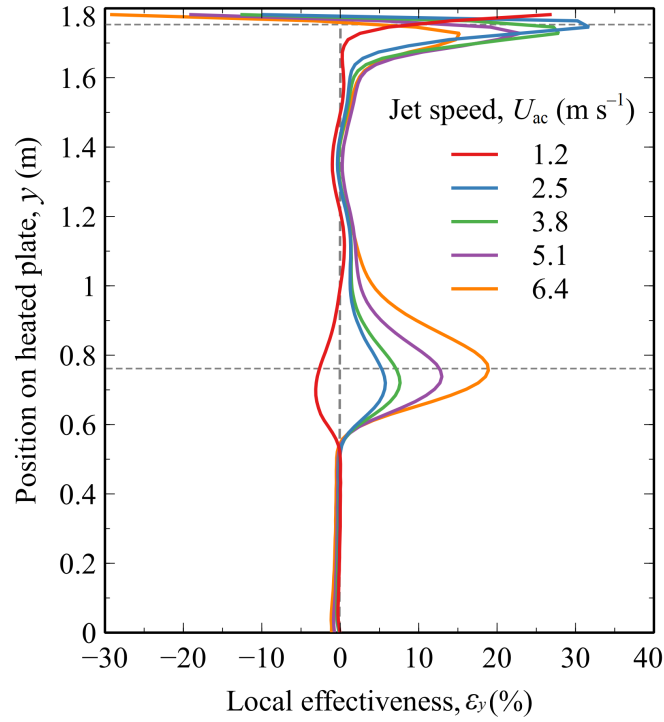


Figure 3.12: Local effectiveness ε_y of a vertical heated wall for different jet speeds when $\alpha = 45^\circ$.

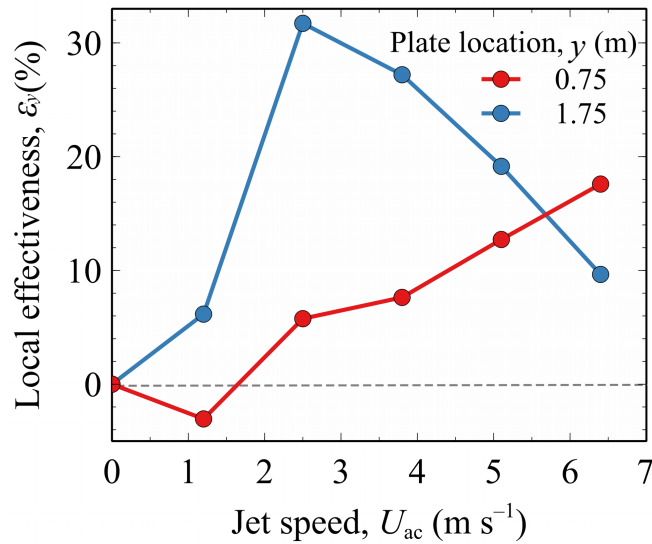


Figure 3.13: Local effectiveness at $y = 0.75$ m and 1.75 m for different jet speeds when $\alpha = 45^\circ$, indicated by grey dashed line in Figure 3.12.

2.5 m s^{-1} , reaching a peak value of 31.2% . Nevertheless, the effectiveness gradually shifted downwards with the increase in jet speed. We found this

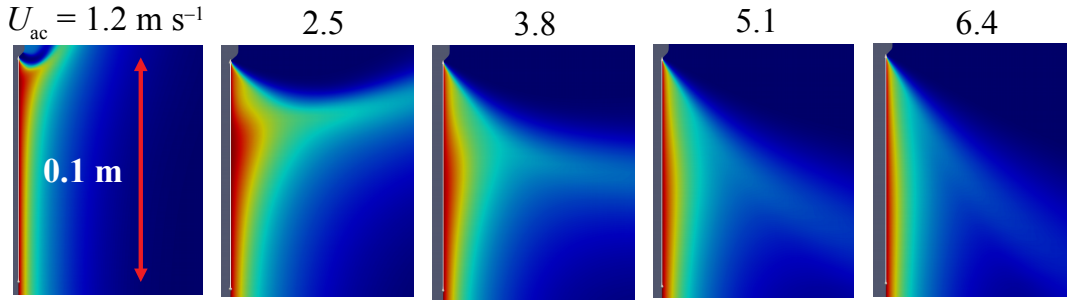


Figure 3.14: Temperature profile of the top 0.1 m of the heated wall, for different jet speeds when $\alpha = 45^\circ$.

has happened because, once the stagnation zone is established, a continuous rise in jet speed gradually disturbs the stagnation zone and reduces effectiveness. This effect can also be observed in the flow visualisation presented in Figure 3.14, where the top 0.1 m of the heated wall is shown for different jet speeds. It is clearly noticeable that the gradual increase in jet speed reduces the stagnation zone slowly. Interestingly, at wall location 0.75 m where the transition of flow occurs from laminar to turbulent, the effectiveness yields an opposite trend. Initially, the effectiveness is slightly negative for a jet speed of 1.2 m s^{-1} , but the effectiveness rises systematically with increasing jet speeds. One possible explanation for this phenomenon is that a high-speed jet appears to be more successful in counteracting the buoyant forces of the convective flow and delaying the transition from laminar to turbulent flow. This effect is well illustrated in Figure 3.9, which demonstrates a gradual reduction in heat flux as the jet velocity increases in the region of laminar to turbulent transition. Consequently, this leads to decreased heat losses and increased effectiveness.

3.2.3 Effect of jet angle on effectiveness

The effect of jet inclination on effectiveness was investigated for $\alpha = 30^\circ$, 35° , 40° and 45° . The jet exit speed was fixed at 5.1 m s^{-1} and the heated wall was vertical ($\theta = 0^\circ$).

The results of this analysis are summarised in Figure 3.15 and 3.16. As seen in Figure 3.15, the 35° , 40° and 45° inclined jet showed a very similar effec-

tiveness trend. However, by carefully examining the data, it is found that a 45° tilted jet shows comparatively better performance than a 35° and 40° inclined jet. Large positive effectiveness regions are present in the stagnation and transition zones in all cases.

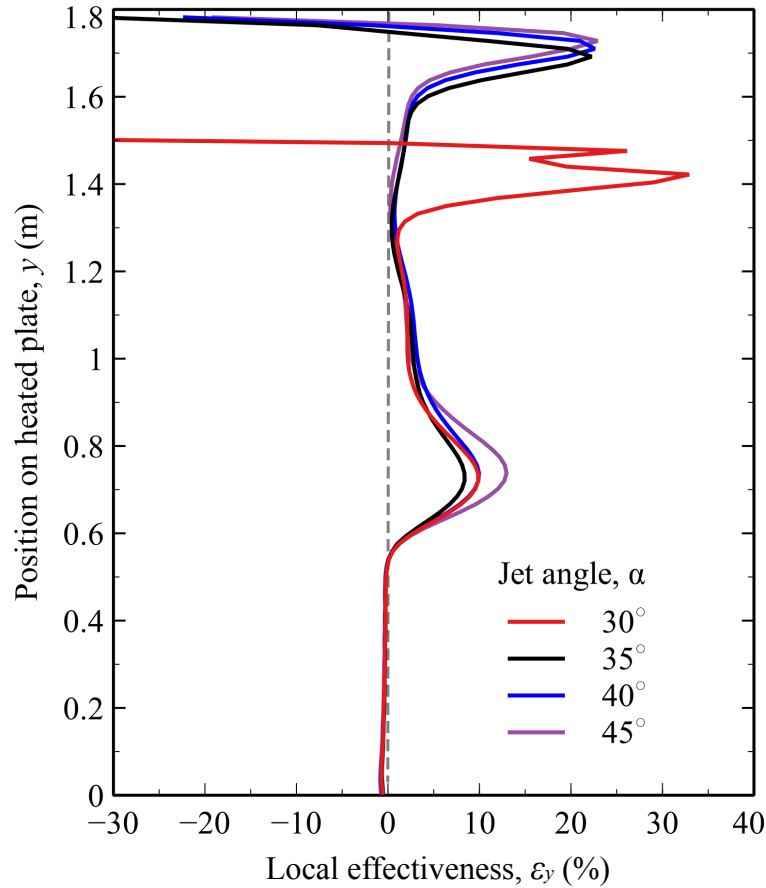


Figure 3.15: Local effectiveness for different jet angles when $U_{ac} = 5.1 \text{ m s}^{-1}$.

A dramatic departure from the behaviour explained above is apparent for a jet inclination $\alpha = 30^\circ$. The stagnation zone is significantly displaced down the heated wall. This occurs due to the attachment of the jet to the heated wall, which is shown in more detail in Figure 3.16. The jet curves toward the heated wall and starts to force the buoyant plume down the wall, leading to a wall-attached jet. This flow condition is known as the Coandă effect (Miozzi et al., 2010). However, where the jet momentum flux and buoyancy forces balance, a stagnation zone is generated. Despite having the highest local effectiveness for this jet inclination, this configuration might not be useful in

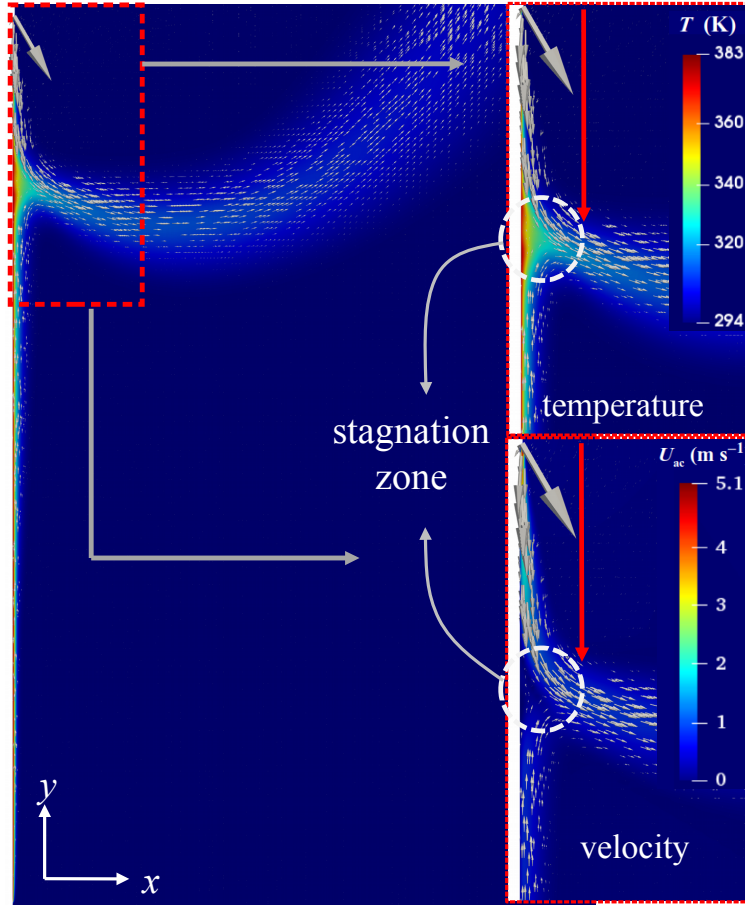


Figure 3.16: Temperature contour with velocity vector for a 30° tilted air curtain with jet speed of 5.1 m s^{-1} .

CSP systems due to the large negative effectiveness near the jet outlet. The jet did not attach to the wall when the inclination angle α between the jet and heated wall was greater or equal to 35° .

3.2.4 Effect of wall inclination on effectiveness

Here we determine the impact of the heated wall inclinations on effectiveness, while keeping the jet angle and velocity constant at $\alpha = 45^\circ$ and $U_{ac} = 5.1 \text{ m s}^{-1}$. This was done by comparing the effectiveness of a heated wall inclined at $\theta = 0^\circ$, 30° , and 45° respectively.

Inclining the heated wall can have a local negative impact, as shown in Figure 3.17, due to the large increase of convective heat losses relative to the

vertical wall at discrete locations, thus giving negative effectiveness. This is particularly true where the transition zone (0.6 m to 1.0 m) is observed. The effectiveness shows an opposite trend compared to the vertical wall in this region and increases negative effectiveness with increasing wall inclination. However, from a wall height of 1 m to just below 1.8 m, the air curtain still shows positive effects for each case, and the peak effectiveness appears close to 1.7 m.

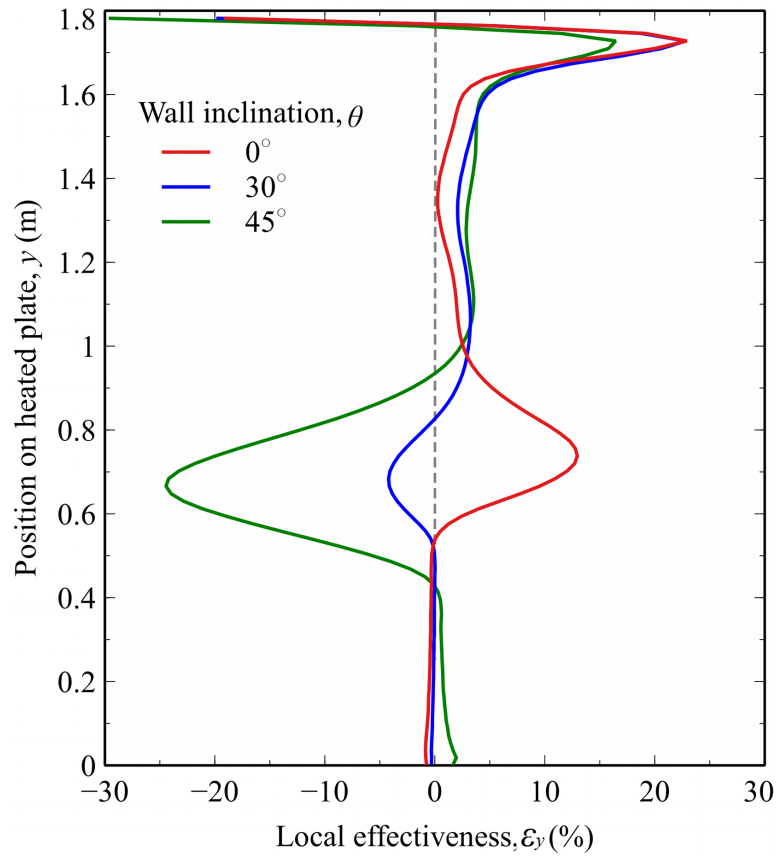


Figure 3.17: Local effectiveness for different wall inclinations when $\alpha = 45^\circ$ and $U_{ac} = 5.1 \text{ m s}^{-1}$.

These were interesting results because the inclined wall and air curtain were expected to perform well due to a reduction in the vertical momentum that might have increased the stagnation region. To understand the poor performance of the inclined wall and air curtain, Figure 3.18 compares the local effectiveness of an inclined heated wall (relative to a vertical wall) for natural convection cases. We can see that the 30° and 45° inclined wall offers a

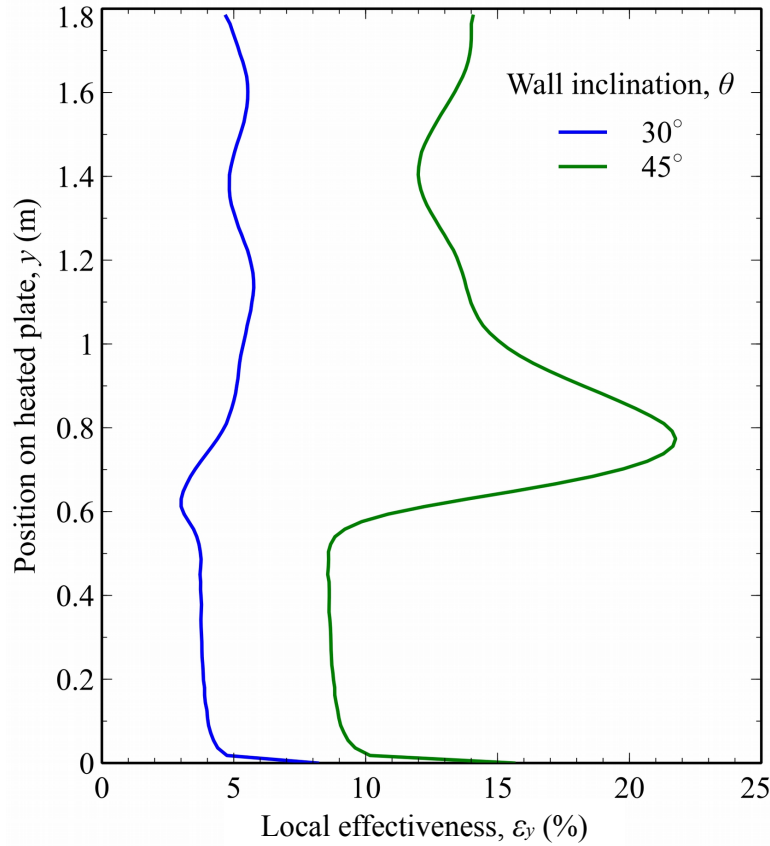


Figure 3.18: Local effectiveness (relative to a vertical heated wall) for different wall inclinations when $U_{ac} = 0 \text{ m s}^{-1}$.

useful performance gain over the length of the heated plate. With an inclination, the flow experiences a weaker buoyant force along the wall, allowing the thermal boundary layer to remain laminar and reducing the convection coefficient. Therefore, it is more difficult for an air curtain to further decrease an already low convection coefficient and give positive effectiveness. Instead, the sensitive buoyant flow is easily disrupted, resulting in adverse effects, i.e. increasing heat loss. This finding can be compared to similar computational studies on cavities receiver air curtains by McIntosh et al. (2014) and Zhang et al. (2015). Both studies found that a horizontal cavity receiver, i.e. with a vertical aperture, had the greatest effectiveness of any inclination. This is due to the high convective heat losses of the cavity at the horizontal without an air curtain. When the air curtain is then applied, the losses are reduced significantly relative to that without the air curtain. In contrast, when the air

curtain is applied with a downward inclined cavity that already has a relatively small convective heat loss, the magnitude of reduction is far less, giving less effectiveness overall. This evaluation is, however, performed for a single jet speed and angle. Further investigations are required to find the actual heat loss reduction by varying jet angles and speeds because of different buoyancy among the walls.

3.2.5 Effect of wall temperature on effectiveness

The heated wall temperatures were investigated for $T_w = 70^\circ\text{C}, 90^\circ\text{C}$, and 110°C while keeping a constant ambient temperature of 21°C . These wall temperatures were chosen in relation to the wall height to effectively generate turbulent natural convection flow on the heated surface, which corresponds to the Grashof number ranges from 3.08×10^{10} to 4.55×10^{10} , characterising the turbulent natural convection. The wall and jet inclination was fixed at $\theta = 0^\circ$ and $\alpha = 45^\circ$, respectively with an exit jet speed of 5.1 m s^{-1} .

Figure 3.19 describes the local effectiveness of an air curtain with jet speed 5.1 m s^{-1} for three wall temperatures. It can be seen that the air curtain was able to produce a similar effectiveness trend for all wall temperatures as that noticed in Figure 3.12, including two peak effectiveness regions in the stagnation and transition zones. In the turbulent region where the stagnation zone exists near the jet outlet (approximately at 1.7 m), the air curtain produced the maximum effectiveness for the highest wall temperature 110°C . The thermal boundary layer remains less disturbed (as noticed in Figure 3.14 at this plate location and increases the stagnation zone by the jet speed of 5.1 m s^{-1} for higher wall temperatures. This suggests that the momentum of the jet and buoyant forces generated by wall temperature 110°C balanced well in this plate location compared to the other wall temperatures. On the contrary, in the intermediate section of the wall, particularly between 0.6 m to 1.6 m where the transition occurs, the air curtain showed better overall performance for lower wall temperatures 70°C and 90°C . This has likely to happen because the buoyancy is comparatively less strong for lower wall temperatures than for higher wall temperatures. It appears that the air curtain can more effectively oppose the buoyant flow for lower wall temperatures at the bottom of

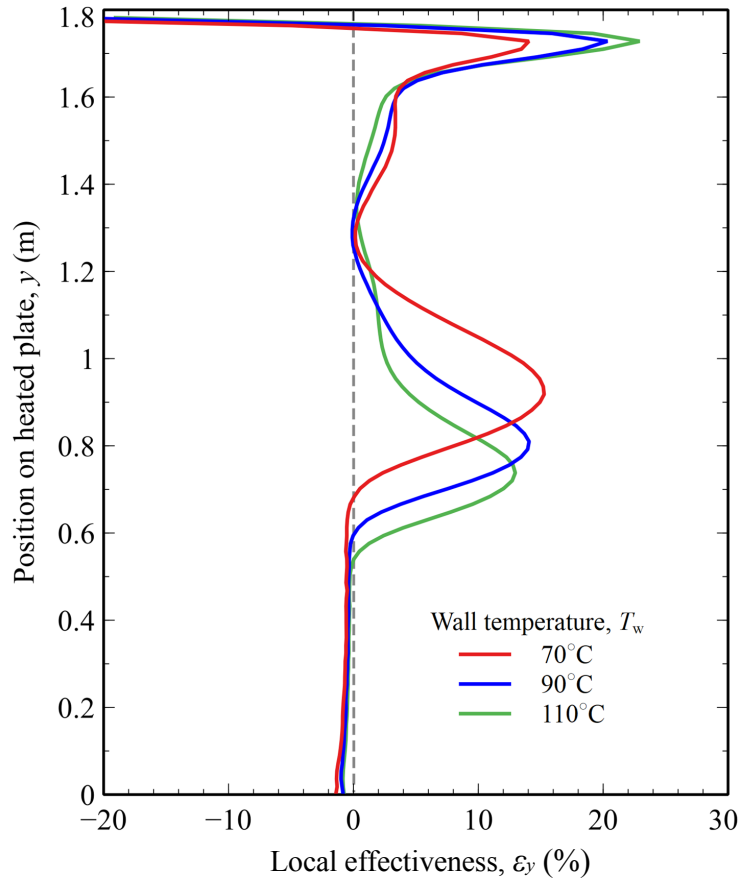


Figure 3.19: Local effectiveness (ϵ) for different wall temperatures when $\alpha = 45^\circ$ and $U_{ac} = 5.1 \text{ m s}^{-1}$.

the wall. Opposing the buoyant flow helps delay the turbulent transition and increase boundary layer thickness, reducing heat loss and increasing effectiveness. Similar to Section 3.2.4, because of the buoyancy differences in the walls for different temperatures, further investigations of the jet angles and speeds are also required to determine the actual effect of wall temperatures on effectiveness.

