WALL PRESSURE AND DUCTING FIN: A STUDY OF THE JOINT EFFECT ON THE EFFICIENCY OF FIN DEVICES IN DUCTING

MADIHAH FARHANA BINTI MOHD FIKRI

B002010195

BACHELOR OF HEATING,VENTILATION & AIR CONDITIONING ENGINEERING TECHNOLOGY (BMKH) WITH HONOURS

2024

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Faculty of Mechanical Engineering Technology

WALL PRESSURE AND DUCTING FIN: A STUDY OF THE JOINT EFFECT ON THE EFFICIENCY OF FIN DEVICES IN DUCTING

MADIHAH FARHANA BINTI MOHD FIKRI

Bachelor of Heating,Ventilation & Air Conditioning Engineering Technology (BMKH) with Honours

2024

WALL PRESSURE AND DUCTING FIN: A STUDY OF THE JOINT EFFECT ON THE EFFICIENCY OF FIN DEVICES IN DUCTING

MADIHAH FARHANA BINTI MOHD FIKRI

A thesis submitted
in fulfillment of the requirements for the degree of
Bachelor of Heating,Ventilation & Air Conditioning Engineering Technology
(BMKH) with Honours

Faculty of Mechanical Engineering Technology

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2024

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **Wall Pressure And Ducting Fin: A Study Of The Joint Effect On The Efficiency Of Fin Devices In Ducting**

SESI PENGAJIAN: **2023-2024 Semester 1**

Saya **MADIHAH FARHANA BINTI MOHD FIKRI**

mengaku membenarkan tesis ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

☐	TERHAD	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia sebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)
☐	SULIT	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
√	TIDAK TERHAD	

Disahkan oleh:

Alamat Tetap:
NO174C, LORONG 7, TAMAN SERI

BAHAGIA, 36000 TELUK INTAN, PERAK

Cop Rasmi:
TS. MOHD FARUQ BIN ABDUL LATIF
Pensyarah
Fakulti Teknologi Dan Kejuruteraan Mekanikal
Universiti Teknikal Malaysia Melaka (UTeM)

Tarikh: 6/2/2024

Tarikh: 6/2/2024

** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebagai SULIT atau TERHAD.

DECLARATION

I declare that this Choose an item. entitled “ **Wall Pressure And Ducting Fin: A Study Of The Joint Effect On The Efficiency Of Fin Devices In Ducting**” is the result of my own research except as cited in the references. The Choose an item. has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature :

Name : MADIHAH FARHANA BINTI MOHD FIKRI

Date : 12/1/2024

APPROVAL

I hereby declare that I have checked this thesis and in my opinion, this thesis is adequate in terms of scope and quality for the award of the Bachelor of Heating, Ventilation and Air Conditioning Engineering Technology (BMKH) with Honours.

TS. MOHD FARUQ BIN ABDUL LATIF
Pensyarah
Fakulti Teknologi Dan Kejuruteraan Mekanikal
Universiti Teknikal Malaysia Melaka (UTeM)

Signature	:	
Supervisor Name	:	TS. MOHD FARUQ BIN ABDUL LATIF
Date	:	12/1/2024

DEDICATION

My heartfelt gratitude and warmest regards for devoting this work to Allah S.W.T., my family and friends, especially my respected parents, Mohd Fikri bin Sulaiman and Mesturah binti Awang who consistently encourage and supporting me. My dissertation is also dedicated to my instructors and my course mates who generously shared their knowledge and words of advices with me as I was working on this thesis.

ABSTRACT

Wall pressure has a significant impact on the pressure distribution and airflow patterns within the ducting system and along the fin surfaces. The importance of doing this study is to see the principle of turbulent entropy generation is used to analyze the influence of wall and reversed pressure gradient on air duct of HVAC system. On the basis of this inquiry, a computational modelling is explored by using Hypermesh software and the simulation was performed and run through ANSYS. This technique consists of six stages: CAD modeling, meshing, CFD calculation and CFD post processing, including testing and analysis. Eventually, in the course of this study, this processes will be used to compare the simulation result of a fin and without fin and physical validation of a benchmark with only fin.

ABSTRAK

Tekanan dinding mempunyai kesan yang ketara terhadap pengedaran tekanan dan corak aliran udara dalam sistem penyaluran dan sepanjang permukaan sirip. Kepentingan dalam melaksanakan kajian ini adalah untuk meneliti prinsip penjanaan entropi turbulen yang digunakan untuk menganalisis pengaruh dinding dan kecerunan tekanan terbalik pada saluran udara sistem HVAC. Atas dasar siasatan ini, pemodelan pengiraan diterokai dengan penggunaan perisian Hypermesh dan simulasi telah dilakukan dan dijalankan melalui ANSYS. Teknik ini terdiri daripada enam peringkat: pemodelan CAD, meshing, pengiraan CFD dan pemprosesan pasca CFD, termasuk ujian dan analisis. Akhirnya, dalam perjalanan kajian ini, proses ini akan digunakan untuk membandingkan hasil simulasi yang mempunyai sirip dan tanpa sirip dan pengesahan fizikal penanda aras yang mempunyai sirip sahaja.

ACKNOWLEDGEMENTS

In the Name of Allah, the Most Gracious, the Most Merciful

First and foremost, I would like to thank and praise Allah the Almighty, my Creator, my Sustainer, for everything I received since the beginning of my life. I would like to extend my appreciation to Universiti Teknikal Malaysia Melaka (UTeM) for providing the research platform. Thank you also to the Malaysian Ministry of Higher Education (MOHE) for the financial assistance.

My utmost appreciation and special thanks goes to my main supervisor, TS. Mohd Faruq Bin Abdul Latif,for all his support, help, advice and inspiration. His constant patience for guiding and providing priceless insights that I received from him will forever be remembered. Also, to Universiti Teknikal Malaysia Melaka (UTeM) who constantly supported my journey.

