

University of Southern Queensland
Faculty of Health, Engineering and Sciences

Effect of Thermal Boundary Layer Below Water Surfaces on Evaporation

A dissertation submitted by

Alex King

In fulfilment of the requirements of

ENG4111 and 4112 Research Project

Towards the degree of

Bachelor of Engineering (Honours) (Civil)

Submitted

14/10/2020

Disclaimer

University of Southern Queensland

Faculty of Health, Engineering and Sciences

ENG4111 & ENG4112 Research Project

Limitations of Use

The Council of the University of Southern Queensland, its Faculty of Health, Engineering and Sciences, and the staff of the University of Southern Queensland, do not accept any responsibility for the truth, accuracy or completeness of material contained within or associated with this dissertation.

Persons using all or any part of this material do so at their own risk, and not at the risk of the Council of the University of Southern Queensland, its Faculty of Health, Engineering and Sciences or the staff of the University of Southern Queensland.

This dissertation reports an educational exercise and has no purpose or validity beyond this exercise. The sole purpose of the course pair entitles “Research Project” is to contribute to the overall education within the student’s chosen degree program. This document, the associated hardware, software, drawings, and any other material set out in the associated appendices should not be used for any other purpose: if they are so used, it is entirely at the risk of the user.

Certification

I certify that the ideas, designs and experimental work, results, analyses and conclusions set out in this dissertation are entirely my own effort, except where otherwise indicated and acknowledged.

I further certify that the work is original and has not been previously submitted for assessment in any other course or institution, except where specifically stated.

Alex King

Student Number



Acknowledgment

I'd like to thank my project supervisor Andrew Wandel for his guidance and support through the preparation of this dissertation. His technical knowledge, patience and understanding aided greatly in this academic endeavour.

I'd also like to present a special acknowledgment to my son. Thomas was born on the 8/03/2020, the start of this project and has given me the emotional boost through this challenging task.

Nomenclature

E_0 = Evaporation Rate (expressed in terms of $\text{kg}/\text{m}^2\cdot\text{h}$)

E = Evaporation (in terms of kg)

p = ratio of actual daily sunlight hours to annual mean daily sunlight hours (expressed in %)

T = Temperature (deg. C)

n/N = The ratio of actual to possible sunshine hours

U_d = Daytime wind speed

RH_{\min} = is the minimum daily relative humidity (expressed in %)

$p_r - p_w$ = difference of density of air between external temperature and surface temperature (expressed in terms of kg/m^3)

$w_w - w_r$ = difference of specific humidity of air between external temperature and water surface temperature (expressed in terms of kg/kg)

CFD = Computational Fluid Dynamics

TBL = Thermal Boundary Layer

Abstract

The thermal boundary layer is a thin layer just below the interface between the air and water. Research has shown that this layer is a lower temperature than the rest of the water body (Wells 2009). This temperature difference is thought to likely cause an impact on the evaporation rates. Many evaporation models have been developed but none are considered a complete representation of evaporation in open body water (Finch and Calver 2008). One element which has not been fully addressed is the described thermal boundary layer.

Thermal boundary layers have been proven to form in open bodies of water such as lakes and dams. To determine the effect of the thermal boundary layer in relation to the evaporation in open reservoirs and characterise which conditions will influence evaporation rates a series of Computational Fluid Dynamic (CFD) models have been created and simulated. Initially a base model was created and was validated against an accepted numerical model. This base model was then modified to simulate several thermal boundary conditions. The extent of evaporation was monitored and documented for each case. This data has been processed, and conclusions made about the effects of the Thermal Boundary Layer on evaporation.

Through this project the key outcomes are to determine under what conditions the thermal boundary layer will affect evaporation. The series of CFD models built on from each other and developed from being simple simulations to understand the capabilities of CFD in modelling evaporation to complex models to simulate close to realistic conditions. These models then had a system to allow for various combinations of the thermal boundary layers depth and temperature difference to be simulated. Results from these simulations were then compared to control simulations for determination of the effect on evaporation.

The general conclusions of the project are that the thermal boundary layer does have an impact on the rate of evaporation under certain conditions. The amount and under what circumstance depends on several factors including; presence of wind and temperature difference of the thermal boundary layer.

Following this project there is scope to further research. Notably due to limitations in the CFD software and time constraints simulations there is scope for extended duration, larger domains and finer mesh. Simulations with these factors may provide for more accurate data and more conclusive outcomes.

Table of Figures

Figure 1 - Crop Field, Goondiwindi: Photo taken by student during a trip to a station at Goondiwindi in November 2019. Photos is of the effects of drought on landowners, during a trip in 2017 this location was a productive crop field.....	1
Figure 2 - Basic Evaporation Parameters (source www.physics.stakeexchange.com).....	2
Figure 3 - Evaporation Pan (Source: Ertek 2011): Photograph of a Pan Evaporation Method Apparatus, note the climate measurement devices and the animal cage.	6
Figure 4 - Radiation Balance at the Earth's Surface (Source: Shuttleworth 1993): Graphic depicts the difference between short-wave and long-wave radiation.	8
Figure 5 - Model of the Surface Thermal Layer of the Ocean (Source: Wells 2009): Simple diagram graphically representing the thermal boundary layer.	10
Figure 6 - Interferometry Experiment Diagram (Source: Coronel 2018): To determine the temperature of a particle a laser is directed at it, depending on the temperature the laser will deflect by a certain degree.	10
Figure 7 - Multiphase Model Setup: Screen shot of the Multiphase method set up screen within Fluent. Not the Thermal Phase Change model has been selected for the phase change between water liquid and water vapor.	13
Figure 8 - Basic Evaporation Model Geometry: Screen shot of the final graphic within the Geometry function of ANSYS.	16
Figure 9 - Basic Evaporation Model Mesh Setup: Screen shot of the final mesh layout, not the mesh is finer at the interface of the air and water bodies.	18
Figure 10 - Solution Methods Extract: Selection of Fluent Solution methods.....	20
Figure 11 - Basic Evaporation Fluent Setup: This screen shot is taken prior to running the calculations, note the number of timesteps (60), Time Step size (1 sec) and Max Iterations/Time Step (50).	22
Figure 12 - Basic Evaporation Model Water Phase Volume Fraction Result: Graphic depicting volume fraction of water at final timestep.....	23
Figure 13 - Thermal Boundary Layer Geometry Setup: Final Geometry graphic, note the inclusion of the extra body to simulate the thermal boundary layer.	25
Figure 14 - Thermal Boundary Layer Mesh Setup: Graphic after final meshing completed.	26
Figure 15 - Thermal Boundary Layer Results: Results graphic showing temperature of the three bodies after the final timestep, note the thermal boundary layer set to -0.6 degrees cooler than the water body.	28
Figure 16 - Sample Results Generation of Thermal Boundary Layer Model: Screen Shot of the function calculator to record the simulation results.	29
Figure 17 - Wind Model Geometry Setup: Graphic after geometry set up complete, note rather than a square as per previous models this shape is a rectangle.....	32
Figure 18 - Wind Model Mesh Setup: Graphic after meshing completed, note it is the same as per previous models.....	33
Figure 19 - Wind Model Fluent Setup Graphic: Opening graphic screen shot in fluent, images shows the inlet and outlets of the model.	36
Figure 20 - Psychrometric Chart (Fly Carpet 2020): This chart was used to determine the pr-pw values for the calculation.	37
Figure 21 - 5mm Thermal Boundary Model Total Evaporation (g) at each Timestep	41
Figure 22 - 1mm Thermal Boundary Model Total Evaporation (g) at each Timestep	44
Figure 23 - 0.1mm Thermal Boundary Model Total Evaporation (g) at each Timestep	47
Figure 24 - Thermal Boundary Layer Control Model Comparison	48

Figure 25 - Thermal Boundary Layer Model 1mm Vs 5mm Depth Comparison.....	49
Figure 26 - Extended Duration Thermal Boundary Layer Model Results	50
Figure 27 - Humidity Validation Results - Water Evaporated (g).....	52
Figure 28 - Humidity Model Comparison – Volume Fraction	52
Figure 29 - Increased Temperature Difference - Volume Fraction Comparison.....	54
Figure 30 - Increased Temperature Difference - Timestep Comparison	55
Figure 31 - Increased Temperature Difference - Water Evaporated (g).....	55
Figure 32 - Control Model Results Graphic.....	60
Figure 33 - Realistic Conditions Result Comparison	62

Contents

Disclaimer.....	ii
Certification.....	iii
Acknowledgment	iv
Nomenclature	v
Abstract.....	vi
Table of Figures.....	vii
Chapter 1: Introduction	1
1.1 Overview	1
1.2 Outline.....	2
1.3 The Problem.....	2
1.4 Research Objectives.....	2
1.5 Conclusion.....	3
Chapter 2: Literature Review	4
2.1 Introduction	4
2.2 Estimation of Evaporation	4
2.2.1 Computer Simulated Evaporation.....	4
2.2.2 Currently Accepted Numerical and Physical Evaporation Models	5
2.3 The Thermal Boundary Layer	9
2.4 Previous Research on the Topic.....	11
2.5 Literature Review Conclusions.....	11
Chapter 3: Methodology.....	12
3.1 Introduction	12
3.2 Project Methodology Overview	12
3.2.1 Selected Software	12
3.2.2 Numerical Equation Selection for CFD Validation	13
3.3 Base Model Simulations and Calculations	14
3.3.1 Base Model Simulations and Calculations Introduction	14
3.3.2 Geometry Set Up.....	15
3.3.3 Base Model Mesh Set Up.....	16
3.3.4 Base Model Fluent Set Up.....	18
3.3.5 Base Model Results System Setup	22
3.3.6 Base Model and Numerical Calculation Comparison.....	23
3.4 Thermal Boundary Layer Simulations	23
3.4.1 Thermal Boundary Layer Simulations Introduction	23
3.4.2 Thermal Boundary Layer Parameters	23

3.4.3 Thermal Boundary Layer Simulation Overview	24
3.4.4 Thermal Boundary Layer Simulation Geometry Setup	24
3.4.5 Thermal Boundary Layer Simulation Mesh Setup	25
3.4.6 Thermal Boundary Layer Simulation Fluent Setup	26
3.4.7 Results System Setup	28
3.4.8 Thermal Boundary Layer Simulation Comparisons.....	29
3.5 Thermal Boundary Layer Simulation Further Modelling	29
3.6 Thermal Boundary Layer Simulations under Realistic Conditions	30
3.6.1 Thermal Boundary Layer Simulations under Realistic Conditions Introduction.....	30
3.6.2 Wind Simulation Parameters	31
3.6.3 Wind Simulation Overview	31
3.6.4 Wind Simulation Geometry Setup	31
3.6.5 Wind Simulation Mesh Setup	32
3.6.6 Wind Simulation Fluent Setup	33
3.6.7 Results System Setup	36
3.6.8 Wind Simulation Comparisons.....	36
Chapter 4: Results and Discussion	37
4.1 Results Overview.....	37
4.2 Base Model Analysis.....	37
4.2.3 Model Validation.....	38
4.3 Thermal Boundary Layer Model Analysis.....	38
4.3.1 5mm Thermal Boundary Layer Model Results.....	38
4.3.2 5mm Thermal Boundary Layer Model Discussion	41
4.3.3 1mm Thermal Boundary Layer Model Results.....	41
4.3.4 1mm Thermal Boundary Layer Model Discussion	44
4.3.5 0.1mm Thermal Boundary Layer Model Results.....	45
4.3.6 0.1mm Thermal Boundary Layer Model Discussion	47
4.3.7 Combined Thermal Boundary Layer Model Discussion	47
4.4 Thermal Boundary Layer Model Further Testing.....	49
4.4.1 Extended Duration Thermal Boundary Layer Model	49
4.4.2 Extended Duration Thermal Boundary Layer Results	49
4.4.3 Initial Humidity Thermal Boundary Layer Model.....	50
4.4.4 Initial Humidity Thermal Boundary Layer Results.....	51
4.4.5 Increased Temperature Difference Thermal Boundary Layer Model.....	53
4.4.6 Increased Temperature Difference Thermal Boundary Layer Results.....	53
4.4.7 Average Brisbane July Thermal Boundary Layer Model	56

4.4.8 Average Brisbane July Thermal Boundary Layer Results	56
4.4.9 Combined Further Testing Discussion	57
4.5 Thermal Boundary Layer Under Realistic Conditions Analysis	58
4.5.1 Thermal Boundary Layer Under Realistic Conditions Results.....	58
4.5.2 Thermal Boundary Layer Under Realistic Conditions Discussion	61
Chapter 5: Conclusion	65
5.1 Review of Goals.....	65
5.2 Conclusions and Outcomes.....	65
5.3 Possible Further Research.....	68
References	70
Appendices.....	72
Appendix A – Project Specification	73
Appendix B – Project Plan	75
Appendix C – Risk Analysis	78
Appendix D – Base Model Results	81
Appendix E – Thermal Boundary Layer Results	83
Appendix F – Further Modelling Results.....	91
Appendix G – Realistic Condition Results	101

Page Left Intentionally Blank

Chapter 1: Introduction

1.1 Overview

At the conceptualisation stage of this project, Australia was being impacted by drought conditions. Due to the nature of the weather patterns in Australia, water is required to be stored in a large open reservoir such as lakes or dams as reliable precipitation cannot be assumed. Depending on the on the weather cycle, extended periods of limited precipitation may occur, due to this with the effects of evaporation can cause the level of these dams to reduce and unsustainable rate. Droughts of various severities have affected Australia, for instance between the years 1864 and 1983 there have been 9 droughts considered to be major and an additional 7 of a lesser severity (ABS 2012). This provides evidence that it needs to be accepted, that droughts will continue to effect Australia. Understanding how evaporation effects storage of water in open bodies is important to ensure water storage can be appropriately managed.

Many evaporation models have been developed but none of which are considered a complete representation of evaporation in open body water (Finch and Calver 2008). One element which has not been fully addressed in these models is the “Thermal Boundary Layer.” Having an accurate understanding on how open bodies of water evaporate is important to countries such as Australia so that losses is stored water can be predicted and a better understanding on how to manage reserves developed.



Figure 1 - Crop Field, Goondiwindi: Photo taken by student during a trip to a station at Goondiwindi in November 2019. Photos is of the effects of drought on landowners, during a trip in 2017 this location was a productive crop field.

The thermal boundary layer forms in open bodies of water and is best described as a thin cool layer situated at the interface between air and water. There has been significant research into this layer, however there is limited regarding its influence on evaporation. This layer and its effects on evaporation form the basis of this dissertation.

Evaporation is affected by wind, air density, humidity and energy. If these parameters are in the right combination evaporation will occur. Generally, evaporation occurs when energy breaks bonds that are holding water molecules together, transforming it into water vapour (Rutledge 2011). This energy is usually in the form of heat. Humidity and air density are also affected by temperature. Therefore, with the effect of the thermal boundary layer decreasing water temperature at the surface the question this dissertation seeks to answer is “does this layer impact the rate of evaporation.”

For this project available evaporation estimation techniques have been used to model the thermal boundary layer and comment on the affects it has on evaporation.

1.2 Outline

Understanding the effect of the “Thermal Boundary Layer” below water surfaces on evaporation was identified from discussions with the Project Supervisor. The purpose and scope of this study is detailed in 1.4 Research Objectives.

1.3 The Problem

Despite significant research and development on models and equations to predict the results of evaporation for open bodies of water, none account for the “Thermal Boundary Layer.” The effects of this layer and in which conditions it may alter evaporation has scope to be further researched. The thermal boundary layer is known to be of a different temperature to the rest of the water body, as temperature can affect evaporation, it was expected that there is a possibility that when accounting for the thermal boundary in a model the rate of evaporation may be different to an estimate or model which does not. Prior to this project the relationship between this layer and evaporation had not yet been determined, this project attempts to better understand this.

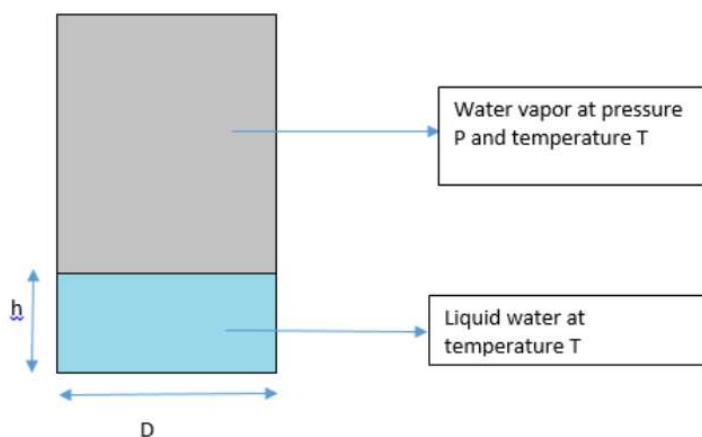


Figure 2 - Basic Evaporation Parameters (source www.physics.stakeexchange.com.)

1.4 Research Objectives

This research is comprised of identifying the problems, trends and needs of evaporation models when addressing the Thermal Boundary Layer. This information will then be used to better understand the effects of the thermal boundary layer in relation to evaporation.

The Research Objectives of this project are;

- (a) Define which current evaporation models are used for large open bodies of water

- (b) Review relevant literature relating to evaporation and the effects of the thermal boundary layer. This literature review will be divided into three subparts
 - I. Review into currently accepted evaporation models
 - II. Research into the Thermal Boundary Layer
 - III. Identifying if this topic has already been addressed
- (c) Develop a model which can simulate the effects of the thermal boundary layer on evaporation for an open body of water.
- (d) Interpret the results of the simulation and describe under what circumstances the thermal boundary layer will affect the rate of evaporation.

1.5 Conclusion

This dissertation aims to investigate the effect of the thermal boundary layer in relation to the evaporation in open reservoirs and characterise which conditions will either promote or retard evaporation.

This research is expected to provide greater understanding of which conditions the effects of the thermal boundary layer can be accurately modelled.

The review of literature for this research will identify which existing models will be best suited for open water body evaporation and the inclusion of the effects of the thermal boundary layer.

The outcomes of this study could be used to help improve the development of current open body water evaporation estimation methods.

Chapter 2: Literature Review

2.1 Introduction

In this chapter the literature will seek to establish the need for further modelling to better understand the effects of the thermal boundary layer in relation to the evaporation in open reservoirs as well as provide the relevant background information.

Following the review of currently accepted evaporation models as well as research into the thermal boundary layer and its possible effects of evaporation, this chapter provides the taken directions to progress to a computer aided model which will help determine conditions where evaporation rates are impacted.

Three main areas have been identified for literature review to enable the project to progress. The outcome of review into these areas is to understand the current development of evaporation and the thermal boundary layer. From this understanding this project has expanded on this research and developed a series of computer aided models.

2.2 Estimation of Evaporation

Estimation of evaporation in large open bodies of water is important to help manage water supply. This is especially important for countries or environments such as Australia as outlined in section 1.1. This estimation can be achieved through, physical, numerical or simulated models. Estimating evaporation using computational simulations will be covered in section 2.2.1, while currently accepted numerical and physical models will be explored in section 2.2.2.

2.2.1 Computer Simulated Evaporation

For modelling of evaporation in the project Computational Fluid Dynamics (CFD) software is required. This software can help predict how liquids and gases perform under different sets of circumstances (Simscale 2020). This allows Engineers, designers and modellers reduce the need for physical experiments. There are multiple available reputable CFD programs for use including the Fluent module of ANSYS, OpenFOAM, PowerFlow, COMSOL and Autodesk CFD.

The benefit of CFD software for this project is the ability to easily change parameters without needing to set up time consuming and costly physical experiments. Also due to the range of predicted parameters of the Thermal Boundary Layer using CFD software will allow for multiple models easily be run.

CFD software models evaporation/condensation using multiphase modelling. This is the simulation of transfer of one phase to another. A phase can be an independent material or a phase of a material such as water liquid or water vapor. In this project there will be three phases which need to be accounted for; being air, water liquid and water vapor. The software will begin by creating and dividing a domain into a mesh of elements. The purpose of multiphase modelling is to determine the phase in each element (discrete continuous) or the volume fraction of each phase within an element (continuous - continuous). Using the prescribed equations and methods the software will then determine the phase changes in each element.

In this project evaporation/condensation will be the primary cause of transfer from one phase to another. This can be achieved in two ways, either discrete continuous or continuous – continuous as mentioned above. Discrete-continuous is generally only used when trying to model specific boundary conditions or an interface. For use in this project, the meshing (element size) will need to be extremely fine for accurate results. The chosen option is to use continuous - continuous modelling for which the Eulerian model equations are required. This requires a continuity/mass

conservation equation to be solved for each phase, this equation is then multiplied by the volume fraction of the phase (r_q) (Wimshurst 2019). The mass transfer of evaporation and condensation would then occur as a phase change. Using the Eulerian equations built into Fluent's multiphase program should result in accurate estimation of evaporation. The Eulerian equation within fluent is as follows;

$$\frac{\partial(r_q \rho_q)}{\partial t} + \nabla \cdot (r_q \rho_q U_q) = \sum_{p=1}^N (\dot{m}_{pq} - \dot{m}_{qp})$$

From review of the capabilities and methodology of CFD software it is determined that it is a suitable instrument for the use of this project. CFD software will allow for accurate modelling of multiphase change and has the flexibility to input the relevant parameters to model the thermal boundary layer.

2.2.2 Currently Accepted Numerical and Physical Evaporation Models

Prior to developing a model, it is important to first understand how evaporation in open body water is currently being estimated. This section seeks to review the current models and equations and understand which ones may be applicable to this research project. These models will also be useful in providing validation to the simulated models.

Depending on its application there are differently accepted evaporation models currently in use. As this project is concerned with open body reservoirs, there are three main areas which have been researched; evaporation for dams, evaporation for swimming pools and evaporation in crops. Each of these areas generally has different parameters for calculating evaporation.

Dam evaporation is currently estimated using one of a number of accepted methods, models or equations. Depending on the need for accuracy and available data one of the following is generally selected;

- (a) Pan Evaporation
- (b) Mass Balance
- (c) Energy Budget
- (d) Bulk or Mass Transfer
- (e) Combinations equations

2.2.2.1 Pan Evaporation

Pan evaporation is one of the oldest and prolific methods, however, is the least relevant to this project. This method requires the monitoring of a standard size pan (generally the USSR GGI-3000 pan) and the environment conditions. The monitored rate can be then use as a reference for a nearby dam's evaporation. Being a physical monitoring station, it has been excluded for use in this project. There is also a high degree of inaccuracy (as much as 20%) some of which can be affected by a number of unaccountable variables, such as animals drinking from the water. This model the parameters of the thermal boundary layer for this method would be extremely difficult to control, therefore is not acceptable for this project.



Figure 3 - Evaporation Pan (Source: Ertek 2011): Photograph of a Pan Evaporation Method Apparatus, note the climate measurement devices and the animal cage.

2.2.2.2 Mass Balance

The Mass balance method is based on the simple principal of measuring the change in volume of water stored and comparing it to the inflow and the outflow. The mass balance equation for this method is defined as;

$$E = P + \frac{(Q_{ri} + Q_{gq}) - (Q_{ro} + Q_{go}) - dV/dt}{A_s}$$

In this equation E is the rate of evaporation, P is the mean rate of precipitation, Q_{ri} is the surface inflow rate, Q_{ro} is the surface outflow, Q_{gi} being the rate of groundwater inflow (seepage) and Q_{go} is the groundwater outflow (seepage), V is the stored amount of water and A_s is the surface area of the body of water.

This model is renowned for its accuracy, being within 5% -12% of the true value (Finch and Calver 2008) however would be unsuitable for this project. Due to the required inputs, this model would be overly complicated to set up corresponding conditions in a CFD model and does not directly have a temperature variable.

2.2.2.3 Energy

The Energy budget method estimates the amount of evaporation based on the amount of energy available (Finch and Calver 2008). The energy components are defined as the latent heat of vaporisation or the heat needed to change water from a liquid phase to a vapour phase and the energy from the water vapour molecules displaced from the body of water. The equation for the method is written as;

$$N = S(1 - \alpha_s) + L_{\downarrow}(1 - \alpha_L) - L_{\uparrow} - E\lambda - c(T_s - T_b)E - H + F_{in} - F_{out} + F_p - G$$

Where N= change in energy, S and L are short and long wave radiation, λE is the latent heat of vaporisation, E is evaporation, c is the temperature of the water, while T_s and T_b regarding he temperature of the evaporated water and the base temperature, H is the flux of heat, F_{in} and F_{out} are the heat fluxes of the water in and out flows, F_p is the heat inflow due to precipitation and G is the heat conduction between water ant the substrate.

This requires all other components of the water body to be known. The main issue of using this method is that energy is in terms of radiation (Singh and Xu 2000) not necessarily temperature. Using radiation measurements will overly complicate the comparison to the simulated model. While modern radiation measurement devices can provide reliable data, for the purpose of this dissertation the information was considered too difficult to obtain.

2.2.2.4 Bulk or Mass Transfer

Bulk or mass transfer equation is a further derivation of the bulk transfer equation (Finch and Hall 2001). The main inputs are the mass transfer coefficient, wind speed and the saturated pressure of the water and air. The saturated pressure component of the equation is helpful as this is almost directly attributed to temperature. The issue however with this method is that the equation relies on wind speed. The equation is given as;

$$E = Cu(e_s^* - e)$$

C is the mass transfer coefficient, u is the wind speed e_s^* is the saturated vapor pressure of the air at the water surface and e is the vapour pressure of the air.

All the inputs can be easily accounted for in both the CFD model and the numerical equation. The issue with this method however how wind is applied to the rest of the equation. As it is wind (C is essentially a drag coefficient of the wind) multiplied by the difference in vapor pressure, if wind is set to zero the result will be an invalid answer of zero, therefore was not included into this project. For this project it was considered too important to be able validate results without a wind factor, the reasoning of this will be explained latter in this dissertation.