3.3 Conclusions

This study has examined the impact of an obliquely oriented planar jet (air curtain) to mitigate the convective heat losses from an isothermal wall. Numerical simulations confirmed that convective heat losses from a 1.8 m heated

plate could be minimised locally using an air curtain configuration.

The key observations that are made from the simulation results are indicated below:

1. The air curtain achieved peak effectiveness of 31.2% for $T_w = 110^\circ\text{C}$ at the stagnation zone near the maximum length and jet outlet in the heated section while $\alpha = 45^\circ$ and $U_{ac} = 2.5 \text{ m s}^{-1}$. For increasing jet speed, the air curtain effectiveness decreased in the stagnation zone, while the effectiveness in the transition zone increased.
2. An air curtain with 45° inclination yielded higher effectiveness than $\alpha < 45^\circ$.
3. Convection losses are lowered as the wall is inclined from the vertical without a jet. However, the addition of introducing a 45° inclined jet having a fixed speed of 5.1 m s^{-1} was observed to have a relatively small further benefit over much of the wall but can be detrimental overall by modifying the transition zone.
4. For the jet inclinations $\alpha = 45^\circ$, higher wall temperatures showed greater efficiency at the top of the wall, especially in the stagnation region, whereas lower wall temperatures produced greater efficiency in the middle of the wall.

In this research, an air jet was examined, which had a fixed thickness and position with respect to the heated wall. Jet thickness and position could be additional parameters further to reduce convective heat losses from the external heated surfaces. The following Chapter 4 investigates an extensive optimisation of the jet parameters to increase the mitigation of the convective heat loss from the heated surface.

Optimization of air curtain parameters to reduce heat loss from heated surface²

While the previous chapter demonstrated a parametric analysis of different air curtain and heated wall parameters to mitigate convective heat loss from a heated surface, this current chapter optimises air curtain parameters by focusing on its global effectiveness in mitigating convective heat loss from an external heated surface. The same previous model is used to perform this study; however, some minor adjustments were required in the computational domain. The necessary changes are stated below.

1. The computational domain is extended as jet angle (α), thickness (b), and gap (G) were needed to vary simultaneously.
2. The mesh elements are adjusted according to the new computational domain.
3. No inclination of the heated wall is considered as previous Chapter 3 showed air curtain performed better for a vertical wall. Hence, the heated surface is kept vertical ($\theta = 0^\circ$) with same wall and ambient temperature of $T_w = 110^\circ\text{C}$ and $T_\infty = 21^\circ\text{C}$, respectively.
4. The initial condition of this optimization considers the best jet condition,

²Material in this chapter has been submitted as: R. Mondal, J. F. Torres, G. Hughes, and J. Pye. Optimal configuration of oblique air curtains for heat loss suppression from vertical solar-thermal receiver surfaces. *Applied Thermal Engineering*, 2022. An extra section 4.2 is added for more clarification of the results.

i.e., $b = 3 \text{ mm}$, $b = 3 \text{ mm}$, and $\alpha = 45^\circ$ found in Chapter 3.

The revised computational domain used for this investigation is presented in Figure 4.1.

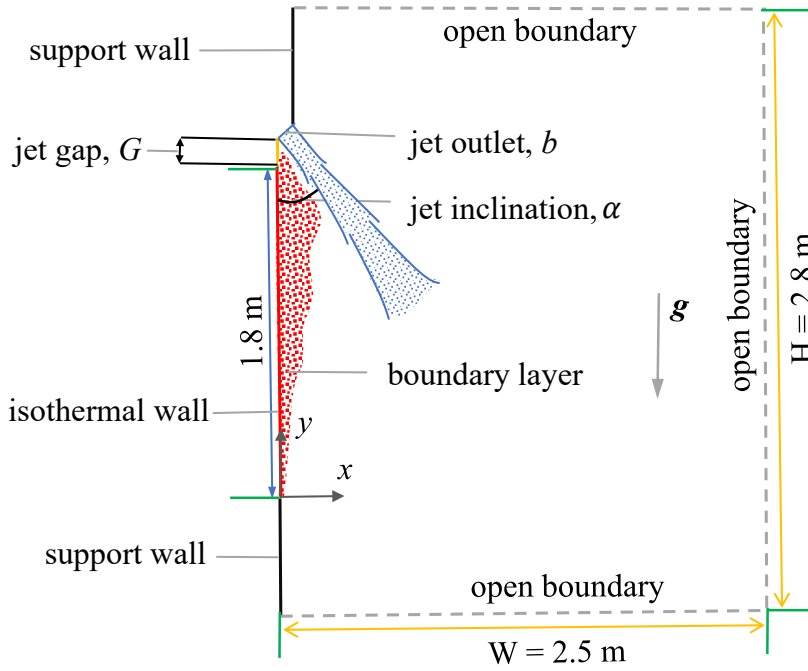


Figure 4.1: Two-dimensional computational domain and associated boundary conditions.

4.1 Grid dependency test

Since the new computational domain has been extended, a new mesh refinement test was required because the right choice of mesh size and distribution is important to obtain accurate flow predictions. Figure 4.2 shows meshing examples of the domain focused on the jet outlet with different widths. As in Figure 2.2, a similar approach was applied to meshing the computational domain. The detailed meshing approach is described in section 2.1.2.

In this study, a structured non-uniform mesh is adopted. As in the meshing process described by Torres et al. (2020), the mesh is created through a combination of *blockMesh* and *snappyHexMesh*, both inbuilt OpenFOAM (2021)

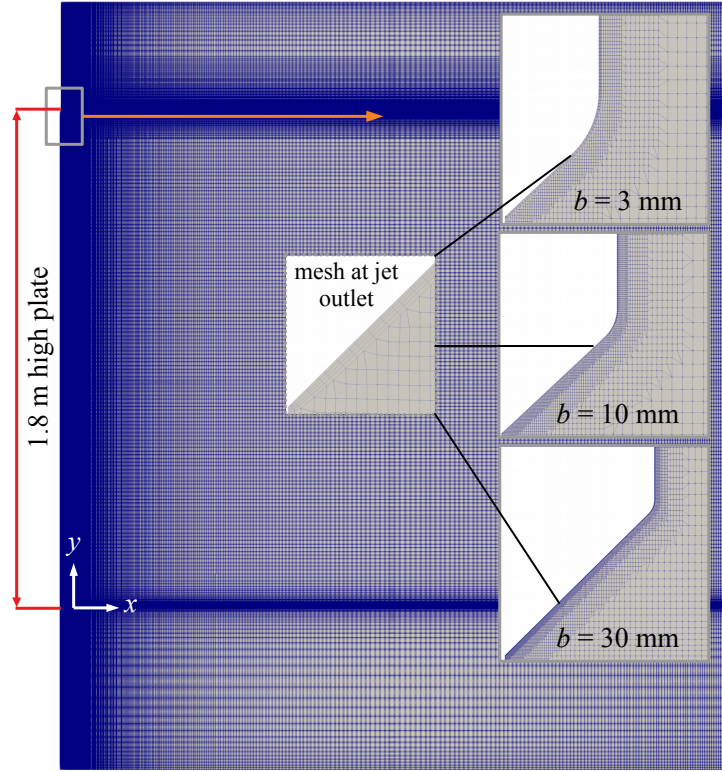


Figure 4.2: Two-dimensional mesh around the heated plate and jet, focusing on different jet widths (b) inclined at $\alpha = 45^\circ$.

meshing utilities, while using Gmsh (Geuzaine and Remacle, 2009) to generate the boundary surface input for *snappyHexMesh*. The mesh was ensured sufficiently fine enough in the vicinity of the jet outlet and heated wall to capture the expected flow. Grid dependency tests were conducted for two separate cases: pure natural convection ($U_{ac} = 0 \text{ m s}^{-1}$) and convection with an air curtain for $\alpha = 45^\circ$ and $U_{ac} = 6 \text{ m s}^{-1}$. This high-speed jet was chosen so that a suitably accurate mesh under these conditions would also be suitable for slower jets.

The grid convergence study for both cases is summarised in Figure 4.3, where Figure 4.3(i) describes the natural convection case, and Figure 4.3(ii) illustrates the natural convection with an air curtain case. In both cases, the heat transfer rate was calculated against six different mesh sizes. It is clear that the first two meshes with less than 5×10^4 cells predict a heat transfer rate from the heated surface that is significantly higher than with a finer mesh. The sug-

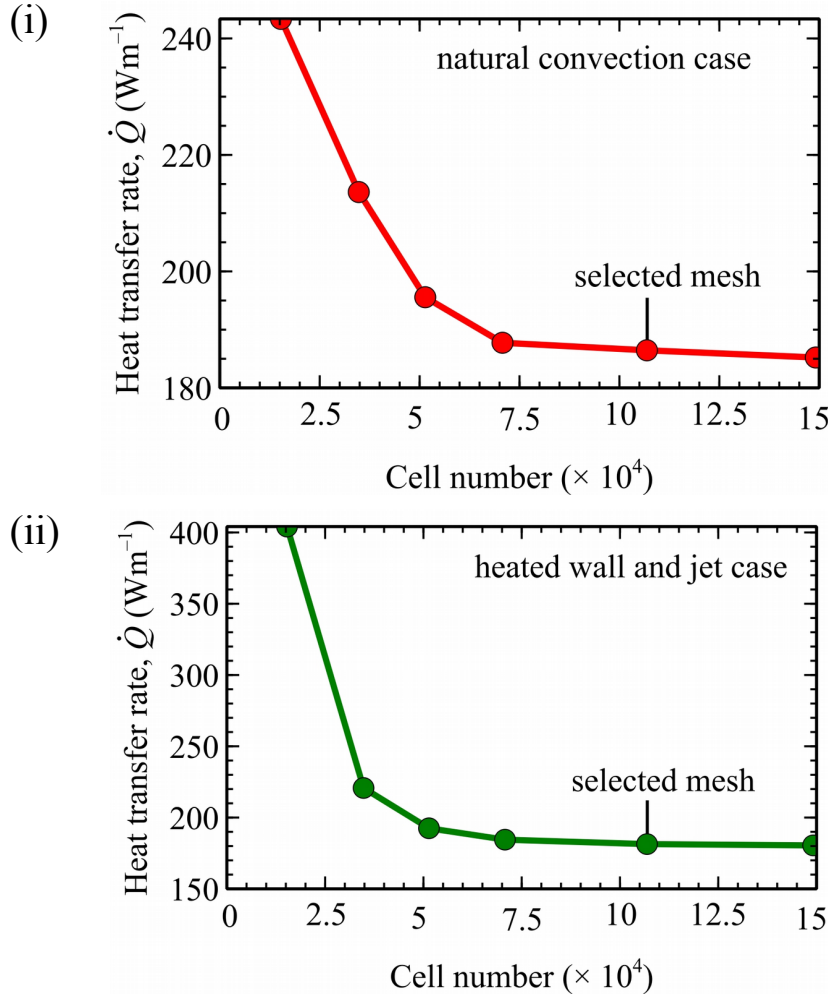


Figure 4.3: Grid dependency test was performed when $T_w = 110^\circ\text{C}$, $b = 3\text{ mm}$, $G = 0\text{ mm}$, and $\alpha = 45^\circ$. (i) natural convection case. (ii) natural convection with jet of speed $U_{ac} = 6\text{ m s}^{-1}$.

gested reason is that the mesh is not fine enough to capture the velocity and thermal boundary layers close to the jet outlet in the vicinity of the heated plate. However, once the number of cells exceeds 7.5×10^4 , the variation in heat transfer rate becomes marginal for both cases, satisfying the mesh dependence test. The percentage difference in heat transfer rate between mesh No. 5 and mesh No. 6 is 0.66% for the natural convection case and 0.50% for the natural convection with an air curtain case. To obtain fast and sufficiently accurate results, mesh No. 5 with 1.06×10^4 cells was selected for performing further simulations.

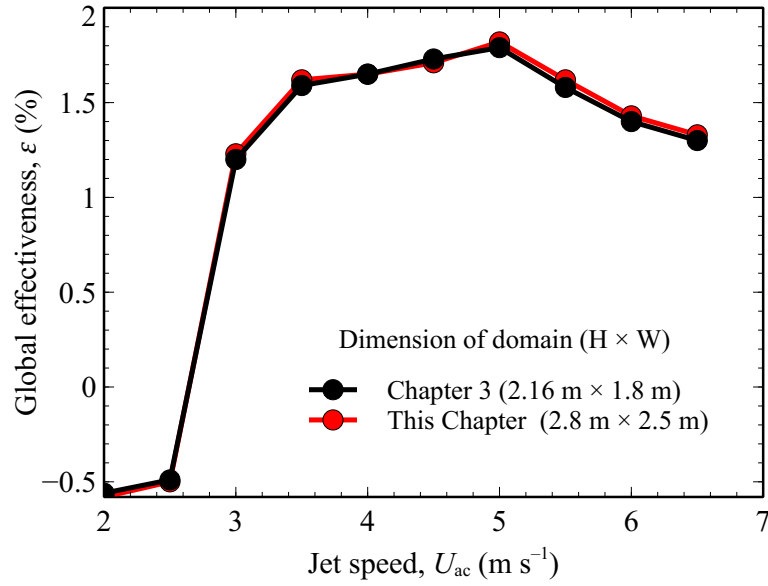


Figure 4.4: Global effectiveness against different jet speeds for two separate domains when $T_w = 110^\circ\text{C}$, $\alpha = 45^\circ$, $b = 3\text{ mm}$, and $G = 0\text{ mm}$.

4.2 Domain dependency test

As this study proposes a relatively large domain, the domain independence test was required to see if the findings obtained in Chapter 3 had changed significantly. To conduct this test, the global effectiveness of various jet speeds is compared between both domains in Figure 4.4. Increasing the domain size had no significant effect (<3.5%) on global effectiveness, as shown in Figure 4.4. The adjustment of different air curtain parameters such as jet angle, thickness, and gap needed to be altered simultaneously to achieve the best conditions; hence this little change in percentage in global effectiveness was deemed acceptable.

4.3 Optimization technique

Optimization of a system is required to get the best design in relation to a set of prioritized criteria and/or constraints. There are numerous available methods, each with various degrees of usefulness. The univariate search method (Pierre, 1986) is utilized to optimize the different air curtain parameters that minimize convection losses from the external heated surface. The set

of parameters is varied through a loop until the solution cannot be improved beyond a pre-established convergence criterion or parameter range. Once all parameters have been varied, the loop is repeated until the goal solution has been achieved.

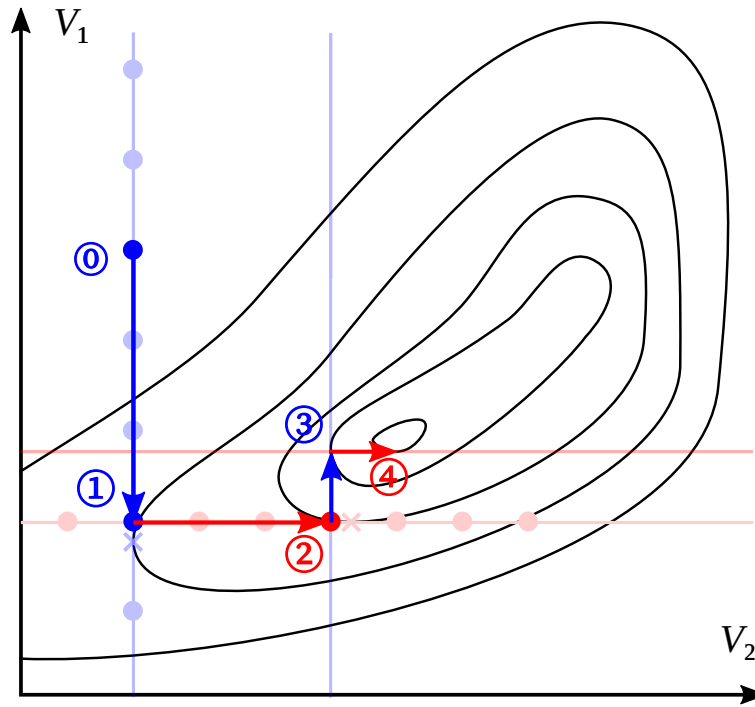


Figure 4.5: Schematic of the univariate search method, for the case of optimization with two variables V_1 and V_2 . From initial point (0), first, V_1 is optimized to reach the point (1), then V_2 , to point (2). In a second loop through the variables, V_1 is optimized to point (3) and V_2 to point (4). Loops are repeated until no further improvements are made. At each step, points are sampled along the search direction (pale circles, shown only for steps 1 and 2), and the best-sampled point is retained, although it might differ slightly from the true line-search optimum (indicated 'x'). The sampling step size for each variable is reduced in each loop.

Figure 4.5 gives a schematic representation of the univariate search method for maximizing an objective function with two variables V_1 and V_2 . An initial guess (point 0) is chosen, with values within the parameter ranges of interest. Then, a one-dimensional search is performed along the V_1 direction to identify a local optimum (point 1), which is then retained as the starting point for future steps. Sampling discretization has the result that the line-search optimum is not exact. Next, starting from point 1, a line search is conducted for

V_2 , to reach point 2. Once a line search has been done for all variables in the problem, this makes one ‘loop’; additional loops are added until no further improvements are found in any of the line searches. The figure shows two such loops.

This work intends to minimize convective heat loss from an isothermally heated vertical flat plate by varying four air curtain parameters: jet speed U_{ac} , angle α , thickness b , and the gap between the plate and jet G . Following the univariate search method, a line search is performed for each jet parameter in turn while keeping other parameters fixed. In the initial loop, the search interval for jet speed, angle, thickness and the gap is chosen manually to be 0.5 m s^{-1} , 5° , 5 mm and 10 mm respectively. Once a (near-)optimal condition is achieved in each parameter direction, these intervals are decreased significantly in the final loop to obtain a better estimate of the maximum heat loss reduction and corresponding optimal jet conditions.

Even though the implementation of this method is straightforward, it is considered less efficient for many cases and has some limitations compared to the other gradient-based or stochastic-based techniques. The final optimum value may vary slightly from that method which implements line searching with an arbitrary direction and variable step size. However, as a preliminary study on this topic, the current finding will still serve as a strong foundation for further investigation. This study also considers reducing the sampling step size in each successive loop to achieve greater precision in locating the optimum.

In systems with multiple optima, this method will only find a single optimum, which will depend on its starting position. It is noted that, in the work of Zhang et al. (2015) looking at an air curtain applied to a heated cavity to minimize heat loss, an extensive two-variable parameter sweep identified two local minima. The potential for these two local minima to arise is hypothesized to be a consequence of the cavity geometry. The second minimum, with a faster jet, relied on the close interaction of the jet with the far-side cavity lip, something that cannot occur with the present geometry. As such, any local optimum found with the univariate search method is expected to be a global optimum for this problem.

4.4 Analysis of results and discussion

This section discusses the key findings when optimising the air curtain parameters of jet speed, angle, thickness, and gap. Two fundamental flows are combined in this study, an isothermal heated wall and a turbulent planar jet. The validation of these models is presented and discussed in chapter 3. That study also performed a parametric study (without optimisation) of different air curtain parameters for reducing convective heat loss from a heated plate.

4.4.1 Loop 1

4.4.1.1 Jet speed

The optimisation procedure begins by determining the optimal effectiveness in the jet speed direction U_{ac} while the jet parameters of angle α , thickness b and gap G are held constant at 45° , 3 mm and 0 mm, respectively. The initial jet conditions were chosen as align with the previous study of Mondal et al. (2022). Figure 4.6 depicts the global effectiveness of air curtain as the jet speed varies. As can be seen in Figure 4.6, which was reproduced from Mondal et al. (2022), the air curtain effectiveness grew monotonically with increasing jet speed until 5 m s^{-1} , where a peak value of 1.82% was reached. The effectiveness then declined monotonically until the upper evaluated limit of 7 m s^{-1} was reached. The monotonic increase (where the jet speed increases towards 5 m s^{-1}) occurred because the jet momentum flux increasingly counters the buoyancy forces generated at the heated surface. This allows the buoyant flow to be effectively slowed or suppressed over a larger section of the heated plate; therefore, heat loss from the wall is reduced. A relatively low-speed jet tends to be significantly deflected by the strong buoyant flow, yielding a smaller global effectiveness because the buoyant flow is modified in a smaller region near the air curtain outlet. In contrast, a high-speed jet is expected to entrain the surrounding air and generate significant levels of turbulence in the vicinity of the heated plate. Therefore, strong jets can disrupt the sensitive stagnation zone created at the top of the wall, increasing local heat transfer and tending to reduce the air curtain effectiveness.

In Figure 4.7, the local Nusselt number Nu_y is compared for natural convec-

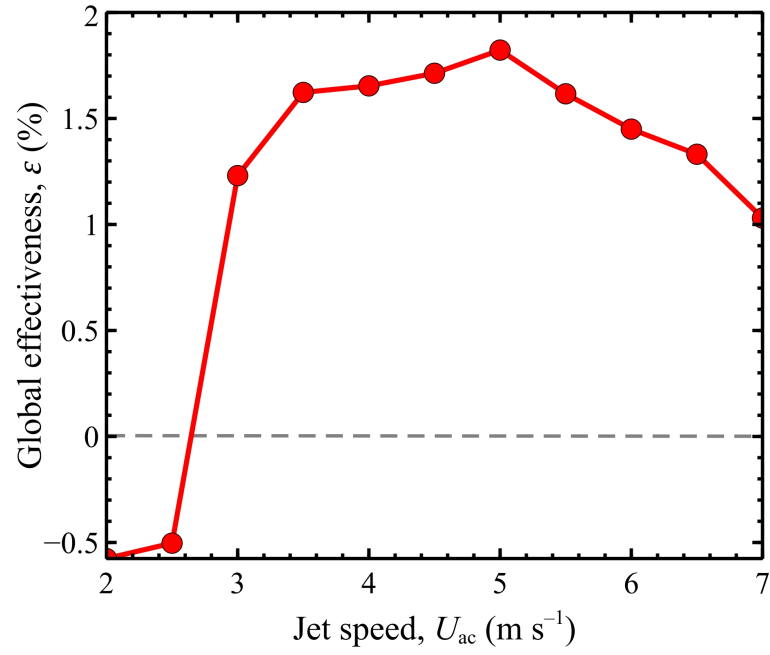


Figure 4.6: Global effectiveness (ε) of air curtain for different jet speeds when $\alpha = 45^\circ$, $b = 3$ mm, and $G = 0$ mm.