Last but not least, from the bottom of my heart a gratitude to my beloved parents and family members for their endless support and love, who always pray for my success. My friends and classmates that I would also like to thank for providing me the assistance, support me mentally and the inspiration to embark on my roller-coaster study life.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS AND ABBREVIATIONS

ACCA	-	Air Conditioning Contractors of America
AMR	-	Adaptive Mesh Refinement
BTU	-	British Thermal Unit
CAD	-	Computer Aided Design
CFD	-	Computational Fluid Dynamic
CFM	-	Cubic Feet per Minute
CSD	-	Computational Structural Dynamics
DMD	-	Dynamic Mode Decomposition
DNS	-	Direct Numerical Simulation
EHD	-	Electrohydrodynamic
FEA	-	Finite Element Analysis
FOE	-	Foreign Object Exclusion
FRP	-	Fiberglass Reinforced Plastic
GCI	-	Grid Convergence Index
HVAC	-	Heating Ventilation and Air Conditioning
LES	-	Large Eddy Simulation
PRISMA	-	Preferred Reporting Items Systematic Reviews and Meta-Analyses
PTV	-	Particle Tracking Velocimetry
PVC	-	Polyvinyl Chloride
RANS	-	Reynolds-Average Navier-Stokes
SLS	-	Selective Laser Sintering

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Background

An efficiency fin devices plays an important role in enhancing heat transfer in a ducting systems. The performance of these devices is controlled by a variety of parameters, including wall pressure and ducting angle. Understanding the interaction of a wall pressure and ducting angle on a fin device efficiency is essential for improving their design and performance.

This backround section discusses the significance of wall pressure and ducting angle in ducting systems and emphasises the need for a complete investigation to determine their combined effect on the efficiency of fin devices.

A wall pressure is known as a pressure imposed by the moving air on the surface of a duct wall. It has a considerable impact on heat transmission between the air and the duct wall in the ducting system. By increasing the contact between the air and the fin surface, higher wall pressure can improve convective heat transfer. Studying the relationship between wall pressure and fin device efficiency is critical for attaining efficient heat transfer and optimising system performance.

A ducting angle is the angle formed by the duct's axis and the plane of the fin. It has a significant impact on the flow and heat transfer characteristics of the ducting system. The airflow pattern and boundary layer growth around the fin surfaces can be influenced by selecting a suitable ducting angle. By increasing air contact with the fin surfaces and lowering flow separation and pressure drop, optimising the ducting angle can promote increased heat transfer performance. (Altwieb et al., 2020)

While earlier research has examined the impacts of wall pressure and ducting angle on the performance of fin devices in ducting systems separately, there has been a paucity of thorough studies investigating their combined effect.

Knowledge about interaction of both of these elements is critical for having a better understanding of the complicated heat transfer mechanisms that occur within the ducting system. A thorough investigation into the combined influence of wall pressure and ducting angle on the effectiveness of fin devices will provide significant information for constructing more efficient heat transfer systems.

In conclusion, understanding how wall pressure and ducting angle interact is vital for optimising the design and function of fin devices in ducting systems. This study intends to contribute to existing knowledge by offering useful insights into the complex heat transfer mechanisms and airflow patterns by analysing these elements concurrently. The findings of this study can help engineers choose the best wall pressure and ducting angle for various applications and aid in the creation of more efficient heat transfer systems.

1.2 Problem Statement

In a ducting system, air or fluid flows through a confined space, typically with the aim of transferring heat between the flowing medium and the surrounding walls. In these duct systems, efficiency fin devices are frequently employed to improve heat transmission. However, factors like as wall pressure and fin ducting can have an impact on airflow performance.

The pressure distribution along the fin surfaces and within the ducting system is heavily influenced by wall pressure. Higher wall pressures can cause increased pressure gradients and fluid flow velocity. This can improve convective heat transfer and overall efficiency of fin devices. The problem statement intends to examine and analyse the pressure distribution along the ducting system's walls when various combinations of wall pressure and ducting angle are used. This issue statement tries to understand how different wall pressures and ducting angles influence the distribution of pressure along the walls. (Qiu et al., 2017)

Additionally, computational fluid dynamics (CFD) simulations can be employed to analyze the pressure distribution within the duct and assess the impact of the fin on flow behavior. Furthermore, the use of optimization techniques and experimental validation can help in developing effective strategies to address and optimize the pressure distribution in ducts with fin devices.

Another issue that always come up from researchers is the change of airflow patterns within the ducting system are influenced by the combined effect of wall pressure and ducting fin. Changes in wall pressure and ducting fin vary flow velocity and direction, influencing convective heat transfer. Optimal wall pressure and fin combinations can generate ideal flow patterns, such as increased mixing and reduced recirculation zones, hence increasing heat transfer efficiency. (McGill AirFlow Corporation, 2003)

Common sources of airflow pattern difficulties in ducts include impediments such as blocked vents and registers, clogged filters, inadequate return air routes, leaky ducts, and incorrectly planned or installed duct systems. These difficulties can result in restricted airflow, hot and cold areas around the room, pressure imbalance, and uneven cooling. To address these issues, inspect the ductwork for obstructions, clean the air ducts, check the air registers or supply vents for any items or dust buildup, and perform computational fluid dynamics (CFD) simulations to study the airflow patterns within the duct.

Besides that, having noise generation can effect every pressure point on airflows in duct system, where it can give big impact on system performance. Pressure points can also contribute to the increasing of noise levels within the duct system as air is forced through restricted areas, leading to turbulence and also vibration.

Finally, by achieving these expected results from all the problem statements enables researchers to improve system efficiency and heat transfer by optimising the design and performance of fin devices.

1.3 Research of Objective

As for this section, the primary objective of this study is to investigate the joint effect of wall pressure and ducting fin in terms on efficiency of fin devices in ducting systems. The specific research objectives are as follows :

1. To develop scale model of square ducting as validation for CFD.
2. To investigate the flow region of square ducting using CFD.
3. To evaluate the relationship between pressure and fin ducting.

1.4 Scope of Research

The scope of this research are as follows:

1. Limitation to scale down model
2. Focusing on vertical angle in square duct.
3. Study on certain angle duct based on wall pressure distribution.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

An essential part of every ducted heating and cooling system is the ductwork itself. The ductwork is in charge of moving air to and from the HVAC system where it distributes clean air by heating and cooling units throughout the enclosed space and eliminates stable air via ducts.

For this section, a systematic review will be conducted to examine the existing knowledge and study findings about the relationship of wall pressure and ducting fin. The influence of the fin at which a duct is linked to a wall on pressure distribution and flow characteristics is the focus of this topic. The first stage in planning a literature review is to explicitly state the study topic or purpose. "What is the effect of ducting fin on wall pressure distribution in fluid flow systems?" for example, could be the research question.

The review's scope will include identifying and reviewing relevant papers that study the link between wall pressure and ducting angle in a variety of contexts, such as fluid dynamics, ventilation systems, or HVAC (Heating, Ventilation, and Air Conditioning) systems. Experimental research, computational simulations, theoretical assessments, and case studies may all be included in the review.