2.2.2.5 Combination Equations

Combination Equations are further improvement of the Bulk or Mass transfer method, commonly The Penman and Priestley-Taylor equations. This method essentially combines the Energy budget method with the Bulk or Mass Transfer method. This method has been widely used for open water evaporation and vegetation (Finch and Calver 2008). The equation is provided as;

$$E = \frac{\Delta R'_n}{\Delta + \gamma} + \frac{\gamma c(u)(e_a^* - e)}{\Delta + \gamma}$$

In this equation R'_n is net radiation, Δ is the grade of the saturated vapour pressure-temperature curve and γ is the psychometric coefficient. Also note, $c(u)(e_a^* - e)$ is as per the Mass Transfer equation defined in section 2.3.2.3.

As this equation takes derives from the energy budget method there is a requirement to account for energy in the form of radiation. This was considered overly complicated for this dissertation and omitted for further use.

2.2.2.6 Currently Accepted Numerical Models for Dams Conclusion

The obvious choice for this project is to research into the evaporation for dams. However, the current models may not be best suited for comparison against a Computational Fluid Dynamic model. This is due to the reliance of “radiant” energy as a major contributing factor. While possible to include instead of, temperature is a much easier factor to use. In many of the models and equations both wind speed and subsequently waves are also major contributing factors. In the model attempted as part of this project both of which are to be excluded. From review of the generally accepted open body reservoir estimation methods it was determined that mentioned above are unlikely to be well suited for this project.

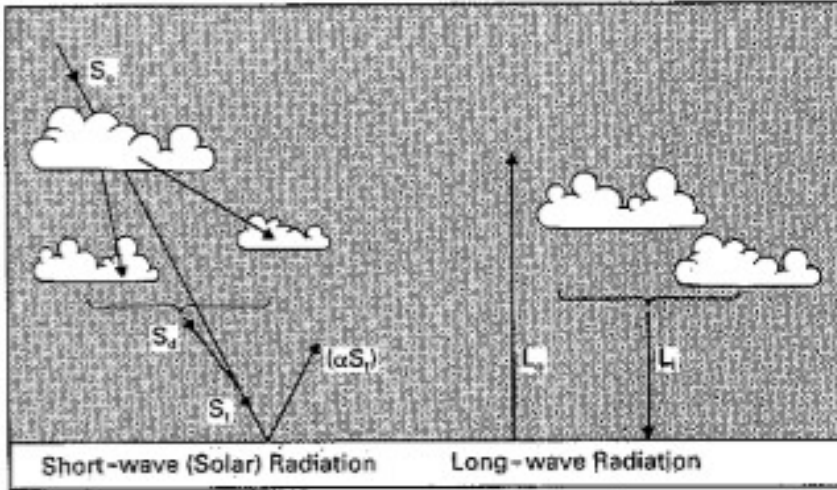


Figure 4 - Radiation Balance at the Earth's Surface (Source: Shuttleworth 1993): Graphic depicts the difference between short-wave and long-wave radiation.

2.2.2.7 Further review of Numerical Models for Evaporation

Evaporation estimation is also important for Agricultural Engineering and understanding evaporation in crops. Review of currently accepted models for this role has been undertaken. Many of the methods used for this purpose are derived from the open water methods however include coefficients for crop cover. While most Temperature-Based equations for this purpose are not commonly used, there are two methods that are accepted: the Hargreaves equation and the Blaney-Criddle Method (Shuttleworth 1993).

The Hargreaves Equation commonly referenced as;

$$E = 0.0023S_o\delta_t(T + 17.8)$$

Unfortunately for this purpose S_o is a measurement of extra-terrestrial radiation. As this would be hard to model, this was considered to not likely be useful. Therefore, use of this equation was excluded from this project.

The Blaney-Criddle however is still commonly used and all inputs can be accounted for in both the mathematical estimations as well as the computer aided modelling. A common form of the equation is;

$$E = a_{BC} + b_{BC}f, \quad (Eq. 1)$$

Where

$$f = p(0.46T + 8.13)$$

$$a_{BC} = 0.0043RH_{min} - \left(\frac{n}{N}\right) - 1.41$$

$$b_{BC} = 0.82 - 0.0041(RH_{min}) + 1.07\left(\frac{n}{N}\right) + 0.0066(U_d) - 0.006(RH_{min})\left(\frac{n}{N}\right) - 0.006(RH_{min})(U_d)$$

p = the ration of actual daily daytime hours to annual mean daily daytime hours (%)

T = air temperature deg. C

$\left(\frac{n}{N}\right)$ = The ratio of actual to possible sunshine hours

$U_d = \text{Daytime wind speed}$

$RH_{min} = \text{is the minimum daily relative humidity in percent}$

This method being temperature based and being able to account for all factors was determined to be acceptable for the use of model calibration.

As windspeed and direct sunlight/radiation are not factors in the proposed model another real-world situation's equations have been reviewed. Indoor swimming pools fit this circumstance, they are large open bodies of water, have limited wind and no direct radiation from the sun. There has been some research undertaken on developing equations for calculations of such situations. The Shah Equation (Eq.2) has undergone a number of developments of recent years but some of it's forms were determined as useful for this project. One equation has been developed to calculate evaporation due to natural convection (Shah 2014). It is stated as;

$$E_o = Cp_w(p_r - p_w)^{\frac{1}{3}}(W_w - W_r), \quad (\text{Eq.2})$$

$C = \text{the coefitecent for either imperail or SI units}$

$p_r - p_w = \text{differnce density of air between room temp. and water surface temp.}$

$W_w - W_r$

$= \text{diffence of specific humidity of air between room temp. and water surface temp.}$

All factors can be accounted for in the model therefore this equation was decided to be used for calibration.

The conclusion of the literature in current evaporation methods is that the ones commonly used for dams cannot be easily used for the purpose of this project. There are however two methods that can be used as they are considered temperature based, these being the Blaney-Criddle method (Eq.1) and the Shah equation (Eq.2).

2.3 The Thermal Boundary Layer

Measurements have shown that just below the surface of an evaporating body of water exists a cool surface layer, which can be as much of 0.6 deg C below that of the surface temperature (Ewing, Gifford, and E. D. McAlister – 1960). Tests have also shown that the although the layer being so thin it can quickly establish itself following disruption, therefore is considered persistent (Katsaros 1980). So, for this project it can be accepted that there is a cooler layer at the air/water interface and the subsequent simulations and estimations will seek to understand its effects on evaporation.

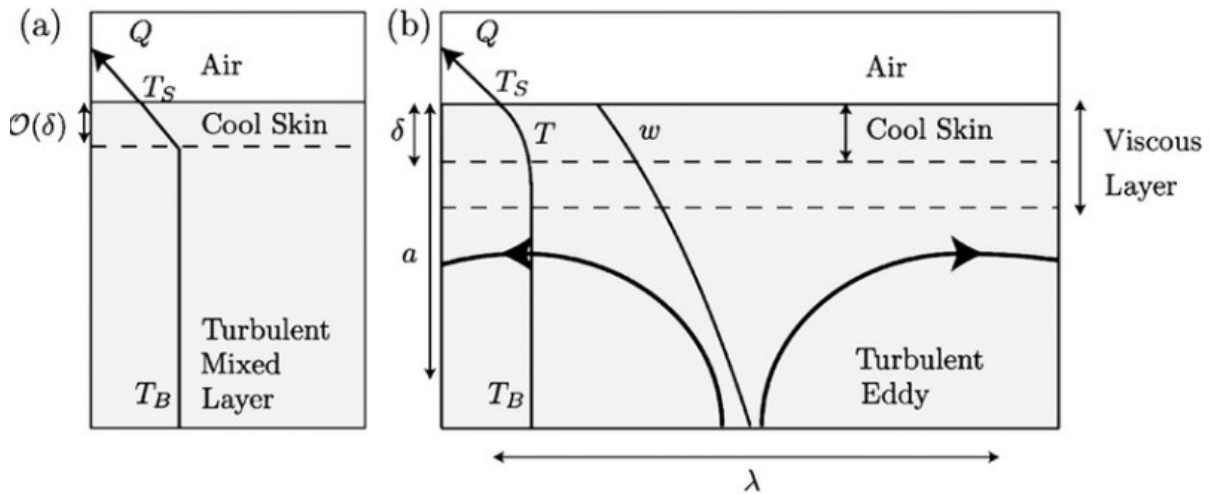


Figure 5 - Model of the Surface Thermal Layer of the Ocean (Source: Wells 2009): Simple diagram graphically representing the thermal boundary layer.

Due to the expected depth only being to be minimal, measuring the exact size is problematic (William Asher – 1996). A technique based on Interferometry has however been able to measure the radiance emitted from water. This technique is then used to determine the depth of the temperature gradients of water. A depth of approximately $65\mu\text{m}$ has been proposed (Mckeown and Asher 1997). While it has been suggested that this depth is limited to a stationary measurement and maybe not accurate for an open body of water such as an ocean. However, more recent tests have concluded that this layer could be between 0.1mm and 5mm , with the depth being greater where in situations where turbulence is low (A Wells 2009). This suggests that in circumstances of limited wind or disturbance a thicker thermal boundary layer can be expected.

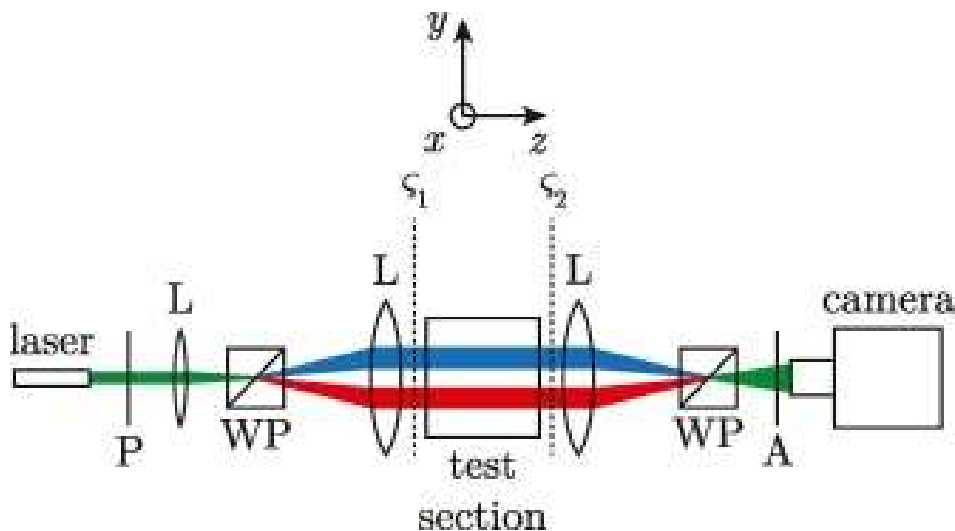


Figure 6 - Interferometry Experiment Diagram (Source: Coronel 2018): To determine the temperature of a particle a laser is directed at it, depending on the temperature the laser will deflect by a certain degree.

In conclusion the literature review into the thermal boundary layer has provided multiple conditions to be tested in the model. These conditions will be combinations of different depths and temperature differences. Temperature ranges will be $0.2 - 0.6$ deg. C, while depth ranges will be $0.1\text{mm} - 5\text{mm}$, using different combinations of these should provide insight under which conditions evaporation is affected. These conditions are considered a typical range of what parameters a thermal boundary layer in an open body of water would be.

2.4 Previous Research on the Topic

From an exhaustive literature review the conclusion has been made that the specific goals of this project have not previously been addressed. This confirms that this project is unique in its goals and methodology. With this confirmation, the decision to continue as planned was made.

2.5 Literature Review Conclusions

The three goals of this literature have been achieved. Current methods for measuring evaporation in open bodies of water have been reviewed. General characteristics of the Thermal Boundary Layer have been identified and this research has also been confirmed to be unique.

The review has extensively outlined how open body water is evaporated as well as identified issues with using currently accepted models for this project. The review however does provide a couple methods and equations which can be used for calibration of the model and possible improvement depending on outcomes of this research. The relevant background information regarding the establishing of a CFD model has also been determined by the literature review. With limited exact data on the characteristics of Thermal Boundary Layer, this research has been successful in providing some baseline conditions to use within the proposed models and simulations.

Following the exhaustive literature review on evaporation modelling and the parameters of the thermal boundary layer a methodology to test effect of the layer have been developed. Details of this methodology are provided in Chapter 3.

Chapter 3: Methodology

3.1 Introduction

To determine the effect of the thermal boundary layer in relation to the evaporation in open reservoirs and characterise which conditions will either promote or retard evaporation a series of Computational Fluid Dynamic (CFD) models have been created and simulated. To develop the data provided and to determine the conclusions in this report, modelling has been carried out in three distinct phases;

- Base-Model development and comparison with accepted numerical calculations.
- Develop and simulate CFD models to compare the evaporation in models with thermal boundary layers of varying size and temperature difference.
- Develop and simulate CFD models to compare the evaporation in models with and without the effects of wind.

This data has then been processed, compared, and conclusions made about the simulated effects of the Thermal Boundary Layer regarding evaporation.

3.2 Project Methodology Overview

3.2.1 Selected Software

The *Fluent* package of the ANSYS modelling software has been used as the program for these simulations. The package which was available for use for this dissertation is ANSY 2020R1/R2 Student Version. It should be noted that the student version does limit the number of elements that can be modelled to approximately 512000, the consequences of this are explained further in this report.

CFD software as described in section 2.2.1 provides a method to simulate multiple different parameters regarding the performance of gasses and liquids. Importantly *Fluent* has the capability to simulate multiphase modelling. For this project, the multiphase modelling will simulate the transfer of different phases of liquids and gases to other phases. The phases required to be modelled in this dissertation are air, water liquid and water vapor. The developed models provide transfer of these phases through both displacement and evaporation/condensation.

ANSYS *Fluent* contains the relevant modelling capacities to model the heat transfer and the phase change for the project. The two primary evaporation models within *Fluent* are the Lee Model and the Terminal Phase Change Model (ANSYS 2020) The Lee model is based of the equation;

$$\text{Where } T_l > T_{sat}$$
$$\dot{m}^m \equiv w_e \alpha_l \rho_l \frac{T_l - T_{sat}}{T_{sat}}$$

Where *l* stands for liquid, *v* stands for vapor, subscript *sat* stands for saturation (temperature evaporation begins) and $w(w_e \neq w_c)$ is the frequency.

The Thermal Phase change model is based on the equations;

Evaporation:

$$\dot{m}''' \equiv \frac{C_l h_l A_i (T_{sat} - T_l) + C_v h_v A_i (T_{sat} - T_v)}{H_v(T_v) - H_l(T_{sat})}$$

Condensation:

$$\dot{m}''' \equiv \frac{C_l h_l A_i (T_{sat} - T_l) + C_v h_v A_i (T_{sat} - T_v)}{H_v(T_v) - H_l(T_{sat})}$$

Where C_l and C_v are scaling factors, h_l and h_v are convection coefficients, H_l and H_v are the phase enthalpies and A_i is the interfacial area concentration.

Due to the Lee Model requiring the w factor to be specified and this will be unknown as part of this project, use of this model is problematic. Also, Volume-of-fluid (VOF) method within *Fluent* is the mentioned Lee model which is considered only a simplified version of the full Eulerian equations used for continuous-continuous phase interactions. For these reasons the Thermal Phase Change Model will be selected as there are no unknown factors and it uses the entire Eulerian method.

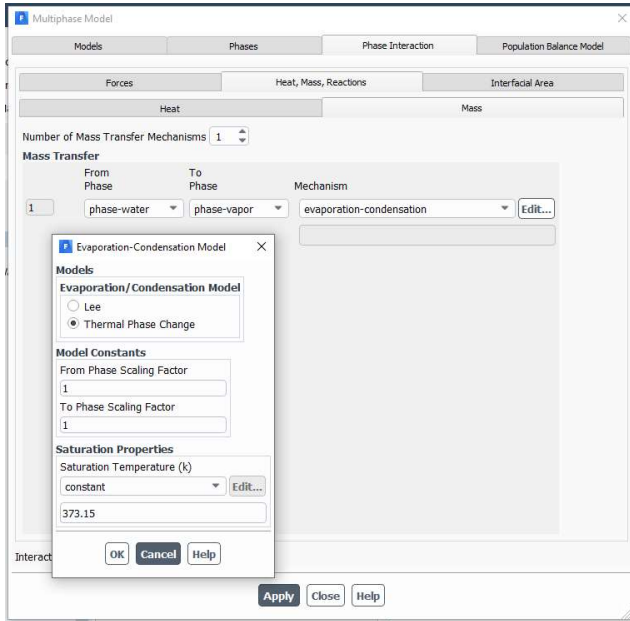


Figure 7 - Multiphase Model Setup: Screen shot of the Multiphase method set up screen within Fluent. Not the Thermal Phase Change model has been selected for the phase change between water liquid and water vapor.

3.2.2 Numerical Equation Selection for CFD Validation

Two separate equations are required to compare the simulated models against. As mentioned in the section 2.2.1 out of the various available equations for estimating evaporation there are only two suitable for this purpose. Both are temperature-based equations however they differ in that one has a variable for wind whereas the other doesn't. For this purpose, the *Shah Equation* (Eq.2) defined as;

$$E_o = C p_w (p_r - p_w)^{\frac{1}{3}} (W_w - W_r)$$

Where;

$C = 35 =$ the coefficient for SI units

$p_r - p_w =$ difference density of air between room temp. and water surface temp.

$W_w - W_r$

$=$ difference of specific humidity of air between room temp. and water surface temp.

The parameters used for this calculation are taken from Brisbane February averages. The table below provides the parameters used.

Parameter	Value	Source
-----------	-------	--------

Average Water Temperature	24 degrees Celsius	Bom 2020
Average Air Temperature	29 degrees Celsius	Bom 2020
Average Air Pressure	1024 mB	Bom 2020
Average Relative Humidity	62%	Bom 2020
Average Density of Air	Refer Psychrometric Charts	Fly Carpet 2020

The parameters in the above table will be used to determine the acceptance of the base model without wind. The *Blaney-Criddle Method (Eq.1)* defined as;

$$E = a_{BC} + b_{BC}f$$

Where;

$$f = p(0.46t + 8.13)$$

$$a_{BC} = 0.0043RH_{min} - \left(\frac{n}{N}\right) - 1.41$$

$$b_{BC} = 0.82 - 0.0041(RH_{min}) + 1.07\left(\frac{n}{N}\right) + 0.0066(U_d) - 0.006(RH_{min})\left(\frac{n}{N}\right) - 0.006(RH_{min})(U_d)$$

will be used in the determination of the models including wind as a factor.

It was expected that the comparison between the numerical calculations and the CFD modelled results were not going to be exact. This is due to the likely discrepancies between both the modelling and the numerical calculations not being exact and known to have some error. With allowance for some discrepancies this method will suffice instead of comparing results against a real time experiment.

3.3 Base Model Simulations and Calculations

3.3.1 Base Model Simulations and Calculations Introduction

The base model is an evaporation without the thermal boundary layer specifically included. The purpose of this model is to provide a basis to test *Fluent's* ability to simulate evaporation for this project and to ensure the user inputs are correct. The results of this model were then calibrated against numerical calculations. This model also provided the basis of the more complex model which includes provisions for the thermal boundary layer.

To develop the CFD models for this dissertation in *Fluent* a number of individual systems need to be created in the *ANSYS* workbench. These systems are

- Geometry – The Geometry system allows the user to create various 2-D or 3-D sketches and convert them into Models. This system will allow the space in which the simulation be undertaken to be created.
- Mesh - Meshing an integral part creating the engineering simulation process, this system allows complex geometries are divided into simple elements so that they be used as discrete local approximations within larger domain. The mesh influences how accurate the model will be as well as how well it will Converge (Ansys 2020)
- Fluent – Is a fluid simulation package used to model several fluid dynamics including for this purpose, heat transfer, energy and multiphase flows.
- Results – This system will allow the complex results to be processed and provided in a usable form for comparisons.

To develop the model which provided evaporation data comparable to that calculated in the Shah Equation (Eq.2), 7 iterations were required before reliable results were obtained. The set up of the successful model are defined in each system below.

3.3.2 Geometry Set Up

As mentioned, this dissertation seeks to replicate results that could be expected in a large open body of water, such as a lake. It is not feasible in this software package to create a 3-d model of large lake, therefore a smaller geometry was chosen. The final dimensions that provided reliable results were;

0.025m wide x 0.025m long, with a water body 0.1m high and a 0.05m high air body

These dimensions have been chosen to allow the largest area while being constrained by the maximum allowable nodes. The student version used has a maximum allowable number of nodes of 512000. This coupled with the required refinement of mesh to test the smallest boundary layers has resulted in these dimensions. The depth of the water bodies and air bodies was selected on the basis that the top and bottom boundary conditions did not interfere with the simulation. These dimensions were developed by a combination of calculations and trial and error.

The sketch used to create the modelled domain was drawn using the basic sketching functions within *Geometry*, while the air and water bodies were created using a combination of the *Extrude* and *Boolean* functions. To ensure that mass transfer and multiphase modelling could occur other provisions were made. When using the Extrude function, it was set to “add frozen” not “add material”. The individual water and air bodies were also combined into a single “part” to allow Fluent to model as a single entity. Using this format allowed flow between the separate bodies to occur.

The process to create the Geometry system component is as follows;

- I. 0.025x0.025m square in XY Plane – Creating this square is the first step in creating our geometric domain.
- II. Generate this square as a sketch – Converting this square to a “sketch” will allow further functions to be applied to it to create our domain.
- III. Extrude 0.15m in y direction by using Add Frozen function – This will develop a 3D structure of 0.15m high, 0.025m Long & 0.025m Wide.
- IV. Extrude 0.1m in y direction by using Add Frozen function – This function sets up to allow for separate bodies to be set up.
- V. Using Boolean subtract function on 0.1m body from 0.15 body, Selecting Preserve Bodies – This will subtract one body from the other creating two separate bodies.
- VI. Names will be set to AirBody & WaterBody – With the two sperate parts they are named depending on their initial phase.
- VII. Select all bodies and combine into single part – This is required to ensure “flow” from one body to the other.
- VIII. Material will of both bodies set to “fluid” – As both bodies will be filled with a liquid phase it needs to be set as a liquid so Fluent can model correctly.

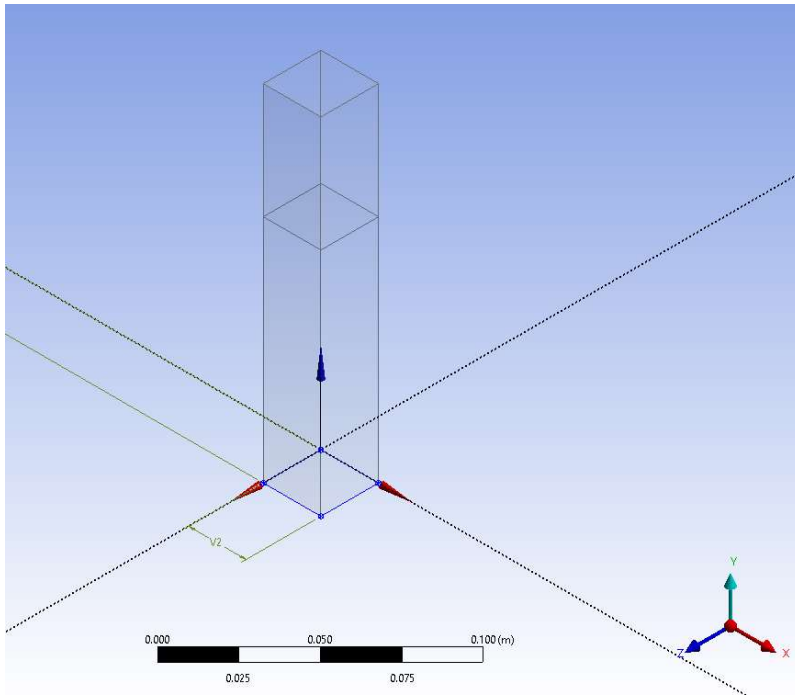


Figure 8 - Basic Evaporation Model Geometry: Screen shot of the final graphic within the Geometry function of ANSYS.

3.3.3 Base Model Mesh Set Up

Meshing directly influences how accurate and how well the simulation will converge. Poor meshing will also occur fatal errors such as floating-point exceptions. The mesh development needs to provide a fine enough mesh to be accurate, but it also constrained by the maximum number of elements, a too fine mesh may also cause calculation durations to become too long to be functional.

Initial iterations had used the “body mesh” function however this resulted in either too many nodes to for the student version or too coarse of a mesh. It is noted that for this purpose the fine mesh is most critical at the Air/Water interface. To achieve this “Face-Meshing” was used. This allows fine mesh at the interface and a courser mesh at the top and bottom boundaries where accuracy is considered less critical.

After multiple iterations of development suitable mesh size to allow for sufficient accuracy and efficient calculations was determined as;

0.0001m at the Air/Water Interface and 0.001m at the top and bottom boundaries

From review of the statistics after the generation of the mesh, it was determined acceptable based on the Orthogonal Quality, Skewness and Element Number. Statics were as follows;

Orthogonal Quality = 0.99999 (min 0.99702, max 1)

Skewness Average = 0.00037449

Element Number = 459680

To ensure reliable simulation in fluent the Orthogonal Quality should be more than 0.7 and the Skewness be less than 0.5 (*Ozgun 2020*). The element number is also required to be less than 512000. From review of the mesh and these parameters the mesh was considered acceptable.

The mesh system also needs various inputs to be made to ensure the Fluent software can appropriately model the space. The individual bodies and faces of the 3d model need to be named at

this stage, this will allow ease of use of the following systems. Importantly, the system needs to be set up for CFD and all bodies set as fluids. Setting the bodies to fluids will ensure the correct calculation methods are imposed during later stages.

The process to create the Mesh system component is as follows;

- I. Create the named sections for the following components;

Body Names

- A. BodyAir
- B. BodyWater

Face Names

- C. Water+z
- D. Water-z
- E. Water+x
- F. Water-x
- G. Air+z
- H. Air-z
- I. Air+x
- J. Air-x
- K. AirWaterInt
- L. Top
- M. Base

Naming these components will be essential when opening in the *Fluent* program. This naming convention will allow for easy setting of Boundary Conditions, Cell Conditions, Initial Conditions, Patching as well as reviewing results.