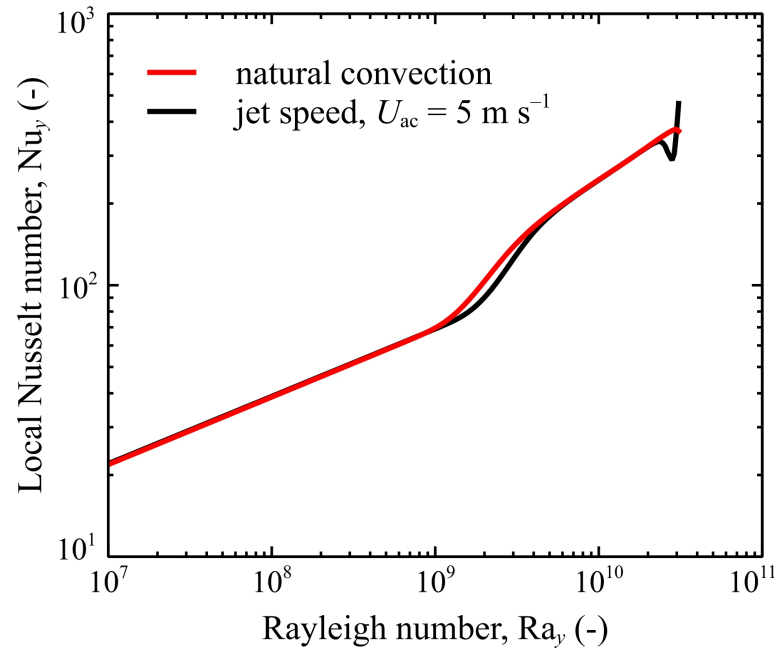


Figure 4.7: Comparison of local Nusselt number along the heated surface for cases without and with a jet when $\alpha = 45^\circ$, $b = 3$ mm, and $G = 0$ mm.

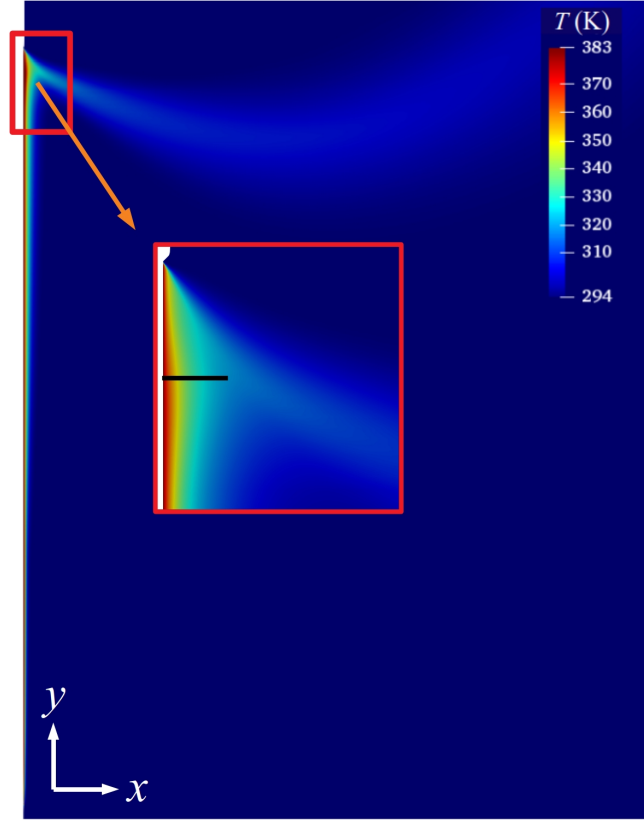


Figure 4.8: Visualization of temperature field along heated surface for $U_{ac} = 5 \text{ m s}^{-1}$, $\alpha = 45^\circ$, $b = 3 \text{ mm}$, and $G = 0 \text{ mm}$.

tion and the optimum jet case over the heated plate. A significant heat loss reduction appears at the top of the heated surface. It is found that a stagnation zone is formed (inset of 4.8) just below the jet outlet, effectively trapping more heat in this region and thickening the thermal boundary layer. To be more precise, a local velocity profile inside the stagnation zone at approximately $y = 1.75 \text{ m}$ is plotted in Figure 4.9, the horizontal location indicated by a solid black line in the top inset of Figure 4.8. The maximum velocity at this discrete location significantly dropped by about 50% for the air curtain case compared to the natural convection case. The effect of jet speed is particularly noticeable for $Ra \geq 10^9$, where the jet successfully reduces convective heat loss (i.e. yielding a positive local effectiveness) in the vicinity of the laminar-turbulent boundary layer transition. Thus, in addition to slowing the buoyant flow near the top of the wall (Figure 4.9), a jet with sufficient momentum also delays the onset of turbulence further down the wall (approximately from 0.6 m to

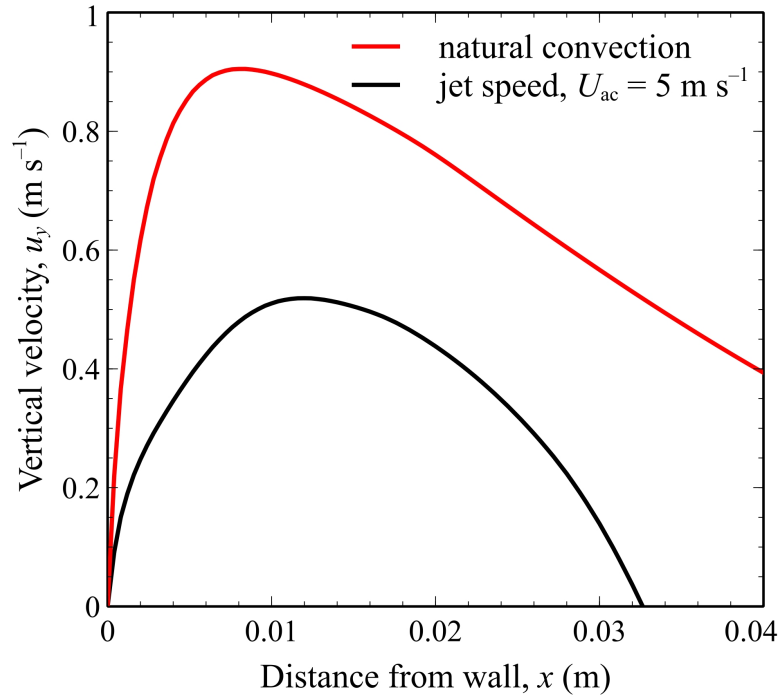


Figure 4.9: Transverse velocity profile at $y = 1.75$ m for $U_{ac} = 5 \text{ m s}^{-1}$, $\alpha = 45^\circ$, $b = 3$ mm, and $G = 0$ mm.

1 m) and reduces the heat loss there.

4.4.1.2 Jet angle

After finding the optimal jet speed of 5 m s^{-1} , the search for the optimal jet inclination is sought by varying the angle between 35° and 60° with a 5° interval.

Interestingly, a jet introduced at relatively acute angles of $\alpha < 40^\circ$ yielded a characteristic flow behaviour for which the heat loss is increased dramatically, as indicated in Figure 4.10 (inset shows the jet angles those are produced positive effectiveness). This behaviour corresponds to flow attachment (Coandă effect) on the upper section of the heated wall, as depicted in Figure 4.11 upon a comparison of the temperature field contours for wall-attached ($\alpha = 30^\circ$) (inset 1) and non-attached ($\alpha = 50^\circ$) (inset 2) jets. The attached jet pushes the buoyant boundary layer flow down the wall by creating the Coandă effect. This results in significant adverse effects by creating a high-temperature gradient in the attachment region. In contrast, the temperature field near the

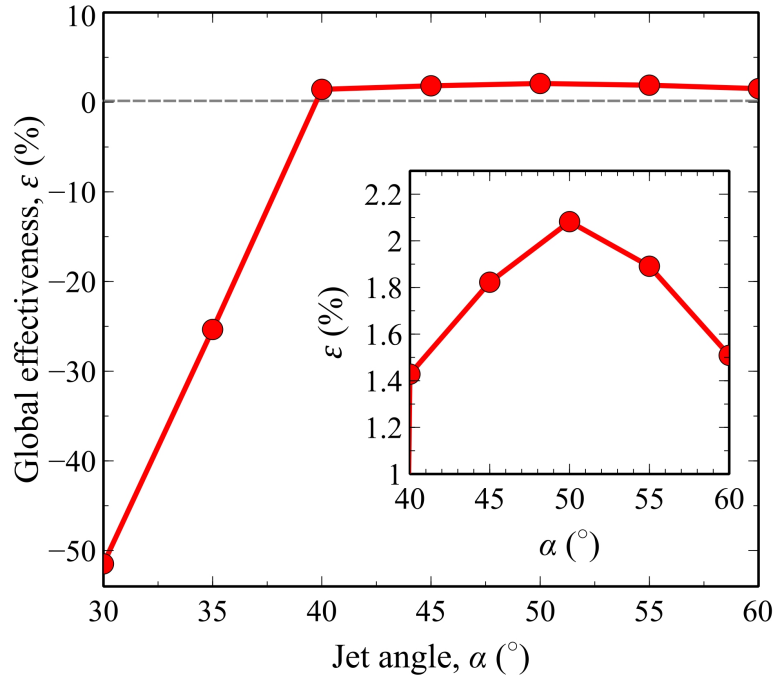


Figure 4.10: Global effectiveness as a function of jet angle when $U_{ac} = 5 \text{ m s}^{-1}$, $b = 3 \text{ mm}$, and $G = 0 \text{ mm}$; inset shows only those jet angles that produced positive effectiveness.

outlet of a non-attached jet in Figure 4.11 shows that the buoyant flow generated along the heated surface is strongly constrained by the planar jet. Note that a jet having the combination of a narrow-angle and a sufficiently high jet speed always tends to attach to the wall (Mondal et al., 2020, 2022). The relationship between jet angle and flow attachment shares similarities in wake flow. An example of this can be observed in hatch-back vehicles with slanted rear ends, where the airflow over the roof exhibits a behavior reminiscent of a jet. At specific slant angles, the flow remains attached to the slanted shape, much like the flow over a jet due to the Coandă effect. However, when the angle reaches a critical threshold, the flow detaches and forms a separation bubble as it continues on its path without remaining attached (Rao et al., 2018).

For $\alpha \geq 40^\circ$, no flow attachment was observed, which is consistent with the improvement of the global effectiveness up to $\alpha \geq 60^\circ$, as shown in Figure 4.10. A maximum of 2.08% heat loss reduction was obtained for a 50° inclined jet, which provides slightly better effectiveness than the jet speed in

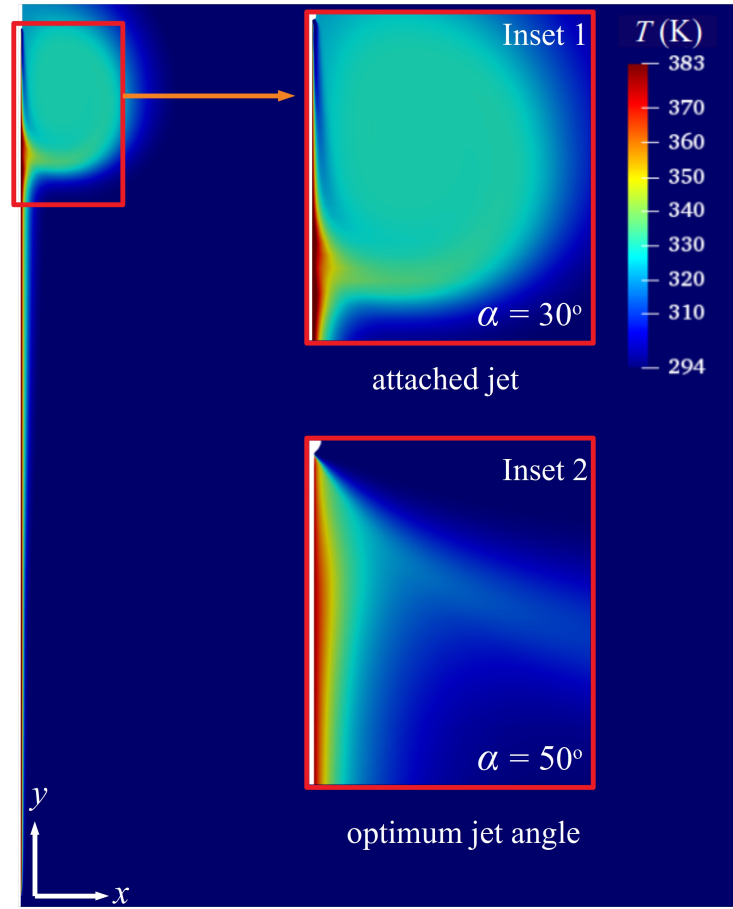


Figure 4.11: Comparison of temperature field when $U_{ac} = 5 \text{ m s}^{-1}$, $b = 3 \text{ mm}$, and $G = 0 \text{ mm}$; inset 1 describes the upper section of the wall for a wall attached jet ($\alpha = 30^\circ$) and inset 2 depicts the upper section of the wall for optimal jet angle in this iteration ($\alpha = 50^\circ$).

the previous subsection. At angles above $\alpha = 50^\circ$ (Figure 4.10, inset 2), the effectiveness drops again. We found that this behaviour occurs because of a change in the relative contributions to heat loss from the regions of the plate adjacent to the jet and where the boundary layer is turbulent. Figure 4.12 compares the wall heat flux (from $y = 1 \text{ m}$ to 1.8 m) for $\alpha = 50^\circ$ and 55° . As can be seen, $\alpha = 55^\circ$ leads to a lower heat flux close to the jet outlet; however, over most of the plate length ($y < 1.75 \text{ m}$), the 50° inclined jet is slightly more effective. Thus, for a jet inclination greater than 50° , the overall effectiveness is reduced by the heat loss from the turbulent region.

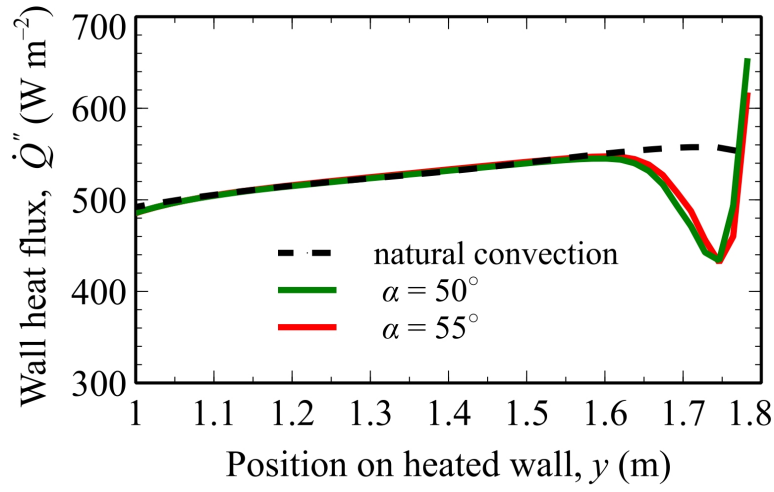


Figure 4.12: Comparison of wall heat flux for $\alpha = 50^\circ$ and 55° when $T_w = 110^\circ\text{C}$, $b = 3\text{ mm}$, $G = 0\text{ mm}$, and $U_{ac} = 5\text{ m s}^{-1}$.

4.4.1.3 Jet thickness

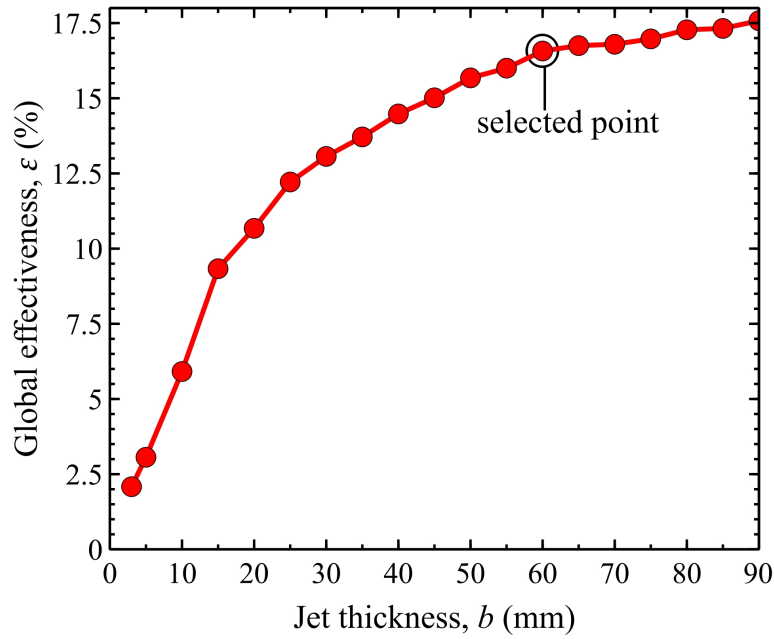


Figure 4.13: Global effectiveness for different jet thicknesses when $\alpha = 50^\circ$, $U_{ac} = 5\text{ m s}^{-1}$, and $G = 0\text{ mm}$.

This section explores the improvement of effectiveness as the jet thickness varied. The jet speed and angle were fixed to 5 m s^{-1} and 50° , respectively, i.e. the optimal values found in the previous search steps. The jet thickness is

varied from 3 mm to 90 mm, which are considered to be practical limits for a design of this scale. Figure 4.13 plots the global effectiveness as a function of jet thickness. Surprisingly, the effectiveness rose dramatically and monotonically with increasing jet thickness. This finding suggests that a thicker jet with a higher volumetric flow rate can further improve confinement of the buoyant flow near the top of the wall, further reducing heat loss and improving air curtain effectiveness. However, the continued improvement in effectiveness is significantly reduced beyond 60 mm by less than 0.5%. We have not attempted further for such a small improvement to consider additional costs associated with the manufacturing and operation of such a thick jet. The practical limit of the jet thickness in this study was considered based on the degree of effectiveness improvement for each 10 mm interval. The increase in effectiveness was more than 1%, which was set as the critical limit for each interval. Thus, a jet thickness of 60 mm with an effectiveness of 16.57% is deemed the ‘optimal’ value for investigating the next parameter.

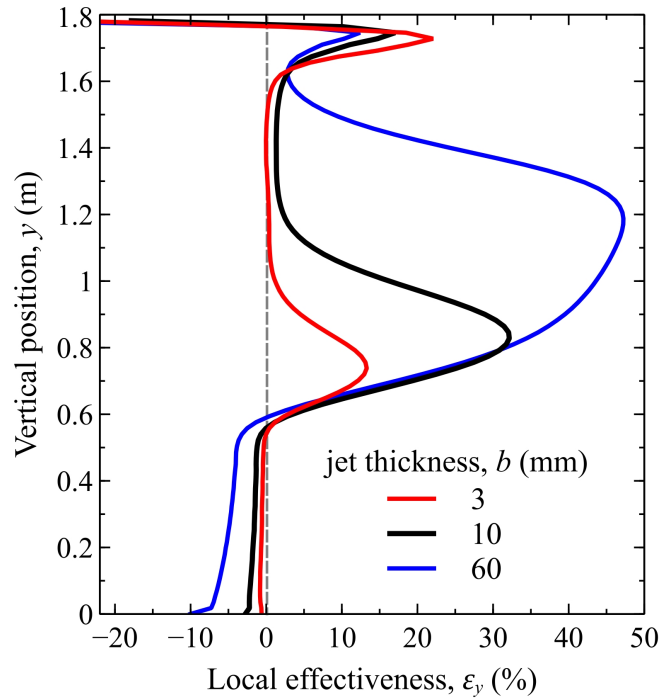


Figure 4.14: Local effectiveness for three different jet thicknesses when $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $G = 0 \text{ mm}$.

To further investigate the interaction between jet thickness and the reduc-

tion of convective heat loss along the plate, we plot the local effectiveness for three selected jet thicknesses in Figure 4.14. Within 20 mm of the jet outlet ($y > 1.78$ m), a negative local effectiveness was consistently observed. Note that the jet is initially positioned adjacent to the heated surface (0 mm gap) with a jet temperature equal to that of the ambient air, $T_\infty = 21$ °C. This local negative effectiveness results from a thinning of the thermal boundary layer caused by a direct interaction between the hot plate and the cold jet (Guyonnaud et al., 2000). However, beyond 20 mm below the jet outlet, a significant reduction of heat loss (i.e. increased effectiveness) was obtained between the approximate heights $y = 1.6$ m and $y = 1.78$ m, where a stagnation zone was established. The thicker jet was slightly less effective in this region and is likely to result from the increase of total momentum in the jet; in a previous study of (Mondal et al., 2022) we observed a similar reduction in local effectiveness in this region as the jet speed increased.

The largest heat loss reduction is achieved in the range $0.6 \text{ m} < y < 1.5 \text{ m}$ (Figure 4.14), with a maximum local effectiveness of 47.2% at $y \approx 1.19$ m on the wall. In this region, the jet notably delayed the laminar-turbulent transition (see Figure 4.7) and reduced the heat loss from the wall. Introducing the jet always disturbed the boundary layer in the laminar region ($y < 0.6$ m), slightly increasing convective losses and resulting in negative effectiveness (up to -10%).

4.4.1.4 Jet position

The final search within Loop 1 to find the near-optimal effectiveness varied the jet gap parameter. The position of the jet is defined by varying the vertical gap (G) between the heated surface and jet (see Figure 2.1). The marginally optimal jet conditions achieved in previous sections were maintained, i.e. $b = 60$ mm, $\alpha = 50^\circ$ and $U_{ac} = 5 \text{ m s}^{-1}$.