The chapter flow for the literature review can be structured starting with introduction, theoretical background, experimental studies, computational simulations, application studies, summaries and conclusions.

2.2 PRISMA Systematic Review

PRISMA method is a widely recognized and accepted protocol for conducting and reporting a systematic review. It provides a standardized and transparent approach to conducting literature reviews and synthesizing evidence from multiple studies.

The PRISMA method is made up of a 27-item checklist and a flow diagram that offers writers with a standardised way to report their review techniques and outcomes. The check method, study selection criteria, data extraction, and risk of bias evaluation. A systematic review of studies by Grimshaw published between 2009 and 2015 found that the overall quality of reporting had improved after the introduction of the PRISMA guideline. (Grimshaw et al., 2021)

Using PRISMA method ensures a rigorous and open review process, lowering the possibility of bias and errors in evidence synthesis. It also makes it easier for others to replicate the review, which improves the reliability and validity of the findings. Researchers, journal editors, and policymakers all across the world have embraced the PRISMA technique. Its application has aided in the improvement of the quality and consistency of systematic reviews and meta-analyses, which are essential for evidence-based decision-making in healthcare and other sectors. (Grimshaw et al., 2021)

Basically, the PRISMA statement is a guideline that provides a standardized approach for authors to report their systematic review and meta-analysis methods and results. Its use has contributed to the transparency and accuracy of systematic reviews and meta-analyses and has become an essential tool in the field of evidence-based medicine.

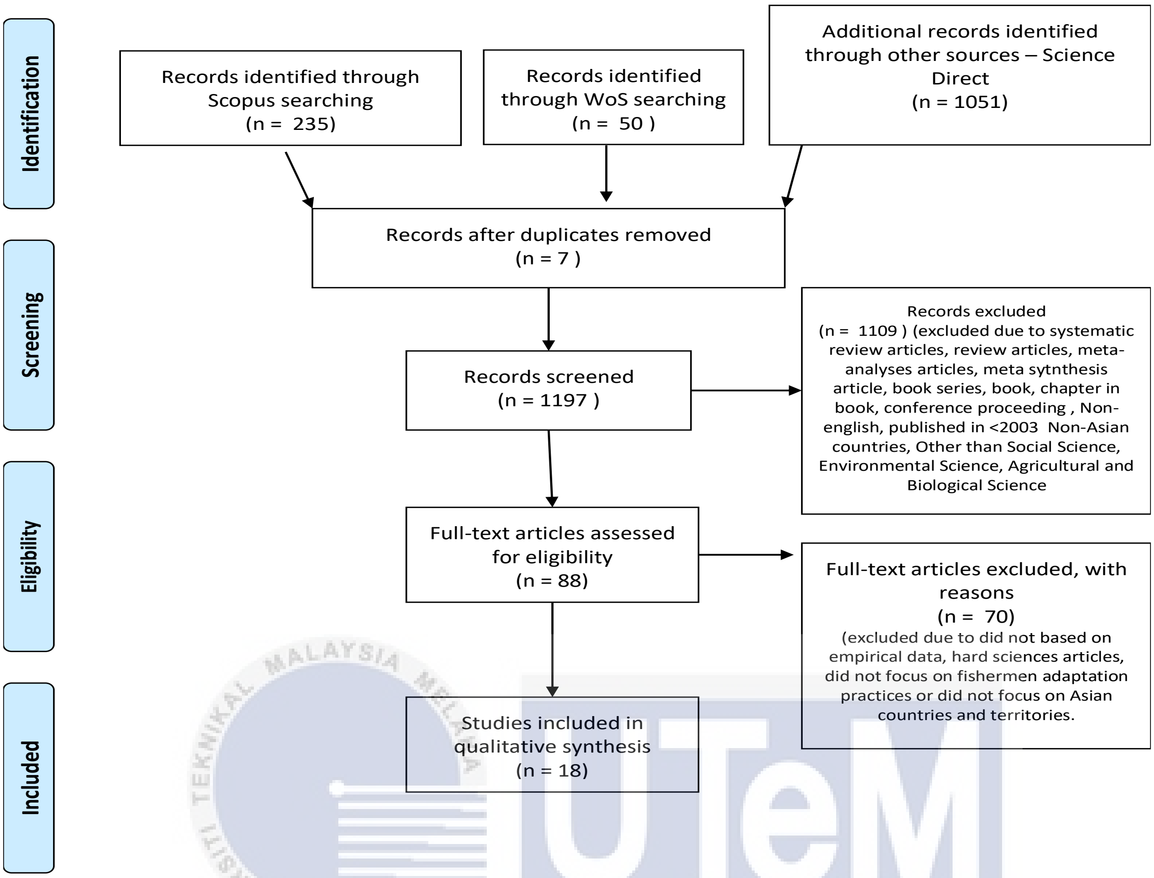

Figure 1 PRISMA Flowchart (Du et al., 2023)

Condcuting a systematic review according to the preferred reporting of items for systematic reviews and meta-analyses (PRISMA) protocol is important with the aim to develop a comprehensive overview of the online trace data approach, which can provide guidance for future research. The example of PRISMA flowchart was shown above. The four-phase flowchart that was presented in Figure 1 describes the actual number of records identified, screened, and ultimately included in the reviewing process. Using the PRISMA flowchart can improve the replicability and transparency of a study. (Du et al., 2023)

2.2.1 Identification Method

Identification method is a process of looking for and selecting relevant studies to include in the systematic review. It is an important step in the review process because it dictates the extent and quality of the evidence that will be synthesized.

The identification method in PRISMA involves several key steps. First, the research question and inclusion/exclusion criteria are defined. It means that we have to identify the important keywords based on the project title. Then, we would searched the synonyms of the keywords or different spelling of these words. These criteria may describe the types of studies to be included or randomised controlled trials, observational studies as well as the research participants' characteristics, interventions, and outcomes of interest.

The primary purpose of the identification method is to minimize bias and increase the likelihood of capturing all relevant studies. By conducting a thorough search using appropriate databases, search terms, and inclusion criteria, researchers aim to locate as many eligible studies as possible. This comprehensive approach helps to reduce the risk of publication bias, where only studies with positive or statistically significant results are included, while ignoring those with neutral or negative findings. Basically, a systematic review attempts to collate all the empirical evidence that fits pre-specified eligibility criteria in order to answer a specific research question. A study from Higgins confirmed that systematic methods that are selected with a view to minimizing bias, thus providing more reliable findings from which conclusions can be made. (Higgins et al., 2020) Next, to discover relevant studies, a detailed search technique is devised. Searching databases (such as Scopus, Web of Science, Science Direct or Google Scholar), hand-searching reference lists and citation tracking, and consulting experts in the field may all be part of this strategy.