- II. Set Model to CFD – This is to ensure ANSYS sets up the modelling to be compatible with Fluent.
- III. Set default mesh size to 0.001m – This will set the default size of elements to be 0.001x0.001mx0.001m
- IV. Create “Face-Sizing” parameters for the following named sections
 - A. AirWaterInt – 0.0001m
 - B. Top – 0.001m
 - C. Base – 0.001m
- V. Check all bodies are set as “fluid” – By default the program should use the material type as defined in Geometry but this is good practice to check.
- VI. Check Statistics – As mentioned above, the statistics need to be checked to ensure the mesh is of good quality.

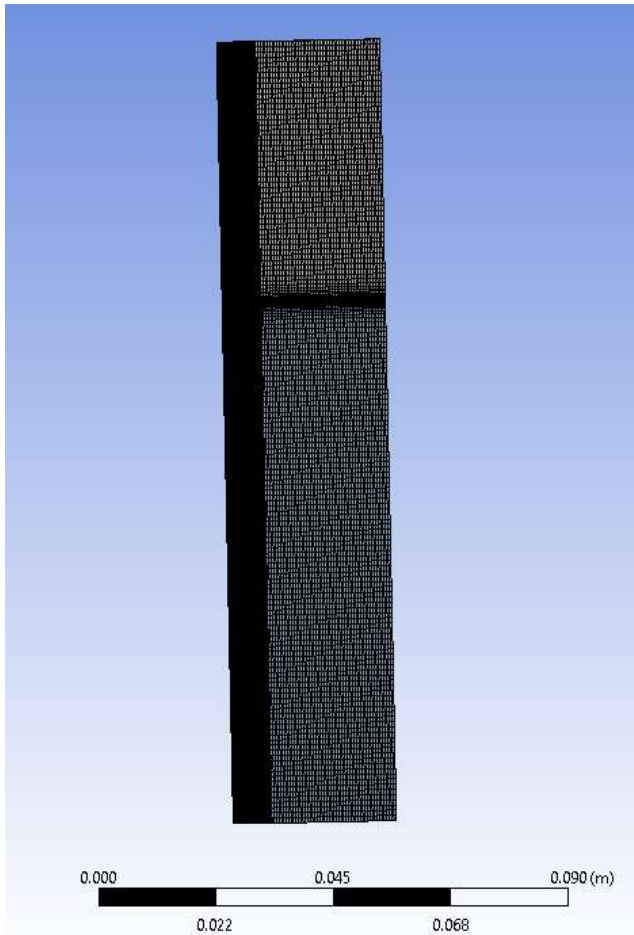


Figure 9 - Basic Evaporation Model Mesh Setup: Screen shot of the final mesh layout, not the mesh is finer at the interface of the air and water bodies.

3.3.4 Base Model Fluent Set Up

The following *Fluent* set up methodology was developed through multiple iterations checking for accuracy, convergence and reliability.

At the initial *Fluent* opening screen, it is important to set the program to double precision, this is due to the very small changes in the volume fractions which may have a substantial impact on the behaviour of the system. The amount of process should reflect the number of available CPU cores of the operating machine. In the instance of this project, generally 4 was used. Issues however did arise later errors and crashes, to prevent these it was reverted back to serial mode (only a single CPU core in use) as this appeared to be the only.

The initial inputs into the *Fluent* system are regarding the general requirements. This is setting the model as transient, meaning the solver will solve for each time step rather than based of iterations. Gravity is also required to be set in the negative 'y' direction to match the used coordinate system. Gravity for this purpose is set to 9.81m/s.

For this model three fluid materials are to be accounted for; air, water liquid and water vapor are all part of this simulation. These materials will form the basis of the three phases. The parameters for these materials are based of the *Fluent* default database have not been altered as the default settings are considered acceptable for this project.

To determine accurate results for this dissertation the following equations are required to be included into the model, these include one for Multiphase, Energy and Viscosity.

The default Fluent Energy Equation has not been modified and is defined as;

$$\rho \frac{De}{Dt} = \frac{\partial u_i p}{\partial x_i} + \frac{\partial u_i T_{ji}}{\partial x_j} - \frac{\partial q_i}{\partial x_i} + S_e$$

For the purpose of this model the K-Epsilon equation is more suitable. Generally, the k-omega model is used for parameters close to wall regions (Argyropoulos 2015), in which this case is not required. Therefore K-Epsilon will be used, and the “scalar wall functions” option has also been selected.

The multiphase equations are where the parameters around the mass transfer of material phases is defined. In this case, the mass transfer of water liquid to water vapor will occur because of evaporation and condensation. There is possibility that material or phases will be transferred to other domains through displacement. This transfer mostly occurs when the water vapour escapes the model and is displaced by air. Displacement also becomes an issue in the wind model, this is described in further detail in later sections. As mentioned in the above sections the “Thermal Phase” Model is required. As the “Thermal Phase” Model is being utilized heat transfer is also required between the material phases. For this purpose, the “two-resistance” model has been included.

The thermal boundary layer as determined as part of previous sections is cooler than the rest of the water body, therefore the model needs to incorporate temperature controls. The simulation also attempts to create real world conditions therefore water and air temperature have been selected based on average temperatures in summer months in Brisbane. Average water temperature is 24 degrees Celsius and average maximum air temperature is 29 degrees Celsius for the month of February (BOM 2020). To incorporate this into the model the Cell Boundary Zone attributed to the water body had the temperature set to a constant of 24 degrees Celsius. The cell boundary zone attributed to the air body did not have a constant set as it is not intended to disturb the cooling of the air near the surface of the water. Backflow for the Air Outlet however is set to 29 degrees Celsius and the wall at the bottom of the water was set to 24 degrees Celsius. This is to ensure these initial temperatures are maintained.

Boundary conditions have been defined to give the most accurate results as well as model an area of water in a middle of larger body as best as possible. Initially a larger model was set up with an inner and outer section for both air and water. This however proved problematic and caused issues during the calculations. Instead the side boundaries of the water and air components have been set to “symmetry” not “wall”. The symmetry boundary condition provides a mirror surface, therefore the conditions with the domain are reflected in the “out of domain” sections. This is done to help simulate the conditions of a small area in the middle of a large open body of water. Early models also had issues with the vapor pressure in the air component impacting results. To rectify the issue the top of the air component has been set to a pressure outlet. It should however be noted that due to this the vapor does escape and measuring the vapour quantity is not reliable in determining evaporation. The boundary conditions also need to be modified to ensure stable relative humidity. This was not correctly modelled the base model but has been addressed in later iterations.

The purpose of this base model to gauge the comparability of the CFD model to a numeric equation. It is also noted that the numeric equation is not completely accurate either (Shah 2008) therefore accuracy gained from number of increasing number of iterations has been determined in this instance as not critical. Also, having an extended duration while desirable is also not considered critical. For this purpose, 50 iterations for 60, one second timesteps is determined to be acceptable.

This will provide residuals of 1×10^{-3} which can be considered suitable as a near enough estimate. This is only in this case as an approximation is sufficient to provide validation.

The method section of the Solutions tab is where the process for each set of equations are set. Firstly, Phase Coupled Simple (PC-Simple) is the chosen Multiphase Coupled solver. This method is only available for Eulerian models but is known for its reliability. This essentially solves the velocity field for each phase separately, but the pressure correction is based on the continuity of all phases. Regarding pressure, PRESTO! is generally considered the best option (Wandel 2020), as there was no issue with its function it is used in this project. The volume fraction was calculated using the Modified HRIC method. For the rest of the Spatial Discretization methods Second Order Upwind has been chosen. This is an improvement over first-order schemes as it includes three data points as opposed to two.

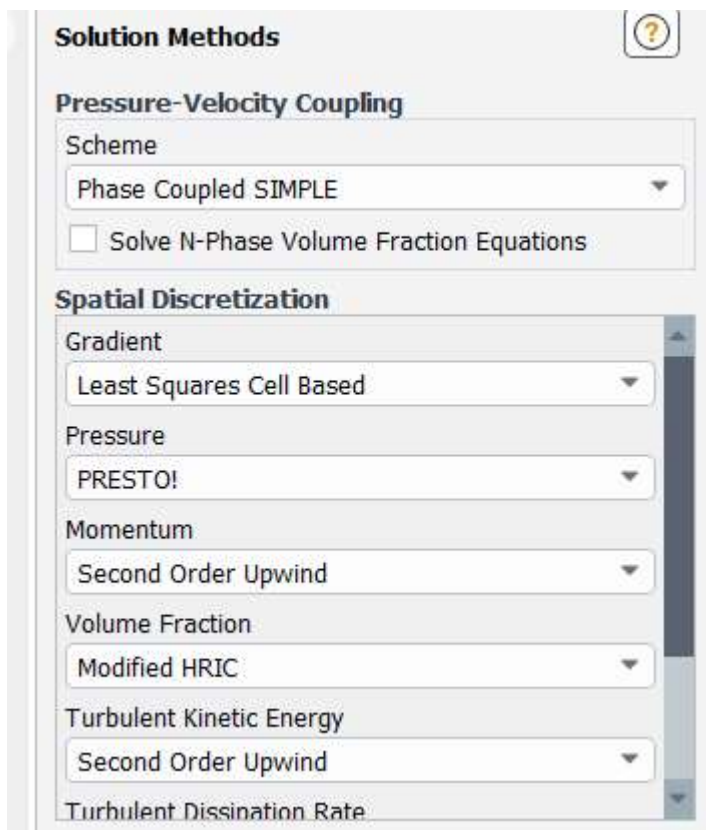


Figure 10 - Solution Methods Extract: Selection of Fluent Solution methods. Note the selected methods for each section.

The process to create the Fluent system component is as follows;

- I. Set from steady to transient
- II. Set Gravity to -9.81 m/s in Y direction
- III. Use Fluent Data base to include Water liquid and Water Vapor in materials
- IV. Turn on Energy Equation
- V. Turn on Viscous Equation and set to K-Epsilon with Scalar Wall Functions
- VI. Set Up Multiphase Model
 - A. Select Eulerian Model
 - B. Set Phases to 3
 - C. Assign each Air, Water, Liquid to a phase and label accordingly
 - D. Turn on Two-resistance Model for Heat Transfer
 - E. Select Mass Transfer and set to 1

- F. Set phase liquid to phase vapor as Evaporation/Condensation using Thermal Phase Change model
- VII. Assign constant temperature of 24 degrees Celsius for *WaterBody*
- VIII. Boundary Conditions
 - A. Water+z - Symmetry
 - B. Water-z - Symmetry
 - C. Water+x - Symmetry
 - D. Water-x - Symmetry
 - E. Air+z - Symmetry
 - F. Air-z - Symmetry
 - G. Air+x - Symmetry
 - H. Air-x - Symmetry
 - I. AirWaterInt - Internal
 - J. Top – Pressure Outlet, thermal backflow set to 29 degrees Celsius
 - K. Base – Wall, thermal temperature set to 24 degrees Celsius
- IX. Methods
 - A. Gradient – Least Squares Cell Based
 - B. Pressure – PRESTO
 - C. Momentum – Second Order Upwind
 - D. Volume – Modified HRIC
 - E. Turbulent Kinetic Energy – Second Order Upwind
 - F. Turbulent Dissipation Rate – Second Order Upwind
 - G. Energy – First Order Upwind
- X. Monitors
 - A. All to be set to 1E-4
- XI. Calculation Activities – record every 1 timestep
- XII. Initialization from all domains
 - A. Initial Temperatures, 24 for water phase and 29 for air phase
 - B. Patch WaterBody as volume function 1 of Water
- XIII. Run Calculation
 - A. 60 Time Steps
 - B. 1 Second Time Steps
 - C. 50 Max Iterations

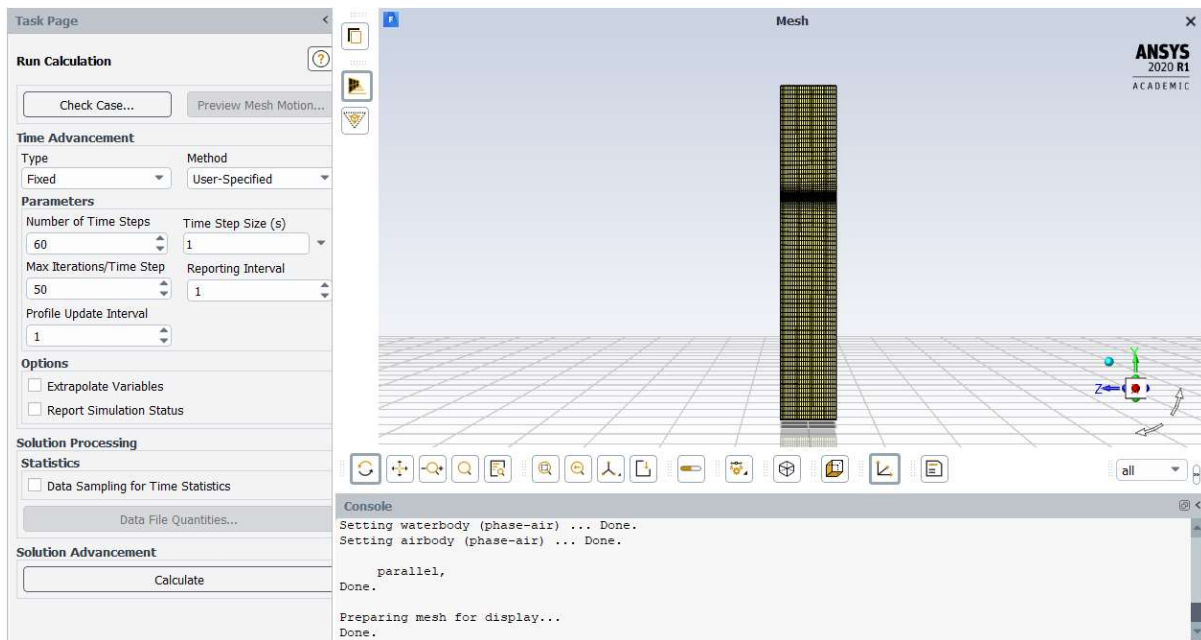


Figure 11 - Basic Evaporation Fluent Setup: This screen shot is taken prior to running the calculations, note the number of timesteps (60), Time Step size (1 sec) and Max Iterations/Time Step (50).

3.3.5 Base Model Results System Setup

To determine the rate of evaporation the volume fraction of the water will be monitored. The domain that the water will be monitored is the water body. The equation used to produce the results is;

$$ave(\text{Phase Water.Volume Fraction})@water$$

A plane is also set up on the x-y axis, at the mid-point of the y axis in the model. Graphics produced are both the volume of water fraction as well as the temperature of the water. The results will be provided in form of the volume fraction of the liquid water phase with the measured domain. This result will be presented as either 1, which is defines the entire domain is of liquid water or a decimal, defining the fraction which the area is of liquid water.

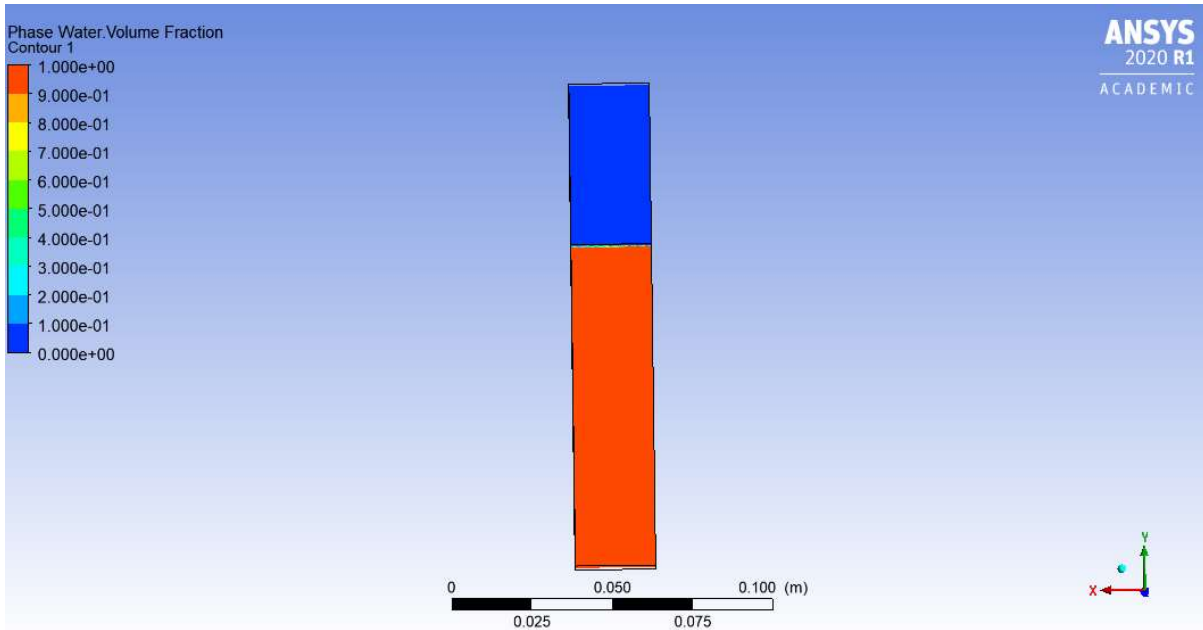


Figure 12 - Basic Evaporation Model Water Phase Volume Fraction Result: Graphic depicting volume fraction of water at final timestep.

3.3.6 Base Model and Numerical Calculation Comparison

The Shah Equation is used using the same parameters as mentioned above. The difference air density and specific humidity's are determined using Psychrometric charts. It was understood to expect some inaccuracy however correlation between the simulated and modelled results was reviewed.

3.4 Thermal Boundary Layer Simulations

3.4.1 Thermal Boundary Layer Simulations Introduction

Following successful set up of the base model, the software was calibrated to simulate evaporation with acceptable accuracy. The next stage of the project was to simulate the effects the thermal boundary layer will have on the rate of evaporation. As per the previous sections it was expected that the cooler layer at the interface will affect the rate of evaporation by either increasing it or decreasing it. To test this, the simulation was built by further developing the base model by including a layer of cooler water at the interface.

3.4.2 Thermal Boundary Layer Parameters

From the literature review it was clear that the dimensions and temperature difference of the thermal boundary layer are not well defined. This is due to the thermal layer having different parameters caused by a number of factors. The literature review provided a range of depth dimensions and temperature differences collated from different sources and preciously conducted experiments by others. To ensure the effects regarding evaporation are better understood a range of various parameters was simulated and compared. The dimensions of the boundary layer varied from 0.01mm to 5mm by altering the geometry and mesh and a temperature difference between 0.2 deg C and 0.6 deg C which was inputted as part of the initialization. The following combinations will be addressed as part of the project;

Table 1 - List of Depth and Temperature Models

Thermal Boundary Layer Depth (mm)	Temperature Difference (deg. C)
-----------------------------------	---------------------------------

5	-0.6
5	-0.2
1.0	-0.6
1.0	-0.2
0.1	-0.6
0.1	-0.2

3.4.3 Thermal Boundary Layer Simulation Overview

To be able to compare evaporation rate, differences between models which include a provision for the thermal boundary layer and those which don't a series of simulations will be run. Initially it was proposed to run a single control model (without the thermal boundary layer) however it was noted that with the dimensions of the thermal boundary layer changing, alterations to the mesh could not be controlled from one model to the next. This was confirmed through a mesh independence study. These mesh alterations caused a difference in simulated evaporation rate therefore results could not be directly compared. To resolve this a control model will be run for each depth dimension.

3.4.4 Thermal Boundary Layer Simulation Geometry Setup

The geometry system set up for the thermal boundary layer simulations will be created similarly to that of the base model, albeit with the inclusion of a separate body for the thermal boundary layer. This body will be created using additional *Extrude* and *Boolean* functions. Geometry set ups will be exactly the same for all models of the same thermal boundary layer depth dimensions and the applicable control model.

Creation of the geometry set up is as follows;

- I. 0.025x0.025m square in XY Plane
- II. Generate this square as a sketch
- III. Extrude 0.15m in y direction by using Add Frozen function
- IV. Extrude 0.1m in y direction by using Add Frozen function
- V. Extrude 0.1m – *thermal boundary layer depth* by using Add Frozen Function
- VI. Using Boolean subtract function on 0.1m body from 0.15 body – Selecting Preserve Bodies
- VII. Using Boolean subtract function on 0.1m – *thermal boundary layer depth* body from 0.1body – Selecting Preserve Bodies
- VIII. Names will be set to AirBody, WaterBody, TBLBody
- IX. Select all bodies and combine into single part
- X. Material will of both bodies set to “fluid”

Different to the base model, more steps are required to be undertaken to create a third domain, being the TBLBody. This body will be used to simulate the thermal boundary layer.

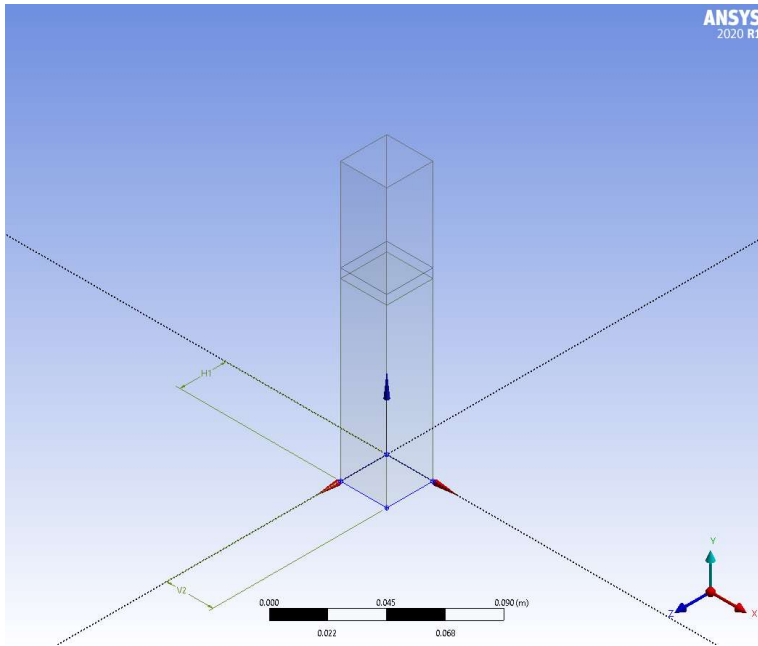


Figure 13 - Thermal Boundary Layer Geometry Setup: Final Geometry graphic, note the inclusion of the extra body to simulate the thermal boundary layer.

3.4.5 Thermal Boundary Layer Simulation Mesh Setup

As mentioned in the overview section the mesh generation will be uncontrollably different from each depth model to the next due to the program accounting for the BodyTBL. The meshing setup however will be similar to that of the control model with a few additional steps. The steps are as follows;

- I. Create the named sections for the following components

Body Names

- A. BodyAir
- B. BodyWater
- C. BodyTBL

The difference compared to the base model is the inclusion of BodyTBL.

Face Names

- D. Water+z
- E. Water-z
- F. Water+x
- G. Water-x
- H. Air+z
- I. Air-z
- J. Air+x
- K. Air-x
- L. TBL+z
- M. TBL-z
- N. TBL+x
- O. TBL-x
- P. AirWaterInt

- Q. Top
- R. Base

While similar to the procedure in section 3.3.3 new named selections are required to account for the TBLBody domain.

- II. Set Model to CFD
- III. Set default mesh size to 0.001m
- IV. Create “Face-Sizing” parameters for the following named sections
 - A. AirWaterInt – 0.0001m
 - B. Top – 0.001m
 - C. Base – 0.001m
- V. Check all bodies are set as “fluid”
- VI. Check Statistics

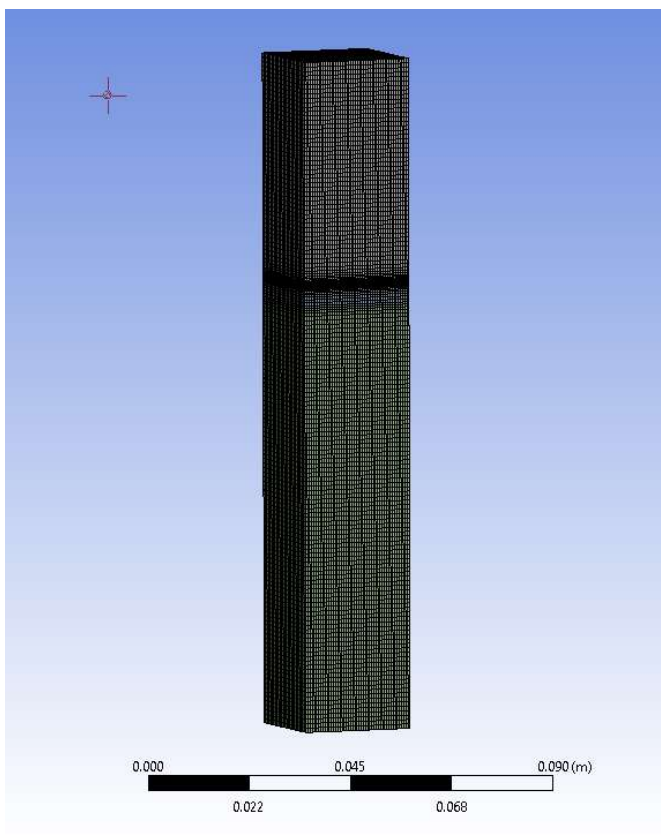


Figure 14 - Thermal Boundary Layer Mesh Setup: Graphic after final meshing completed.

3.4.6 Thermal Boundary Layer Simulation *Fluent* Setup

The base model has provided the foundation for the *Fluent* Setup of the Thermal Boundary Layer. A number of changes in the setup process were however required to allow the software to provide the results needed for the comparison.

The temperature difference within the Thermal Boundary Layer needed to be addressed. As mentioned in the parameters section the temperature difference being simulated ranges from -0.6 degrees Celsius to -0.2 degrees Celsius. The way this project provides this is by making alterations to the “Cell Zone Conditions”. The AirBody and WaterBody will have the same parameters as the Base Model however the new TBLBody will be set to either -0.6, or -0.2 degrees Celsius different to that

of the WaterBody. In the case of the control simulations the TBL and Water body were both set to the same value.