The effectiveness of the air curtain for different jet gaps is shown in Figure 4.15. Although global effectiveness did not show a large improvement, it follows a discernible trend. The effectiveness increased as the gap began to increase, reaching a maximum of 17.17% for $G = 30$ mm, then dropping monotonically until $G = 70$ mm. Figure 4.16 shows the local effectiveness

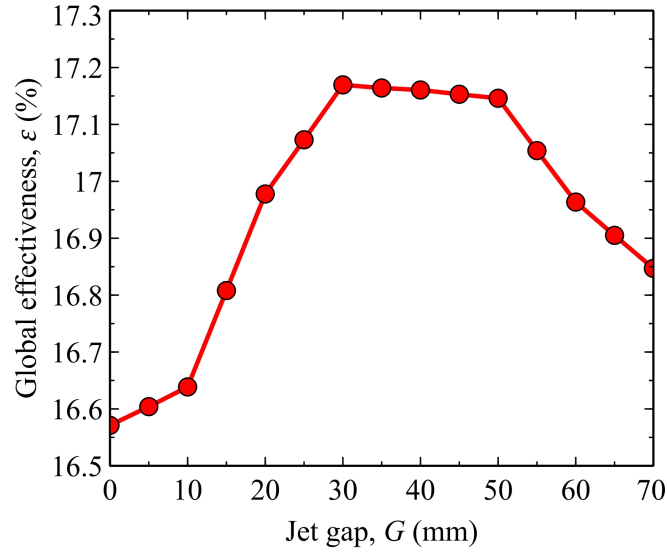


Figure 4.15: Global effectiveness (ε) of air curtain for different jet gaps when $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $b = 60 \text{ mm}$.

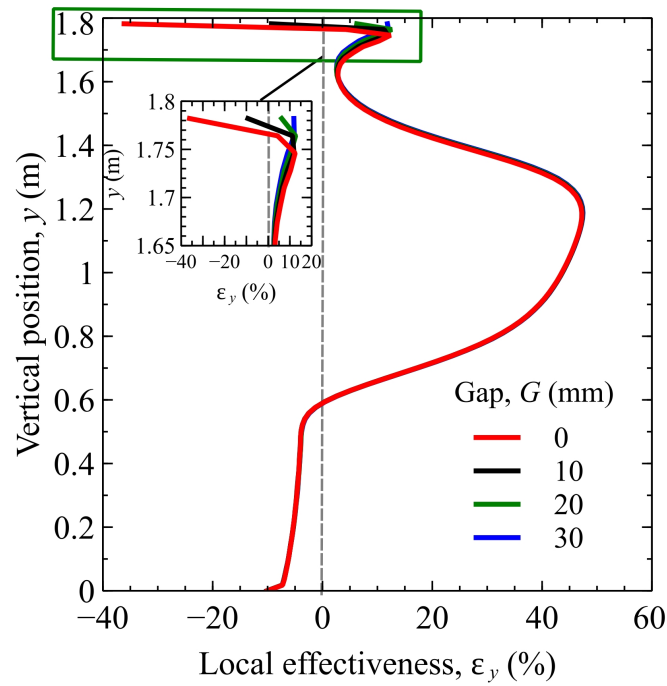


Figure 4.16: Local effectiveness of different jet gaps for $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $b = 60 \text{ mm}$.

for different jet gaps. It is seen that an increasing gap leads to a substantial improvement in local effectiveness for $y \geq 1.78 \text{ m}$ (inset of Figure 4.16). This behaviour is the result of reduced disturbance of the stagnation zone by the

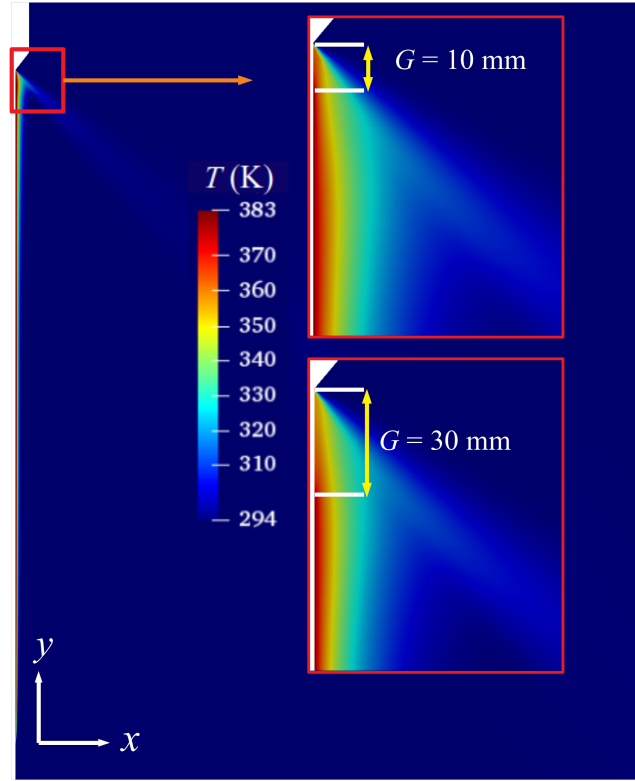


Figure 4.17: Temperature contour for jet gap, $G = 10$ mm and 30 mm when $\alpha = 50^\circ$, $U_{ac} = 5 \text{ m s}^{-1}$, and $b = 60$ mm.

jet; minimal influence is evident over the remainder of the plate length. This hypothesis is supported by the temperature fields in Figure 4.17: the insets show the top of the wall for a jet gap of 0 mm and 30 mm. It is noticeable that when the ambient jet is present adjacent ($G = 0$ mm) to the heated surface, it immediately interacts with the buoyant fluid at the top of the surface to increase the heat loss (i.e. reducing the local effectiveness, to the point that it can become negative). In contrast, when the ambient jet is present at $G = 30$ mm from the heated surface, it interacts first with the pocket of hot air in the gap between the heated wall and the jet outlet, resulting in reduced heat loss (and a less negative, or even positive local effectiveness) at the top of the wall.

Table 4.1: The optimal parameters obtained in Loop 2.

Parameters	Conditions	Effectiveness (%)
Jet speed (m s^{-1})	5.5	17.54
Jet angle ($^\circ$)	45	17.68
Jet thickness (mm)	30	18.37
Jet gap (mm)	30	18.37

4.4.2 Additional loops

As in Section 4.4.1, additional loops were performed to determine the optimal jet conditions. The starting jet conditions for each loop were based on the near-optimal conditions acquired in the preceding loop. Note that the search intervals for each parameter were the same as in Loop 1.

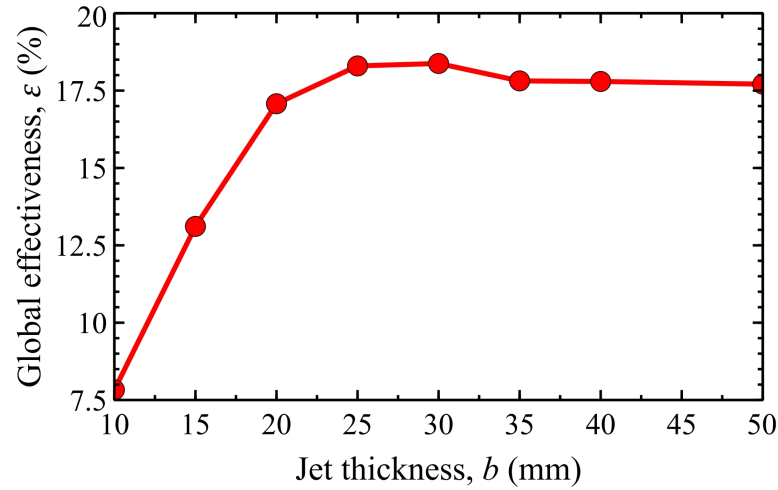


Figure 4.18: Effectiveness as a function of jet thickness in Loop 2 when $U_{ac} = 5.5 \text{ m s}^{-1}$, $\alpha = 45^\circ$, and $G = 30 \text{ mm}$.

Table 4.1 shows that the global effectiveness is increased in Loop 2 by 1.17% from Loop 1. A small improvement in effectiveness was found for each of the jet speed, angle and thickness parameters. However, no further improvement was found for the jet gap parameter. The existence of an effectiveness maximum in the range of jet speeds and angles was observed, as for Loop 1, but this was not the case for the jet thickness. In Loop 1, the effectiveness increased monotonically with jet thickness, but a plateau in effectiveness was observed in Loop 2 for $b > 35 \text{ mm}$, as shown in Figure 4.18.

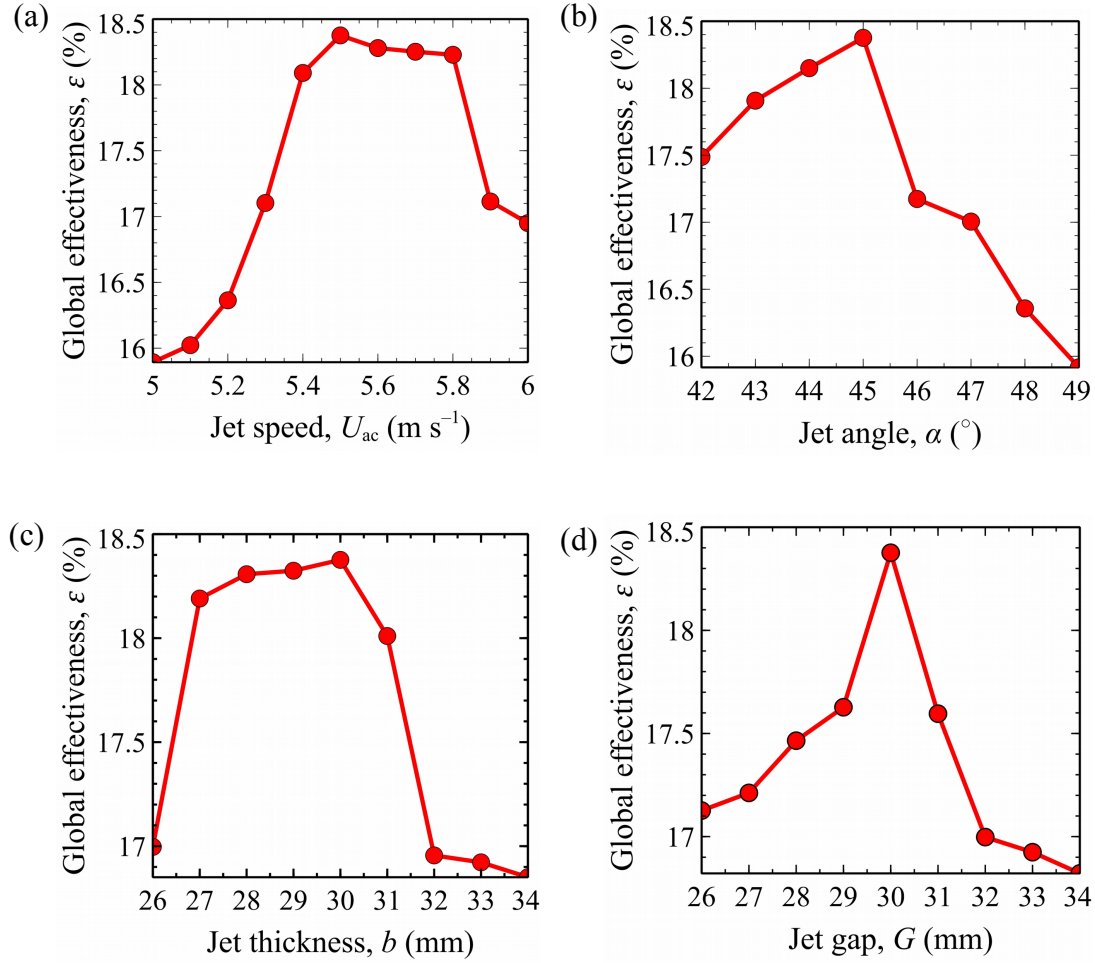


Figure 4.19: Global effectiveness obtained in Loop 3 as a function of the jet parameter: (a) Jet speed when $\alpha = 45^\circ$, $b = 30$ mm and $G = 30$ mm. (b) Jet angle when $U_{ac} = 5.5 \text{ m s}^{-1}$, $b = 30$ mm and $G = 30$ mm. (c) Jet thickness when $U_{ac} = 5.5 \text{ m s}^{-1}$, $\alpha = 45^\circ$ and $G = 30$ mm. (d) Jet gap when $U_{ac} = 5.5 \text{ m s}^{-1}$, $\alpha = 45^\circ$ and $b = 30$ mm.

Although the improvement in effectiveness was relatively small in Loop 2, an additional loop search (Loop 3) was performed. However, the search intervals close to the previously obtained optimal conditions were reduced for each parameter to $U_{ac} = 0.1 \text{ m s}^{-1}$, $\alpha = 1^\circ$, $b = 1$ mm and $G = 1$ mm. Figure 4.19 summarises the results obtained in Loop 3. It is clearly seen that the global effectiveness did not exceed the maximum value of 18.37% obtained in Loop 2. Hence, the optimal jet condition is deemed to be $\alpha = 45^\circ$, $U_{ac} = 5.5 \text{ m s}^{-1}$,

$b = 30 \text{ mm}$ and $G = 30 \text{ mm}$.

4.4.3 Power consumption of the air curtain

After completing the final loop search, the air curtain achieved a maximum of 18.37% effectiveness in reducing convective heat loss from the heated surface. Nevertheless, there is some energy loss in operating the air curtain, which needs to be considered for calculating the actual air curtain performance. This section attempted to estimate the energy required to operate the air curtain under optimal conditions achieved in the final loop. The actual power savings and modified effectiveness of the air curtain is also reported.

The estimation of total energy consumed by an air curtain depends on a few parameters, such as the energy required to accelerate the fluid from the jet outlet, the efficiency of the fan, motor, and the power block of the system. The power required to accelerate the fluid from the jet outlet can be estimated from its kinetic energy (Alipourtarzanagh et al., 2020c).

$$P_{\text{jet}} = \frac{1}{2} \dot{m} U_{\text{ac}}^2, \quad (4.1)$$

where P_{jet} is the power consumption at the air curtain nozzle exit, $\dot{m}$ is the mass of the fluid (air), and U_{ac} is the speed of the air curtain.

In equation 4.1, the mass flow rate at the jet outlet is calculated from its density, cross-sectional area, and jet speed.

$$\dot{m} = \rho_a A_{\text{ac}} U_{\text{ac}}, \quad (4.2)$$

Here ρ_a is the density of the fluid (air), A_{ac} is the cross-sectional area of the jet outlet, and U_{ac} is the speed of the air curtain.

Substituting the equation 4.2 in equation 4.1,

$$\begin{aligned} P_{\text{jet}} &= \frac{1}{2} \dot{m} U_{\text{ac}}^2 \\ &= \frac{1}{2} \rho_a A_{\text{ac}} U_{\text{ac}}^3 \end{aligned}$$

Hence, the power needed at the air curtain outlet to operate under optimal jet conditions is,

$$P_{\text{jet}} = \frac{1}{2} \rho_a A_{\text{ac}} U_{\text{ac}}^3 \quad (4.3)$$

$$= \frac{1}{2} (1.039 \text{ kgm}^{-3}) (0.03 \text{ m} \times 1 \text{ m}) (5.5 \text{ ms}^{-1})^3 \quad (4.4)$$

$$= 2.5 \text{ W} \quad (4.5)$$

The efficiency of the fan (η_{fan}), motor (η_{motor}), and power block (η_{PB}) can be estimated from the following equations 4.6, 4.7, and 4.8, respectively.

$$\eta_{\text{fan}} = \frac{P_{\text{jet}}}{P_{\text{fan}}} \quad (4.6)$$

$$\eta_{\text{motor}} = \frac{P_{\text{fan}}}{P_{\text{elec}}} \quad (4.7)$$

$$\eta_{\text{PB}} = \frac{P_{\text{elec}}}{\dot{Q}_{\text{th}}} \quad (4.8)$$

where P_{fan} is the power used by the fan, P_{elec} is the electricity power to operate the fan, and $\dot{Q}_{\text{th}}$ is the thermal energy required to produce electricity.

Combining the equations 4.6, 4.7, and 4.8, we get

$$\dot{Q}_{\text{th}} = \frac{P_{\text{jet}}}{\eta_{\text{fan}} \cdot \eta_{\text{motor}} \cdot \eta_{\text{PB}}} \quad (4.9)$$

In the experiment, four Sanyo Denki DC fans (Model no. 9GA0612P7H01) were installed to provide the necessary airflow for the planar jet. The fans are suggested as very power efficient (no exact efficiency rating found) (Osawa, 2015). On the other hand, though the efficiency of the power block in a CSP system largely depends on the temperature, the gross efficiency lies between 35%-45% (Hirsch and Khenissi, 2014). Therefore, an air curtain having a fan efficiency of 75%, a motor efficiency of 95%, and a power block efficiency of 40% will consume energy,

$$\dot{Q}_{th} = \frac{P_{jet}}{\eta_{fan} \cdot \eta_{motor} \cdot \eta_{PB}} \quad (4.10)$$

$$= \frac{2.5 \text{ W}}{0.75 \cdot 0.95 \cdot 0.40} \quad (4.11)$$

$$= 8.77 \text{ W} \quad (4.12)$$

Hence, the total power savings by the air curtain is,

$$P_s = \dot{Q}_{NC} - (\dot{Q}_{AC} + \dot{Q}_{th}) \quad (4.13)$$

$$= 186.11 \text{ W} - (152.07 + 8.77) \text{ W} \quad (4.14)$$

$$= 25.27 \text{ W} \quad (4.15)$$

Finally, the actual effectiveness of the air curtain reduces to (using equation 2.26),

$$\epsilon^* = \frac{\dot{Q}_{NC} - (\dot{Q}_{AC} + \dot{Q}_{th})}{\dot{Q}_{NC}} \times 100\% \quad (4.16)$$

$$= \frac{186.11 \text{ W} - (152.07 + 8.77) \text{ W}}{186.11 \text{ W}} \times 100\% \quad (4.17)$$

$$= 13.58\% \quad (4.18)$$

4.4.4 Effect of jet outlet temperatures

In this section, the effect of jet outlet temperature on effectiveness is examined. This option was considered because of the potential availability of waste heat streams in a CSP plant that could be provided to give a warm jet with a hypothesised 'blanket' effect. The optimal jet parameters of $U_{ac} = 5.5 \text{ m s}^{-1}$, $\alpha = 45^\circ$, $b = 30 \text{ mm}$ and $G = 30 \text{ mm}$ are chosen for this investigation. Figure 4.20 shows that the jet effectiveness gradually declines with increasing jet outlet temperature. This result is perhaps surprising as a heated air curtain would be expected to decrease the thermal contrast across the boundary layer.

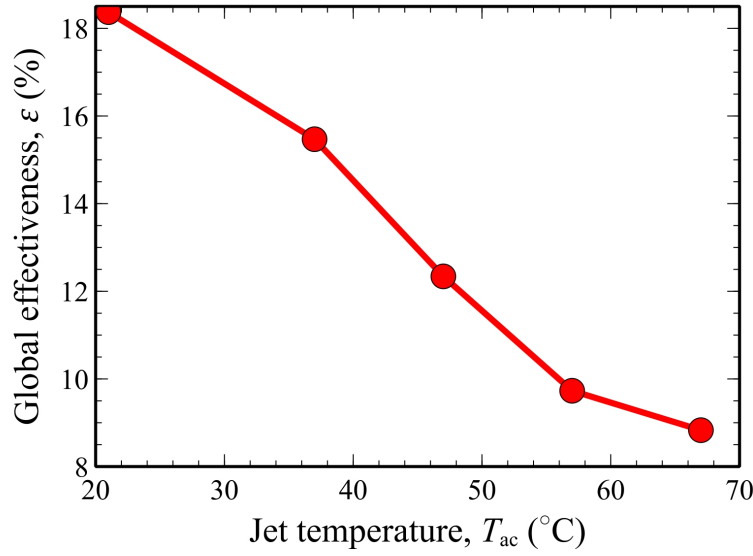


Figure 4.20: Global effectiveness for different jet outlet temperatures when $T_w = 110^\circ\text{C}$, $b = 30$ mm, $G = 30$ mm.

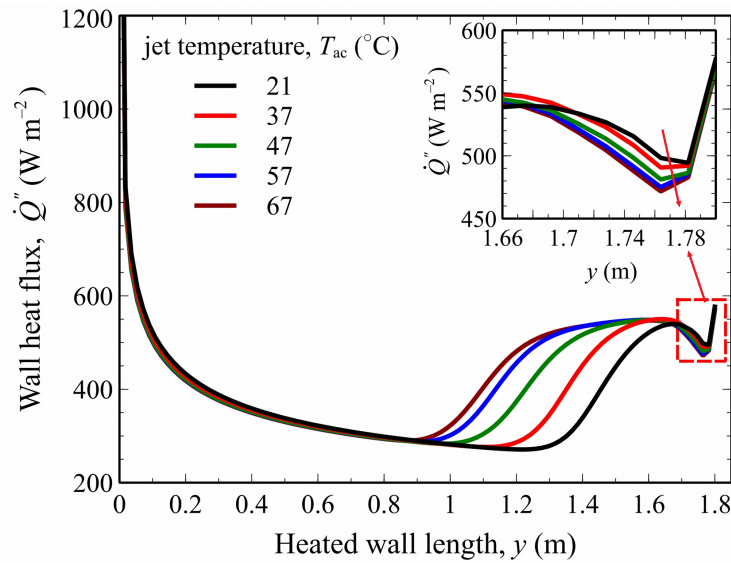


Figure 4.21: Wall heat flux for different jet outlet temperatures when $T_w = 110^\circ\text{C}$, $b = 30$ mm, $G = 30$ mm.