After that, to guarantee that all relevant studies are identified, the search method should be thoroughly documented, including the databases examined, the search words used, and the date the search was carried out. Following the completion of the search, the identified studies are screened for eligibility based on the inclusion/exclusion criteria. Typically, this is done in two stages: first, titles and abstracts are screened to find possibly relevant research, and then full-text publications are evaluated to see if they fit the inclusion or exclusion criteria.

2.2.2 Screening Method

Screening method is a process of selecting relevant studies from a pool of potentially eligible research identified during the identification phase. The screening method in PRISMA involves two stages: a title or an abstract screening stage and a full-text screening stage. (Grimshaw et al., 2021)

The reviewers assess the relevance of each study during the title/abstract screening stage based on the information supplied in the title and abstract. This is a fast and effective approach that permits reviewers to screen a large number of papers. Another study of systematic reviews from China in orthopedics found that the use of two or more independent reviewers for screening and data extraction was associated with higher reporting quality scores. (Zhi et al., 2017)

Studies that do not fit the inclusion criteria are discarded, but those that do meet the requirements or are potentially relevant are kept for full-text screening. During the full-text screening stage, the reviewers obtain the full-text articles of the retained studies and assess them for eligibility based on the inclusion and exclusion criteria. This stage is more extensive and time-consuming since it entails a full examination of the study methods, findings, and conclusions. Studies that match the inclusion criteria are kept for data extraction and analysis, while those that don't are excluded.

2.2.3 Eligibility Method

Eligibility method refers to the process of determining which studies meet the pre-specified inclusion criteria and will be included in the systematic review or meta-analysis. The eligibility method in PRISMA involves a thorough evaluation of each study that has passed the screening stage, based on the inclusion and exclusion criteria. In most cases, the inclusion criteria describe the types of studies to be included (e.g., randomised controlled trials, observational studies) as well as the features of the research participants, interventions, and outcomes of interest. Exclusion criteria, on the other hand, identify the types of studies that are not acceptable for inclusion, such as those having a high risk of bias, incomplete or insufficient data, or those are irrelevant to the research topic. (Page et al., 2016)

The PRISMA guideline emphasizes transparency and reproducibility in the eligibility method. Authors are encouraged to clearly specify the inclusion and exclusion criteria used in the review and provide a detailed rationale for their choices. This transparency allows readers to assess the validity and appropriateness of the included studies.

The eligibility method also involves assessing the risk of bias in individual studies. This evaluation helps to gauge the quality and reliability of the evidence. Various tools and checklists, such as the Cochrane Risk of Bias Tool, can be utilized to assess the risk of bias in different study designs. (Moher et al., 2009)

During the eligibility method, the reviewers assess each study against the inclusion and exclusion criteria to determine whether it should be included or excluded from the review. This evaluation often includes a thorough examination of the study's methodologies, findings, and conclusions, as well as any supplementary information pertinent to the research topic.

2.2.4 Inclusion Method

Inclusion method refers to the process of selecting studies that have passed the eligibility criteria and will be included in the final analysis of the systematic review or meta-analysis. Another study of systematic reviews published in high-impact medical journals found that the PRISMA guideline was associated with higher reporting quality scores. (Kamioka, 2019)

The inclusion method in PRISMA involves a thorough assessment of each eligible study to determine whether it meets the criteria for inclusion in the final analysis. This evaluation often includes a thorough examination of the study's methodologies, findings, and conclusions, as well as any supplementary information pertinent to the research topic. The eligible studies that match the inclusion requirements are included in the final analysis, and their data are collected and synthesised to answer the research question using the included method.

This method may be analysed using a variety of statistical methodologies, such as meta-analysis, qualitative synthesis, or narrative synthesis, depending on the research question and the nature of the data. This method is crucial for ensuring the final analysis's quality and dependability. The inclusion method ensures that the final analysis is based on high-quality data that is relevant to the research issue by selecting studies that match the pre-specified eligibility criteria and excluding those that do not. (Kamioka, 2019)

2.3 History of Findings

On this section, the history of findings provides a summary and review of the important historical research and findings connected to the topic under investigation. It intends to provide a chronological summary of major investigations, theories, and discoveries that have led to the current understanding of the subject. It starts by identifying the early studies or key research publications that lay the foundation for the field of investigation. It emphasises the major findings, methodology used, and any significant discoveries that resulted from those early investigations.

The section then moves forward in time, summarising future investigations and advances in the subject. It could go into detail about how theories changed, methodology were refined, and new approaches or technology were introduced. Important experiments, empirical findings, or theoretical frameworks that have shaped contemporary understanding of the issue may be highlighted in this section.

In summary, the history of findings portion of a literature review chapter provides a chronological overview of the field's major studies and discoveries. It emphasises the progression of knowledge and the contributions of previous researchers, laying the groundwork for current research and establishing its relevance in light of previous findings.

2.3.1 Fishbone Diagram

Fishbone diagram is known as a cause-and-effect diagram or an Ishikawa diagram, is a tool that used to identify and organize causes of a specific pro blem or effect. It is referred to as a fishbone diagram because of its design, which resembles a fish's skeleton, with the effect or problem as the "head" and potential causes branching out as the "bones.". (Coccia, 2018) Figure 2 shows a fishbone diagram along with the findings from various authors.