The solution methods were required to be modified for both accuracy and stability of simulation. As second order upwind in Fluent uses two points for the computation rather than one as in the first order upwind it results in more accurate results. Where possible the methods have been set to second order upwind. It is necessary to however provide two exceptions. Second order upwind for the energy calculation is problematic and causes “floating point exceptions”. The volume fraction is to be calculated using the Modified HRIC method. This is because it is using two upwind and one downwind point for the calculation, similar to the second order upwind methods. As the Eulerian model is used for the multiphase flow, the Phase Coupled algorithm is also required.

The default residual monitor of 0.001 is not considered sufficient for this model. Using the default criteria will not guarantee an accurate solution. For the purpose of this project a tolerance of 0.0001 is considered acceptable. All monitors had the Absolute Criteria set to 0.0001.

The number of time steps is also altered from the base model. Due to the need for greater accuracy the number of max iterations had also needed to be substantially increased. To rectify this issue the time steps selected at 60 seconds. With the increase in iterations processing durations it became unfeasible to provide for 60 timesteps, therefore this was reduced to 10. With a greater need for accuracy the maximum iterations have been set to 2000, however this was reduced to 1000 after crashing issues required Fluent to be run in serial mode. 2000 iterations allowed for consistent residuals of approximately 1-E5 which in many cases is considered accurate (Kuron 2015), does result in reliable convergence and is balanced regarding processing times (10 hours, which increased to close to 24 hours when run in serial mode).

The process to create the Fluent system component is as follows;

- I. Set from steady to transient
- II. Set Gravity to -9.81m/s in Y direction
- III. Use Fluent Data base to include Water liquid and Water Vapor in materials
- IV. Turn on Energy Equation
- V. Turn on Viscous equation and set to K-Epsilon with Scalar Wall Functions
- VI. Set Up Multiphase Model
 - A. Select Eulerian Model
 - B. Set Phases to 3
 - C. Assign each Air, Water, Liquid to a phase and label accordingly
 - D. Turn on Two-resistance Model for Heat Transfer
 - E. Select Mass Transfer and set to 1
 - F. Set phase liquid to phase vapor as Evaporation/Condensation using Thermal Phase Change model
- VII. Assign constant temperature of 24 degrees Celsius for *WaterBody* and 24 degrees – (0.6,0.4 or 0.2) for TBLBody
- VIII. Boundary Conditions
 - A. Water+z - Symmetry
 - B. Water-z - Symmetry
 - C. Water+x - Symmetry
 - D. Water-x - Symmetry
 - E. Air+z - Symmetry
 - F. Air-z - Symmetry

- G. Air+x - Symmetry
 - H. Air-x - Symmetry
 - I. AirWaterInt - Internal
 - J. Top – Pressure Outlet, thermal backflow set to 29 degrees Celsius
 - K. Base – Wall, thermal temperature set to 24 degrees Celsius
- IX. Methods
- A. Gradient – Least Squares Cell Based
 - B. Pressure – PRESTO!
 - C. Momentum – Second Order Upwind
 - D. Volume – Modified HRIC
 - E. Turbulent Kinetic Energy – Second Order Upwind
 - F. Turbulent Dissipation Rate – Second Order Upwind
 - G. Energy – First Order Upwind
- X. Monitors – All Residual monitor’s Absolute Criteria is set to 0.0001
- XI. Calculation Activities – record every 1 timestep
- XII. Initialization from all domains
- A. Initial Temperatures, 24 for water phase in and 29 for air phase
 - B. Patch WaterBody as volume function 1 of Water
- XIII. Run Calculation
- A. 10 Time Steps
 - B. 1 Second Time Steps
 - C. 2000 Max Iterations

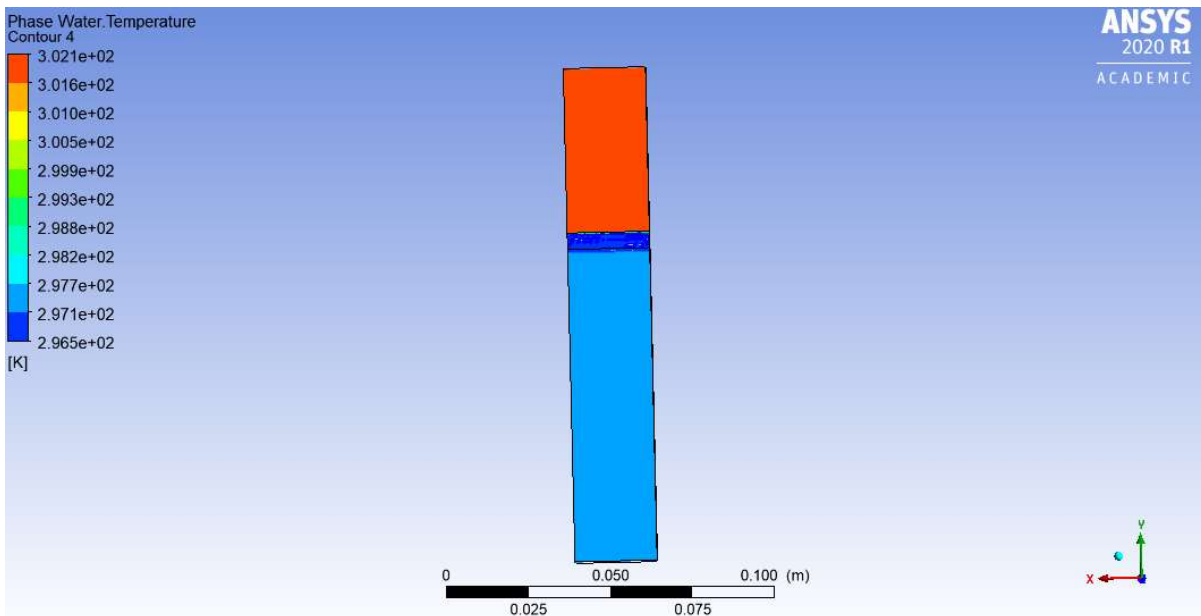


Figure 15 - Thermal Boundary Layer Results: Results graphic showing temperature of the three bodies after the final timestep, note the thermal boundary layer set to -0.6 degrees cooler than the water body.

3.4.7 Results System Setup

To determine the rate of evaporation the volume fraction of the water will again be monitored. The domain that the water will be monitored however will instead a “Plane” set up on the XY axis halfway through the model on the Z axis. This method was chosen as it removed the requirement for post results calculations. The other option would require the TBL Body and the Water Body to be recorded separately and the results combined. As there is no flow of water this plane provides an

accurate representation of the water volume for the entire model. The equation used to produce the results is;

$$\text{ave}(\text{Phase Water.Volume Fraction})@\text{plane3}$$

A plane is also set up on the x axis, at the mid-point of the y axis in the model. Graphics produced are both the volume of water fraction as well as the temperature of the water. The result output is in terms of the liquid phase water function.

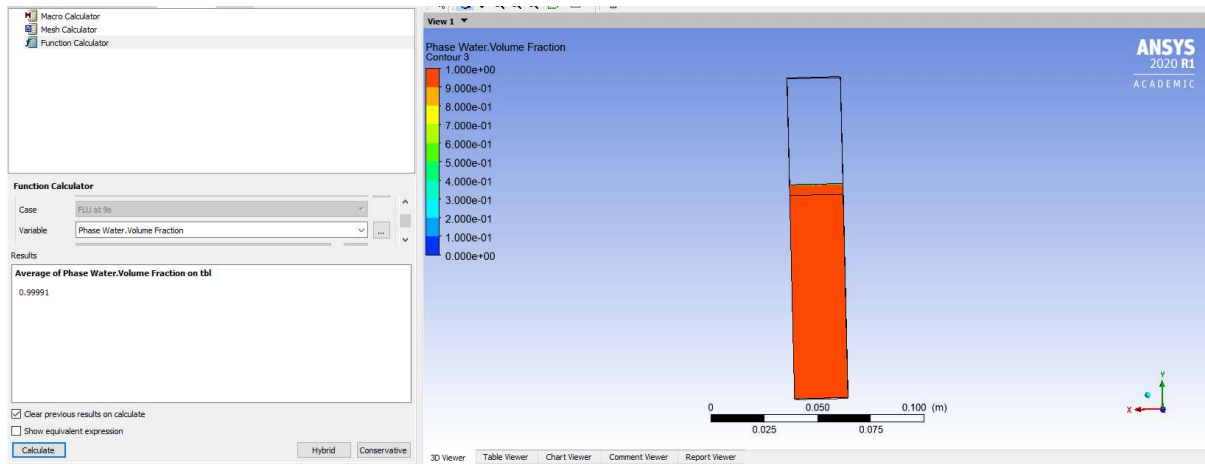


Figure 16 - Sample Results Generation of Thermal Boundary Layer Model: Screen Shot of the function calculator to record the simulation results.

3.4.8 Thermal Boundary Layer Simulation Comparisons

The rates of evaporation are to be compared from the thermal boundary layer simulations to that of the control models. The results are provided in the Water Volume Fraction of the thermal boundary layer. These volume fractions are to be recorded for each time step and for the entire duration. These can then be mathematically converted into rate of evaporation in terms of kg/m^2 or grams evaporated if required. The percentage difference from the individual parameter simulations to the controls is also calculated. This information is then graphed for easy interpretation of data.

3.5 Thermal Boundary Layer Simulation Further Modelling

After running the initial thermal boundary layer models, it was the case that the selected parameters did not show any notable or conclusive difference between the models that incorporate a thermal boundary layer and those that do not. Due to the limited scope of the modelling it cannot be considered conclusive that the thermal boundary does not have effect on the evaporation rate. Therefore, as there was no notable difference in the results of the initial simulations further modelling was undertaken.

It was hypothesized that the lack of difference in results could stem from a number of variables;

- The rate difference is too small to measure over the proposed duration
- The model does not account for initial humidity
- The model only measures a single set of environmental parameters.
- The parameters of the Thermal Boundary Layer need to be adjusted

Increasing the duration while maintaining the timestep as selected in section 3.4.6 is unfeasible as the process times are too long. From various attempts it has been noted that 30 second timesteps are the greatest standard number that will not cause floating point errors. The same setup will be run as discussed above albeit with 30 second timesteps. This will provide results over a 15-minute

time period. This greater duration is to check if the results between the models will diverge over time.

An overlooked parameter of the initial models was the initial humidity was not properly accounted for. The volume fraction of water vapor can be patched following initialization. The backflow volume fraction is also to be set to ensure the water vapor to air volumes remain within the same ratio. The ratio of water vapor within the air domain will be set relative to the average relative humidity for the specified month. The average humidity for February is 62% (BOM 2020), this however needs to be converted to actual humidity to be inputted in as volume fraction. A relative humidity accounts for 1.588% water volume in the air, therefore the during the initialization the volume fraction of water vapour in the air body is to be set to 0.01588. Due to the limited time, minimal vapour creation and size of the air domain any vapour created will only have a negligible effect on the relative humidity. For this reason, the backflow volume fraction of the air vapour will be set to 1. Due to time constraints this test was only run for the 1mm thermal boundary model.

Building on from the humidity test, the next simulation was to combine a number of factors. This includes temperature, iterations and duration. The effects might be either negligible or too small to accurately measure with the proposed methodology. As evaporation rates could not be measured using the prescribed parameters detailed in section 2.2.2 testing beyond these was required. Section 2.2.2 provides a range of depths and temperatures to be modelled. Temperature is seen as the critical factor for this project, so the temperature difference was needed to be extended beyond this range. Initially this temperature difference was doubled to 1.2 degrees cooler. As from the previous iterations, it was determined that an extended duration is required, to achieve this the time step is increased to 10 seconds. However, due to time constraints the number of iterations was reduced to 5. Setting the backflow volume to 1 as per the previous test this only works in very limited durations, if the timestep is extended to 10 seconds, the backflow volume fraction of the air vapour needs to be reduced to 0.95. This will still cause the humidity to fluctuate however it will remain relatively stable. Note, that this number was determined from an iterative method. From the initial simulations it was determined that 1000 max iterations are too few, so this was also increased to 2000. Also, due to time constraints this test was only run for the 1mm thermal boundary model as time was a major constraint.

The model is based on average Brisbane Summer (Month of February) air and water temperatures. This is a very narrow scope of environmental parameters. The purpose of this project is to find under what parameters the thermal boundary layer will affect evaporation, so to understand if the surrounding air and water temperatures may alter the result a Winter model will be run if required. The air temperature will be set to 21 degrees Celsius which is the average mean maximum temperature of the Months of June and July (Bom 2020), the water temperature of Wivenhoe Dam gets as low as 15 degrees Celsius (McJannet 2008) so this temperature will be selected. The relative humidity will also be set to 50% (Weather Atlas 2020) The model was run in the same format as described in the paragraph above however with the initial temperatures altered and by error only 4 timesteps run instead of 5.

3.6 Thermal Boundary Layer Simulations under Realistic Conditions

3.6.1 Thermal Boundary Layer Simulations under Realistic Conditions Introduction

Once the effects of the Thermal Boundary Layer had been established in a simplified model the factor of wind was included to create a more realistic model. Wind is a significant contributing to evaporation and the project is to understand evaporation in open bodies of water it is important to understand how it functions when the thermal boundary layer is included.

3.6.2 Wind Simulation Parameters

Following the results of the thermal boundary layer simulations, the set of parameters with the highest difference between the thermal boundary layer model and the control model will be chosen. The wind speed chosen for this simulation will be 10km/h, this was selected as the average wind speed of 18.5 km/h for February Brisbane (BOM 2020) is considered too high for such a small domain.

3.6.3 Wind Simulation Overview

To compare evaporation rate results three models were set up. The first model was set up as a control with the thermal boundary layer not modelled, the other two have a thermal boundary layer model set to 0.6 degrees and 1.2 degrees Celsius cooler than the rest of the body of water. This provided the relevant information to compare the effect of the thermal boundary layer under realistic circumstances.

3.6.4 Wind Simulation Geometry Setup

The geometry system was set up to be similar to the other models described in sections 3.3 and 3.4 however differed slightly, being more rectangular than square. The long axis of the rectangle was then coincided with the direction of the wind. This will be set up similarly using the *Extrude* and *Boolean* functions. Geometry set ups will be exactly the same for the control and the wind model

Creation of the geometry set up is as follows;

- I. 0.005x0.1m rectangle in XY Plane – This is different to that of the previous models as the longer axis will be in the direction of the wind
- II. Generate this rectangle as a sketch
- III. Extrude 0.15m in y direction by using Add Frozen function
- IV. Extrude 0.1m in y direction by using Add Frozen function
- V. Extrude 0.1m – *thermal boundary layer depth* by using Add Frozen Function
- VI. Using Boolean subtract function on 0.1m body from 0.15 body – Selecting Preserve Bodies
- VII. Using Boolean subtract function on 0.1m – *thermal boundary layer depth* body from 0.1body – Selecting Preserve Bodies
- VIII. Names will be set to AirBody, WaterBody, TBLBody
- IX. Select all bodies and combine into single part
- X. Material will of both bodies set to “fluid”

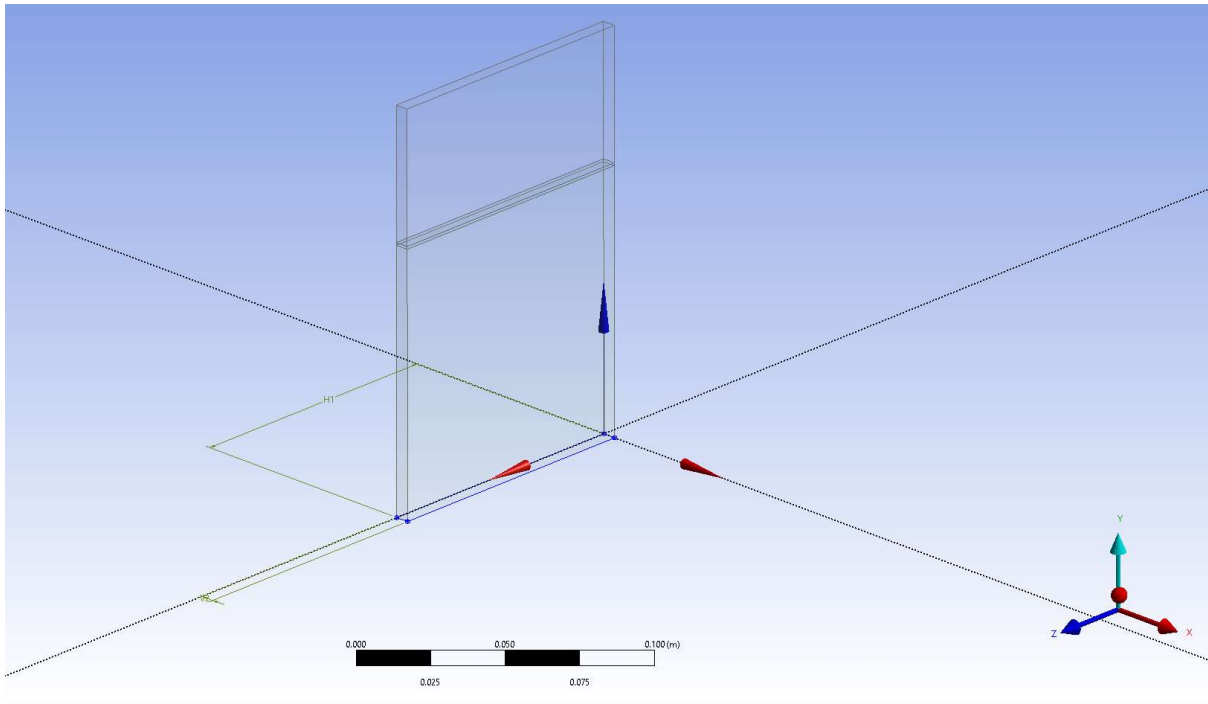


Figure 17 - Wind Model Geometry Setup: Graphic after geometry set up complete, note rather than a square as per previous models this shape is a rectangle.

3.6.5 Wind Simulation Mesh Setup

There is essentially no difference in the Mesh Setup when compared to the Thermal Boundary Layer setup as shown in section 2.2.3.3. It should however be noted that the wind will act on the z axis. This why it was important to include the +z or -z nomenclature as it will easily allowed for developing the model and recording results.

- I. Create the named sections for the following components

Body Names

- A. BodyAir
- B. BodyWater
- C. BodyTBL

Face Names

- D. Water+z
- E. Water-z
- F. Water+x
- G. Water-x
- H. Air+z
- I. Air-z
- J. Air+x
- K. Air-x
- L. TBL+z
- M. TBL-z
- N. TBL+x
- O. TBL-x

- P. AirWaterInt
- Q. Top
- R. Base
- II. Set Model to CFD
- III. Set default mesh size to 0.001m
- IV. Create “Face-Sizing” parameters for the following named sections
 - A. AirWaterInt – 0.0001m
 - B. Top – 0.001m
 - C. Base – 0.001m
- V. Check all bodies are set as “fluid”
- VI. Check Statistics

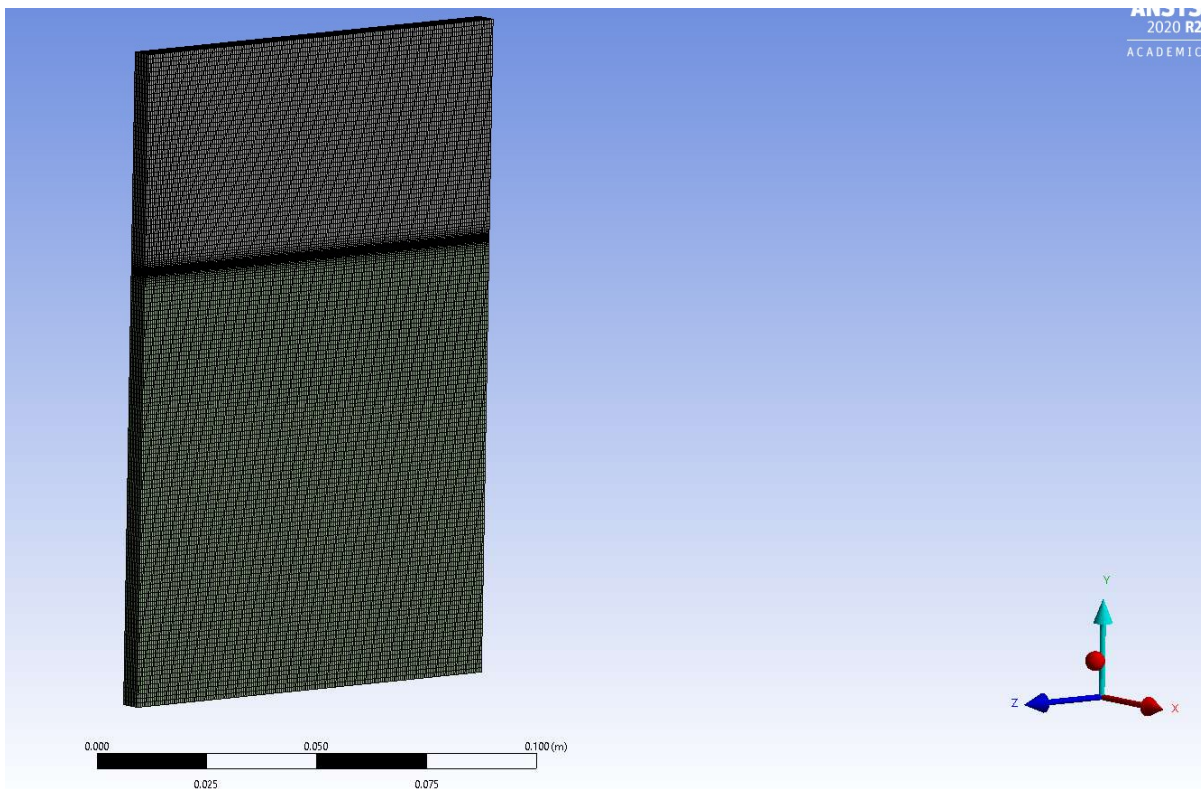


Figure 18 - Wind Model Mesh Setup: Graphic after meshing completed, note it is the same as per previous models.

3.6.6 Wind Simulation *Fluent* Setup

As the wind model is only further developing from the Thermal Boundary Layer model the majority of the *Fluent* setup will be similar. The notable difference between the two models is the boundary conditions of the “air+z” and “air-z” faces. These faces are to be set up a velocity inlet and pressure outlet. As it is proposed for the air to flow in the positive z direction, air-z will be set as the pressure inlet. Using the velocity function, the wind speed will be set to 10km/h (2.7778m/s).

From data gathered from the further modelling defined in section 3.5 and the subsequent results in section 4.4 the *Fluent* Setup methodology has progressed to create more reliable data. The improvements noted from alterations to the model were noted as part of these tests and have been incorporated into the Wind Simulation Model.

Several thermal boundary layer tests in section 3.4 were run with a maximum number of iterations of 1000. The results of the further modelling however provided evidence that 2000 produces a more accurate result, therefore had been adopted. Due to the processing time constraints the number of

timesteps needed to be reduced to 5 as per the further modelling tests, it was considered more beneficial to have fewer more accurate results than more with inherent inaccuracies. Due to the increased complexity of this model, possibly more would have been beneficial however time constraints did not allow for this. To test some validity of the results, a separate simulation was run with the extended number of time steps of ten.

The initial humidity errors of section 3.4 were also validated and therefore to improve on the model the boundary conditions have been modified to provide stable humidity for the duration of the simulation. To achieve this the top outlet has had the backflow volume for the water vapor phase set to 0.95, while not achieving a perfectly stable level of humidity it is considered enough for this task, this was confirmed through review of the humidity volume fraction results. The initial humidity is also set by using the volume fraction patch function and setting 0.01588 as per previous models to simulate a relative humidity of 62%, or the average for Brisbane in the month of February (Bom 2020)

Timesteps of some of the further modelling tests in section 3.5 were set to 10 seconds. While this allowed for a greater duration to be able to be recorded with limited impact on processing times from initial tests with the wind model results were unreliable due to the increased complexity of the model. It was determined that 5 second timesteps allowed for the best balance between duration and accuracy. A 5 second timestep was adopted for this simulation.

From the results in section 4.4 the modelling showed measurable difference in evaporation rate of the models with 0.6-degree temperature difference in the thermal boundary layer but a notable difference was identified when tested with the expanded parameter of 1.2 degree temperature difference. Therefore, to test this influence attributed to the thermal boundary layer this model was run with a thermal boundary layer temperature difference of 1.2 degrees. As this is outside the normal parameters of the thermal boundary layer determined in section 2.4 a model with a 0.6-degree difference was also run.

Primarily due to from what is known about the thermal boundary layer and from the results of previous testing it was determined that only modelling will only include a modelled thermal boundary layer of 1mm thick. This decision was also made due to time constraints. The 0.1mm model was excluded from the realistic condition testing due to the inherent inaccuracies of the model. From testing in section 3.4 it was identified that a thermal boundary layer of such small dimensions caused issues with the meshing and providing results. While a relative comparison regarding the effects of the boundary could still be undertaken in section 4.4, in this instance where the results will also need to be compared to a numeric model it cannot be included. A thermal boundary layer of 5mm depth was also omitted due to the intention of this test being to mimic realistic conditions. From what is known about the thermal boundary layer is that it thins out due to wind (Mckeown 1997), as this test will have significant wind a maximum depth thermal boundary layer would unlikely occur in realistic conditions. For these reasons all tests regarding the wind/realistic model occurred with a 1mm deep thermal boundary layer.