To explain the poor performance of the heated air curtain, we analyse the local heat flux of the wall for different jet temperatures in Figure 4.21. Two different effects were observed for $1.6\text{ m} < y < 1.8\text{ m}$, where the stagnation zone is formed, and $0.6\text{ m} < y < 1.6\text{ m}$, where the laminar-turbulent transition and turbulent flow occur. By employing an air curtain in the stagnation

zone, the convective heat loss from the heated surface is reduced as the temperature rises, improving the overall effectiveness (see inset of Figure 4.21). This behaviour is consistent with our expectation that the hotter jet decreases the temperature contrast in the thermal boundary layer. However, below the stagnation zone, the heat loss can be seen to increase significantly as the air curtain temperature increases. The reason is that the hotter jet exhibits more buoyancy and is lighter than the ambient jet because of reduced density. As a result, it is more easily deflected by the buoyant flow rising along the surface (Ogasawara et al., 2021), thus permitting a stronger boundary layer flow over much of the plate. The jet deflection, therefore, results in a significant increase in overall convective heat losses and a reduction of global effectiveness.

It should be noted that this investigation was performed under optimum jet conditions for an ambient jet. Since the hotter jet has more buoyancy and is easily deflected by buoyant flow rising along the heated wall, it may require a higher-speed jet for a heated air curtain to create a potential barrier for natural convection flow. Therefore, our results suggest that re-optimising is needed in a heated air curtain to get the exact effect of jet outlet temperature.

4.5 Non-dimensional Analysis

This thesis focuses on reducing convective heat loss from an isothermal receiver surface with a temperature of 110 °C and a height of 1.8 m. However, the receiver's height can reach up to 10 m to 20 m with a maximum temperature of 800 °C (Dennis and Kraabel, 1984). To simplify this complex technical problem, a non-dimensional analysis is conducted using non-dimensional parameters. The Buckingham π theorem (Gupta et al., 2014) is employed to establish the number of dimensional groups in this study. This theorem is commonly used due to its easier application.

The heat transfer coefficient (h) of the receiver surface is affected by various factors, including the length of the heated surface y , temperature difference ΔT , fluid viscosity μ , fluid density ρ , fluid thermal conductivity κ , fluid specific heat capacity C_p , acceleration due to gravity g , thermal expansion coefficient β , jet speed U_{ac} , jet thickness b , jet gap G , and jet angle α .

The dimensions for each dependent and independent variable are,

$$\begin{aligned} h &= MT^{-3}K^{-1} & \beta &= K^{-1} \\ \Delta T &= K & y &= L \\ \mu &= ML^{-1}T^{-1} & U_{ac} &= LT^{-1} \\ \rho &= ML^{-3} & b &= L \\ \kappa &= MLT^{-3}K^{-1} & G &= L \\ C_p &= L^2T^{-2}K^{-1} & \alpha &= M^0L^0T^0K^0 \\ g &= LT^{-2} \end{aligned}$$

The functional relationship between the dependent and independent variables can be written as,

$$h = f(y, g, \beta, \Delta T, \mu, \rho, \kappa, C_p, U_{ac}, b, G, \alpha) \quad (4.19)$$

The Buckingham π theorem states that the number of π terms is equal to the number of variables (n) minus the number of reference dimensions (m). For this particular problem, we have $n = 13$ variables, but in natural convection, $g\beta\Delta T$ is treated as a single parameter because the fluid velocity is a function of these three variables, reducing the number of variables to $n = 11$. The reference dimensions for this problem are Mass (M), Length (L), Time (T), and Temperature (K), which means that $m = 4$. Therefore, the total number of π terms can be calculated as $n - m = 7$.

Then the equation 4.19 can be rewritten as,

$$h = f(y, (g\beta\Delta T), \mu, \rho, \kappa, C_p, U_{ac}, b, G, \alpha) \quad (4.20)$$

The equation expressing the relationship among the variables has a solution of the form,

$$F(\pi_1, \pi_2, \pi_3, \pi_4, \pi_5, \pi_6, \pi_7) = 0 \quad (4.21)$$

The solution would be either

$$F(\pi_1, \pi_2, \pi_3, \pi_4, \pi_5, \pi_6) = 0 \text{ or } \pi_1 = f(\pi_2, \pi_3, \pi_4, \pi_5, \pi_6, \pi_7) \quad (4.22)$$

The repeating variables considered in this analysis are dynamic viscosity μ , thermal conductivity κ , fluid density ρ , and heated plate length y .

Therefore, the first dimensionless group can be expressed as,

$$\pi_1 = \mu^{a_1} \kappa^{b_1} \rho^{c_1} y^{d_1} h \quad (4.23)$$

Writing down the fundamental dimensions of each variable in the above equation,

$$M^0 L^0 T^0 K^0 = (ML^{-1}T^{-1})^{a_1} \cdot (MLT^{-3}K^{-1})^{b_1} \cdot (ML^{-3})^{c_1} \cdot (L)^{d_1} \cdot MT^{-3}K^{-1} \quad (4.24)$$

$$= M^{a_1+b_1+c_1+1} \cdot L^{-a_1+b_1-3c_1+d_1} \cdot T^{-a_1-3b_1-3} \cdot K^{-b_1-1} \quad (4.25)$$

Comparing the power of M , L , T , and K ,

$$a_1 + b_1 + c_1 + 1 = 0 \quad (4.26)$$

$$-a_1 + b_1 - 3c_1 + d_1 = 0 \quad (4.27)$$

$$-a_1 - 3b_1 - 3 = 0 \quad (4.28)$$

$$-b_1 - 1 = 0 \quad (4.29)$$

The above relations can easily be solved to obtain $a_1 = 0$, $b_1 = -1$, $c_1 = 0$, and

$d_1 = 1$. Using these, the equation 4.23 will be,

$$\pi_1 = \mu^0 \kappa^{-1} \rho^0 y^1 h \quad (4.30)$$

$$= \frac{yh}{\kappa} \quad (4.31)$$

$$= \text{Nu} \quad (4.32)$$

The second dimensionless group can be written as,

$$\pi_2 = \mu^{a_2} \kappa^{b_2} \rho^{c_2} y^{d_2} C_p \quad (4.33)$$

Writing down the fundamental dimensions of each variable in the above equation,

$$M^0 L^0 T^0 K^0 = (ML^{-1}T^{-1})^{a_2} \cdot (MLT^{-3}K^{-1})^{b_2} \cdot (ML^{-3})^{c_2} \cdot (L)^{d_2} \cdot L^2 T^{-2} K^{-1} \quad (4.34)$$

$$= M^{a_2+b_2+c_2} \cdot L^{-a_2+b_2-3c_2+d_2+2} \cdot T^{-a_2-3b_2-2} \cdot K^{-b_2-1} \quad (4.35)$$

Similarly, comparing the power of M , L , T , and K ,

$$a_2 + b_2 + c_2 = 0 \quad (4.36)$$

$$-a_2 + b_2 - 3c_2 + d_2 + 2 = 0 \quad (4.37)$$

$$-a_2 - 3b_2 - 2 = 0 \quad (4.38)$$

$$-b_2 - 1 = 0 \quad (4.39)$$

Solving these above relations, we can obtain $a_2 = 1$, $b_2 = -1$, $c_2 = 0$, and

$d_2 = 0$. Using these, the equation 4.33 will be,

$$\pi_2 = \mu^1 \kappa^{-1} \rho^0 y^0 C_p \quad (4.40)$$

$$= \frac{\mu C_p}{\kappa} \quad (4.41)$$

$$= \text{Pr} \quad (4.42)$$

The third dimensionless group can be written as,

$$\pi_3 = \mu^{a_3} \kappa^{b_3} \rho^{c_3} y^{d_3} (g \beta \Delta T) \quad (4.43)$$

Writing down the fundamental dimensions of each variable in the above equation,

$$M^0 L^0 T^0 K^0 = (ML^{-1}T^{-1})^{a_3} \cdot (MLT^{-3}K^{-1})^{b_3} \cdot (ML^{-3})^{c_3} \cdot (L)^{d_3} \cdot LT^{-2}K^{-1}K \quad (4.44)$$

$$= M^{a_3+b_3+c_3} \cdot L^{-a_3+b_3-3c_3+d_3+1} \cdot T^{-a_3-3b_3-2} \cdot K^{-b_3} \quad (4.45)$$

Comparing the power of M , L , T , and K ,

$$a_3 + b_3 + c_3 = 0 \quad (4.46)$$

$$-a_3 + b_3 - 3c_3 + d_3 + 1 = 0 \quad (4.47)$$

$$-a_3 - 3b_3 - 2 = 0 \quad (4.48)$$

$$-b_3 = 0 \quad (4.49)$$

Solving these above relations, we can obtain $a_3 = -2$, $b_3 = 0$, $c_3 = 2$, and

$d_3 = 3$. Using these, the equation 4.43 will be,

$$\pi_3 = \mu^{-2} \kappa^0 \rho^2 y^3 g \beta \Delta T \quad (4.50)$$

$$= \frac{g \beta \Delta T y^3 \rho^2}{\mu^2} \quad (4.51)$$

$$= \frac{g \beta \Delta T y^3}{\nu^2} \quad (4.52)$$

$$= \text{Gr} \quad (4.53)$$

The fourth dimensionless group can be written as,

$$\pi_4 = \mu^{a_4} \kappa^{b_4} \rho^{c_4} y^{d_4} U_{ac} \quad (4.54)$$

Writing down the fundamental dimensions of each variable in the above equation,

$$M^0 L^0 T^0 K^0 = (ML^{-1}T^{-1})^{a_4} \cdot (MLT^{-3}K^{-1})^{b_4} \cdot (ML^{-3})^{c_4} \cdot (L)^{d_4} \cdot LT^{-1} \quad (4.55)$$

$$= M^{a_4+b_4+c_4} \cdot L^{-a_4+b_4-3c_4+d_4+1} \cdot T^{-a_4-3b_4-1} \cdot K^{-b_4} \quad (4.56)$$

Comparing the power of M , L , T , and K ,

$$a_4 + b_4 + c_4 = 0 \quad (4.57)$$

$$-a_4 + b_4 - 3c_4 + d_4 + 1 = 0 \quad (4.58)$$

$$-a_4 - 3b_4 - 1 = 0 \quad (4.59)$$

$$-b_4 = 0 \quad (4.60)$$

Solving these above relations, we can obtain $a_4 = -1$, $b_4 = 0$, $c_4 = 1$, and

$d_4 = 1$. Using these, the equation 4.54 will be,

$$\pi_4 = \mu^{-1} \kappa^0 \rho^1 y^1 U_{ac} \quad (4.61)$$

$$= \frac{\rho y U_{ac}}{\mu} \quad (4.62)$$

$$= \text{Re} \quad (4.63)$$

Similarly, the fifth, sixth, and seventh dimensionless groups containing jet thickness b , jet gap G , and jet angle α will be,

$$\pi_5 = \frac{b}{y} \quad (4.64)$$

$$\pi_6 = \frac{G}{y} \quad (4.65)$$

$$\pi_7 = \alpha \quad (4.66)$$

Substituting the values of π_1 , π_2 , π_3 , π_4 , π_5 , π_6 , and π_7 in equation 4.22, we get

$$\text{Nu} = f(\text{Pr}, \text{Gr}, \text{Re}, \frac{b}{y}, \frac{G}{y}, \alpha) \quad (4.67)$$

We can simplify equation 4.67 further by neglecting the Prandtl number since the working fluid for this study is air, and its value is fixed. Additionally, we can obtain another important physical group $\frac{\text{Gr}}{(b/y) * \text{Re}^2}$ by manipulating terms π_3 , π_4 , and π_5 . This group estimates the ratio of buoyancy and momentum forces in the vertical momentum equation. Based on the findings obtained in this study, it is suggested that $\frac{b}{G}$ is a better alternative than $\frac{G}{y}$ since the jet dynamics near the heated plate are responsible for any dependence. Also, the value of $\frac{G}{y}$ is approximately close to zero and not significant.

Finally, the equation 4.67 can be reduces to

$$\text{Nu} = f\left(\frac{\text{Gr}}{(b/y) * \text{Re}^2}, \frac{b}{G}, \alpha\right) \quad (4.68)$$

$$= f(\text{Ri}^*, \frac{b}{G}, \alpha) \quad (4.69)$$

Here, Ri^* is the modified Richardson number.

4.6 Conclusion

This work considered the use of a forced external planar jet or air curtain to reduce convective heat loss from an external solar-thermal receiver surface, modelled here as an isothermal vertical plate of height 1.8 m and temperature $T_w = 110^\circ\text{C}$. Numerical simulations were conducted to optimise the performance of the air curtain, which was represented in terms of five parameters.

The findings confirmed that a significant reduction of convective heat loss from the plate could be achieved using an obliquely introduced air curtain. More than 18% heat loss reduction was obtained for an air curtain with a jet width $b = 30\text{ mm}$, a vertical gap above the plate $G = 30\text{ mm}$, inclination angle $\alpha = 45^\circ$ and speed $U_{ac} = 5.5\text{ m s}^{-1}$. The jet thickness was found to be the key parameter responsible for significant improvement of the effectiveness. The effect of heating the air curtain was investigated under otherwise optimal jet conditions, and surprisingly, an air curtain at ambient temperature was the most effective at reducing heat loss. A non-dimensional analysis has also been performed to simplify the problem in terms of real-world applications. Thus, our results serve as a strong motivation for further investigation of air curtain design, including experimental implementation of airflow control strategies, for application in solar-thermal technologies and other fields where control of convective heat loss from external surfaces is needed. Since this study is conducted in a two-dimensional space, a comparison between 2D and 3D models has been performed in Chapter 5 to identify the potential discrepancies between them. In addition, Chapter 5 also includes an experimental validation of the 3D model to gain more confidence in this work.

Validation of CFD modelling against experimental data³

The objective of this chapter is twofold. One is to show the comparison between 2D and 3D simulation results for the natural convection case to understand the difference between them, and another is the validation of the 3D CFD model against experimental data for both natural convection and natural convection with an air curtain case. The experiment was conducted by Abbasi-Shavazi (2019), a PhD student at Australian National University. He first experimentally demonstrated how an air curtain may be utilised to reduce natural convection heat loss from a heated surface. This work, however, lacks corresponding flow field data, which is required to comprehend heat transport mechanisms from the wall. The flow behaviour accompanying the interaction between buoyant fluid emerging from a heated wall and air curtain jet flow will also be demonstrated.

5.1 Geometrical set up

The geometry of the numerical model was built to be analogous to the experimental configuration. Figure 5.1 and 5.2 demonstrate the experimental and modelling setup of the isothermal heated wall and air curtain, respectively. A vertical heated wall and an air curtain are assembled in both configurations. The plate of dimensions $1.8\text{ m} \times 0.22\text{ m}$ (height $\times$ width) is constructed by

³Experimental material in this chapter has been collected from: Ehsan Abbasi-Shavazi. Experimental investigation of external convection from scale-model solar thermal receivers. PhD thesis (Draft), The Australian National University.

assembling 18 modular units, where each unit consists of a $0.1\text{ m} \times 0.22\text{ m}$ plate. Each unit of the plate is heated at 110°C . The Grashof number (Gr) based on the plate height and temperatures was 4.55×10^{10} , indicating fully turbulent flow on the plate's surface. A planar jet of height 3 mm is formed to create an air curtain. The width of the air curtain was set to 0.22 m to match the width of the heated plate. The air curtain was positioned at the top of the plate with inclination, $\alpha = 45^\circ$ (similar to that in Figure 5.2). No offset gap was maintained between the heated plate and the air curtain. The air curtain temperature was considered similar to the ambient fluid temperature of 21°C .

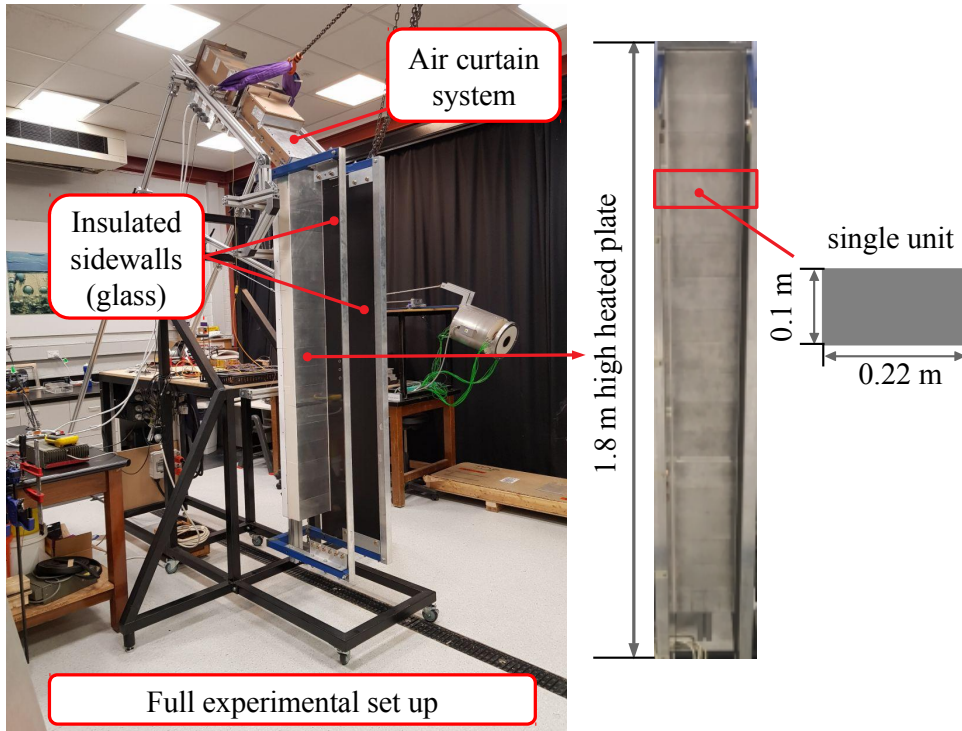


Figure 5.1: Experimental configuration of heated plate and air curtain when $T_w = 110^\circ\text{C}$ and $\alpha = 45^\circ$.

The boundaries of the numerical model were defined to approximate the experimental setup. Two adiabatic sidewalls with no-slip boundary conditions were attached in the front and back of the domain. These sidewalls help to prevent any side flows and allow the boundary layer to grow along the heated plate. Similar insulated boundaries with no-slip boundaries were also placed at the bottom and top of the plate. Open boundaries are implemented on the

top, bottom, and right gaps between the front and back walls so that fluid can enter and exit the domain freely. As described in 2.10, the same boundary conditions are imposed at the jet outlet. The jet outlet was switched on only when the natural convection boundary layer reached a quasi-steady state condition.

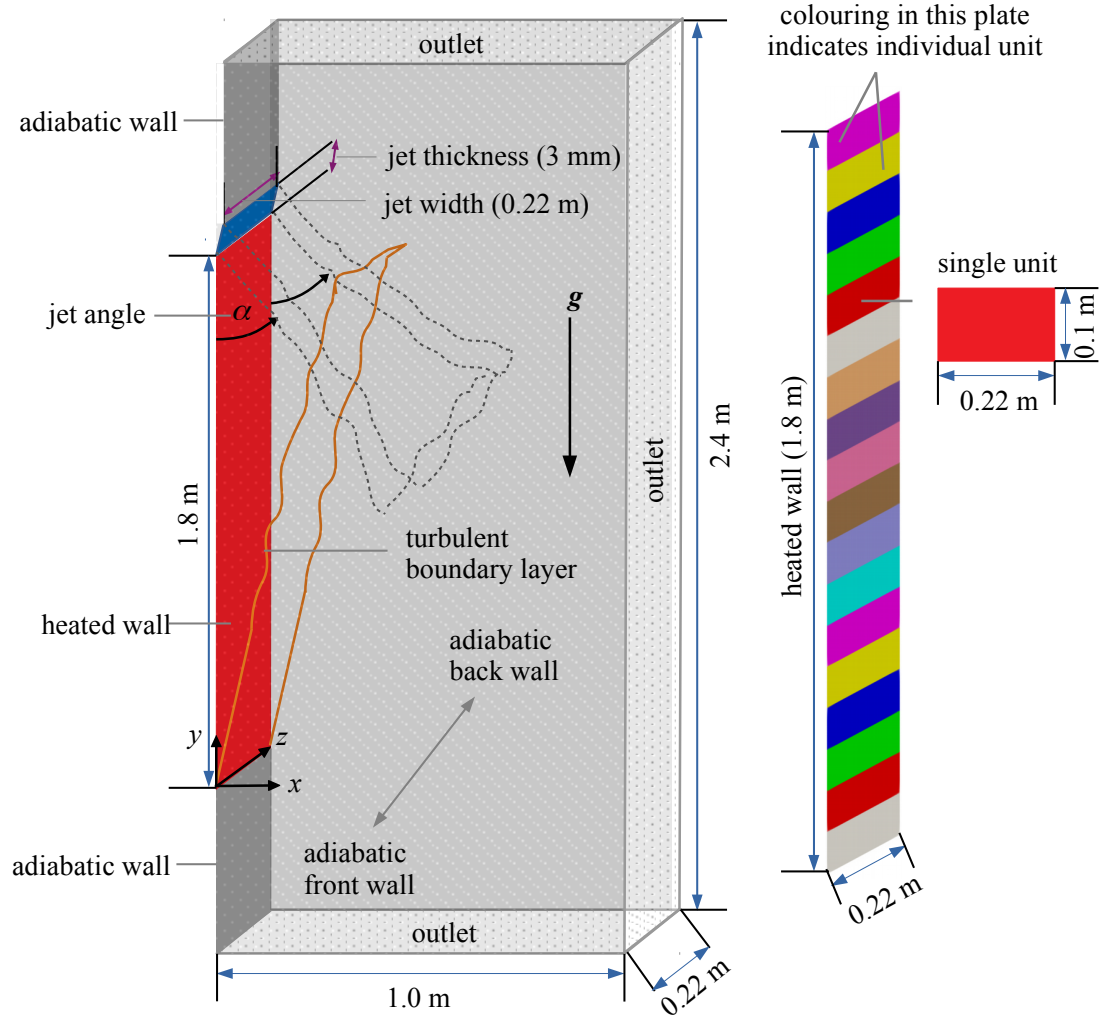


Figure 5.2: 3D modelling for heated plate and air curtain when $T_w = 110^\circ\text{C}$ and $\alpha = 45^\circ$.