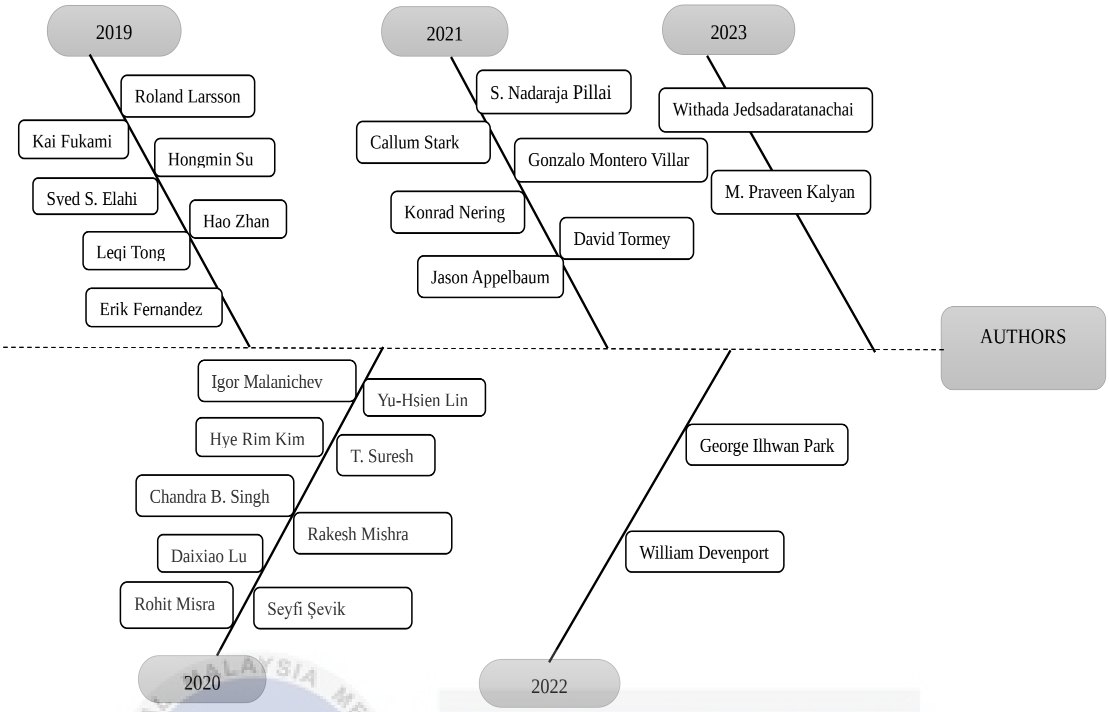

Figure 2 Fishbone Diagram

The diagram refers to the outcomes or results obtained from using a fishbone diagram to analyze a problem or investigate the root causes of an issue. The findings are the insights and conclusions derived from the diagram. The major consequence or problem is identified and placed at the top of the fishbone diagram when it is created. Then, as 'bones' radiating from the main line, categories or branches are drawn. These categories are often used to describe various features or causes that may contribute to the situation. Examples of common categories include people, processes, equipment, materials, environment, and management.

Following the establishment of the categories, the following stage is to identify possible causes within each category. These causes are depicted as smaller branches branching from the appropriate category lines. Causes can be generated through brainstorming, gathering input from stakeholders, or relying on accessible facts or evidence.

Besides that, the diagram also refer to the list of causes identified and organized within the fishbone diagram. These findings highlight the potential factors contributing to the problem or effect under investigation. The causes identified in the fishbone diagram serve as a starting point for further analysis and problem-solving efforts.

The fishbone diagram findings, which represent potential causes and their relationships, provide a clear and systematic summary of the various sources of the problem. This aids in comprehending the issue's complexities and allows teams or researchers to choose areas for further inquiry or corrective action. (Coccia, 2018)

In brief, "Fishbone diagram findings" refer to the discovered causes or prospective components that contribute to a certain problem or impact, as represented by a fishbone diagram. The findings give an organised approach for problem analysis and resolution, as well as a visual representation of the likely reasons.

2.3.2 Table of Finding

For this section, it started with Table 1 where it shows how table of findings been created and what are the findings that each author had discovered.

Table 1 Authors and Findings

YEAR	AUTHORS	FINDINGS
2019	• Marko Toši´c • Roland Larsson • Janko Jovanovic • Thomas Lohner	➢ Investigation of fluid behaviour in thermal elastohydrodynamic line contacts, with a focus on understanding the shearing mechanisms involved using CFD.
	• Kai Fukami, • Yusuke Nabae, • Ken Kawai	➢ Machine learning-based approach for generating realistic turbulent inflow

	• Koji Fukagata	conditions using simulations and algorithms.
	• Guiyu Cao • Hongmin Su • Jinxiu Xu • Kun Xu	➢ Proposal of a high-order implicit gas kinetic system for modelling turbulent flows
	• Syed S. Elahi • Eric A. Lange • Stephen P. Lynch	➢ Investigation regarding impact of Reynolds number on turbulent flow and heat transfer in a junction configuration.
	• Baigang Mi • Hao Zhan	➢ Improvement of the effectiveness of foreign object exclusion (FOE) in the intake system of a turboprop engine.
	• Leqi Tong • Jun Gao • Zhiwen Luo	➢ Achievement of uniform exhaust airflow in a central air exhaust ventilation system.
	• Michael Elmore • Erik Fernandez • Jayanta Kapat	➢ Analyzing the heat transfer characteristics of turbulence-generating ribs using a technique called dynamic mode decomposition (DMD)
2020	• Igor Malanichev • Fail Akhmadiev	➢ Improvement efficiency of airflow in ventilation systems.

	• Yu-Hsien Lin • Xian-Chen Li	➢ Investigation effectiveness of a sliding mesh model for conducting hydrodynamic analysis of a SUBOFF model in turbulent flow fields.
	• Torben Eggers • Hye Rim Kim • Simon Bittner • Jens Friedrichs	➢ A study of aerodynamic and aeroelastic effects of design-based geometry variations on a low-pressure compressor.
	• O. Szulc • P. Doerffer • P. Flaszynski • T. Suresh	➢ A study regarding control of shock wave-boundary layer interaction through passive wall ventilation.
	• Shubham Subrot Panigrahi • Chandra B. Singh • John Fielke	➢ Analyzing the physical velocity and anisotropic resistance components in a peaked stored grain with aeration ducting systems using CFD.
	• Tongqing Guo • Daixiao Lu • Zhiliang Lu • Binbin Lyu	➢ Development a flutter prediction method for experimental models in a transonic wind tunnel with a porous wall.
	• Rohit Misra • Jagbir Singh • Sheetal Kumar Jain • Pradeep K. Goyal	➢ Prediction behavior of a triangular solar air heater duct using a V-down rib.