The process to create the Fluent system component is as follows;

- I. Set from steady to transient
- II. Set Gravity to -9.81m/s in Y direction
- III. Use Fluent Data base to include Water liquid and Water Vapor in materials
- IV. Turn on Energy Equation
- V. Turn on Viscous equation and set to K-Epsilon with Scalar Wall Functions

- VI. Set Up Multiphase Model
 - D. Select Eulerian Model
 - E. Set Phases to 3
 - F. Assign each Air, Water, Liquid to a phase and label accordingly
 - G. Turn on Two-resistance Model for Heat Transfer
 - H. Select Mass Transfer and set to 1
 - I. Set phase liquid to phase vapor as Evaporation/Condensation using Thermal Phase Change model
- VII. Assign constant temperature of 24 degrees Celcius for *WaterBody* and 24 degrees – (0.6 & 1.2) for *TBLBody*
- VIII. Boundary Conditions
 - J. Water+z - Symmetry
 - K. Water-z - Symmetry
 - L. Water+x - Symmetry
 - M. Water-x - Symmetry
 - N. Air+z – Pressure Outlet;
 - The water backflow volume needs to be set to 1 to stop liquid water leaving the domain.
 - O. Air-z – Velocity Inlet
 - Volume fraction of the water vapor is to be set to 0.01588 and velocity to 2.7778m/s
 - Air velocity to be set to 2.7778m/s
 - P. Air+x - Symmetry
 - Q. Air-x - Symmetry
 - R. AirWaterInt - Internal
 - S. Top – Presure Outlet, thermal backflow set to 29 degrees Celsius
 - T. Base – Wall, thermal temperature set to 24 degrees Celsius
- IX. Methods
 - U. Gradient – Least Squares Cell Based
 - V. Pressure – PRESTO
 - W. Momentum – Second Order Upwind
 - X. Volume – Modified HRIC
 - Y. Turbulent Kinetic Energy – Second Order Upwind
 - Z. Turbulent Dissipation Rate – Second Order Upwind
 - AA. Energy – First Order Upwind
- X. Monitors – All Residual monitor’s Absolute Criteria is set to 0.0001
- XI. Calculation Activities – record every 1 timestep
- XII. Initialization from all domains
 - BB. Initial Temperatures, 24 for water phase in and 29 for air phase
 - CC. Patch *WaterBody* as volume function 1 of Water
 - DD. Patch Thermal Boundary Layer Temperature as required (24, 23.4 or 22.8 deg. C)
 - EE. Patch *AirBody* as volume function of 0.01588
- XIII. Run Calculation
 - FF. 5 Time Steps
 - GG. 5 Second Time Steps
 - HH. 2000 Max Iterations

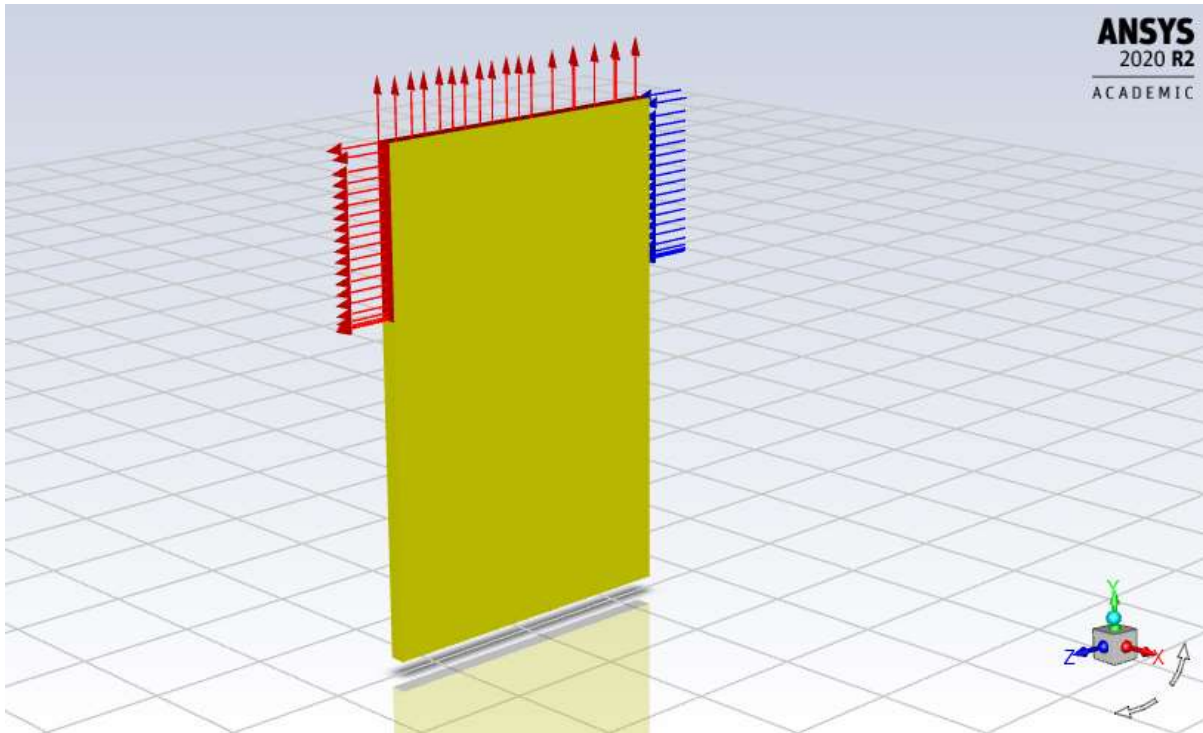


Figure 19 - Wind Model Fluent Setup Graphic: Opening graphic screen shot in fluent, images shows the inlet and outlets of the model.

3.6.7 Results System Setup

To determine the rate of evaporation the volume fraction of the water will again be monitored. The domain that the water will be monitored however will instead be a new plan set up as “plane4”. Plane 4 will run along the zy axis in the middle of the domain on the x axis. Unlike the previous plane 4 in section 3.5 this plan will be through the entire domain, including the air body. The equation used to produce the results is;

$$\text{ave}(\text{Phase Water.Volume Fraction})@\text{plane4}$$

A plane is also set up on the x axis, at the mid-point of the y axis in the model. Graphics produced are both the volume of water fraction as well as the temperature of the water. A plan set up on the z axis at the mid-point of the x axis is also required. This will monitor the water volume again.

The results will be presented as a volume fraction of the “Plane4”, that is why it is important to be in the direction of the wind otherwise displacement caused by the wind will alter the results. It should also be noted then that the initial water volume won’t be 1 as in previous models but 0.645182 as this is the initial volume fraction of water in the entire model.

3.6.8 Wind Simulation Comparisons

To determine the effect of the Thermal Boundary Layer under realistic conditions the results of the wind model and the control model were compared. Again, this was done by comparing the change in the volume fraction. The percentage differences from the Wind Model versus the control were then compared. For an overall understanding of the accuracy of this model the Blaney-Cridde Method is to be used to calculate the numerical evaporation rate under the same parameters. The Blaney-Cridde method estimates evaporation in terms of mm/day, this will need to be converted to grams/sec. The volume fraction results from the simulation will also need to be converted to grams/sec to allow for simplified comparison.

Chapter 4: Results and Discussion

4.1 Results Overview

The results of the simulations will be split into the three different stages of the model development. Firstly, the results of the model validation, followed by the thermal boundary layer simulations and finally the results for the model under realistic conditions. As mentioned in previous sections and due to expected differences in results between models' results cannot of different models cannot be directly compared. Each component result was examined individually, however the overall trends were compared. Conclusions were drawn from this analysis.

4.2 Base Model Analysis

The purpose of this stage of the project was to validate the computer simulated model against an accepted numerical calculation. The results of which were directly compared to each other. Some margin of error was expected but correlation between the results was also considered a required.

4.2.1 Numerical Calculation

As mentioned in the methodology section, the Shah Equation has been used to develop a calculated rate of evaporation. This result will provide the basis for the numerical component of the comparison. The calculation performed for this task is as follows;

$$E_o = Cp_w(p_r - p_w)^{\frac{1}{3}}(W_w - W_r)$$

C = the coefiecent for either imperail or SI units

$p_r - p_w$ = differnce density of air between room temp. and water surface temp.

$W_w - W_r$

= diffence of specific humidity of air between room temp. and water surface temp.

$C = 35$; Coeffiennt from (Shah 2014)for SI units

$p_r - p_w = 0.019\text{kg}/\text{m}^3$ (Fly Carpet 2020)

$W_w - W_r = 0.0161\text{kg}/\text{kg}$

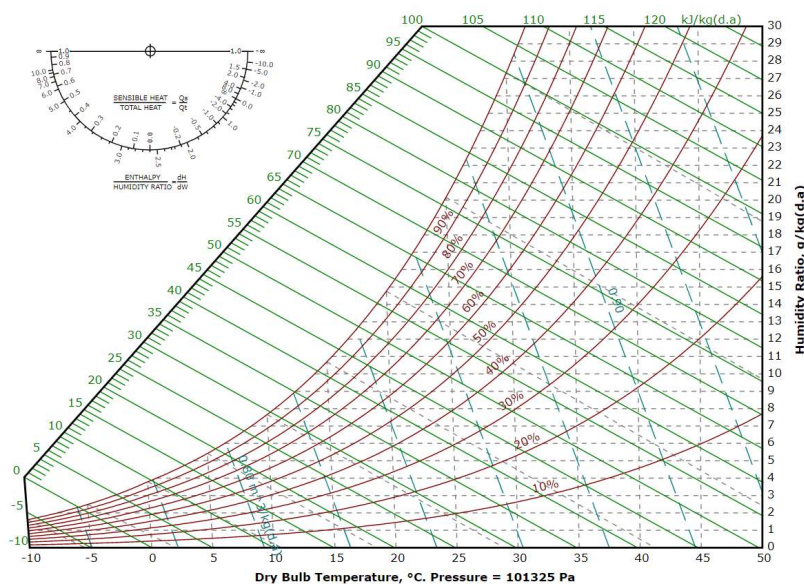


Figure 20 - Psychrometric Chart (Fly Carpet 2020): This chart was used to determine the $p_r - p_w$ values for the calculation.

$$E_o = 35 \cdot 1.175(0.019)^{\frac{1}{3}}(0.0161)$$

$$E_o = 0.17667 \text{ kg/m}^2 \cdot \text{h}$$

4.2.2 Base Model Simulation

A CFD simulation was undertaken in the format described in the methodology section. The water volume fraction of the final timestep in the water body was taken to determine total evaporation; 61st (60 seconds) timestep had a water volume of 0.999946 within the water body

For ease of comparison this is then converted into figures of kg/m².h;

Determine total loss of mass (kg) by multiplying volume fraction by volume

$$M = 0.999946 \times 0.00625$$

$$M = 0.062496625 \text{ kg}$$

Convert to kg/m²

$$E_o = 0.00062496625 / 0.00625 \times 60$$

$$E_o = 0.16665 \text{ kg/m}^2 \cdot \text{h}$$

4.2.3 Model Validation

The calculation using the Shah Equation (Eq.2) resulted in an evaporation rate of 0.176678kg/m².h while the simulated model provided a rate of 0.16665kg.m².h. This results in a roughly 9% difference. As this is less than 10%, a figure of which most evaporation measuring techniques have approximately as a margin of error (Finch 2008), the model simulation is considered validated. The difference in results could also be attributed to the approximate nature of the simulation used for this test. This simulation was only intended to provide a rough estimation for validation. Accuracy could have been increased with the simulation occurring over a larger duration or more iterations per timestep. However, as this would result in extended durations to run the simulation the provided results are considered acceptable for this project.

4.3 Thermal Boundary Layer Model Analysis

Following successful validation of the model and multiphase simulation in section 4.2, simulations accounting for the thermal boundary layer in standard configurations were run. The parameters of these models are provided section 3.4.2 of this report. The results of these models are reviewed and compared in this section. The ultimate goal of this comparison is to determine whether the thermal boundary layer has impact on the rate of evaporation.

4.3.1 5mm Thermal Boundary Layer Model Results

The first set of models run include a thermal boundary layer of 5mm deep. The models have been run in three varieties; a control (where the thermal boundary layer temperature is set to the same as the below water temperature), a -0.6-degree model and a -0.2-degree model. The results provided in the table below are in terms of water volume fraction within the domain.

Table 2 - TBL 5mm Control Results

Name	5mm Control
Version	17
Date	30/08/2020
Air Temp. (deg. Kelvin)	302.15

Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	297.15
Max Iterations	2000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999983	0.000017
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999994	0.000002
3	0.999991	0.000003
4	0.999989	0.000002
5	0.999987	0.000002
6	0.999986	0.000001
7	0.999985	0.000001
8	0.999984	0.000001
9	0.999984	0.000000
10	0.999983	0.000001

Table 3 - TLB 5mm -0.6deg. Results

Name	5mm06
Version	17
Date	1/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	2000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999983	0.000017
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999994	0.000002
3	0.999991	0.000003
4	0.999989	0.000002
5	0.999987	0.000002
6	0.999986	0.000001
7	0.999985	0.000001
8	0.999984	0.000001

9	0.999984	0.000000
10	0.999983	0.000001

Table 4 - TBL 5mm -0.2 deg. Results

Name	5mm02
Version	17
Date	3/9/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.95
Max Iterations	2000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999983	0.000017
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999994	0.000002
3	0.999991	0.000003
4	0.999989	0.000002
5	0.999987	0.000002
6	0.999986	0.000001
7	0.999985	0.000001
8	0.999984	0.000001
9	0.999984	0.000000
10	0.999983	0.000001

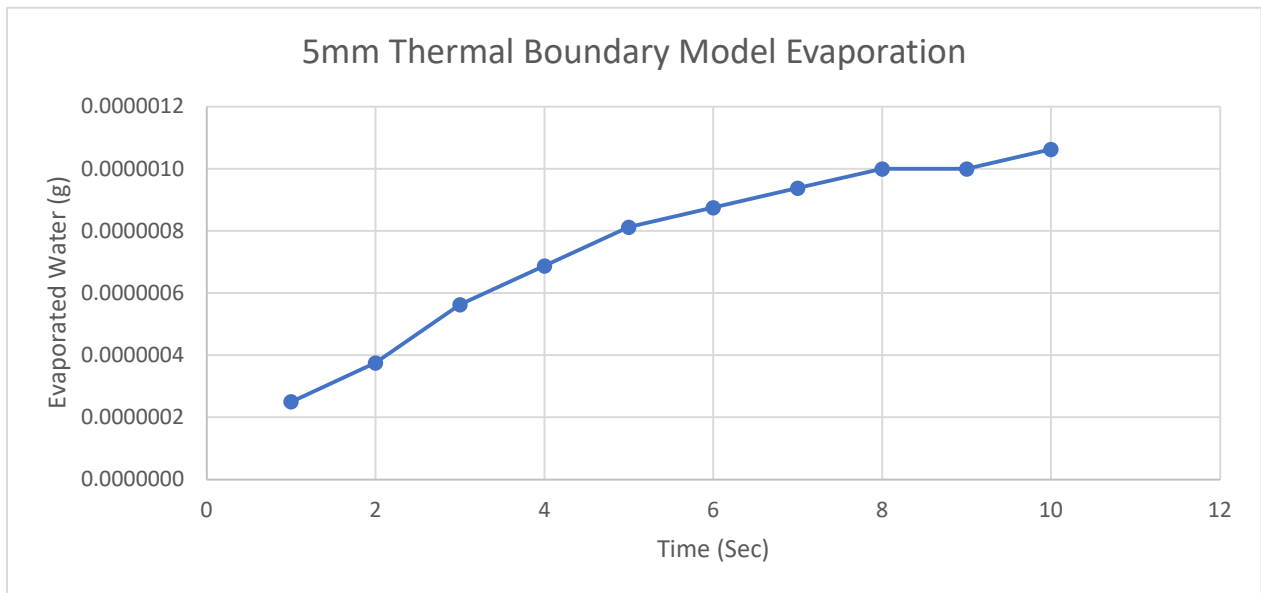


Figure 21 - 5mm Thermal Boundary Model Total Evaporation (g) at each Timestep

4.3.2 5mm Thermal Boundary Layer Model Discussion

With results from all three model set ups for a 5mm thermal boundary layer, comparisons and notes regarding the model can be drawn. The initial finding is that there is no difference regarding the rate of evaporation. The 5mm model was initially run as it was suspected to be the most likely of causing an altered evaporation rate. However, all three models provided the exact same figures across the entire simulation.

From review of the results other items need to be considered. It appears that the evaporation rate appears to drop over time. The initial difference in the water volume fraction is 0.000004 accounting for an evaporation rate of $1.44\text{kg/m}^2\cdot\text{h}$. It should be noted that this rate is much higher than the model validation result and the calculated result, which were 0.166665 and $0.1776678\text{kg/m}^2\cdot\text{h}$ respectively. However, this rate rapidly decreased after the first few timesteps and began approaching that of the validation model and numerical calculation in the later timesteps. Also, the zero difference between timesteps at the ninth timestep is likely not that zero evaporation occurred but rather the amount was too small to be rounded to 0.000001.

It is suspected that if the model was run for a greater duration the evaporation rate for the total time would converge of that in the validation simulation and numerical calculation. However, due to suspected issues with the *Fluent* software at the time of these calculations, running in parallel mode was no longer possible. This has resulted in calculation times being of greater than 24 hours for 10 iterations, a longer duration simulation with the same time step was simply considered no longer feasible. It is evident that the numerical values returned by the results are on the limit of what can be acceptable due their very small size. Regarding accuracy however the residuals for the volume fraction of the Fluent Calculation sit at approximately $1\text{E-}5$ therefore this is considered adequate for the purpose of this dissertation.

4.3.3 1mm Thermal Boundary Layer Model Results

Following the unexpected result of the 5mm thermal boundary layer test showing no measurable difference in the evaporation rate, the 1mm thermal boundary layer tests where run. It is important to note that due to the limitations with *Fluent* at this time all tests have been run in serial mode. To allow for the same number of timesteps to be run, the max iterations has been reduced to 1000

aside from the 5mm model which was run at 2000 iterations. The models have been run in three varieties; a control (where the thermal boundary layer temperature is set to the same as the below water temperature), a -0.6-degree model and a -0.2 degree model.

Table 5 - TBL 1mm Control Results

Name	1mm Control
Version	17
Date	18/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	297.15
Max Iterations	2000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999987	0.000013
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999993	0.000003
3	0.999992	0.000001
4	0.999991	0.000001
5	0.999991	0.000000
6	0.999990	0.000001
7	0.999989	0.000001
8	0.999989	0.000000
9	0.999988	0.000001
10	0.999987	0.000001

Table 6 - TBL 1mm -0.6 deg. Results

Name	1mm06
Version	17
Date	19/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999979	0.000021

0	1.000000	0.000000
1	0.999996	0.000004
2	0.999984	0.000012
3	0.999983	0.000001
4	0.999983	0.000000
5	0.999982	0.000001
6	0.999981	0.000001
7	0.999981	0.000000
8	0.999980	0.000001
9	0.999980	0.000000
10	0.999979	0.000001

Table 7 - TBL 1mm -0.2 deg Results

Name	1mm02
Version	17
Date	20/9/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999980	0.000020
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999988	0.000008
3	0.999987	0.000001
4	0.999984	0.000003
5	0.999983	0.000001
6	0.999982	0.000001
7	0.999981	0.000001
8	0.999981	0.000000
9	0.999980	0.000001
10	0.999980	0.000000

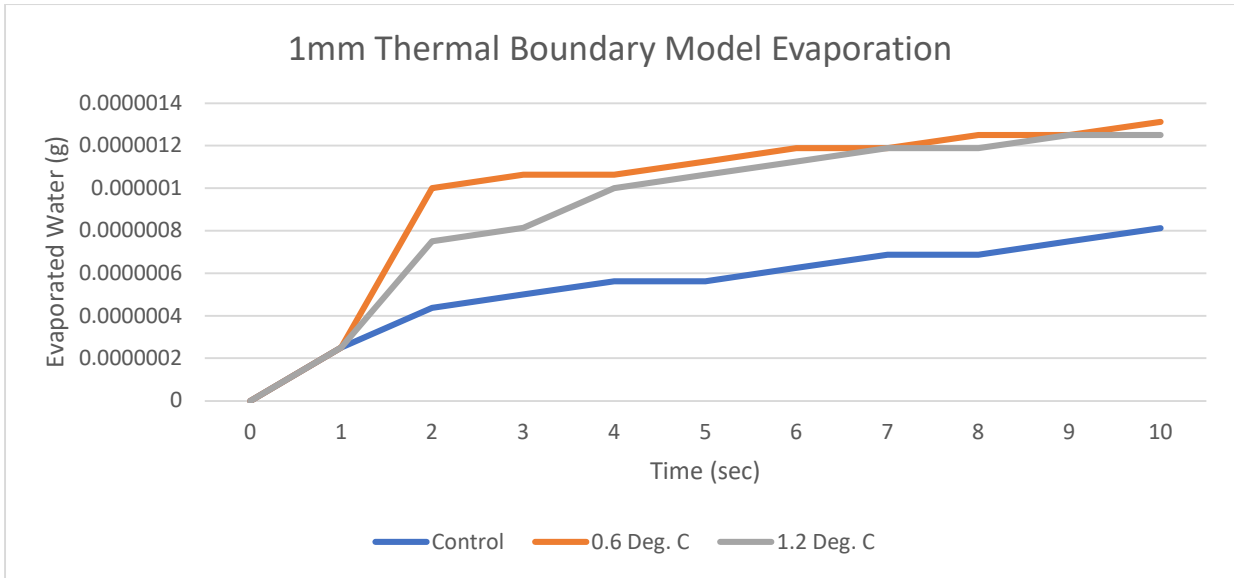


Figure 22 - 1mm Thermal Boundary Model Total Evaporation (g) at each Timestep

4.3.4 1mm Thermal Boundary Layer Model Discussion

From initial review of the results their appears to be significant difference between the three models. These differences are shown in table 8 below.

Table 8 - 1mm Model Results Comparison

	Control	0.6 Deg		0.2 Deg	
Timestep	Volume Fraction	Volume Fraction	Difference	Volume Fraction	Difference
Total	0.999987	0.999979	0.000008	0.999980	0.000007
0	1.000000	1.000000	0.000000	1.000000	0.000000
1	0.999996	0.999996	0.000000	0.999996	0.000000
2	0.999993	0.999984	0.000009	0.999988	0.000005
3	0.999992	0.999983	0.000009	0.999987	0.000005
4	0.999991	0.999983	0.000008	0.999984	0.000007
5	0.999991	0.999982	0.000009	0.999983	0.000008
6	0.999990	0.999981	0.000009	0.999982	0.000008
7	0.999989	0.999981	0.000008	0.999981	0.000008
8	0.999989	0.999980	0.000009	0.999981	0.000008
9	0.999988	0.999980	0.000008	0.999980	0.000008
10	0.999987	0.999979	0.000008	0.999980	0.000007

From further review it is considered that this conclusion may not be accurate. It appears as a result of the reduced number of iterations for each timestep there may be inconsistencies with the first few results. The largest change in volume fraction for each result occurs at the 2nd timestep. This is expected to be caused by the simulation not achieving complete convergence at this point due to the reduced iterations. It can also be noted that the final 5 timesteps of each model produce the same amount of change in volume fraction, being 0.000004. When converting this to evaporation rate in terms of kg/m².h the result is 0.288. The downward trend of evaporation rate should also be

noted. It is expected that if iterations were continued to be run this would converge on the calculated results in section 4.2.1.

To validate this assumption a new simulation was run for the Control and the -0.6 Degree model with the maximum iterations per timestep increased to 2000 but only for the first 2 timesteps. The results are confirming the assumption as both simulations provided results of 0.999992 and 0.999988 for the first and second timestep respectively. Following this test, it is clear that a number between 1000 and 2000 iterations are required to achieve convergence in the initial timesteps.

From review of the data, it is expected that the thermal boundary layer in this configuration and surrounding parameters does not have any or very minimal effect on the evaporation rate.

4.3.5 0.1mm Thermal Boundary Layer Model Results

Following the result of the 1mm thermal boundary layer test which still indicated none or a very minor difference in the evaporation rate, the 0.1mm thermal boundary layer tests were run. The issues with the parallel mode persisted in Fluent for these tests and therefore this simulation again needed to be run in serial mode. Following the same format as the 1mm depth model, the maximum iterations have been set to 1000. The models have been run in three varieties; a control (where the thermal boundary layer temperature is set to the same as the below water temperature), a -0.6-degree model and a -0.2 degree model.

Table 9 – TBL 0.1mm Control Results

Name	0.1mm Control
Version	17
Date	22/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	297.15
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999332	0.000668
0	1.000000	0.000000
1	0.999763	0.000237
2	0.999795	-0.000032
3	0.999691	0.000104
4	0.999602	0.000089
5	0.999528	0.000074
6	0.999466	0.000062
7	0.999418	0.000048
8	0.999381	0.000037
9	0.999353	0.000028
10	0.999332	0.000021

Table 10 - TBL 0.1mm 0.6deg. Results

Name	0.1mm06
Version	17
Date	24/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	difference/Time Step
Total Time	0.999488	0.000512
0	1.000000	0.000000
1	0.999922	0.000078
2	0.999902	0.000020
3	0.999865	0.000037
4	0.999794	0.000071
5	0.999717	0.000077
6	0.999652	0.000065
7	0.999595	0.000057
8	0.999548	0.000047
9	0.999510	0.000038
10	0.999488	0.000022

Table 11 - TBL 0.1mm 0.2deg. Results

Name	0.1mm02
Version	17
Date	25/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	difference/Time Step
Total Time	0.999485	0.000515
0	1.000000	0.000000
1	0.999827	0.000173

2	0.999811	0.000016
3	0.999798	0.000013
4	0.999753	0.000045
5	0.999689	0.000064
6	0.999657	0.000032
7	0.999591	0.000066
8	0.999545	0.000046
9	0.999507	0.000038
10	0.999485	0.000022

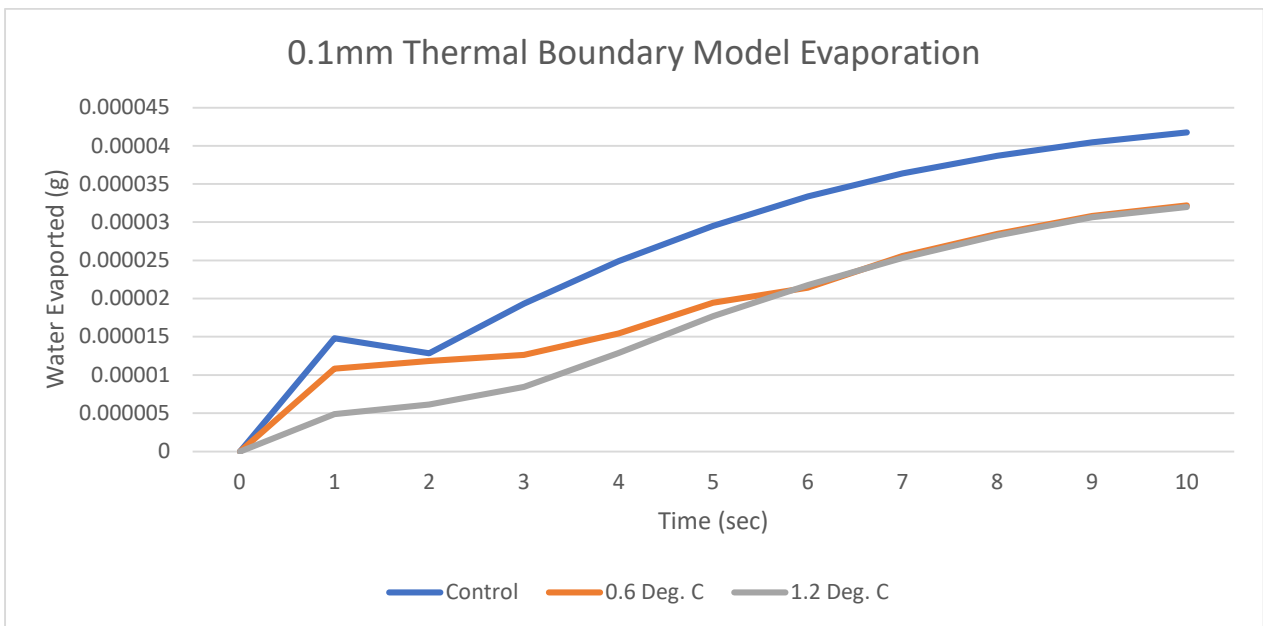


Figure 23 - 0.1mm Thermal Boundary Model Total Evaporation (g) at each Timestep

4.3.6 0.1mm Thermal Boundary Layer Model Discussion

From initial review there appears to be a notable difference between the evaporation rates of the models. However, as per section 4.3.4 there appears to be a similar correlation that as the simulation progresses the difference in timesteps converge. The differences between the 0.6- & 0.2-degree models appear to converge around the 9th timestep. Neither however converging with the control test. The validation test in section 4.3.4 would however be relatable to these results. It would be expected that if either more iterations were run or the simulation went for further timesteps the results would converge.