The mesh surrounded by a three-dimensional domain is presented in Figure 5.3. Fine mesh was created close to the heated plate to resolve the boundary layer. The mesh close to the air curtain outlet was also refined until the

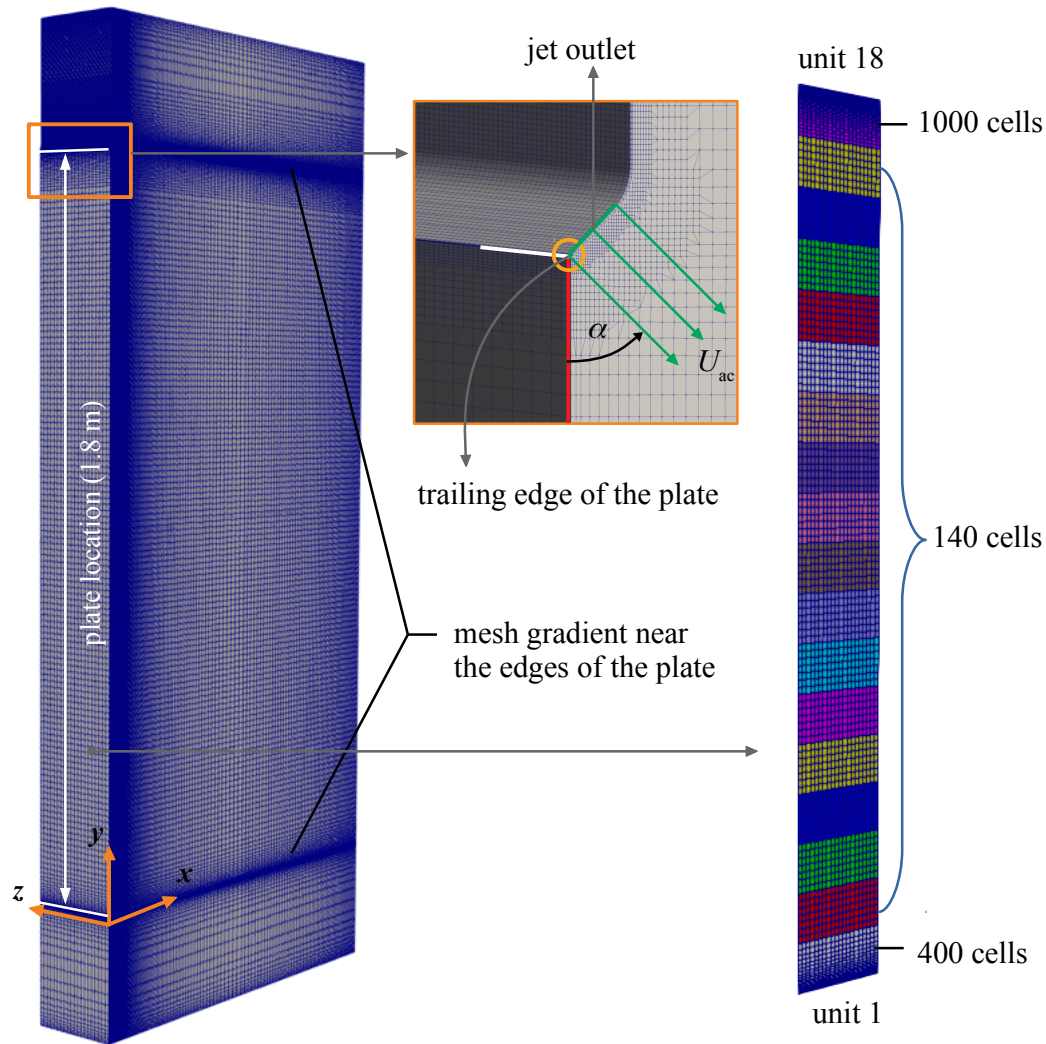


Figure 5.3: 3D mesh surrounded by the domain for $\alpha = 45^\circ$.

expected jet flow was found. Special care was taken to provide equal cell numbers in units 2 to 17 so that it does not affect the heat transfer rate from each unit of the plate. The cell numbers for unit 1 and unit 18 were not the same as other units. The maximum heat transfer occurred in the leading edge of the plate. This is why mesh gradient was used close to the leading edge to create a high-resolution mesh, resulting in increased cell numbers in unit 1. On the other hand, the top unit (unit 18) was attached with an air curtain outlet, where a more refined mesh was created.

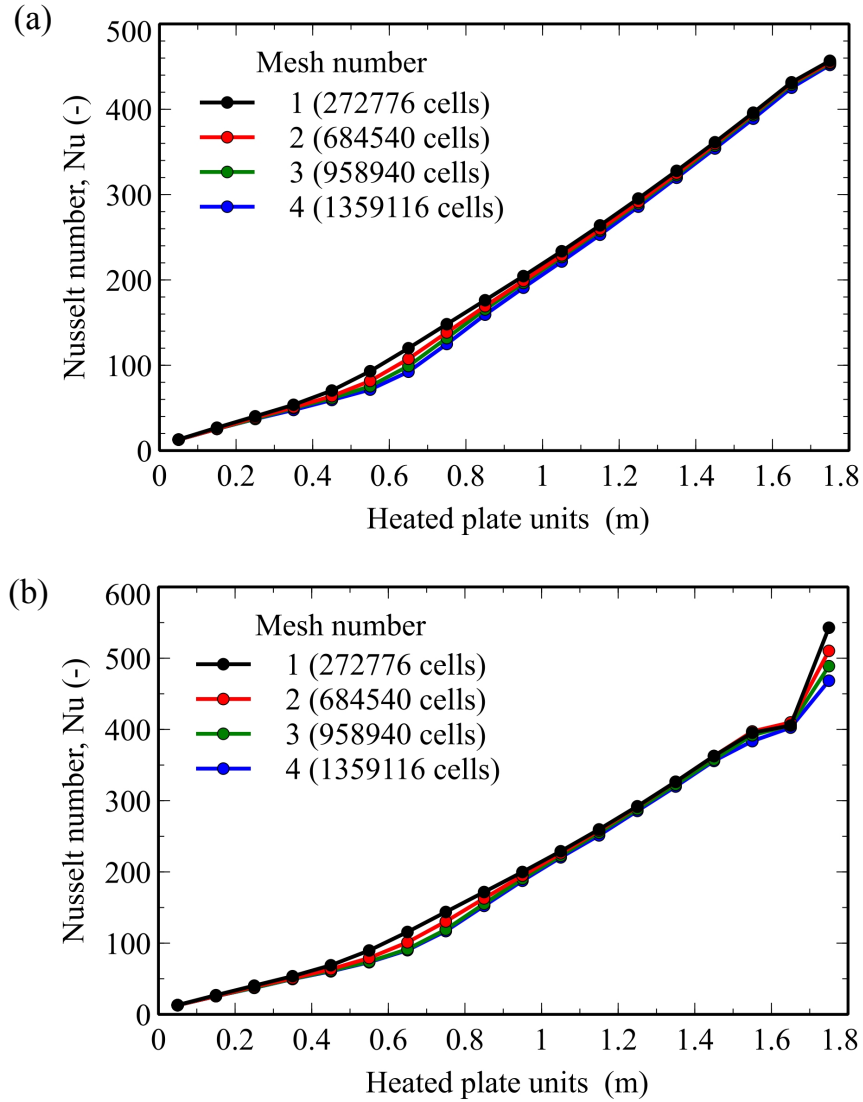


Figure 5.4: Mesh refinement test for four different mesh sizes when $T_w = 110^\circ\text{C}$, $\alpha = 45^\circ$. (a) natural convection. (b) natural convection with air curtain of speed $U_{ac} = 5.75 \text{ m s}^{-1}$.

It is essential that the mesh is suitably fine in order for the simulation results to be satisfactory. As a result, a mesh refinement test was performed for this investigation, as illustrated in Figure 5.4. The test was conducted in two configurations: natural convection and natural convection with an air curtain. Figure 5.4(a) compares the computed Nusselt number of each plate unit among four different mesh sizes for the natural convection case. The

selected mesh sizes did not reveal a significant difference in the bottom 5 units (from 0 m to 0.5 m) and top 8 units (from 1 m to 1.8 m), where laminar and turbulent flow are predicted to occur respectively. Meshes, on the other hand, have a substantial influence from units 5 to 9, when the transition from laminar to turbulent flow may occur. For the first two meshes (Mesh No. 1 and 2), the difference was larger in this region compared to mesh No. 3 and 4. When transitioning from mesh No. 3 to mesh No. 4, a slight change in the Nusselt number was observed, indicating that the mesh is refined enough to deliver reliable results. A similar pattern was discovered in the natural convection and air curtain cases. Mesh No. 3 and 4 produced the lowest difference. Hence, mesh No. 3 was judged to produce acceptable results.

5.2 Validation of natural convection simulations

In this section, a comparison is made between modelling (for both 2D and 3D) and experimental results for natural convection flow along the isothermal wall. A typical natural convection flow is observed in Figure 5.5 (3D simulation), where the boundary layer grows from the leading edge all the way up to the plate and exits from the top outlet boundary. No unstable flow behaviours were noticed. For comparison, the Nusselt number, a non-dimensional parameter, was calculated for each unit of the plate by using the heat transfer rate. The calculation technique was similar to that used in obtaining experimental data where the heat transfer rate was calculated at the middle of each unit. More details of the calculation method of dimensionless numbers can be found in Chapter 2.

Figure 5.6 reveals the Nusselt number as a function of the Rayleigh number for both simulations and experimental data. An additional set of experimental data obtained by Tsuji and Nagano (1988) is also included in this comparison. The comparison of 2D and 3D simulation results showed a very good match with no significant changes in the Nusselt number. Slight discrepancies appeared in the transition and turbulent regions. This is because the 2D simulation is unable to capture the spanwise roll structure of the flow from the plate. The spanwise roll structure has a significant effect on producing the instabilities in the thermal boundary layer to obtain transition and turbulent

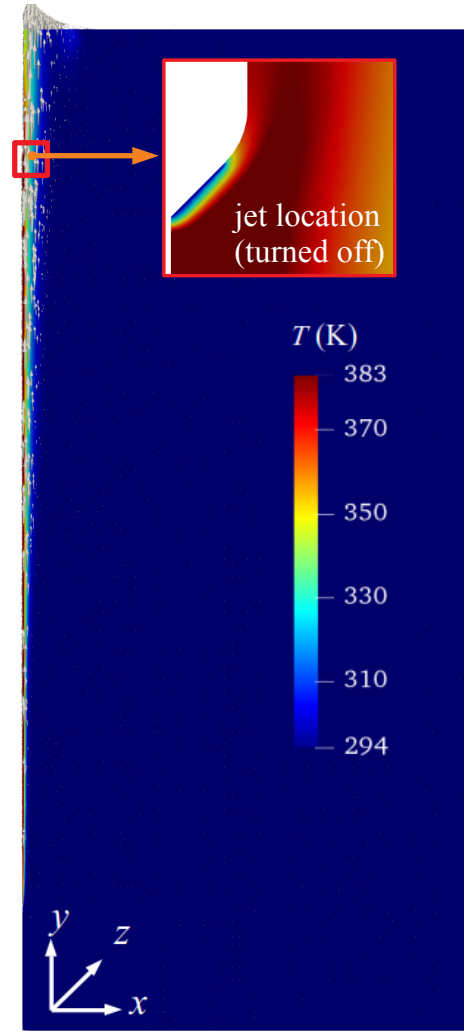


Figure 5.5: Temperature contour and velocity vector of the natural convection flow (2D view) developed along an isothermal wall when $T_w = 110^\circ\text{C}$ and $\alpha = 45^\circ$. Note the domain has been sliced along z -axis to show the velocity vector.

flows. Therefore, a lower Nusselt number value in 2D simulations is observed than in 3D in these regions. Although there are slight differences in results, the 2D simulation performed in this thesis still provides insightful findings considering the high computational cost associated with 3D simulations.

Both simulation results predicted well with the experimental data conducted by Tsuji and Nagano (1988). However, a noticeable difference is observed over the plate length between Abbasi-Shavazi (2019) experimental and 3D simula-

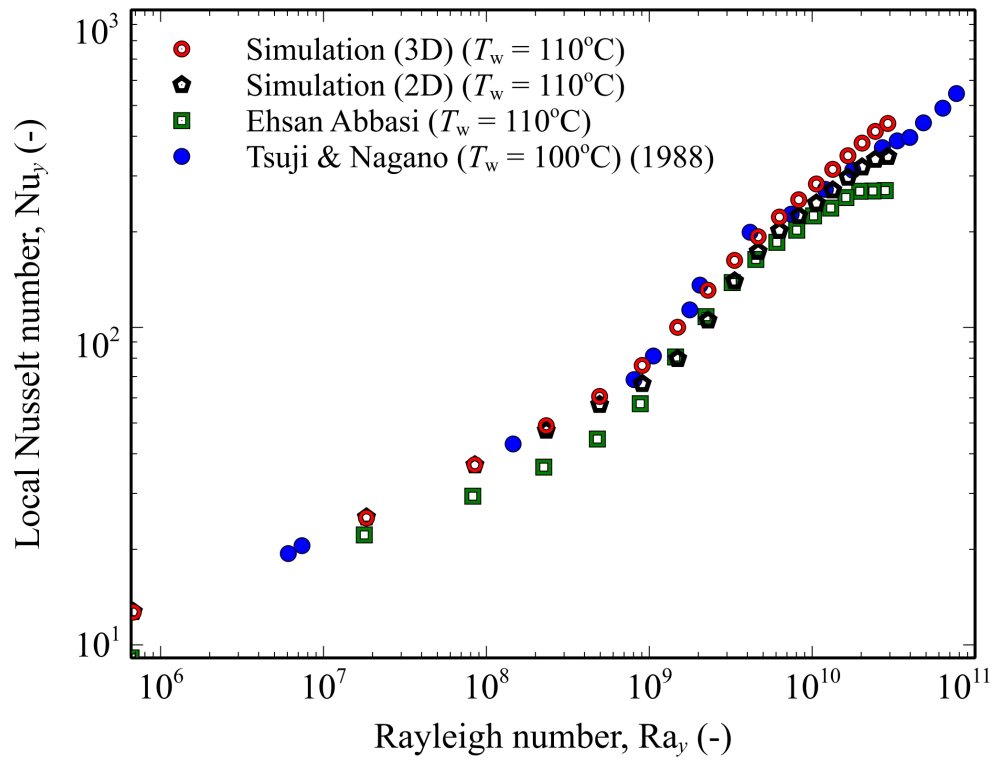


Figure 5.6: Nusselt number as a function of Rayleigh number for natural convection flow when $T_w = 110^\circ\text{C}$.

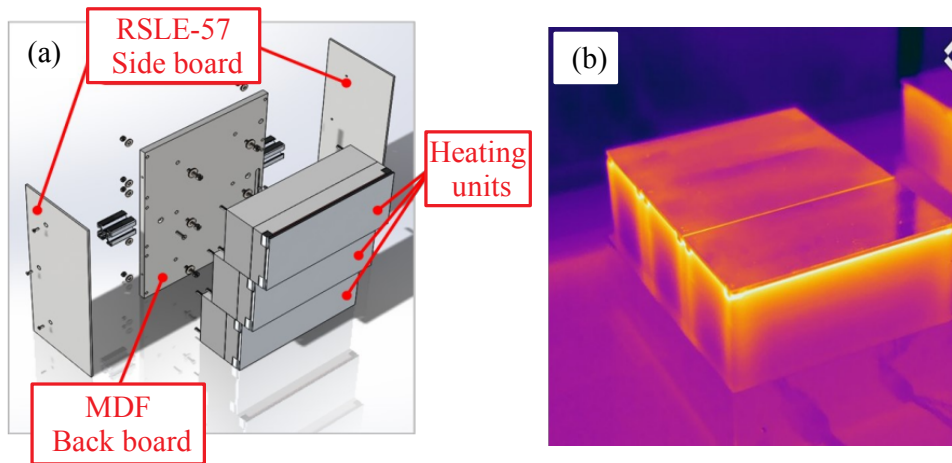


Figure 5.7: Heat losses through the edge of the plate unit, reported in the experimental setup. Image was taken from (Abbasi-Shavazi, 2019).

tion results. A maximum of 40% difference has appeared between simulation and experimental data (Figure 5.8) in the plate's top section, notably in the last

three units; the experimental analyses showed significantly lower convection loss than the simulation. This might occur due to the presence of an oblique air curtain face on top of the plate (similar to the inset of Figure 5.5). The air curtain outlet was 45° inclined above the plate. This oblique air curtain face may cause a blockage effect for natural convection flow, resulting in decreased convection loss from these units. This reduction of heat loss from the top unit is also seen in the modelling result; however, very minimal. Another substantial deviation of 30% between simulation and experiment is detected in Figure 5.8 near the transition region where laminar flow tends to become turbulent. It is seen that the onset of transition in the experiment is earlier than in the simulation. The experimental surface is made by combining 18 modular units. While blending, small edges between the units created irregularities on the surface, which tripped the flow early and caused a quicker transition. The plate used in the simulations in this thesis is flat and smooth because the different units of the plate are created by an OpenFOAM utility ‘topoSet’ (OpenFOAM, 2021). The ‘topoSet’ creates a patch by selecting the group of cell/face sets. It does not create any perturbations on the surface. The minor disturbances that are seeded in simulated flow only come about based on the RANS model.

Another evident difference between experimental and simulation results is that the heat transfer rate of every plate unit is lower in the experimental investigation compared to the modelling. In experimental settings, for a given temperature in the middle of each unit, some heat loss is expected through the edges of the units (as seen in Figure 5.7). Therefore, the perspex side-walls reach the buoyancy and flow in the boundary layers, causing a lower heat transfer rate from the units. Apart from that, other possible aspects, such as external disturbance and the different aspect ratios of the computational domain, might have contributed to the discrepancies. It should also be mentioned that the simulated domain was unable to model everything in the experimental setup, which may have led to these differences. Hence, having these variations mentioned above between experimental and simulation setups, this comparison was deemed acceptable for this validation study.

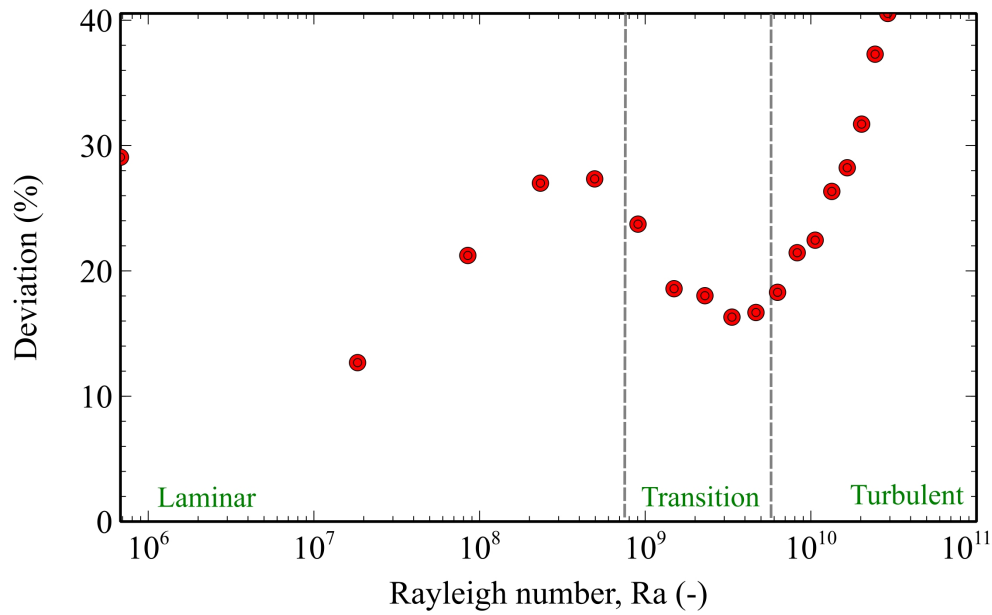


Figure 5.8: Deviation of simulated data from the experimental data produced by Abbasi-Shavazi (2019).

5.3 Air curtain effectiveness

This section illustrates how an air curtain interacts with the natural convection flow from a heated plate while the insulated sidewalls are attached and show its relative effectiveness in reducing convective heat losses from the plate. The effectiveness of an air curtain is defined as the ratio of heat loss reduction with an air curtain as compared to without an air curtain, i.e., the natural convection case. The definition of effectiveness has already been described in Chapter 2. Air curtain effectiveness for two separate jet angles ($\alpha = 25^\circ$ and 45°) are chosen for the comparison. The selection is based on the best-case and worst-case scenarios regarding heat loss reduction.

5.3.1 Jet angle, $\alpha = 45^\circ$

This jet angle is found to be the most effective setting to minimise convective heat loss from the plate (Mondal et al., 2020, 2022). Analogous to the experiment, effectiveness was computed for two different jet speeds of $U_{ac} = 3.45 \text{ m s}^{-1}$ and 5.75 m s^{-1} .

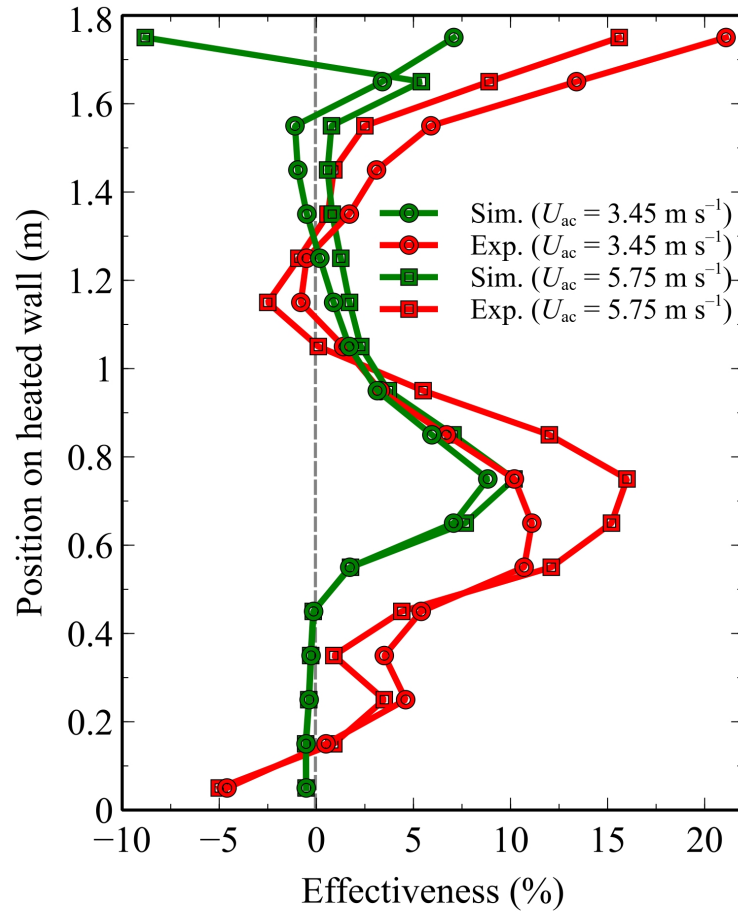


Figure 5.9: Comparison of effectiveness for air curtain speeds of $U_{ac} = 3.45 \text{ m s}^{-1}$ and 5.75 m s^{-1} .