	• Seyfi Şevik • Mesut Abuşka	➢ Improvement of thermal efficiency of a solar air heater through the implementation of single-pass semi-flexible foil ducts.
	• Miftah Altwieb • Krzysztof J.	➢ Optimizing the efficiency of fluid-to-air multi-fin heat exchangers using CFD model.
. 2021	• S. Nadaraja Pillai • Aakhash Sundaresan • Abdul Zubar Hameed • Abdul Gani Abdul Jameel	➢ Estimation of chaotic surface pressure characteristics of airfoils with ice accretion using a 0-1 test approach.
	• Callum Stark • Weichao Shi • Mehmet Atlar	➢ Effects of bio-inspired leading-edge tubercles on the hydrodynamic performance of a benchmark ducted propeller.
	• Sudharsan Vasudevan • Sassan Etemad • Lars Davidson	➢ Estimation of heat flux in subcooled flow boiling and indicating the interaction of vapor bubbles.
	• Konrad Nering • Krzysztof Nering	➢ Validation of a modified algebraic model for airflow prediction

		characteristics during transitional flow in HVAC.
	• Mohammad Reza Kadivar • David Tormey • Gerard McGranaghan	➢ A comprehensive overview of the fundamental aspects and theoretical frameworks related to turbulent flow over rough surfaces.
	• Jason Appelbaum • Duncan Ohno • Ulrich Rist • Christoph Wenzel	➢ Conducting Direct Numerical Simulation (DNS) simulations of a turbulent boundary layer using inflow conditions derived from 4D-PTV (Four-Dimensional Particle Tracking Velocimetry) data.
2022	• Withada Jedsadaratanachai • Amnart Boonloi	➢ Investigation of heat transfer characteristics and fluid flow structure in a square duct with modified wavy baffles.
	• P.S. Arshi Banu • D.N.S.Ramesh Lohith • Dilip Sai Vempati • B. Hemanth Sai	➢ Simulation and validating the performance of a fin and tube heat exchanger using CFD analysis.
2023	• Xiaohan Hu • Imran Hayat	➢ A study of characteristics of a three-dimensional turbulent boundary layer

	• George Ilhwan Park	in a bent square duct to using wall-modelled LES.
	• N. Agastya Balantrapu • W. Nathan Alexander • William Devenport	➢ Investigation of a characteristics of wall-pressure fluctuations in an axisymmetric boundary layer under a strong adverse pressure gradient.

For the first article, (Tošić et al., 2019), it shows that the findings from these authors as about the application of computational fluid dynamics (CFD) to comprehend fluid behaviour in thermal electrohydrodynamic (EHD). They also investigating fluid behaviour in thermal elastohydrodynamic line contacts, with a focus on understanding the shearing mechanisms involved. Article by (Fukami et al., 2019), they presented a machine learning-based approach for generating realistic turbulent inflow conditions where they focusing on the difficulty of effectively turbulent flow patterns that used a machine to overcome challenges and producing synthetic turbulent inflow data. The article from (Cao et al., 2019) stated that simulation turbulent flows using a gas kinetic approach that addresses the challenge of accurately represent turbulent behavior in CFD. It tells that the integrating gas kinetic principles with an implicit formulation where it stated that the technique has the potential to improve the accuracy of turbulence simulations. From (Elahi et al., 2019b), the authors investigated the impact of Reynolds number on turbulent flow and heat transfer in a junction configuration where they mentioned the relationship between Reynolds number and essential flow parameters by investigating fluid dynamics and heat transfer characteristics at varied Reynolds numbers. Another one from (Mi & Zhan, 2019), (Elmore et al., 2019) and (Tong et al., 2019) stated about the acievement of uniform exhaust airflow in a central air exhaust ventilation system that addresses the challenge of uneven airflow distribution and inadequate exhaust in such systems and also mentioned about analyzing the heat transfer characteristics of turbulence-generating ribs using a technique called dynamic mode decomposition (DMD) to understand the heat transfer mechanisms and optimize the design of ribs for enhanced heat transfer in various applications.

In 2020, there were 9 authors and most of them stated on how efficiency of airflow and the development method for experiment models. But like (Szulc et al., 2020), they stated about the study of controlling shock wave boundary layer interction through passive wall ventilation. (Malanichev & Akhmadiev, 2020) mentioned about the improvement efficiency of airflow in ventilation systems that focused on reducing pressure loss in ventilation ducts through shape optimization of removable profiled components. While (Lin & Li, 2020) and (Eggers et al., 2020) said about the investigation of sliding mesh model effectiveness along with the study of aerodynamics and aeroelastic effects of design-based geometry variations on a low-pressure compressor where it shows the relationship between the geometric design variations and the aerodynamic and aeroelastic behavior of the low-pressure compressor that contributes to the optimization of compressor design, enabling improved efficiency, stability, and reliability. (Subrot et al., 2020) mentioned in his article about the use of CFD to analyze the physical velocity and anisotropic resistance components in a peaked stored grain with aeration ducting systems. For (Guo et al., 2020) and (Misra et al., 2020) are different where they mentioned about combination of CFD and CSD to accurately predict the occurrence and characteristics of flutter in the wind tunnel experiments and also the analyzing the thermal and fluid flow characteristics of a triangular solar air heater duct using CFD simulations. Lastly, for (Şevik & Abuşka, 2020) and (Altwieb et al., 2020) stated that investigating the efficiency of the heat exchanger by considering factors such as the geometry of the fins, flow rates, and heat transfer coefficients and the improvement of thermal efficiency of a solar air heater through the implementation of single-pass semi-flexible foil ducts.

In 2021, the authors made more research regarding test approach in development of improvement design such as (Stark et al., 2021) and (Vasudevan et al., 2021) where numerically investigation effects of bio-inspired leading-edge tubercles on the hydrodynamic performance of a benchmark ducted propeller. Besides numerical model estimating heat flux in subcooled flow boiling and indicating the interaction of vapor bubbles. Meanwhile, (Nering, 2021) and (Kadivar et al., 2021) mentioning about a comprehensive overview of the fundamnetal aspects and theoretical frameworks related to turbulent flow over rough surfaces. An article of (Appelbaum et al., 2021) stated that conducting Direct Numerical Simulation (DNS) simulations of a turbulent boundary layer using inflow conditions derived from 4D-PTV (Four-Dimensional Particle Tracking Velocimetry) data.

In 2022, there were only 2 authors that stated about the investigstion of heat transfer characteristics and fluid flow structure in a square duct with modified wavy baffles and the simulation and validating the performance of a fin and tube heat exchanger using CFD analysis to understand the heat transfer characteristics and validate the accuracy of the simulation results by (Boonloi & Jedsadaratanachai, 2022) and (Banu et al., 2022).

Lastly, for 2023, the other two authors from (Hu et al., 2023) and (Balantrapu et al., 2023) stated that they used the wall-modelled large-eddy simulation (LES) to study the characteristics of a three-dimensional turbulent boundary layer in a bent square duct to understand the complex flow behavior and turbulence dynamics in such ducts and also mentioning about the knowledge of impact of adverse pressure gradients on the pressure fluctuations near the wall.

2.3.3 Table of Analysis

For this section, it started with Table 2 where it shows how table of analysis been created and what are the parameters that each author used for their studies.