It should also be noted that with the 0.1mm depth model the meshing layer at the thermal boundary layer was not as consistent as the 5mm model. This is understood to be ANSYS attempting the mesh in the small model area of the thermal boundary layer domain.

From review of the data, it is expected that the thermal boundary layer in this configuration and surrounding parameters does not have any or very minimal effect on the evaporation rate.

4.3.7 Combined Thermal Boundary Layer Model Discussion

The data provided by the thermal boundary layer test methodology did not provide conclusive evidence of the evaporation rate being affected by the thermal boundary layer. It, however, also

cannot be concluded that the thermal boundary layer does not affect it, only if there is an effect, it is too minor to be measured in the developed methodology.

There is a specific point that need to be discussed with the data. It should be noted that the provided results for each thermal boundary layer depth are substantially different. The biggest difference is between the 0.1mm depth layer and the 1mm depth. This comparison is shown in the graph below, figure 15. The cause of this has been identified as a result of the meshing within ANSYS. Due to the thin domain the mesh needed to create uneven shapes to fit within the geometric area. As the nodes could not grow evenly the elements in this area are substantially larger than in the other models. This issue with meshing was discussed in section 3.2.4 and the use of separate control models was used to ensure the wrong conclusions were not drawn.

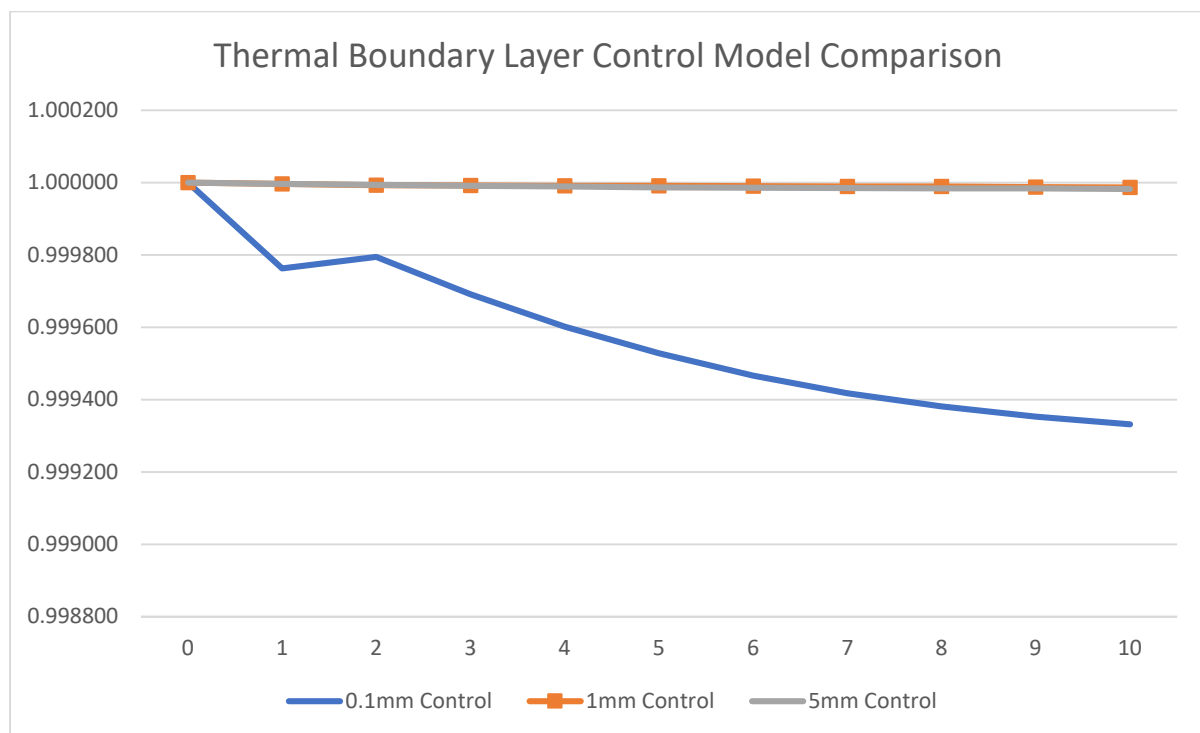


Figure 24 - Thermal Boundary Layer Control Model Comparison

Overall, all models provided mass transfer of phases through evaporation and no substantial difference in pattern is noted. To highlight this the 1mm and 5mm models are compared in the below figure 25. The noted difference at the second timestep has been concluded in section 4.3.4 as an issue with the simulation not converging due to the limitations of the number of iterations.

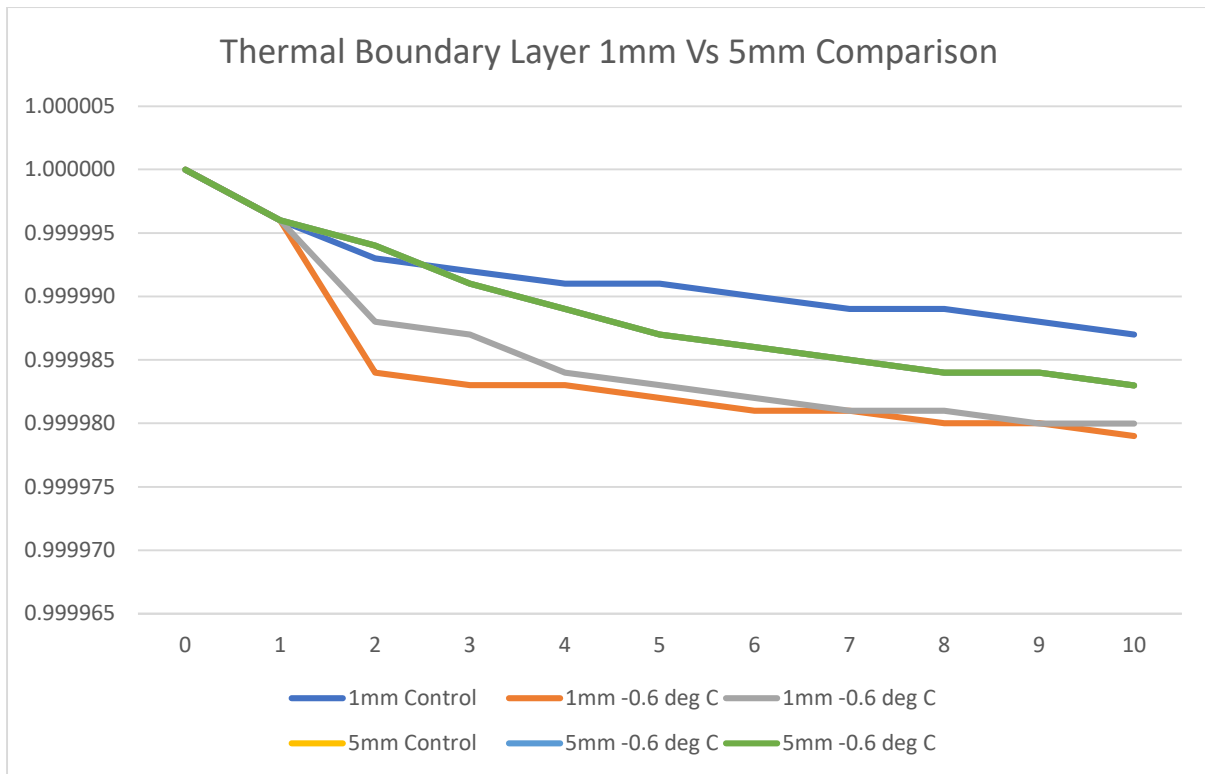


Figure 25 - Thermal Boundary Layer Model 1mm Vs 5mm Depth Comparison

These models only test very specific parameters over a short duration. Therefore, it is not considered to be able to provide conclusive evidence.

4.4 Thermal Boundary Layer Model Further Testing

Following the results of the thermal boundary layer models as detailed in section 4.3, the decision that further modelling was required was made. With only a single set of surrounding temperature conditions tested, an extremely short duration and humidity not modelled it is considered possible that there might be a combination that does detail significant effect of the thermal boundary layer on evaporation. The parameters and variable were expanded as part of the Thermal Boundary Layer model testing. This included review of a simulation taken over a much greater duration. The above results were taken as an average, Brisbane February day but initial humidity was set to essentially zero therefore a model with humidity set to the monthly average of a relative 62% or 1.588% actual volume was also run. Following this the Thermal Boundary Layer temperature will be set to double the previous maximum amount to 1.2-degree Celsius difference.

4.4.1 Extended Duration Thermal Boundary Layer Model

As the difference in evaporation is predicted to be extremely small between the control and the thermal boundary layer models, it was considered possible that due to the small duration this small difference could not be identified. To address this issue a previously ran model was reviewed. This model was for a 30-minute duration. Each time step was 60 seconds and ran for 30 timesteps.

4.4.2 Extended Duration Thermal Boundary Layer Results

The results of this simulation provided for exact figures across all three models, being the control - 0.6 & -0.2 degrees Celsius. The results are provided in Appendix F or summarised in Figure 26 below. The results provided in the graph are in terms of water volume fraction within the domain.

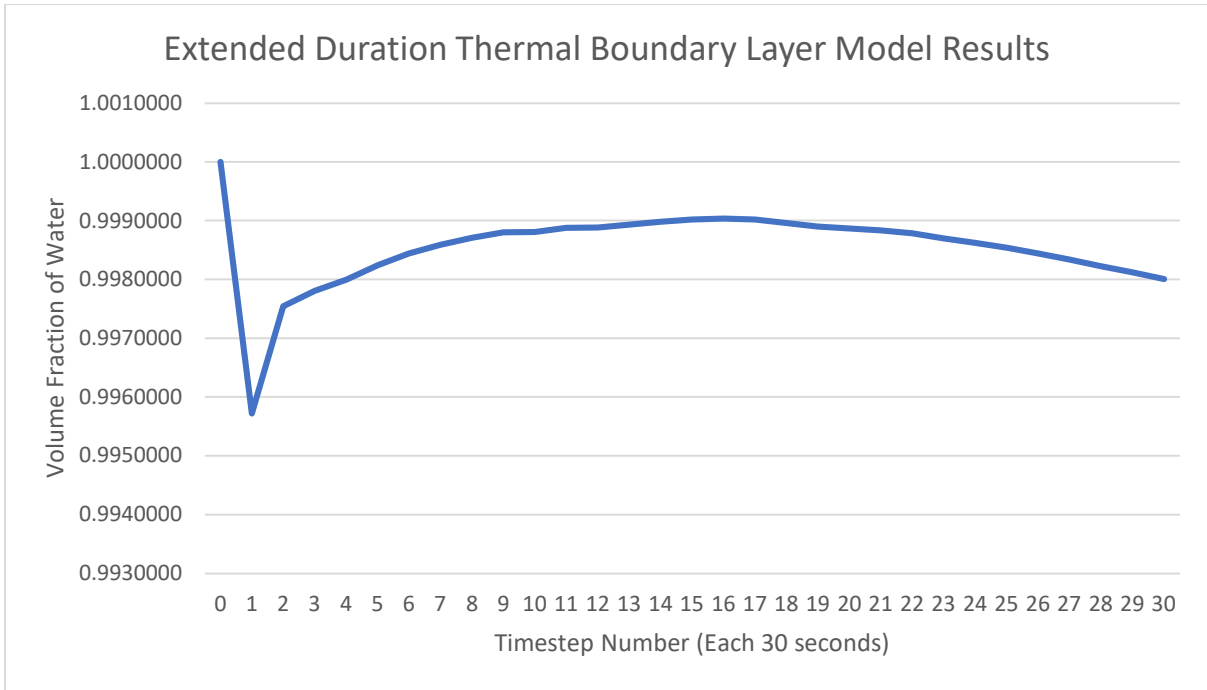


Figure 26 - Extended Duration Thermal Boundary Layer Model Results

As shown in the results of this simulation, the inclusion of the thermal boundary layer did not provide measurable difference to the rate of evaporation. It should however be noted that the first timestep reported a large change in the volume fraction of water. The volume fraction of water then increased until the 16th timestep before beginning to fall again. The initial drop is likely caused by the simulation not converging due to the large timestep. The following timesteps then attempt to rectify the issue. However, by the 24th timestep the rate of evaporation normalises. The results for these 6 timesteps are shown below in table 11. As seen from the results they are sufficiently consistent. It is noted that this is similar to the results provided in the above thermal boundary layer calculations when adjusted for the longer timestep.

Table 12 - Extended Duration Thermal Boundary Layer Extracted Results (TS 26-30)

Time Step	Water Volume	Difference/Time Step
26	0.9984410	0.0001000
27	0.9983380	0.0001030
28	0.9982250	0.0001130
29	0.9981190	0.0001060
30	0.9980060	0.0001130

4.4.3 Initial Humidity Thermal Boundary Layer Model

Humidity has a significant effect on the rate of evaporation. This can be easily explained mathematically using Eq. 2 (Shah 2014) as the difference in humidity at the water surface and that of the surrounding air, multiplies the rest of the equation. As the initial simulations do not inherently account for humidity, at least not at the initialization of the simulation this test is to validate the impact realistic humidity does have on the results. The results of first timesteps of the simulations defined in section 4.3 show a much greater rate of evaporation than the later timesteps. This is hypothesised to be in part a result of the humidity normalising.

The model was set up as detailed in section 3.5. For the purpose of time conservation and as this test is only seeking to validate the results of humidity only the 1mm model will be run. It should also be noted that for this same reason, timesteps were reduced to 5 but the iterations were increased to 2000. The increase in iterations is due to the findings in section 4.3.

4.4.4 Initial Humidity Thermal Boundary Layer Results

Table 13 - 1mm Model Humidity Control Results

Name	HumControlV3	
Version	18	
Date	28/09/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	1	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
Total Time	0.999980	0.000020
0	1.000000	0.000000
1	0.999986	0.000014
2	0.999983	0.000003
3	0.999982	0.000001
4	0.999981	0.000001
5	0.999980	0.000001

Table 14 - 1mm Model Humidity -0.6 Degrees C. Results

Name	Hum -06	
Version	18	
Date	27/09/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	296.55	
Max Iterations	2000	
Time Step Length (seconds)	1	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
Total Time	0.999980	0.000020
0	1.000000	0.000000
1	0.999986	0.000014
2	0.999983	0.000003

	3	0.999982	0.000001
	4	0.999981	0.000001
	5	0.999980	0.000001

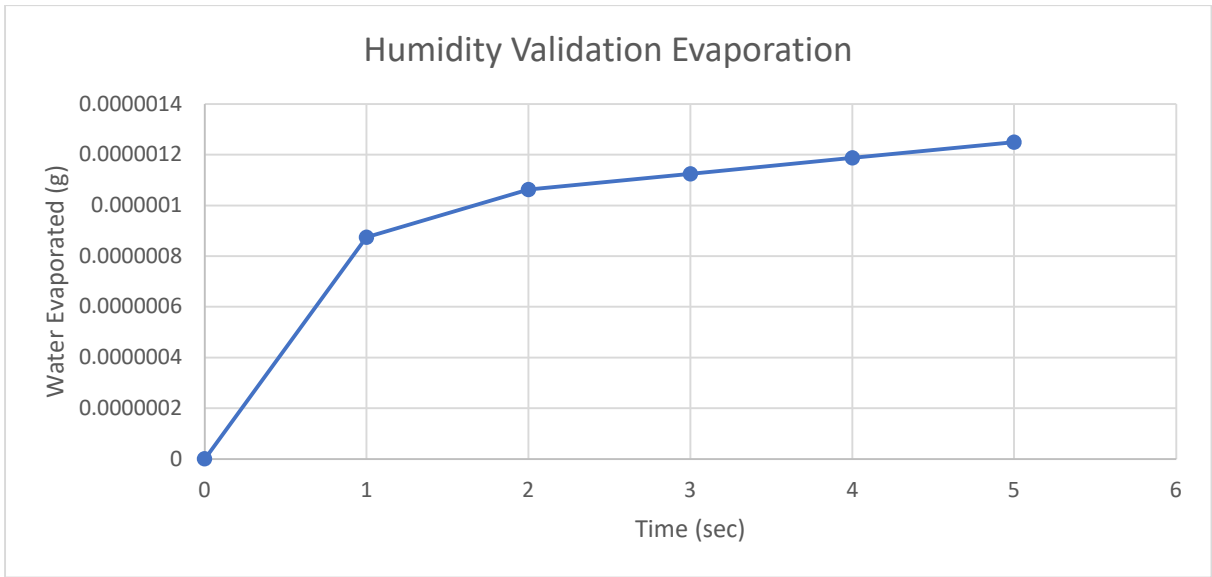


Figure 27 - Humidity Validation Results - Water Evaporated (g)

From review of the results of this model important information can be gathered. Firstly, the purpose of the test was to evaluate the effect of properly accounted for humidity. This can be seen in the figure 28. As can be understood from the graph, while there is still a much greater than expected evaporation over the first-time step, the model that accounts for humidity normalises much quicker. By the third timestep the model achieves a steady rate of evaporation. It should also be noted as this model was ran with 2000 iterations both the control and the 0.6-degree difference model provide the same results (hence why only 1 set of data is shown for the humidity model on figure 19). This also confirms the theory in section 4.3 that using only 1000 iterations for the simulation does not provide accurate results.

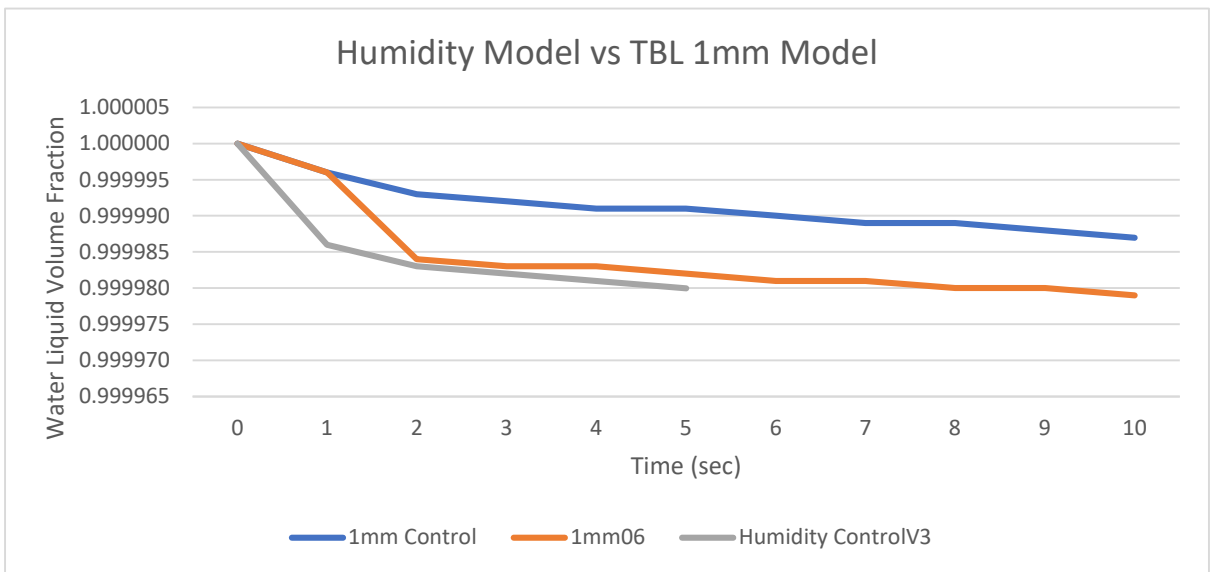


Figure 28 - Humidity Model Comparison – Volume Fraction

4.4.5 Increased Temperature Difference Thermal Boundary Layer Model

The main objective of this simulation is to test if an increase in temperature difference between the thermal boundary layer and the rest of the water body would induce different evaporation results. Previous testing within expected thermal boundary layers as defined in section 2.4 have not provided any conclusive results that either increases or decreases evaporation. As per section 3.5 the temperature difference in this simulation has been increased to 1.2 Degrees Celsius. Improvements to the model which have come about as part of the iterative process of the previous “further modelling” results have also been included in this test. This test runs for 5 timesteps, each of 10 seconds and with 2000 max/iterations per step.

4.4.6 Increased Temperature Difference Thermal Boundary Layer Results

Table 15 - Increased Temperature Control

Name	Increased Temperature Difference	
Version	19	
Date	30/09/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	1.000000	0.000000
1	0.999988	0.000012
2	0.999985	0.000003
3	0.999984	0.000001
4	0.999977	0.000007
5	0.999960	0.000017
	Total	0.000040

Table 16 - Increased Temperature Difference TBL Model

Name	Increased Temperature Difference TBL	
Version	18	
Date	1/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	295.95	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step

0	1.000000	0.000000
1	0.999988	0.000012
2	0.999986	0.000002
3	0.999982	0.000004
4	0.999977	0.000005
5	0.999954	0.000023
Total		0.000046

As seen from the results this is the first simulation that provides notable difference between the control and the thermal boundary layer model. Over the total simulation the thermal boundary layer model had greater amounts of evaporation. It should be noted that it is not consistently higher across all timesteps, but the overall amount is higher. In total the control model had 0.00249 grams of water evaporate while the thermal boundary layer had 0.00287 grams. This equates to an approximately a 14% difference. Without the consistently higher rate of evaporation this result was initially met with some scepticism. However, from all the testing, which was undertaken up to this point, this model is the best candidate to provide parameters with results indicating the thermal boundary layer will affect evaporation. These results are summarized in Figure 29 and 30.