Figure 5.9 illustrates the comparison of air curtain effectiveness between simulated and experimental data. Although the differences in effectiveness for both jet velocities over the length of the plate is significant, a common effectiveness trend is found for both simulation and experimental measurement. Two peak effectiveness regions were revealed for both configurations. The largest heat loss reduction appeared at the top of the plate generating a stagnation region. Figure 5.10 shows the associated flow behaviour when an air curtain interacts with buoyant flow arising from a heated plate. It can be seen that the air curtain successfully produced a stagnant zone of hot fluid below the jet outlet, resulting in reduced heat loss significantly.

Another significant reduction of heat loss has appeared near the transition

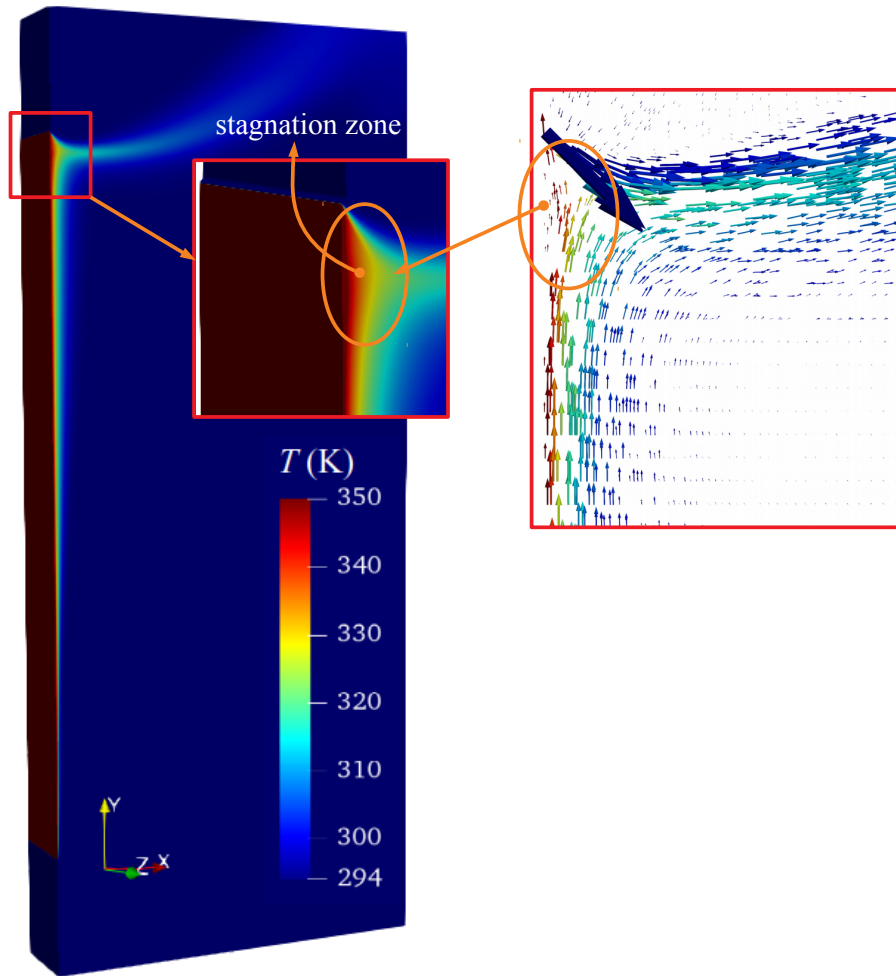


Figure 5.10: Temperature contour (left) and velocity vector (right) when an air curtain of speed $U_{ac} = 3.45 \text{ m s}^{-1}$ interacts with the natural convection of a heated wall. The temperature of the wall is scaled at 350 K for better viewing of the contour.

region, where laminar flow tends to become turbulent. The transition from laminar to turbulent flow has occurred at vertical positions between 0.5 m to 1.2 m of the plate for both simulation and experimental investigations. To understand what causes the large reduction of heat loss from the transition region in the simulated result, the heat transfer rate from the different plate units (transition region) is drawn in Figure 5.11 with and without an air curtain case. It is clearly visible that the jet significantly delayed the transition of laminar to turbulent boundary layer flow. Practically, the oblique air curtain creates an aerodynamic barrier on top of the plate for natural convective flow,

similar to that in Figure 5.10. This barrier opposes the natural convective boundary layer from being developed, and thereby, transitioning from laminar flow to becoming turbulent is slowed, which allows the thermal boundary layer to have remained laminar for much of the wall and reduces the convection coefficient. This effect can also be explained by the visualization of the heat flux distribution of the entire plate for with and without air curtain cases, shown in Figure 5.12. As can be seen, the transition location shifted towards the upper section of the wall when an air curtain was introduced.

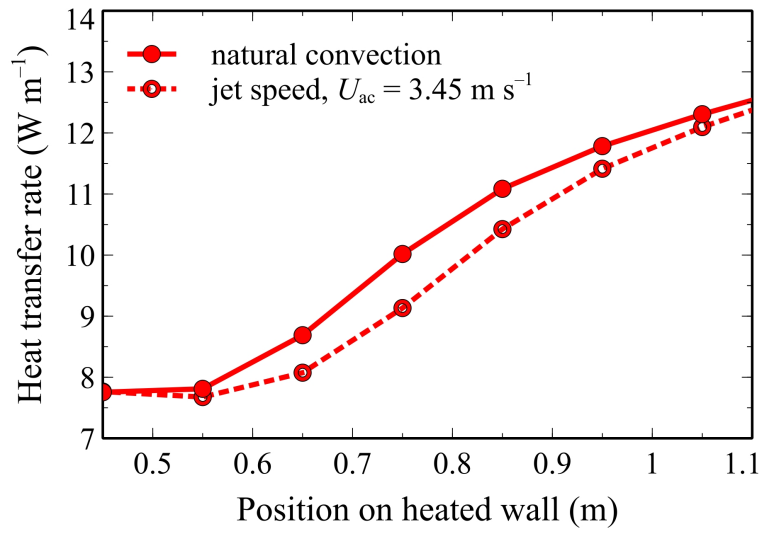


Figure 5.11: Heat transfer rate from different units of the plate in transition region for $U_{ac} = 3.45 \text{ m s}^{-1}$.

Some major discrepancies between modelling and experimental results have been revealed. First of all, the modelling result showed large negative effectiveness in the top unit for the jet speed of $U_{ac} = 5.75 \text{ m s}^{-1}$, which is completely opposite to the experimental finding. Evidence found in Chapter 3 that the modelling results always produced negative effectiveness at the top of the wall, particularly for the higher-speed jets. The higher speed jet completely reversed the buoyant plume on top of the plate, and heat transfer in this region increased instead of decreased (Mondal et al., 2022). The most significant differences in effectiveness appeared in $y \geq 1.6 \text{ m}$ and $y \leq 0.7 \text{ m}$ of the plate. This difference is associated with the comparison of natural convection cases. In Figure 5.6, it was seen that these two regions produced the largest discrepancy in experimental and simulation results. The heat loss in

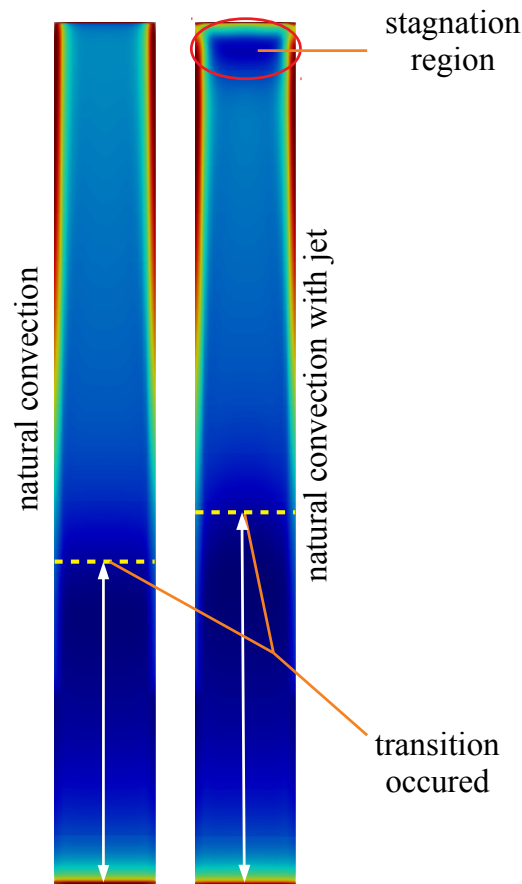


Figure 5.12: Heat flux distribution of the plate for the case of natural convection and natural convection with an air curtain of speed $U_{ac} = 3.45 \text{ m s}^{-1}$.

the experimental case was comparatively lower than the simulation in these two regions.

5.3.2 Jet angle, $\alpha = 25^\circ$

In some cases, the air curtain produced a significantly larger negative effectiveness at the top of the heated wall. This generally occurs due to the combination of a smaller jet angle and higher jet speed. This section discussed one of the examples of when a jet was 25° inclined and had the speed of $U_{ac} = 5.75 \text{ m s}^{-1}$.

Figure 5.13 compares the effectiveness of air curtain for both simulation and experiment. The comparison was only made for the top 10 units (from 0.9 m

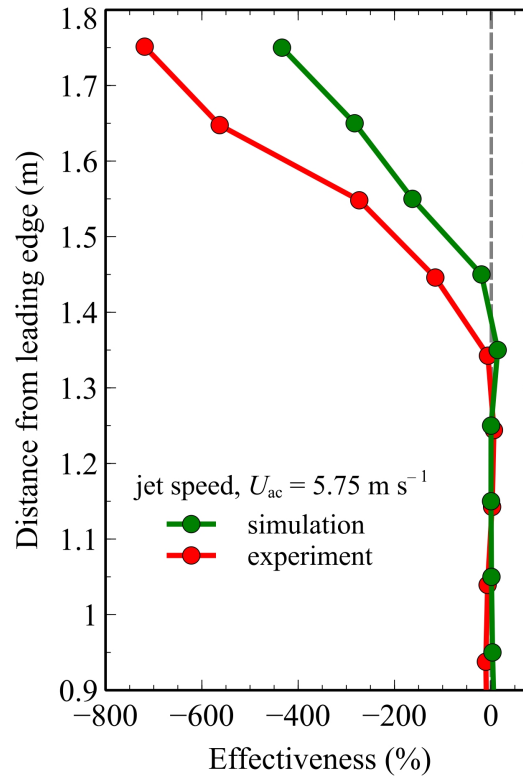


Figure 5.13: Effectiveness of air curtain for the combination of $\alpha = 25^\circ$ and $U_{ac} = 5.75 \text{ m s}^{-1}$.

to 1.8 m) of the plate where transition and turbulent flow are expected to have occurred. Also, the case with the air curtain yields significant negative effectiveness from the last 3 units of the plate. Both results showed a good agreement up to $y = 1.35 \text{ m}$ of the plate. Although a noticeable difference is found in the last 4 units of the plate, the trends of both results are the same. The sizeable negative effectiveness generally happened because of the flow attachment with the wall. A jet with a narrow inclination relative to the plate and a large speed always tends to attach to the plate from the top of the wall. This is because the jet entrains fluid between itself and the heated surface. The cold stream starts pushing the buoyant flow down the wall, creating a high-temperature gradient and producing large negative effectiveness. Note that the higher the jet speed, the quicker it curves into the heated surface. The narrow jet angles have larger initial horizontal velocity components so they drive a greater heat flux when completely curled in. The accompanying flow behaviour of temperature contour and velocity vector is shown in Figure 5.14.

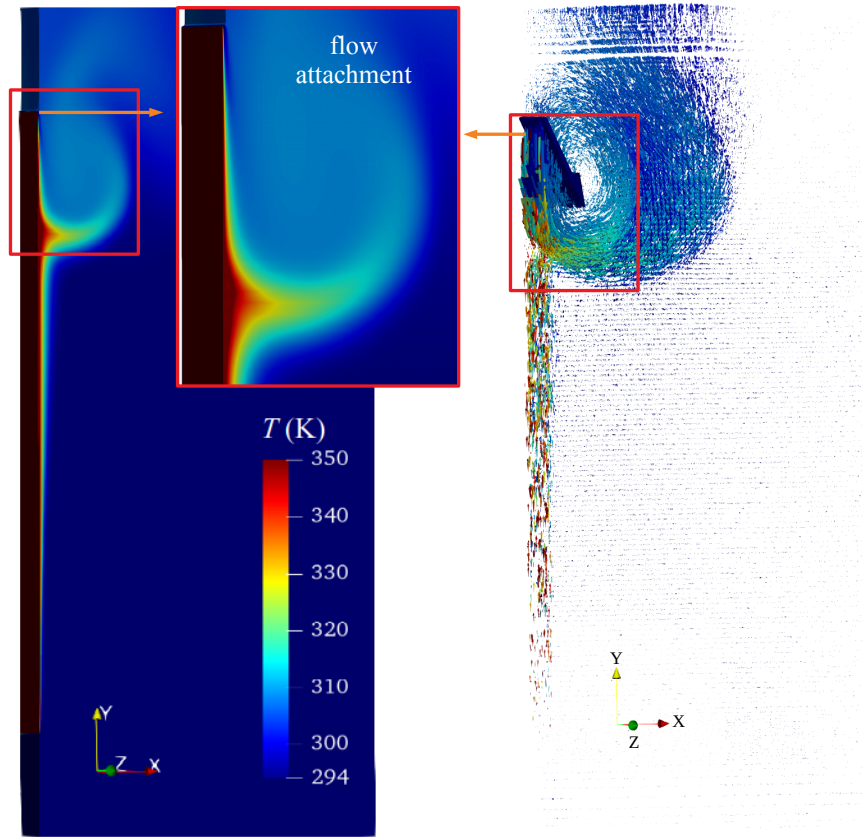


Figure 5.14: Temperature contour (left) and velocity vector (right) of the flow attachment case when $\alpha = 25^\circ$ and $U_{ac} = 5.75 \text{ m s}^{-1}$. Note the maximum temperature scaled at 350 K for better visualisation.

It is clearly visible that this large negative effectiveness is produced because of the wall-attached jet.

5.4 Conclusion

This study compared the results of 2D and 3D simulations and validated the 3D model against experimental data. The comparison of the natural convection flow revealed no significant discrepancies between the results obtained from the 2D and 3D simulations. The validation study considered both natural convection and natural convection with air curtain cases. In both cases, the results showed substantial variances over the plate length but generated a similar pattern. When the jet effectiveness was high, both experimental and

numerical studies identified a region of high effectiveness near the jet and a second region further down the plate, associated with changes in the point at which laminar to turbulent transition occurs. Similarly, the experimental and numerical studies both identified a large negative effectiveness region on top of the surface for the case when an air curtain was attached to the wall, showing good qualitative agreement between the two. Tentatively, any quantitative discrepancies are attributed to differences in experimental and modelling settings. To fully confirm this assessment, further experimental data is required, including measurements of parasitic losses from the units, the effect of the oblique jet face on top of the wall, and the surface roughness of the plate. Additional numerical studies could be applied to introduce irregularities on the surface and complete modelling of the air curtain device.

Conclusions and Recommendations

This chapter summarizes the key findings and contributions of the research conducted in this PhD work, highlighting its significance and potential implications. The research presented in this thesis has addressed the research questions and objectives and has made contributions to the field of CSP technology. The recommendations for future potential work for this project have also been discussed in this chapter.

6.1 Conclusions

The results of this research demonstrate for the first time the effectiveness of an air curtain in minimizing convective heat losses from an external heated plate, which is a crucial component of an external CSP receiver surface. This study employed a detailed computational fluid dynamics (CFD) model in OpenFOAM (OpenFOAM, 2021) to design an external receiver surface with the aid of a 1.8 m high isothermal wall and an air curtain generated by a planar air jet.

The validation of the isothermal heated wall and turbulent planar jet against existing literature preceded a parametric analysis on the heat transfer from the isothermal plate with an air curtain, varying four parameters: jet speed, jet angle, plate temperature, and plate inclination. The results confirm that an air curtain effectively reduces local convective heat loss from the heated wall. At the 1.8 m scale with a surface temperature of 110 °C, an air curtain provides up to 31.2% reduction in local heat loss by creating a stagnation zone beneath the jet outlet for a vertical wall with a jet speed and angle of 2.5 m s^{-1}

and 45° , respectively. Another heat loss reduction region occurs due to the jet interfering with the laminar-to-turbulent transition, speculated to be due to the delay of transition. The air curtain's effectiveness in the stagnation region decreased with increasing jet speeds, while the effectiveness increased near the laminar-to-turbulent transition region with increasing jet speed. Smaller air curtain angles relative to the wall resulted in lower effectiveness, and a 45° air curtain on a vertical wall provided performance benefits similar to inclining a wall from the vertical. Higher wall temperatures were associated with better effectiveness near the jet outlet, particularly in the stagnation region, while lower wall temperatures produced higher effectiveness further from the jet.

Furthermore, different air curtain parameters, including jet speed, angle, thickness, and gap, were optimized to increase the reduction of convection loss from a vertical external heated surface, focusing on global effectiveness. An optimum jet with a speed of $U_{ac} = 5.5 \text{ m s}^{-1}$, angle of $\alpha = 45^\circ$, thickness of $b = 30 \text{ mm}$, and gap of $G = 30 \text{ mm}$ achieved a maximum global effectiveness of more than 18%. The air curtain was most beneficial in reducing heat loss in two regions of the surface: at the top of the heated wall by creating a local stagnation zone and over the intermediate section of the wall by delaying the transition from laminar to turbulent boundary layer flow. The air curtain tended to increase local heat loss at the bottom of the wall, below 0.6 m. The effect of jet outlet temperatures on effectiveness was also examined for the optimum case; the effectiveness gradually decreased with increasing jet outlet temperature.

Finally, the simulation results for both the 2D and 3D models were compared, but no significant changes were observed. The 3D CFD model was further validated against an experimental setup conducted by a PhD student Abbasi-Shavazi (2019) at the Australian National University. The air curtain's effectiveness was found to be more beneficial in turbulent regions by generating a stagnation region and near transition regions by delaying the transition in laminar-turbulent boundary layer flow. The effectiveness of the air curtain decreased significantly when it was attached to the top of the wall. The subsequent flow behaviours between the interaction of buoyant plumes and air curtain flows were discussed for modelling results, which lacked in the

experimental research.

Overall, the findings of this thesis have contributed to the understanding of how convection loss from an external CSP receiver can be tackled by implementing an air curtain. Thus, these results serve as a strong motivation for further investigation of air curtain design, including experimental implementation of airflow control strategies for application in solar-thermal technologies and other fields where control of convective heat loss from external surfaces is needed.

6.2 Recommendations

Due to the limited scope of a single PhD thesis, not all areas of receiver design verification were covered in this study. Further research is recommended before the air curtain may potentially be used on an external receiver design for real-world settings. The following is a suggested approach.

- This study analyzed the performance of the air curtain on a small-scale receiver design. However, further testing is necessary to evaluate the air curtain's performance on a large-scale receiver design using non-dimensional parameters obtained in this study.
- The receiver is likely expected to operate in unsteady winds. Nevertheless, this study did not take into account the effect of wind speed on air curtain performance, which also needs to be performed before doing any experimental or large-scale investigations.
- The surface temperature of an external receiver varies in real-world settings as it depends on the irradiation profile generated by the heliostats, aiming strategy, etc. Therefore, it is necessary to justify the air curtain performance for a variable plate temperature.
- The performance of the air curtain is shown on a single surface of a receiver panel for this study. The air curtain implementation needs to be performed over a complete receiver design.
- This thesis optimises the gap between the heated wall and jet in the vertical direction; however, the investigations of the jet gap towards the

horizontal direction need to be checked.

- This thesis used a RANS (Reynolds-averaged Navier-Stokes) model formulation with the widely used two-equation eddy viscosity turbulence closure $k-\omega$ SST (Shear Stress Transport) approach to perform an extensive parametric and optimisation study in a feasible manner; but more computationally expensive models, including direct numerical simulations (DNS) and large eddy simulation (LES), can be used to get more accurate flow prediction.

References

- Turbulence intensity. https://www.cfd-online.com/Wiki/Turbulence_intensity#Estimating_the_turbulence_intensity. Accessed 10 May 2023.
- ABBASI-SHAVAZI, 2019. Experimental investigation of external convection from scale-model solar thermal receivers, phd thesis (draft). The Australian National University.
- ABBASI-SHAVAZI, E.; TORRES, J. F.; HUGHES, G.; AND PYE, J., 2020. Experimental correlation of natural convection losses from a scale-model solar cavity receiver with non-isothermal surface temperature distribution. *Solar Energy*, 198 (2020), 355–375.
- AHMADI, M. H.; GHAZVINI, M.; SADEGHZADEH, M.; NAZARI, M. A.; KUMAR, R.; NAEIMI, A.; AND MING, T., 2018. Solar power technology for electricity generation: A critical review. *Energy Science & Engineering*, 6, 5 (2018), 340–361.
- AHRENS, J.; GEVECI, B.; AND LAW, C., 2005. Paraview: An end-user tool for large data visualization. *The visualization handbook*, 717, 8 (2005).
- ALIPOURTARZANAGH, E.; CHINNICI, A.; NATHAN, G. J.; AND DALLY, B. D., 2020a. Experimental insights into the mechanism of heat losses from a cylindrical solar cavity receiver equipped with an air curtain. *Solar Energy*, 201 (2020), 314–322.
- ALIPOURTARZANAGH, E.; CHINNICI, A.; NATHAN, G. J.; AND DALLY, B. D., 2020b. Experimental investigation on the influence of an air curtain on the convective heat losses from solar cavity receivers under windy condition. In *AIP Conference Proceedings*, vol. 2303.