Table 2 Table of Analysis

AUTHOR & YEAR	CFD	DUCTING	FIN	TURBULENCE / MESH	WALL PRESSURE
(Tosic et al., 2019)	/				
(Altwieb et al., 2020)	/		/		
(Stark et al., 2021)	/	/			
(Tong et al., 2019)		/			
(Eggers et al., 2020)					/
(Guo et al., 2020)	/				
(Boonloi & Jedsadaratanachai, 2022)		/			
(Subrot et al., 2020)	/				
(Appelbaum et al., 2021)				/	
(Şevik & Abuşka, 2020)		/			
(Pasha et al., 2021)			/		
(Cao et al., 2019)				/	
(Elmore et al., 2019)		/	/		
(Elahi et al., 2019a)				/	

(Vasudevan et al., 2021)					/
(Kadivar et al., 2021)				/	/
(Banu et al., 2022)	/		/		
(Szulc et al., 2020)	/				/
(Mi & Zhan, 2019)	/				
(Fukami et al., 2019)				/	
(Jain et al., 2019)	/	/			
(Malanichev & Akhmadiev, 2020)	/	/			/
(Lin & Li, 2020)	/			/ **(MESH)**	
(Nering, 2021)	/	/			
(Hu et al., 2023)		/		/	
(Balantrapu et al., 2023)					/

From the table above, the most article that been studied all time is about CFD and the relationship between wall pressure and ducting angle. Most of them used turbulence modelling as it is more accurate when it comes to development of efficiency of a system. But the author (Lin & Li, 2020) had mentioned about using MESH method in (Fang et al., 2020) to investigating a slide mesh model for hydrodynamic analysis of a SUBOFF model in turbulent flow fields. In summary, these researchers in this section helps situate the current research within the broader context of past studies. It allows the reader to understand the progression of knowledge, the development of key concepts, and any shifts or controversies that have occurred over time

2.4 Ducting Design

Ducting systems are essential compinents of HVAC systems,enabling controlled airflow and efficient operation. This section provides a comprehensive review of essential ducting design elements such as layout, form, material, insulation, and sizing. The pattern and form of ducting systems affect airflow distribution and pressure losses, highlighting the need of well-designed duct layouts with smooth bends.

Due to the choice of duct materials, it influences thermal conductivity, sound transmission, and fire resistance, necessitating consideration of aspects such as durability, heat transfer, and safety. Adequate insulation, such as fibre glass or foam insulation, contributes to energy efficiency by reducing heat loss or gain. Accurate duct sizing calculations that adhere to industry standards enable optimal airflow rates and minimise inefficiencies. Furthermore, CFD simulations enable extensive study and optimisation of ducting systems. Engineers may design efficient ducting systems for better HVAC performance by taking these considerations into account. (A. Bhatia, 2012)

The effective and controlled airflow for HVAC systems is made possible by ducting systems, which are essential in many sectors. In order to maximise energy efficiency, achieve ideal airflow distribution, and reduce pressure losses, ducting system design is crucial. The aim in designing ducting is to investigate the important elements impacting ducting designs, such as duct layout, shape, material, insulation, and sizing, by looking at pertinent studies and study results. (McGill AirFlow Corporation, 2003)

2.4.1 Duct Material and Insulation

2.4.1.1 Metallic Duct

Aluminium is the next most popular metal duct material. Although aluminium ducts are lightweight, their basic cost per pound is higher than that of galvanised steel. Copper and stainless steel are also utilised in particular conditions, while nonmetallic ducts may comprise glass fibre, compressed paper, plastic, cement-asbestos, vitrified clay, and concrete. Each material of duct has properties that may make it suitable for use in specialised applications. Following is a list of key characteristic of duct materials: (McGill AirFlow Corporation, 2003)

1. Galvanized Steel - Widely used as a duct material for most air handling systems; not recommended for corrosive product handling or temperatures above 400ºF. Advantages include high strength, rigidity, durability, rust resistance, availability, non-porosity, workability, and weldability.
2. Carbon Steel (Black Iron) - Applications include flues, stacks, hoods, other high temperature duct systems, and ducts requiring paint or special coating. Advantages include high strength, rigidity, durability, availability, weldability, and non-porosity. Some limiting characteristics are corrosion resistance and weight.
3. Aluminium - Aluminium ducting is most commonly used for clean room applications. These are also preferred systems for moisture laden air, special exhaust systems and ornamental duct systems. Some advantages include weight and resistance to moisture corrosion. Limiting characteristics include low strength, material cost, weldability, and thermal expansion.
4. Stainless Steel - Used in duct systems for kitchen exhaust, moisture laden air, and fume exhaust. Advantages include high resistance to corrosion from moisture and most chemicals and the ability to take a high polish. Limiting characteristics include labour and material costs, workability, and availability.
5. Copper - Copper applications include duct systems exposed to outside elements and moisture laden air, certain chemical exhaust, and ornamental ductwork. Advantages are durability and corrosion resistance and that it accepts solder readily and is nonmagnetic. Limiting characteristics are cost, ductility, electrolysis, thermal expansion, and stains.

2.4.1.2 Non-Metallic Duct

1. Fibreglass Reinforced Plastic (FRP): Applications include chemical exhaust, scrubbers, and underground duct systems. Limiting characteristics include cost, weight, range of chemical and physical properties, brittleness and fabrication. Fibreglass duct board is insulated and sealed as part of its construction. It provides excellent sound attenuation, but its longevity is highly dependent on its closure and fastening systems. Resistance to corrosion and ease of modification are advantages of FRP. It is usually used to form rectangular supply and return trunks, branches, and plenums, although it can be used for run outs as well.
2. Polyvinyl Chloride (PVC): Applications are exhaust systems for chemical fumes and underground duct systems. Advantages include resistance to corrosion, weight, weldability, and ease of modification. Limiting characteristics include cost, fabrication, code acceptance, thermal shock, and weight.
3. Concrete: Concrete can be used for underground ducts and air shafts. Advantages include compressive strength and corrosion resistance. Cost, weight, porosity, and fabrication (requires forming processes) are some limiting characteristics.
4. Rigid Fibrous Glass: Fibrous glass ducts are fabricated from sheets of materials that have been manufactured from resin bonded inert and inorganic glass fibbers. A factory applied facing (typically aluminium or reinforced aluminium) is applied to one face, and serves as a finish and a vapour barrier. Fibrous glass air ducts have been limited to 2 in- WG pressure and below.