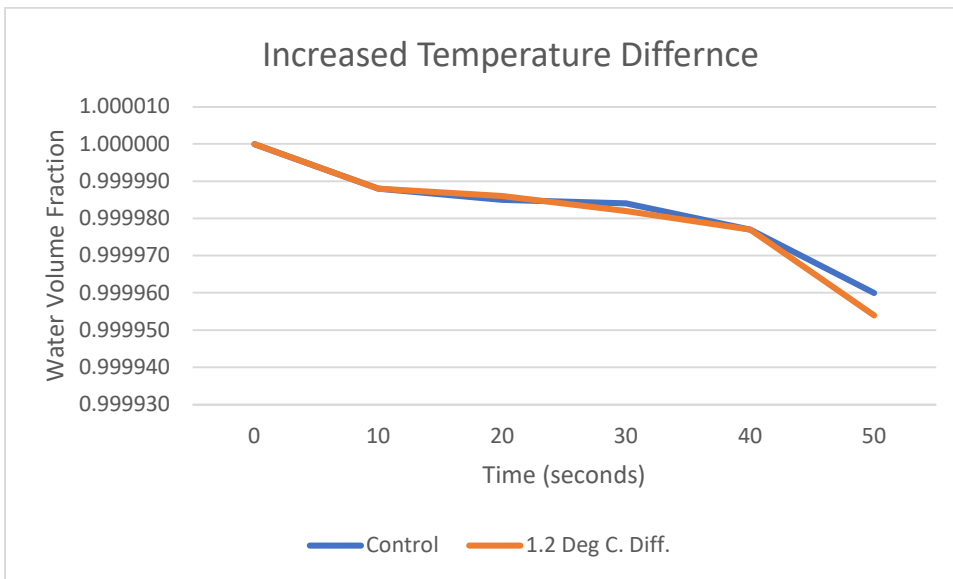


Figure 29 - Increased Temperature Difference - Volume Fraction Comparison

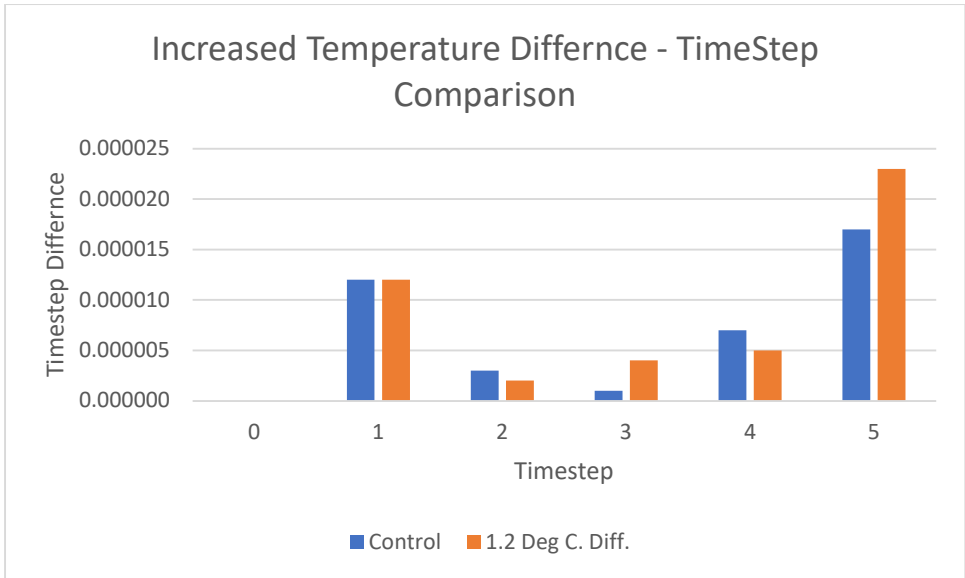


Figure 30 - Increased Temperature Difference - Timestep Comparison

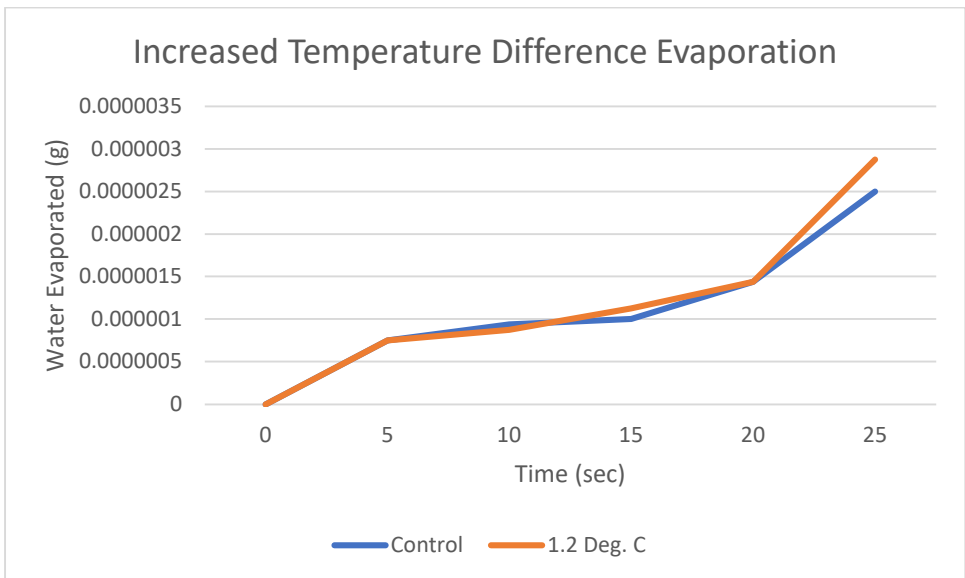


Figure 31 - Increased Temperature Difference - Water Evaporated (g)

Following the review of the results and noting an increase in the rate of evaporation due to the addition of a thermal boundary layer a hypothesis has been developed to explain this. As part of this dissertation an opinion has been put forward regarding the reasoning behind this, the increase in the rate of evaporation could be attributed to the increase of temperature difference between that of the water surface and that of the surrounding air. Temperature effects both the density of air as well as specific humidity. The lower temperature attributed to the thermal boundary layer will increase the difference in density of air and the difference in specific humidity between the surrounding air and that of the water surface temperature. This is best explained mathematically using the Shah Equation (Eq.2). Using the thermal boundary layer's temperature as opposed to the water of the surrounding body of water will result in a greater rate of evaporation. A comparison of which is shown in the below table 17.

Table 17 - Shah Equation comparison between control parameters and parameters including a -1.2-degree thermal boundary layer

	C	p_w	p_r-p_w	W_w-W_r	E_o
Control	35	1.175	0.019	0.0161	0.176678
TBL -1.2 Deg C	35	1.181	0.025	0.0247	0.298535

The results from the Shah Equation calculations however account for an approx. 41% difference in evaporation rate. The simulation only had a rate of 14%. Even comparing the largest timestep difference between the two simulations only accounts for an approx. 26% increase. Therefore, to further add to the hypothesis there is likely an unknown counteracting factor or due to the thin depth it does not induce the full effect of the temperature difference. Providing further evidence for this hypothesis is out of scope of this dissertation however will be further explored in section 5 of this report.

4.4.7 Average Brisbane July Thermal Boundary Layer Model

The purpose of this dissertation is to test the effects under different conditions. All models to this point were conducted using an average February set of environmental conditions. To widen the scope of testing a test was undertaken with an average July set of conditions. These conditions are notably cooler and have less relative humidity. The test was run within normal expected parameters of the thermal boundary layer being, a depth of 1mm and a temperature difference of 0.6 degrees Celsius.

4.4.8 Average Brisbane July Thermal Boundary Layer Results

From review of the results both the control and the 0.6-degree thermal boundary layer model both returned the same evaporation rates. This result provides insight that the findings in earlier simulations may be constant across a wider range of environmental conditions. The results of this simulation are provided in Table 18 and 19 below.

Table 18 - July Conditions Control

Name	July Control	
Version	19	
Date	9/10/2020	
Air Temp. (deg. Kelvin)	294.15	
Water Temp. (deg. Kelvin)	288.15	
TBL Temp. (deg. Kelvin)	288.15	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	4	
Time Step	Water Volume	Difference/Time Step
0	1.000000	0.000000
1	0.999984	0.000016
2	0.999982	0.000002
3	0.999982	0.000000
4	0.999981	0.000001
	Total	0.393788

Table 19 - July Conditions -0.6 Deg. C TBL

Name	July -0.6 Deg. C	
Version	19	
Date	10/10/2020	
Air Temp. (deg. Kelvin)	294.15	
Water Temp. (deg. Kelvin)	288.15	
TBL Temp. (deg. Kelvin)	287.55	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	4	
Time Step	Water Volume	Difference/Time Step
0	1.000000	0.000000
1	0.999984	0.000016
2	0.999982	0.000002
3	0.999982	0.000000
4	0.999981	0.000001
	Total	0.393788

4.4.9 Combined Further Testing Discussion

Following the results of the initial thermal boundary layer simulations defined in section 4.3 it was identified that further testing was required to validate assumptions, correct the modelling and test a wider range of parameters. Duration required number of iterations, humidity controls, thermal boundary parameters and climatic conditions were all further explored as part of these simulations.

Duration was expanded to test the theory, that divergence in evaporation rate between thermal boundary layer models and the controls did not occur as the difference was too small to measure over a short time period. While the model run for this did have short comings regarding absolute accuracy, it did validate that there was no difference as all results were exactly the same between models. The theme of duration did however continue through other tests and eventually the timestep interval was increased to 10 seconds for subsequent tests (reduced again to 5 seconds in the July test and later wind tests).

The initial simulations for the 1mm and 0.1mm thermal boundary models tested in section 4.3 were confirmed to have some inherent inaccuracy due to the reduced number of iterations per timestep. Increasing from the maximum number of iterations from 1000 to 2000 in the further testing in this section did validate that a higher number provided for more reliable data. It should however be noted to achieve this, due to the processing limitations the number of timesteps had to be reduced to 5. This is to a degree problematic, as in some cases the first and second timestep data may not be considered reliable, depending on the simulation. This means for some models there are only 3 reliable data points available, which such a small is not desirable. This is further explored in the "Possible Further Research" section 5.2.

Humidity was a fundamental factor which was not addressed completely as part of the initial modelling. The assumptions regarding this was confirmed and conditions created to rectify this issue. It is noted that the boundary conditions to control humidity are not perfect and need to be

appropriately adjusted depending on timestep duration. However, the set up allows for results with relative accuracy.

From the extensive testing in sections 4.3 and the first tests of 4.4 difference in evaporation rates attributed to the thermal boundary layer could not be confirmed. Increasing the temperature difference of the thermal boundary layer however did provide notable results. It is theorised under the simulated parameters the thermal boundary layer increases the humidity and air density differences between the surface of the water and the surrounding air and therefore increases evaporation rate. This is further discussed in Chapter 5.

The last test was to test in different climate conditions. This test did not provide any notable further data when compared to the other tests. It does however provide some validation to the overall results.

The further testing component did successfully accomplish its goals. The parameters for all further modelling have been refined and sources of inaccuracy have been rectified. The important other success is that the thermal boundary layer has been shown to influence evaporation. While the data is minimal it does provide a starting point for the testing in section 4.5. It also does confirm the overall trends of the effect of the thermal boundary layer in conditions without wind.

4.5 Thermal Boundary Layer Under Realistic Conditions Analysis

Baseline data has provided both the outline for the realistic model as well as the areas of interest for the realistic model which were tested. The notable difference between this model and the previous in sections 4.3 and 4.4 is the inclusion of wind. Due to this parameter the overall geometry has also been altered. This section will compare the results of the models with the inclusion of wind to determine how the thermal boundary layer influences evaporation under these conditions.

4.5.1 Thermal Boundary Layer Under Realistic Conditions Results

Tabled results from Simulations

Table 20 - Results for Realistic Conditions Control

Name	Wind Control	
Version	19	
Date	1/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625766	0.019416
2	0.614409	0.011357
3	0.609887	0.004522
4	0.606212	0.003675
5	0.604543	0.001669
	Total	0.040639

Table 21 - Results of -1.2 Deg. C. diff. under Realistic Conditions

Name	Wind -1.2 deg. C.	
Version	19	
Date	2/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	295.55	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625537	0.019645
2	0.615234	0.010303
3	0.612886	0.002348
4	0.612402	0.000484
5	0.612372	0.000030
	Total	0.032810

Table 22 - Results of -0.6 Deg. C. diff. under Realistic Conditions

Name	Wind -06 deg. C.	
Version	19	
Date	4/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	296.55	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625710	0.019472
2	0.614905	0.010805
3	0.611899	0.003006
4	0.611640	0.000259
5	0.611347	0.000293
	Total	0.033835

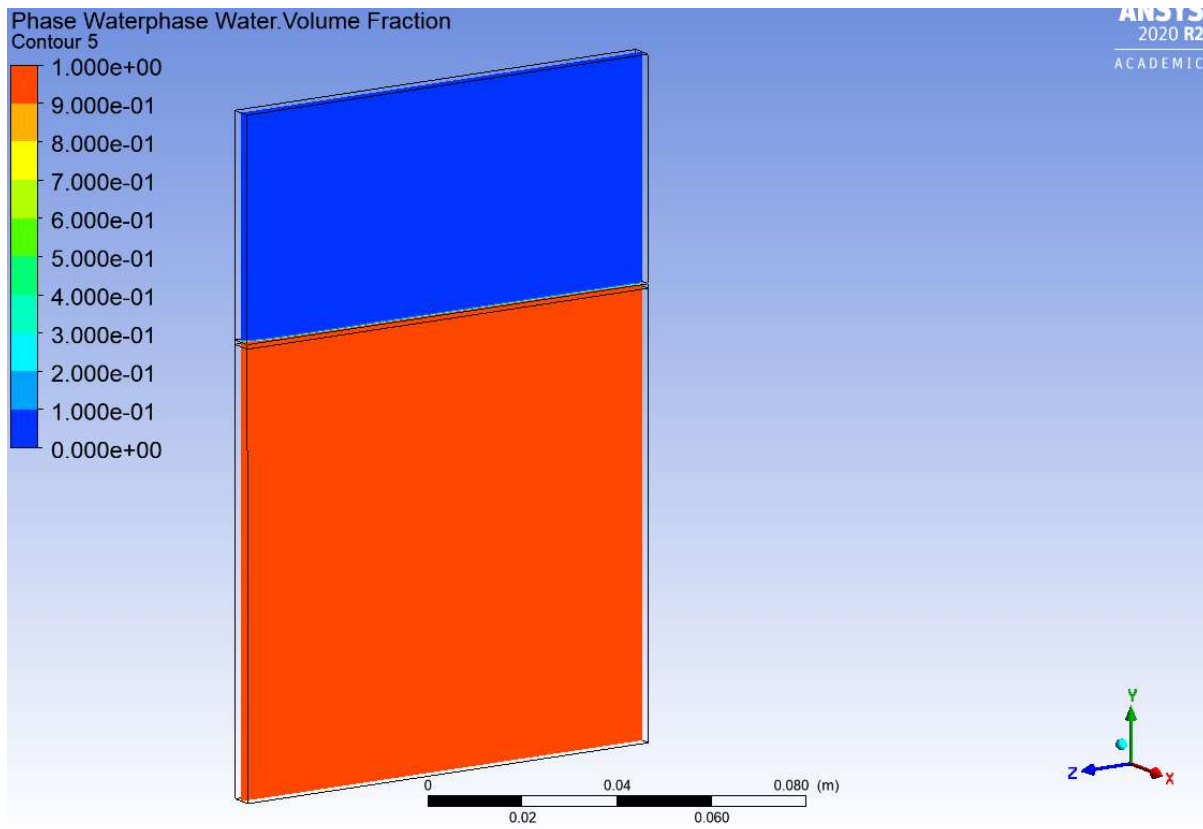


Figure 32 - Control Model Results Graphic

Blaney-Criddle Calculation using the Shah Equation (Eq.1)

$$E = a_{BC} + b_{BC}f$$

$$f = p(0.46T + 8.13)$$

$$f = 1(0.46(29) + 8.13) = 21.47$$

$$a_{BC} = 0.0043RH_{min} - \left(\frac{n}{N}\right) - 1.41$$

$$a_{BC} = 0.0043(0.62) - 1 - 1.41 = -2.407334$$

$$b_{BC} = 0.82 - 0.0041(RH_{min}) + 1.07 \left(\frac{n}{N}\right) + 0.0066(U_d) - 0.006(RH_{min}) \left(\frac{n}{N}\right) - 0.006(RH_{min})(U_d)$$

$$b_{BC} = 0.82 - 0.0041(0.62) + 1.07 + 0.0066(2.777) - 0.006(0.62) - 0.006(0.62)(2.777) = 1.901$$

$$E = -2.407334 + 1.901(21.47) = 38.407mm/d$$

To compare the numerically calculated rate to the simulated results, the result of the blaney-criddle method needs to be converted into grams. This is achieved by:

Convert mm/d to mm/sec;

$$\frac{38.407}{(24 \times 60 \times 60)} = 0.000444525 \text{ mm/sec}$$

Convert from mm/sec to g/sec

$$0.000444525 \times (5 \times 100) \times 0.001 = 0.0002222627 \text{ g/s}$$

Multiply to determine total loss for simulation length;

$$0.0002222625 \times 25 \times = 0.00555656 \text{ g}$$

Time Taken for Entire Domain to be Evaporated;

$$\frac{10}{38.407} \times 24 = 6.975325 \text{ hours} = 25111.1624 \text{ seconds}$$

Convert to grams per second (domain is 50 grams; 5mm x 100mm x 100mm)

$$\frac{50}{25111.1624} = 0.00199115 \text{ g/s}$$

4.5.2 Thermal Boundary Layer Under Realistic Conditions Discussion

From the results of section 4.4 it was expected that the thermal boundary layer would continue to increase the rate of evaporation, however the opposite occurred. The significant point in the data is that the inclusion of the thermal boundary layer reduced the rate of evaporation. It was expected that the thermal boundary layer would also have less of an effect in a simulation with the inclusion of wind as the wind component of evaporation is generally more significant to that of the temperature, this proved to also be untrue.

Following the control model, the 1.2-degree difference model was run as it provided the most promising results in section 4.3. The inclusion of the thermal boundary layer in this simulation accounted for a total of 19.25% less evaporation. When comparing the two results it is evident that the thermal boundary layer model slows in evaporation at a much quicker rate than that of the control model over the timesteps. Initially, this was thought to be an issue with the humidity conditions but from review of the humidity for the air domain while not staying completely steady at the initial value there is no extreme variance. These results are provided in table 23 below, comparing volume fraction of water vapor in the air domain. While some variance could create inaccuracies, the results are contradictory to the humidity levels being a significant effect on the different rates. This is due to that in most cases the humidity in the control model is higher than the thermal boundary model, but the evaporation rate is also higher in the control, where normally a higher humidity should slow evaporation.

Table 23 - Fraction of Water Vapor in Air Domain Comparison

Timestep	Control (volume fraction)	1.2 Deg. Difference (volume fraction)
0	0.01588	0.01588
1	0.0155621	0.015545
2	0.0159725	0.01565
3	0.0157316	0.015675
4	0.0158444	0.0157218
5	0.0155784	0.0157377

A concern with the model is the rapid evaporation at the start of the simulation. This rapid evaporation is not considered an accurate reflection and appears to be of a similar issue found in the previous models with 1000 max iterations. As per the theory stated in section 4.2, this is believed to be due to insufficient iterations. Due to the more complex model being run the number of iterations may not be enough to convergence at the initial timesteps. To maintain reasonable process times and model accuracy without needing to increase the number of maximum iterations, the timestep was reduced from 10 to 5 seconds, this following review of the results it is considered likely not enough if an extended duration is required. To achieve this the number of iterations would need to be increased.

Following the results of the 0.6 degree difference the three models of the realistic conditions were compared, the results of which are summarised in figure 26. It is important to note that it is that the -0.6 model follows a similar trend to that of the -1.2-degree model however with results closer to that of the control model. This offers some validation that the bigger the temperature difference the larger the effect on evaporation. This is also the first time a -0.6-degree model has provided results that differ to that off the control model.

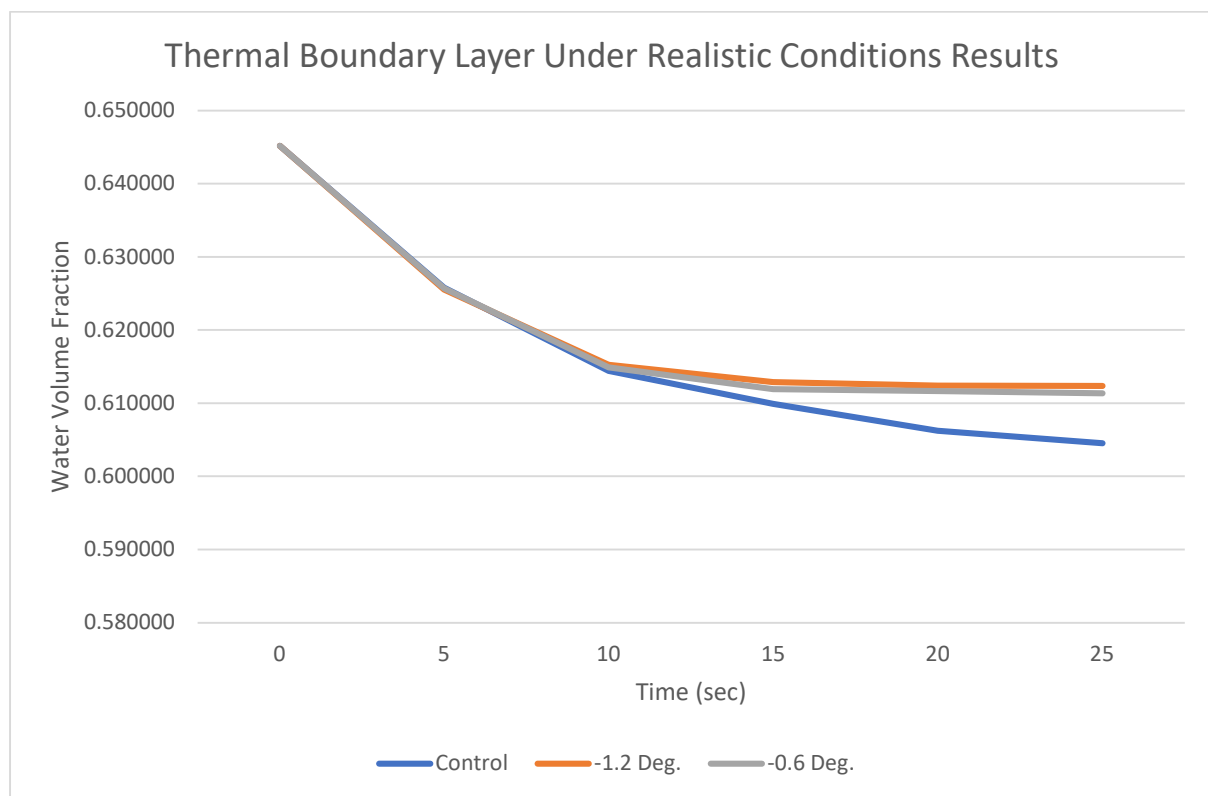


Figure 33 - Realistic Conditions Result Comparison

The reason why the thermal boundary layer decreases evaporation in these parameters needs to be explored. This dissertation puts forward a hypothesis on why this might occur. As discussed in section 1.1 evaporation occurs when energy allows free water molecules to escape (Rutledge K 2011), also discussed in section 2.4 was that the thermal layer is persistent (Katsaros 1980). Therefore, while the wind continually breaks the surface of the water, the thermal boundary layer continues to reform cooling the air at the surface of the water and itself, reducing the energy available. Validating this theory is out of the scope of this project but will be discussed further in Chapter 5.

As per the methodology of this project, the realistic condition results were required to be tested against a reliable numerical model. The method chosen for this estimation was the Blaney-Criddle method. When comparing the results, the Blaney-Criddle method estimation provided for a total evaporation of 0.04977875g or approx. 0.001% of the total amount of liquid water. If the results from the simulations are taken and converted into grams for comparison, a serious issue is found. If the change in volume fraction noted in the control experiment of 0.035569 which equates to approx. 5.5% or 2.7565g which is incorrect by a large magnitude. For the other models the result is similar. However, this appears to be an issue with the first few timesteps with the later ones appearing to begin to stabilise. When taking the last value of each model the results are 0.0124771g, 0.092997g and 0.0117796g evaporated over the time step for the control, -1.2 deg and -0.6 deg models respectively. These figures are much closer to the 0.009956g evaporated for the timestep as calculated by the Blaney-Criddle method.

The results show a downward trend in the rate of evaporation as the timesteps progress, the variance between the evaporation rate of each timestep also appears to stabilise. It was thought that if the timesteps were to increase this trend would continue. To test this theory another simulation was carried out for the control model, however the timesteps were increased to 10. The results are shown in table 24 below. The results of this test did show that over an extended time the results did start becoming closer to the calculated values.

Table 24 - Wind Control Extended Simulation Results

Name	Wind Control Extended	
Version	19	
Date	12/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	10	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625766	0.019416
2	0.614409	0.011357
3	0.609887	0.004522
4	0.606212	0.003675
5	0.604543	0.001669
6	0.603307	0.001236
7	0.602443	0.000864
8	0.601789	0.000654
9	0.601322	0.000467
10	0.600992	0.000330
	Total	0.044190

From review of the results between the numerical calculation and the simulated models it was concluded that there is still scope to achieve more accurate results however for a model of this complexity the number of iterations need to be greatly increased and the timestep reduced significantly. The possibility to improve accuracy is discussed further in section 5.3. If this is undertaken accuracy of the model could be improved however the provided results of this dissertation are considered to provide a general overview of the trend in evaporation rates when the thermal boundary layer is included. The purpose of this task was to determine if the thermal boundary layer and under what circumstances the rate of evaporation will be affected, which these results are sufficient.

Chapter 5: Conclusion

5.1 Review of Goals

As discussed in section 1.4, this dissertation set out to accomplish the following tasks;

- (a) Define which current evaporation models are used for large open bodies of water
- (b) Review relevant literature relating to evaporation and the effects of the thermal boundary layer.
- (c) Develop a model which can simulate the effects of the thermal boundary layer on evaporation for an open body of water.
- (d) Interpret the results of the simulation and describe under what circumstances the thermal boundary layer will affect the rate of evaporation.

The currently accepted evaporation models are explored in section 2.2 of this dissertation. The objective of this review is to understand how evaporation is currently estimated and how these methods applied to this project. This dissertation was successful in this goal as it defined a method to model the thermal boundary layer as well as a numeric model to validate the results. The conclusion of this goal was the proposal to use CFD modelling to run the necessary simulations for to achieve the relevant data. This review also provided to accepted methods to validate the computer model to, being Eq.1 and Eq.2.

Before being able to develop models to test the effects on evaporation, parameters of the thermal boundary layer needed to be established. This dissertation was successful in researching and determining a set of parameters to include into the subsequent models. These parameters were provided by reviewing published research by others. From this literature review it was determined that the depth range of the thermal boundary layer for this project was to be between 0.1mm and 5mm, while the temperature difference between the thermal boundary layer and the rest of the body of water was to be between -0.2 and -0.6 degrees Celsius.

To simulate effects of the thermal boundary layer on evaporation for an open body of water a series of CFD models were set up. The methodology of these models is defined in Chapter 3. To accomplish this goal multiple models were set up to simulate specific conditions or validate results. Validation of the models using accepted numeric calculations also occurred in sections 4.2.3 and 4.4.2. This goal is determined to be successfully completed.

Following development of models for simulation and recording of results the data was interpreted providing insight into the effects of the thermal boundary layer. This goal is also considered completed.

5.2 Conclusions and Outcomes

From accomplishing the goals as defined in section 1.4, insight into the influence of the thermal boundary layer on evaporation can be presented. Each stage of the methodology set out to provide either model development or to test the effects of the thermal boundary layer under a certain circumstance.

The base model simulations as defined in section 3.3 were successful in validating the fundamentals of the CFD methodology to be utilised throughout the subsequent simulations. As shown in section 4.2 the developed base model allowed for reliable evaporation in the proposed domain. This model and CFD set up was validated using the Shah equation (Eq.2). The comparison between the results from the simulated model and the numerically estimated model are defined as relatively comparable, thus validating the modelling. As mentioned in section 4.2.3 there was some variance

between the results, however due to the inherent inaccuracies of both systems this was to be expected.