-
- ALIPOURTARZANAGH, E.; CHINNICI, A.; NATHAN, G. J.; AND DALLY, B. D., 2020c. Impact of flow blowing and suction strategies on the establishment of an aerodynamic barrier for solar cavity receivers. *Applied Thermal Engineering*, 180 (2020), 115841.
- ASSELINEAU, C.-A.; LOGIE, W.; PYE, J.; VENN, F.; WANG, S.; AND COVENTRY, J., 2022. Design of a heliostat field and liquid sodium cylindrical receiver for the gen3 liquids pathway. In *AIP Conference Proceedings*, vol. 2445, 110001. AIP Publishing LLC.
- BAHAROON, D. A.; RAHMAN, H. A.; OMAR, W. Z. W.; AND FADHL, S. O., 2015. Historical development of concentrating solar power technologies to generate clean electricity efficiently—a review. *Renewable and Sustainable Energy Reviews*, 41 (2015), 996–1027.
- BANK, W., 2021. Concentrating solar power: Clean power on demand 24/7. (2021). <https://www.greenindustryplatform.org/research/concentrating-solar-power-clean-power-demand-247>.
- BARAKOS, G.; MITSOULIS, E.; AND ASSIMACOPOULOS, D. O., 1994. Natural convection flow in a square cavity revisited: laminar and turbulent models with wall functions. *International journal for numerical methods in fluids*, 18, 7 (1994), 695–719.
- BARLEV, D.; VIDU, R.; AND STROEVE, P., 2011. Innovation in concentrated solar power. *Solar energy materials and solar cells*, 95, 10 (2011), 2703–2725.
- BISHOP, E. H.; MACK, L. R.; AND SCANLAN, J. A., 1966. Heat transfer by natural convection between concentric spheres. *International Journal of Heat and Mass Transfer*, 9, 7 (1966), 649–662.
- BISOI, M.; DAS, M. K.; ROY, S.; AND PATEL, D. K., 2017. Large eddy simulation of three-dimensional plane turbulent free jet flow. *European Journal of Mechanics-B/Fluids*, 65 (2017), 423–439.
- BODENSCHATZ, E.; PESCH, W.; AND AHLERS, G., 2000. Recent developments in rayleigh-bernard convection. *Annual review of fluid mechanics*, 32, 1 (2000), 709–778.

-
- BOEREMA, N.; MORRISON, G.; TAYLOR, R.; AND ROSENGARTEN, G., 2013. High temperature solar thermal central-receiver billboard design. *Solar Energy*, 97 (2013), 356–368.
- BRADSTOCK, R.; CLARKE, H.; COLLINS, L.; CLARKE, M.; NOLAN, R. H.; AND PENMAN, T. A staggering 1.8 million hectares burned in “high-severity” fires during australia’s black summer. *The Conversation*. Retrieved from <https://theconversation.com/a-staggering-1-8-million-hectares>, Accessed 04 May 2023.
- CHEESEWRIGHT, R., 1968. Turbulent natural convection from a vertical plane surface. *Journal of Heat Transfer*, 90, 1 (1968), 1–6.
- CHEN, T. S. AND YUH, C. F., 1980. Combined heat and mass transfer in natural convection along a vertical cylinder. *International Journal of Heat and Mass Transfer*, 23, 4 (1980), 451–461.
- CHEN, Y. G. AND YUAN, X. L., 2005. Simulation of a cavity insulated by a vertical single band cold air curtain. *Energy Conversion and Management*, 46, 11-12 (2005), 1745–1756.
- CHOI, S. K. AND KIM, S. O., 2012. Turbulence modeling of natural convection in enclosures: A review. *Journal of Mechanical Science and Technology*, 26, 1 (2012), 283–297.
- CHURCHILL, S. W. AND CHU, H. H. S., 1975. Correlating equations for laminar and turbulent free convection from a vertical plate. *International journal of heat and mass transfer*, 18, 11 (1975), 1323–1329.
- COSTA, J. J.; OLIVEIRA, L. A.; AND SILVA, M. C. G., 2006. Energy savings by aerodynamic sealing with a downward-blowing plane air curtain—a numerical approach. *Energy and buildings*, 38, 10 (2006), 1182–1193.
- DENNIS, D. S. L. AND KRAABEL, J. S., 1984. Estimating convective energy losses from solar central receivers. Technical report, Sandia National Lab.(SNL-CA), Livermore, CA (United States).

-
- DEO, R. C.; NATHAN, G. J.; AND MI, J., 2007. Comparison of turbulent jets issuing from rectangular nozzles with and without sidewalls. *Experimental Thermal and Fluid Science*, 32, 2 (2007), 596–606.
- DESRAYAUD, G.; CHÉNIER, E.; JOULIN, A.; BASTIDE, A.; BRANGEON, B.; CALTAGIRONE, J. P.; CHERIF, Y.; EYMARD, R.; GARNIER, C.; GIROUX-JULIEN, S.; ET AL., 2013. Benchmark solutions for natural convection flows in vertical channels submitted to different open boundary conditions. *International journal of thermal sciences*, 72 (2013), 18–33.
- FAN, Y.; ZHAO, Y.; TORRES, J. F.; XU, F.; LEI, C.; LI, Y.; AND CARMELIET, J., 2021. Natural convection over vertical and horizontal heated flat surfaces: A review of recent progress focusing on underpinnings and implications for heat transfer and environmental applications. *Physics of Fluids*, 33, 10 (2021), 101301.
- FANG, J.; TU, N.; TORRES, J. F.; WEI, J.; AND PYE, J., 2019. Numerical investigation of the natural convective heat loss of a solar central cavity receiver with air curtain. *Applied Thermal Engineering*, 152 (2019), 147–159.
- FERZIGER, J. H. AND PERIC, M., 2002. *Computational methods for fluid flow*. Springer, third edition edn.
- FILKOV, A. I.; NGO, T.; MATTHEWS, S.; TELFER, S.; AND PENMAN, T. D., 2020. Impact of australia’s catastrophic 2019/20 bushfire season on communities and environment. retrospective analysis and current trends. *Journal of Safety Science and Resilience*, 1, 1 (2020), 44–56.
- FLESCH, R.; GROBBEL, J.; STADLER, H.; UHLIG, R.; AND HOFFSCHMIDT, B., 2016. Reducing the convective losses of cavity receivers. In *AIP Conference Proceedings*, vol. 1734, 030014.
- FRANK, D. AND LINDEN, P. F., 2014. The effectiveness of an air curtain in the doorway of a ventilated building. *Journal of fluid mechanics*, 756 (2014), 130–164.
- GANESAN, P. AND RANI, H. P., 1998. Transient natural convection along vertical cylinder with heat and mass transfer. *Heat and Mass transfer*, 33, 5 (1998), 449–455.

-
- GEBHART, B., 1962. Effects of viscous dissipation in natural convection. *Journal of fluid Mechanics*, 14, 2 (1962), 225–232.
- GEBHART, B., 1973. Instability, transition, and turbulence in buoyancy-induced flows. *Annual review of fluid mechanics*, 5, 1 (1973), 213–246.
- GEUZAIN, C. AND REMACLE, J. F., 2009. Gmsh: A 3-d finite element mesh generator with built-in pre-and post-processing facilities. *International journal for numerical methods in engineering*, 79, 11 (2009), 1309–1331.
- GOUBRAN, S.; QI, D.; SALEH, W. F.; WANG, L. L.; AND ZMEUREANU, R., 2016. Experimental study on the flow characteristics of air curtains at building entrances. *Building and Environment*, 105 (2016), 225–235.
- GUPTA, G. S.; SARKAR, S.; CHYCHKO, A.; TENG, L.; NZOTTA, M.; AND SEETHARAMAN, S., 2014. Process concept for scaling-up and plant studies. In *Treatise on process metallurgy*, 1100–1144. Elsevier.
- GUYONNAUD, L.; SOLLIEC, C.; DE, V. M. D.; AND REY, C., 2000. Design of air curtains used for area confinement in tunnels. *Experiments in Fluids*, 28, 4 (2000), 377–384.
- HAYES, C. AND STOECKER, W. F., 1969. Design data for air curtains. *Ashrae Transactions*, 75, 2 (1969), 168–180.
- HENNESSY, K.; LAWRENCE, J.; AND MACKEY, B., 2022. Ipcc sixth assessment report (ar6): Climate change 2022-impacts, adaptation and vulnerability: Regional factsheet australasia. (2022).
- HIRSCH, T. AND KHENISSI, A., 2014. A systematic comparison on power block efficiencies for csp plants with direct steam generation. *Energy Procedia*, 49 (2014), 1165–1176.
- HO, C. K. AND IVERSON, B. D., 2014. Review of high-temperature central receiver designs for concentrating solar power. *Renewable and Sustainable Energy Reviews*, 29 (2014), 835–846.
- HOLMAN, J. P., 2010. *Heat transfer*. McGraw Hill Higher Education, tenth edn.

-
- HUGHES, G.; PYE, J.; KAUFER, M.; ABBASI-SHAVAZI, E.; ZHANG, J.; MCINTOSH, A.; AND LINDLEY, T., 2016. Reduction of convective losses in solar cavity receivers. In *AIP Conference Proceedings*, vol. 1734, 030023.
- IRENA, 2018. Renewable power generation costs in 2018. techreport, International Renewable Energy Agency. <https://www.irena.org/publications/2019/May/Renewable-power-generation-costs-in-2018>.
- JIA, H. AND GOGOS, G., 1996. Laminar natural convection heat transfer from isothermal spheres. *International Journal of Heat and Mass Transfer*, 39, 8 (1996), 1603–1615.
- KALENDAR, A. A. AND OOSTHUIZEN, P. H., 2011. Numerical and experimental studies of natural convective heat transfer from vertical and inclined narrow isothermal flat plates. *Heat and Mass Transfer*, 47, 9 (2011), 1181–1195.
- KALOGIROU, S. A., 2004. Solar thermal collectors and applications. *Progress in energy and combustion science*, 30, 3 (2004), 231–295.
- KALOGIROU, S. A., 2013. *Solar energy engineering: processes and systems*. Academic Press.
- KELLY, B. D. AND ROBERT, R. L., 1990. Solar receiver having wind loss protection. US Patent 4,913,129.
- KROTHAPALLI, A. AND GRESKA, B., 2011. Concentrated solar thermal power. *Handbook of Climate Change Mitigation and Adaptation*, 2 (2011), 1503–1536.
- KURUP, P. AND TURCHI, C., 2015. Initial investigation into the potential of CSP industrial process heat for the southwest united states. Technical report. doi:10.2172/1227710. <https://www.osti.gov/biblio/1227710>.
- LEE, H. R.; CHEN, T. S.; AND ARMALY, B. F., 1988. Natural Convection Along Slender Vertical Cylinders With Variable Surface Temperature. *Journal of Heat Transfer*, 110, 1 (1988), 103–108.
- LILLIESTAM, J.; LABORDENA, M.; PATT, A.; AND PFENNINGER, S., 2017. Empirically observed learning rates for concentrating solar power and their responses to regime change. *Nature Energy*, 2, 7 (2017), 1–6.

-
- LOVEGROVE, K.; FERNANDEZ, R. B. D.; AND URKALAN, K., 2018. Concentrating solar thermal technology status: Informing a csp roadmap for australia. Technical report, IT P Thermal Pty Limited.
- LOVEGROVE, K. AND STEIN, W., 2021. Introduction to concentrating solar power technology. In *Concentrating Solar Power Technology*, 3–17. Elsevier.
- MARCONDES, F. AND MALISKA, C. R., 1999. Treatment of the inlet boundary conditions in natural-convection flows in open-ended channels. *Numerical Heat Transfer: Part B: Fundamentals*, 35, 3 (1999), 317–345.
- MCINTOSH, A.; HUGHES, G.; AND PYE, J., 2014. Use of an air curtain to reduce heat loss from an inclined open-ended cavity. *Proceedings of the 19th Australasian Fluid Mechanics Conference, Melbourne, Australia*, (Nov. 2014).
- MEHOS, M.; TURCHI, C.; VIDAL, J.; WAGNER, M.; MA, Z.; HO, C.; KOLB, W.; ANDRAKA, C.; AND KRUIZENGA, A., 2017. Concentrating solar power gen3 demonstration roadmap. Technical report, National Renewable Energy Lab.(NREL), Golden, CO (United States).
- MENTER, F. R.; KUNTZ, M.; AND LANGTRY, R., 2003. Ten years of industrial experience with the sst turbulence model. *Turbulence, heat and mass transfer*, 4, 1 (2003), 625–632.
- MIOZZI, M.; LALLI, F.; AND ROMANO, G., 2010. Experimental investigation of a free-surface turbulent jet with coanda effect. *Experiments in Fluids*, 49 (2010), 341–353.
- MONDAL, R.; TORRES, J. F.; HUGHES, G.; AND PYE, J., 2020. Analysis of air curtains for natural convection heat-loss mitigation. *Proceedings of the 22nd Australasian Fluid Mechanics Conference, Queensland, Australia*, (Dec. 2020).
- MONDAL, R.; TORRES, J. F.; HUGHES, G.; AND PYE, J., 2022. Air curtains for reduction of natural convection heat loss from a heated plate: A numerical investigation. *International Journal of Heat and Mass Transfer*, 189 (2022), 122709.
- MOTYKA, M. W. AND JONES, D., 2022. Global electricity mid-year insights. techreport, EMBER. <https://ember-climate.org/>

-
- insights/research/global-electricity-mid-year-insights-2022/
#supporting-material-downloads.
- NAVAZ, H. K.; HENDERSON, B. S.; FARAMARZI, R.; POURMOVAHED, A.; AND TAUGWALDER, F., 2005. Jet entrainment rate in air curtain of open refrigerated display cases. *International Journal of Refrigeration*, 28, 2 (2005), 267–275.
- NOCK, I.; LOGIE, W.; COVENTRY, J.; PYE, J.; ET AL., 2016. A computational evaluation of convective losses from bladed solar thermal receivers. In *Proceedings of the 2016 Asia-Pacific Solar Research Conference, Australian PV Institute*.
- OGASAWARA, N.; TORRES, J. F.; KANDA, Y.; KOGAWA, T.; AND KOMIYA, A., 2021. Resonance-driven heat transfer enhancement in a natural convection boundary layer perturbed by a moderate impinging jet. *Physical Review Fluids*, 6, 6 (2021), L061501.
- OLIVEIRA, L. A.; COSTA, J. J.; CARVALHO, M. G.; GERHARDT, H. J.; AND KRAMER, C., 1991. On aerodynamic sealing for industrial applications. *Journal of Wind Engineering and Industrial Aerodynamics*, 37, 3 (1991), 255–268.
- OPENFOAM, 2021. OpenFOAM user guide. Ninth edn. <https://doc.cfd.direct/openfoam/user-guide-v9/index>.
- OSAWA, H., 2015. Fan basics and selection criteria (how to use). *SANYO DENKI Corporation, Ltd., Tech. Rep*, 40 (2015).
- PAITOONSURIKARN, S.; LOVEGROVE, K.; HUGHES, G.; AND PYE, J., 2011. Numerical investigation of natural convection loss from cavity receivers in solar dish applications. *Journal of Solar Energy Engineering*, 133, 2 (2011).
- PIERRE, D. A., 1986. Optimization theory with applications. Courier Corporation.
- PYE, J.; COVENTRY, J.; HO, C.; YELLOWHAIR, J.; NOCK, I.; WANG, Y.; ABBASI, E.; CHRISTIAN, J.; ORTEGA, J.; AND HUGHES, G., 2017. Optical and thermal performance of bladed receivers. In *AIP Conference Proceedings*, vol. 1850, 030040. AIP Publishing LLC.

-
- RAO, A.; MINELLI, G.; BASARA, B.; AND KRAJNOVIĆ, S., 2018. On the two flow states in the wake of a hatchback ahmed body. *Journal of Wind Engineering and Industrial Aerodynamics*, 173 (2018), 262–278.
- RICHTER, C.; TESKE, S.; AND SHORT, R., 2009. Concentrating solar power global outlook 09 - why renewable energy is hot. Technical report, Greenpeace International, SolarPACES, ESTELA, Amsterdam.
- ROGELJ, J.; ELZEN, M. D.; HÖHNE, N.; FRANSEN, T.; FEKETE, H.; WINKLER, H.; SCHAEFFER, R.; SHA, F.; RIAHI, K.; AND MEINSHAUSEN, M., 2016. Paris agreement climate proposals need a boost to keep warming well below 2 c. *Nature*, 534, 7609 (2016), 631–639.
- SÁNCHEZ, M. R., 2015. *On the design of solar external receivers*. Ph.D. thesis, Ph.D. Thesis, University Carlos Madrid, Madrid, Spain.
- SHAVAZI, E. A.; TORRES, J.; HUGHES, G.; AND PYE, J., 2018. Convection heat transfer from an inclined narrow flat plate with uniform flux boundary conditions. *Proceedings of the 21st Australasian Fluid Mechanics Conference, Adelaide, Australia*, (Dec. 2018).
- SIEBERS, D. L. AND KRAABEL, J. S., 1984. Estimating convective energy losses from solar central receivers. Technical report, Sandia National Lab.(SNL-CA), Livermore, CA (United States).
- SIRÉN, K., 2003. Technical dimensioning of a vertically upwards blowing air curtain—part i. *Energy and Buildings*, 35, 7 (2003), 681–695.
- SONG, Z.; LIU, J.; AND YANG, H., 2021. Air pollution and soiling implications for solar photovoltaic power generation: A comprehensive review. *Applied Energy*, 298 (2021), 117247. doi:10.1016/j.apenergy.2021.117247.
- SUN, J.; TSAMOS, K. M.; AND TASSOU, S. A., 2017. CFD comparisons of open-type refrigerated display cabinets with/without air guiding strips. *Energy Procedia*, 123 (2017), 54–61.
- TAUMOEFOLOU, T., 2004. *Experimental investigation of convection loss from a model solar concentrator cavity receiver*. Master's thesis, The Australian National University.

- TAUSSIG, R. T., 1984. Aerowindows for central solar receivers. *American Society of Mechanical Engineers (ASME)*, 12p (1984), 9–14.
- THIRUGNANASAMBANDAM, M.; INIYAN, S.; AND GOIC, R., 2010. A review of solar thermal technologies. *Renewable and sustainable energy reviews*, 14, 1 (2010), 312–322.
- TORRES, J. F.; GHANADI, F.; NOCK, I.; ARJOMANDI, M.; AND PYE, J., 2018. Mixed convection around a tilted cuboid with an isothermal sidewall at moderate Reynolds numbers. *International Journal of Heat and Mass Transfer*, 119 (2018), 418–432.
- TORRES, J. F.; GHANADI, F.; WANG, Y.; ARJOMANDI, M.; AND PYE, J., 2020. Mixed convection and radiation from an isothermal bladed structure. *International Journal of Heat and Mass Transfer*, 147 (2020), 118906.
- TSUJI, T. AND NAGANO, Y., 1988. Characteristics of a turbulent natural convection boundary layer along a vertical flat plate. *International journal of heat and mass transfer*, 31, 8 (1988), 1723–1734.
- VAALMA, C.; BUCHHOLZ, D.; WEIL, M.; AND PASSERINI, S., 2018. A cost and resource analysis of sodium-ion batteries. *Nature reviews materials*, 3, 4 (2018), 1–11.
- VAN, N. Q. AND HOWELL, R. H., 1976. Influence of initial turbulence intensity on the development of plane air-curtain jets. *Transaction of the ASHRAE*, (1976).
- VAN BELLEGHEM, M.; VERHAEGHE, G.; T'JOEN, C.; HUISSEUNE, H.; DE JAEGER, P.; AND DE PAEPE, M., 2012. Heat transfer through vertically downward-blowing single-jet air curtains for cold rooms. *Heat transfer engineering*, 33, 14 (2012), 1196–1206.
- VERSTEEG, H. K. AND MALALASEKERA, W., 2007. An introduction to computational fluid dynamics: the finite volume method. Pearson education.
- WANG, Q.; YAO, Y.; HU, M.; CAO, J.; QIU, Y.; AND YANG, H., 2021. An air curtain surrounding the solar tower receiver for effective reduction of convective heat loss. *Sustainable Cities and Society*, (2021), 103007.

-
- WANG, Z., 2021. Blue book of china's concentrating solar power industry 2021. (2021).
- WELLER, H. G.; TABOR, G.; JASAK, H.; AND FUREBY, C., 1998. A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in physics*, 12, 6 (1998), 620–631.
- YANG, S.; WANG, J.; LUND, P. D.; WANG, S.; AND JIANG, C., 2018. Reducing convective heat losses in solar dish cavity receivers through a modified air-curtain system. *Solar Energy*, 166 (2018), 50–58.
- ZHANG, H. L.; BAEYENS, J.; DEGRÈVE, J.; AND CACÈRES, G., 2013. Concentrated solar power plants: Review and design methodology. *Renewable and sustainable energy reviews*, 22 (2013), 466–481.
- ZHANG, J. J.; PYE, J.; AND HUGHES, G. O., 2015. Active air flow control to reduce cavity receiver heat loss. In *Energy Sustainability*, V001T05A023. American Society of Mechanical Engineers (ASME).
- ZHAO, Y.; LEI, C.; AND PATTERSON, J. C., 2013. Resonance of the thermal boundary layer adjacent to an isothermally heated vertical surface. *Journal of Fluid Mechanics*, 724 (2013), 305–336.
- ZHAO, Y.; LEI, C.; AND PATTERSON, J. C., 2017. The k-type and h-type transitions of natural convection boundary layers. *Journal of Fluid Mechanics*, 824 (2017), 352.
- ZHAO, Y.; ZHAO, P.; LIU, Y.; XU, Y.; AND TORRES, J. F., 2019. On the selection of perturbations for thermal boundary layer control. *Physics of Fluids*, 31, 10 (2019), 104102.