The next stage of the project was to incorporate conditions within the model to simulate the thermal boundary layer. As defined in section 2.4, the thermal boundary layer did not have an exact set of parameters but rather a range. A combination of depth and temperature difference parameters were selected for testing. The objective of this model was to model the thermal boundary layer in a simplistic manner; therefore, the condition of wind was not included. The methodology for these simulations is defined in section 3.4. The main conclusion of this test and the results presented in section 4.3, was that the thermal boundary layer did not influence the evaporation rate to any measurable degree for the specified parameters. It should however be clearly noted that it was not conclusive that the thermal boundary layer did not influence the evaporation rate. This is discussed in greater detail in section 4.3.7. There were however several shortcomings with these tests and areas that needed validation, to ensure the wrong conclusions were not drawn the requirement for further testing was determined.

Following completion of the modelling defined in section 3.4 and review of the results in section 4.3, it was decided further testing was required. Issues with the model included a reduction in the maximum number of iterations, humidity controls were not properly implemented, only very short durations being measured and that the results only provided for a single set of environmental conditions. To address these issues a series of new simulations were developed as defined in section 3.5. From review of the results of this further testing however validated the conclusion that the thermal boundary layer in the selected parameters did not provide any measurable influence. The results of this are specified in section 4.3. Using standard parameters of the thermal boundary layer results which indicated alteration to the evaporation rate could not be achieved, to further test this the parameters were expanded to include a 1.2-degree temperature difference, doubling what was defined in the literature review. The conclusion of this test and review of the results in section 4.4.6 is that the increased temperature did cause the evaporation rate to be altered. This confirms that the thermal boundary layer has the capacity to affect evaporation rates, it also confirms that drawing any conclusion that from the previous tests that the thermal boundary layer does not alter evaporation would not be correct.

The thermal boundary layer parameters that did cause a change of evaporation rate in the simulation without wind, caused the evaporation rate to increase. It is thought that this is due to the increase in difference between the air density and humidity between the water surface and the surrounding air. This has been explained mathematically using (Eq.2) however the rate it effects the evaporation is not consistent with the simulated results. The results of (Eq.2) for the tested for parameters are shown below in table 25. Therefore, it is theorised the full extent of this effect is not realised due to the thin nature of the thermal boundary layer or there is an unaccounted-for counteracting factor. Further investigation into this is discussed in section 5.3.

Table 25 - Shah Equation (Eq.2) results for tested for conditions

	C	pw	pr-pw	Ww-Wr	Eo	% Difference from Control
Control	35	1.175	0.019	0.0161	0.176678	
TBL -1.2 Deg C	35	1.181	0.025	0.0247	0.298535	40.82
TBL -0.6 Deg C	35	1.178	0.022	0.0204	0.235677	25.03
TBL -0.2 Deg C	35	1.176	0.02	0.0175	0.19552	9.64

The final set of modelling for this project was to test the results as defined in the previous models in realistic conditions. The main component missing from the previous tests was the inclusion of wind. The methodology set out in section 3.6, defines how wind will be included into model. From the review of the results set out in section 4.4 the effect of the thermal boundary layer could be further understood. The modelling showed significant reduction in the rate of evaporation in models including the thermal boundary layer when compared to the control model. The variance between the -0.6-degree difference and the -1.2-degree difference models was also in line with what was to be expected, the lower temperature difference provided for a result closer to the control. It is however important to note that while the results for the model that didn't include wind in section 4.4.6 increased evaporation, the results in the realistic model showed a decrease in evaporation rates. It is hypothesised that this decreases evaporation due to the wind constantly disturbing the thermal boundary layer, which in turn causes it to establish itself. This re-stabilising and subsequent cooling is affecting the evaporation rate.

This project set out to understand in what circumstances the thermal boundary layer effects evaporation, in that aspect this dissertation is successful. The unusual outcome however is that in conditions without wind the thermal boundary layer is expected to increase evaporation, while in circumstances with wind it can decrease the rate of evaporation. This highlights the complexity of attempting to model evaporation for open bodies of water. Suggestions for the cause of this phenomenon have been put forward, however further validation is considered out of scope.

The final conclusions of this dissertation are;

Modelling of open bodies of water can be achieved accurately using CFD software.

In average Brisbane, February and July conditions a thermal boundary layer of expected depth and temperature difference, if any will have only limited effect on evaporation rates in the absence of wind.

In the absence of wind in average Brisbane, February conditions any effect the thermal boundary layer will have, would be that the rate of evaporation increases.

The thermal boundary is likely to influence the rate of evaporation, with the greater the temperature difference the greater the effect.

In average Brisbane, February conditions when accounting for significant wind a thermal boundary layer of expected depth and temperature difference will have the effect of decreasing the rate of evaporation.

5.3 Possible Further Research

While this dissertation does provide insight under what circumstance the Thermal Boundary Layer effects the rate of evaporation there is scope to further expand this research. The scope of conditions can be increased, the modelling extents increased, inclusion these results into other methods explored, more accurately understanding the extent of the effect and further validation applied. Theories into the cause of the increasing and decreasing evaporation rates were provided in sections 4.4.6 and 4.5.2, these could form basis of further research. These items are however placed out of scope of the purpose of this project due to software limitations, lack of resources, time constraints or simply not included as part of the original project planning.

While this project sought to define conditions which the thermal boundary layer affected the rate of evaporation and was successful in determining this for specific Brisbane conditions, this could be greatly expanded. The modelling for the most part was provided for the average environmental conditions of February. This could be expanded to include other months, day or night-time averages or even other geographic locations. Fringe conditions could also be further investigated such as conditions where the water body is the same temperature as the air, in this instance evaporation would be minimal to none but would the thermal boundary layer affect this.

Due to the use of a student version of *ANSYS* the domain size and mesh sizing was constrained. If the full version of the software was utilised a larger size domain with finer meshing could be set up. This would provide more accurate results and may rectify the issues with the 0.1mm thermal boundary model defined in section 4.3.5. Time constraints have also been an issue through the duration of this project. With many of the models required to be run in serial mode, process times neared 24 hours. It was shown that accuracy increased in the later timesteps. Also, issues to resolve limited duration by expanding the time per timestep did impact reliability. Longer duration and smaller timesteps would likely provide more accurate data.

To further validate the simulated data a physical model could be created. Data provided from this model could directly relate to the validation of the simulation as well as the accuracy of the numeric models used in this project. This would require extensive resources and time, hence was omitted from this dissertation.

As discussed throughout Chapters 4 and 5 of this dissertation, while the accuracy of the modelling is considered sufficient to understand if the thermal boundary layer is affecting the rate of evaporation, it may not be adequate to accurately state to what extent. To provide conclusive results in relation to the extent of the affect, further refinement and validation would be required. With the limited resources available providing definite values was not considered possible for this project, therefore there is potential for further research.

In section 4.4.6 a position was provided on why evaporation might decrease due to the thermal boundary layer in the absence of wind. Further research into the validation or disproving of this theory might be a possibility. In section 4.4.6 and again in section 5.2 it was suggested that the full effect was not evident due to other factors which at this stage are unknown. This project sought to understand “if” the thermal boundary layer and under what circumstances evaporation would be affected and not “why”. However, highlighting there is an impact opens the possibility into further research into why it is the case.

Similarly, to the above paragraph in section 4.5.2 a theory was put forward to why the evaporation rate decreases in situation of significant wind. There is a possibility of further research into the cause of this.

The last item for possible further research is regarding using this data to help further develop open water evaporation techniques. This project did demonstrate that the thermal boundary layer has potential to affect the rate of evaporation. Further research into the accuracy of these methods and if accounting for the thermal boundaries layer's effects can improve them could be investigated.

References

- Australian Bureau of Statistics (ABS). 2012, *Drought in Australia, Year Book Australia*. <https://www.abs.gov.au/ausstats/abs@.nsf/lookup/1301.0feature%20article151988>
- Finch J and Calver A. 2008, *Methods for Quantification of Evaporation from Lakes*, Wallingford, CEH Wallingford
- Finch J W and Hall R L. 2001, *Estimation of Open Water Evaporation*. Bristol. Environmental Agency
- Rutledge K. 2011, *The Process of Evaporation*, National Geographic Resource Library
- Simscale 2020, *What is CFD| Computational Fluid Dynamics?*, www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/
- Wimshurst A. 2019, *Eulerian Multiphase Modeling*. www.fulidmechanics101.com
- Shuttleworth J W. 1993, *Handbook of Hydrology – Chapter 4*. Wallingford. Institute of Hydrology
- Ewing G and McAliseter E D. 1960, *On the Thermal Boundary Layer of the Ocean*. Vol 131, *Science*
- Singh V P and Xu C Y. 2000, *Evaluation and Generalization of Radiation-Based Methods for Calculating Evaporation*. Vol 14, *Hydrological Processes*
- Katsaros K B. 1980, K.B. *The aqueous thermal boundary layer*. *Boundary-Layer Meteorol* **18**, 107–127
- Katsaros K B, Lie T W, Tillman E J and Businger J A. 1976, *Heat Transport and Thermal Structure in the Interfacial Boundary Layer Measured in an Open Tank of Water in Turbulent Free Convection*. Vol 83, *J. Fluid Mech*
- Mckeown W and Asher W. 1997, *A Radiometric Method to Measure the Concentration Boundary Layer Thickness at an Air-Water Interface*. *Joint Institute for the Study of Atmospheric and Ocean Contribution*, Number 423
- Coronel A S 2018, *Temperature Field Measurements of Thermal Boundary Layer and Wake of Moving Hot Spheres using Interferometry*, *Experimental Thermal and Fluid Science*, Vol 90
- Wells A. 2009, *Variations in Ocean Surface Temperature due to Near-Surface Flow: Straining the Cool Skin Layer*. Vol 39, *Journal of Physical Oceanography*
- Wandel A. 2020, *Computational Fluid Dynamics – Study Guide*. University of Southern Queensland.
- Shah M M. 2014, *Methods for Calculation of Evaporation from Swimming Pools and Other Water Surfaces*. Vol 120, *ASHRAE Transactions*
- Shah M M. 2008, *Analytical Formulas for Calculating Water Evaporation from Pools*. Vol 114, *ASHRAE Transactions*
- Gokhale P. 2007, *Mixed Convective Heat Transfer and Evaporation at the Air-Water Interface*. Vol 272, *All Theses*.
- Monteith J L. 1981, *Evaporation and Surface Temperature*. Vol 107, *Quarterly Journal of The Royal Meteorological Society*
- Ertek, A. 2011, *Importance of Pan Evaporation for Irrigation Scheduling and Proper Use of Crop-Pan Coefficient, Crop Coefficient and Pan Coefficient*. Vol 6, *African Journal Of Agricultural Research*

ANSYS 2020, ANSYS Meshing Solutions, ANSYS Resource Library

Ozgun 2020, Is Your Mesh Good Enough, MechHead, Online Journal August 2020

Argyropoulos CD. 2015, Recent Advances on the Numeric Modelling of Turbulent Flows, Applied Mathematic Modelling, Vol 39, Issue 2

BOM 2020, Bureau of Meteorology, Australian Government, www.bom.gov.au

Fly Carpet 2020, Free Online Interactive Psychrometric Chart, FlyCarpet Inc, www.flycarpet.net

Kuron M 2015, 3 Criteria for Assessing CFD Convergence, Engineering.com, Design Software articles

McJannet D, Freeman J C, Kinght J H, Burn S 2008,Evaporation Reduction by Monolayers, Urbane Water Security Research Alliance Technical Report, No. 6

Weather Atlas 2020, <https://www.weather-atlas.com/en/australia/brisbane-climate>

Appendices

Appendix A – Project Specification

ENG4111/4112 Research Project

Project Specification

For: Alex King

Title: Effect of Thermal Boundary Layer below Water Surfaces on Evaporation (*Note: This is different to the previous title "Effect of Thermal Boundary Layer below Water Surfaces on Evaporation"*)

Major: Civil Engineering

Supervisors: Associate Professor Andrew Wandel

Enrolment: ENG4111 - Ext S1, 2020
ENG4112 – Ext S2, 2020

Project Aim: To investigate the effect of the thermal boundary layer in relation to the evaporation in open reservoirs and characterise which conditions will either promote or retard evaporation.

Programme: Version 1, 18/3/2020

1. Research background information and current findings for contribution of the thermal boundary layer to evaporation effects in open reservoirs and research currently accepted evaporation models.
2. Define evaporation models numerical simulations will be based on. Complexity of system to be increased throughout programme:
 - a. Basic Evaporation (no current or wind factors)
 - b. Evaporation incorporating water current
3. Research and define experimental data
4. Simulate the transition between the air temperature and the bulk water
5. Simulate the effect of the thermal boundary layer on evaporation by undertaking a simulation with the thermal boundary layer and without
6. Validate simulations by analysing data compared with experimental data
7. Repeat steps 4 - 6 for models b & C
8. Provide detailed simulation of the thermal boundary layer to define its behaviour

If time and resources permit;

9. Conduct own field test
10. Compare results of field test to model simulation
11. Attempt to redesign model for alternative shape (i.e. evaporation pan)

Appendix B – Project Plan

Communication Plan

Communication is an important aspect of the project plan to ensure timeframes are being met and academic process is being followed. At this stage no third parties are to be required to be included in this plan. Important University Contacts are;

Andrew Wandel - Associate Professor; Project Supervisor

Andrew will be the contact for all discussions regarding progress and technical aspects of the project.

Chris Snook; Course Examiner

Chris Snook will be the contact for administration queries and discussions for the course.

Communication will be mainly conducted via email using all parties USQ email address.

Weekly communication with the supervisor is required. Progress emails are to be sent out to the supervisor on Mondays. Questions regarding the project can be asked as part of this process.

During critical research components were further discussions are required, real time communication can be conducted via "ZOOM". It is expected that the student provides the supervisor sufficient notice to arrange this appointment. This can be arranged by email.

Appendix C – Risk Analysis

Risk/Hazard	Risk Mitigation Measures	Risk Score Before	Risk Score After
Preparation Phase			
Approval not given by USQ supervisor	Begin discussions early with USQ staff	E1	M7
Incorrect Software	Discuss software requirements with Supervisor	E1	M7
Modelling Phase			
Unable to effectively use software	Ensure available time to research software use and obtain relevant guides	M6	L3
Incorrect modelling parameters or inputs	Seek advice from Supervisor when possible and update on model changes	H4	M3
Initial Data Analysis Phase			
Inconsistent data	Attempt to collect data from as many sites as possible, undertake proven statistical analysis techniques	H4	M3
Too much data to process	Ensure all data collected is relevant	M6	L3
Loss of data	Use multiple storage locations	H3	M1
Further required Information Phase			
Refer above phases			
Final Analysis phase			
Insufficient data	Have time-frame built into program for further data collection if required	H3	M1
Write Up - Presentation Phase			
Insufficient time to complete	Begin draft dissertation as early as possible	H3	M1

Appendix D – Base Model Results

Name	Basic Evap.
Version	6
Date	2/07/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	N/A
Max Iterations	50
Time Step Length (seconds)	1
No. of Time Steps	60
Time Step	Water Volume
Total Time	0.999946
0	1
1	0.999996
2	0.999994
3	0.999991
4	0.999991
5	0.999987
6	0.999986
7	0.999985
8	0.999984
9	0.999984
10	0.999983
11	0.999982
12	0.999981
13	0.999981
14	0.99998
15	0.999979
16	0.999978
17	0.999977
18	0.999977
19	0.999977
20	0.999976
21	0.999976
22	0.999975
23	0.999974
24	0.999974
25	0.999973
26	0.999973
27	0.999972
28	0.999971

29	0.999971
30	0.999971
31	0.99997
32	0.999969
33	0.999968
34	0.999968
35	0.999966
36	0.999966
37	0.999955
38	0.999955
39	0.999955
40	0.999954
41	0.999953
42	0.999953
43	0.999953
44	0.999953
45	0.999953
46	0.999952
47	0.999952
48	0.999951
49	0.999951
50	0.999951
51	0.99995
52	0.99995
53	0.99995
54	0.999949
55	0.999949
56	0.999949
57	0.999948
58	0.999947
59	0.999947
60	0.999946

Appendix E – Thermal Boundary Layer Results

Name	5mm Control
Version	17
Date	30/08/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	297.15
Max Iterations	2000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999983	0.000017
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999994	0.000002
3	0.999991	0.000003
4	0.999989	0.000002
5	0.999987	0.000002
6	0.999986	0.000001
7	0.999985	0.000001
8	0.999984	0.000001
9	0.999984	0.000000
10	0.999983	0.000001

Name	5mm06
Version	17
Date	1/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	2000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999983	0.000017
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999994	0.000002
3	0.999991	0.000003
4	0.999989	0.000002
5	0.999987	0.000002
6	0.999986	0.000001
7	0.999985	0.000001
8	0.999984	0.000001
9	0.999984	0.000000
10	0.999983	0.000001

Name	5mm02
Version	17
Date	3/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.95
Max Iterations	2000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999983	0.000017
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999994	0.000002
3	0.999991	0.000003
4	0.999989	0.000002
5	0.999987	0.000002
6	0.999986	0.000001
7	0.999985	0.000001
8	0.999984	0.000001
9	0.999984	0.000000
10	0.999983	0.000001

Name	1mm Control
Version	17
Date	18/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	297.15
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999987	0.000013
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999993	0.000003
3	0.999992	0.000001
4	0.999991	0.000001
5	0.999991	0.000000
6	0.999990	0.000001
7	0.999989	0.000001
8	0.999989	0.000000
9	0.999988	0.000001
10	0.999987	0.000001

Name	1mm06
Version	17
Date	19/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999979	0.000021
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999984	0.000012
3	0.999983	0.000001
4	0.999983	0.000000
5	0.999982	0.000001
6	0.999981	0.000001
7	0.999981	0.000000
8	0.999980	0.000001
9	0.999980	0.000000
10	0.999979	0.000001

Name	1mm02
Version	17
Date	20/09/2020
Air Temp. (deg. Kelvin)	302.15
Water Temp. (deg. Kelvin)	297.15
TBL Temp. (deg. Kelvin)	296.55
Max Iterations	1000
Time Step Length (seconds)	1
No. of Time Steps	10

Time Step	Water Volume	Difference/Time Step
Total Time	0.999980	0.000020
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999988	0.000008
3	0.999987	0.000001
4	0.999984	0.000003
5	0.999983	0.000001
6	0.999982	0.000001
7	0.999981	0.000001
8	0.999981	0.000000
9	0.999980	0.000001
10	0.999980	0.000000

Name	1mm02 Iteration Validation	
Version	17	
Date	25/09/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	296.55	
Max Iterations	2000	
Time Step Length (seconds)	1	
No. of Time Steps	2	
Time Step	Water Volume	Difference/Time Step
Total Time	0.999980	0.000020
0	1.000000	0.000000
1	0.999996	0.000004
2	0.999992	0.000004

Appendix F – Further Modelling Results

Name	Extended Duration Control	
Version	14	
Date	22/08/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	500	
Time Step Length (seconds)	30	
No. of Time Steps	30	
Time Step	Water Volume	Difference/Time Step
Total Time	0.9980060	0.001994
0	1.0000000	0.000000
1	0.9957200	0.004280
2	0.9975440	-0.001824
3	0.9978040	0.002196
4	0.9979970	1.000000
5	0.998242	-0.000245
6	0.9984390	-0.000197
7	0.9985880	0.001412
8	0.9987070	2.000000
9	0.9987990	-0.000092
10	0.9988050	-0.000006
11	0.9988760	0.001124
12	0.9988810	3.000000
13	0.9989340	-0.000053
14	0.9989780	-0.000044
15	0.9990190	0.000981
16	0.9990330	4.000000
17	0.9990160	0.000017
18	0.9989570	0.000059
19	0.9989010	0.001099
20	0.9988660	5.000000
21	0.9988360	0.000030
22	0.9987820	0.000054
23	0.9986980	0.001302
24	0.9986220	6.000000
25	0.9985410	0.000081
26	0.9984410	0.000100
27	0.9983380	0.001662
28	0.9982250	7.000000
29	0.9981190	0.000106
30	0.9980060	0.000113

Name	Extended Duration 5mm - 0.6 deg. c	
Version	14	
Date	23/08/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	500	
Time Step Length (seconds)	30	
No. of Time Steps	30	
Time Step	Water Volume	Difference/Time Step
Total Time	0.9980060	0.001994
0	1.0000000	0.000000
1	0.9957200	0.004280
2	0.9975440	-0.001824
3	0.9978040	0.002196
4	0.9979970	1.000000
5	0.998242	-0.000245
6	0.9984390	-0.000197
7	0.9985880	0.001412
8	0.9987070	2.000000
9	0.9987990	-0.000092
10	0.9988050	-0.000006
11	0.9988760	0.001124
12	0.9988810	3.000000
13	0.9989340	-0.000053
14	0.9989780	-0.000044
15	0.9990190	0.000981
16	0.9990330	4.000000
17	0.9990160	0.000017
18	0.9989570	0.000059
19	0.9989010	0.001099
20	0.9988660	5.000000
21	0.9988360	0.000030
22	0.9987820	0.000054
23	0.9986980	0.001302
24	0.9986220	6.000000
25	0.9985410	0.000081
26	0.9984410	0.000100
27	0.9983380	0.001662
28	0.9982250	7.000000
29	0.9981190	0.000106
30	0.9980060	0.000113

Name	Extended Duration 5mm -0.2 deg. c	
Version	14	
Date	25/08/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	500	
Time Step Length (seconds)	30	
No. of Time Steps	30	
Time Step	Water Volume	Difference/Time Step
Total Time	0.9980060	0.001994
0	1.0000000	0.000000
1	0.9957200	0.004280
2	0.9975440	-0.001824
3	0.9978040	0.002196
4	0.9979970	1.000000
5	0.998242	-0.000245
6	0.9984390	-0.000197
7	0.9985880	0.001412
8	0.9987070	2.000000
9	0.9987990	-0.000092
10	0.9988050	-0.000006
11	0.9988760	0.001124
12	0.9988810	3.000000
13	0.9989340	-0.000053
14	0.9989780	-0.000044
15	0.9990190	0.000981
16	0.9990330	4.000000
17	0.9990160	0.000017
18	0.9989570	0.000059
19	0.9989010	0.001099
20	0.9988660	5.000000
21	0.9988360	0.000030
22	0.9987820	0.000054
23	0.9986980	0.001302
24	0.9986220	6.000000
25	0.9985410	0.000081
26	0.9984410	0.000100
27	0.9983380	0.001662
28	0.9982250	7.000000
29	0.9981190	0.000106
30	0.9980060	0.000113

Name	HumControlV3	
Version	18	
Date	28/09/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	1	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
Total Time	0.999980	0.000020
0	1.000000	0.000000
1	0.999986	0.000014
2	0.999983	0.000003
3	0.999982	0.000001
4	0.999981	0.000001
5	0.999980	0.000001

Name	Hum -06	
Version	18	
Date	27/09/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	296.55	
Max Iterations	2000	
Time Step Length (seconds)	1	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
Total Time	0.999980	0.000020
0	1.000000	0.000000
1	0.999986	0.000014
2	0.999983	0.000003
3	0.999982	0.000001
4	0.999981	0.000001
5	0.999980	0.000001

Name	Increased Temperature Difference	
Version	19	
Date	30/09/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	1.000000	0.000000
1	0.999988	0.000012
2	0.999985	0.000003
3	0.999984	0.000001
4	0.999977	0.000007
5	0.999960	0.000017
	Total	0.000040

Name	Increased Temperature Difference	
Version	18	
Date	1/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	295.95	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	1.000000	0.000000
1	0.999988	0.000012
2	0.999986	0.000002
3	0.999982	0.000004
4	0.999977	0.000005
5	0.999954	0.000023
	Total	0.000046

Name	July Control	
Version	19	
Date	9/10/2020	
Air Temp. (deg. Kelvin)	294.15	
Water Temp. (deg. Kelvin)	288.15	
TBL Temp. (deg. Kelvin)	288.15	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	4	
Time Step	Water Volume	Difference/Time Step
0	1.000000	0.000000
1	0.999984	0.000016
2	0.999982	0.000002
3	0.999982	0.000000
4	0.999981	0.000001
	Total	0.393788

Name	July -0.6 Deg. C	
Version	19	
Date	10/10/2020	
Air Temp. (deg. Kelvin)	294.15	
Water Temp. (deg. Kelvin)	288.15	
TBL Temp. (deg. Kelvin)	287.55	
Max Iterations	2000	
Time Step Length (seconds)	10	
No. of Time Steps	4	
Time Step	Water Volume	Difference/Time Step
0	1.000000	0.000000
1	0.999984	0.000016
2	0.999982	0.000002
3	0.999982	0.000000
4	0.999981	0.000001
	Total	0.393788

Appendix G – Realistic Condition Results

Name	Wind Control	
Version	19	
Date	1/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625766	0.019416
2	0.614409	0.011357
3	0.609887	0.004522
4	0.606212	0.003675
5	0.604543	0.001669
	Total	0.040639

Name	Wind -1.2 deg. C.	
Version	19	
Date	2/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	295.55	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625537	0.019645
2	0.615234	0.010303
3	0.612886	0.002348
4	0.612402	0.000484
5	0.612372	0.000030
	Total	0.032810

Name	Wind -06 deg. C.	
Version	19	
Date	4/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	296.55	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	5	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625710	0.019472
2	0.614905	0.010805
3	0.611899	0.003006
4	0.611640	0.000259
5	0.611347	0.000293
	Total	0.033835

Name	Wind Control Extended	
Version	19	
Date	12/10/2020	
Air Temp. (deg. Kelvin)	302.15	
Water Temp. (deg. Kelvin)	297.15	
TBL Temp. (deg. Kelvin)	297.15	
Max Iterations	2000	
Time Step Length (seconds)	5	
No. of Time Steps	10	
Time Step	Water Volume	Difference/Time Step
0	0.645182	0.000000
1	0.625766	0.019416
2	0.614409	0.011357
3	0.609887	0.004522
4	0.606212	0.003675
5	0.604543	0.001669
6	0.603307	0.001236
7	0.602443	0.000864
8	0.601789	0.000654
9	0.601322	0.000467
10	0.600992	0.000330
	Total	0.044190