2.4.2 Duct Sizing and Method

Accurate duct sizing is crucial to maintain the desired airflow rate and minimize pressure losses. Several studies emphasize the importance of following industry standards and guidelines, such as those provided by the Air Conditioning Contractors of America (ACCA) Manual D, for duct sizing calculations (Edition & Manual, 2013). Proper sizing ensures that the airflow velocity remains within acceptable limits to avoid excessive noise, pressure drops, and inefficient HVAC operation.

The main goal of designing HVAC duct systems is to use the lowest cost (read smallest) duct sizes that can be used without violating certain sizing constraints. First and operating cost considerations dictate that duct systems should be designed to operate at the lowest possible static pressure. The most widely used method to size duct is constant friction loss method. The other methods are velocity reduction method and static regain method.

2.4.2.1 Constant Friction Lose Method

Duct systems in small buildings are generally sized using the equal friction or modified equal friction method. The equal friction method, as its name implies, is based on maintaining the same pressure drop per unit of duct length (or friction rate) throughout the system (ACCA 1990). The duct size is based on the flow rate through a particular section of duct, and design value for the friction rate. Each section is sized using the design friction rate criterion, and the total pressure drop for each run is simply the sum of the pressure drop of each individual section. The duct sections pressure drop includes straight duct friction loss, pressure losses through fittings such as elbows, takeoffs, and registers or diffusers. In the sections entering and leaving the HVAC unit, pressure losses associated with the flow transitions entering the leaving the unit (the system effect) are also included. The unit fan speed is selected to provide the design cfm and produce enough pressure difference to overcome pressure losses in the supply and return branches having the greatest pressure drop. Note that duct systems designed using the equal friction method is not self-balancing so balancing dampers must be installed in lower pressure loss branches to balance the system. (NAIMA, 2003)

In duct systems with branches having widely varying pressure losses, the modified equal friction method is used to design systems that are closer in balance (Edition & Manual, 2013). Design friction rates fro shorter duct runs are increased in an attempt to design each branch with the same total pressure loss. This method provides a design that is better balanced, but balance dampers must still be installed since it is not possible to provide a truly self balanced system using this method. Also, duct velocities is shorter runs must be checked for noise problems.

2.4.2.2 Velocity Reduction Method

The velocity criterion for sizing duct is fairly simple and straightforward. With this method, the ducts are sized fixing the speed in the duct immediately downstream from the delivery fan and empirically reducing this speed over subsequent duct trunks, normally close to each branch. Velocity limits are commonly used as a surrogate for limiting duct breakout noise. Many argue it is a poor indicator since noise is more likely to result from turbulence than velocity; example like a high velocity system with smooth fittings may make less noise than a low velocity system with abrupt fittings. Nevertheless, limiting velocity to limit noise is a common practice. It is important to consult with the project’s acoustical engineer on this issue. Many rules-of-thumb for velocity limits exist depending on the noise criteria of the spaces served and the location of the duct.

2.4.2.3 Static Regain Method

This method refers to increase or regain of static pressure in the ductwork when the air velocity decreases. The Static Regain method of duct sizing is based on Bernoulli's equation, which states that when a reduction of velocities takes place, a conversion of dynamic pressure into static pressure occurs. With this method, the air speed in the duct is reduced near each branch or diffuser so that the dynamic pressure conversion obtained exactly balances the pressure drop of the air in the trunk of the next duct. This means there is the same static pressure near all the branches and all the diffusers, thereby obtaining an intrinsically balanced air distribution system without having to use throttling devices.

Compared to the two previous methods, this method usually involves a larger surface area of the panels, but lower electric fan power and easier balancing of the plant. For complex plants, it may be advisable to apply two methods simultaneously; the constant pressure loss method for sizing the main trunk, with insertion of adjustment air locks on the branches; the static pressure recovery method for sizing the branches fitted with terminals to obtain the same operating pressure in the latter.

2.5 Meshing and Turbulence

Method meshing in CFD is a technique used to optimize the meshing process in order to obtain accurate and efficient simulations of fluid flow. Meshing refers to the process of dividing the computational domain into a collection of discrete cells or elements, which are used to represent the fluid domain. The quality and resolution of the mesh play a crucial role in the accuracy and computational efficiency of CFD simulations. This method involves the application of different meshing techniques to different regions of the computational domain based on their specific characteristics.

Method meshing optimises the computational resources necessary for accurate simulations by using structured meshes in areas with regular flow behaviour and unstructured meshes in places with complex flow features. It enables finer resolution in crucial areas while preserving a coarser representation in less important areas, lowering processing costs. (Lintermann, 2021)

Overall, method meshing in CFD provides a powerful approach to optimize the meshing process and improve the accuracy and efficiency of fluid flow simulations. By selectively applying different meshing methods, such as structured and unstructured meshes, and incorporating adaptive mesh refinement, method meshing allows for a more precise representation of flow physics while minimizing computational resources. (Li et al., 2017)

Turbulence is a complex fluid dynamics phenomenon characterised by irregular variations in flow characteristics such as velocity, pressure, and temperature. It is important in CFD because it impacts simulation accuracy and dependability. Understanding and effectively modelling turbulence is critical for many technical and scientific applications, including aerodynamics, heat transfer, combustion, and environmental studies.

In CFD simulations, turbulence models aim to capture the statistical behavior of turbulent flows. These models can be broadly categorized into two main types: Reynolds-averaged Navier-Stokes (RANS) models and large eddy simulation (LES) models. (Pei & Rim, 2021). RANS models are the most widely used turbulence models in CFD. They employ time-averaging techniques to separate the flow variables into time-averaged and fluctuating components. The time-averaged Navier-Stokes equations, combined with additional turbulence closure equations, are solved to obtain the mean flow field. The turbulence closure equations provide closure for the turbulent stresses and other relevant quantities. (Casartelli et al., 2022) RANS models assume that the flow can be decomposed into large-scale mean flow and small-scale turbulence, which allows for the efficient simulation of a wide range of turbulent flows. Examples of RANS models include the popular k-ε model, which provides estimates of turbulence kinetic energy and its dissipation rate, and the Reynolds stress models, which directly solve for the Reynolds stresses. (Boonloi & Jedsadaratanachai, 2022)

2.6 Wall Pressure

Wall pressure refers to the force per unit area exerted by a fluid (such as air or a gas) on the surface of a solid boundary, typically a wall or a duct wall. When fluid flows through a duct, it exerts pressure on the walls due to the momentum and energy of the flowing fluid. In this section, a wall pressure specifically refers to the pressure exerted by the flowing air on the walls of the duct. It is an important parameter to consider because it can influence the performance and efficiency of fin devices installed in the ducting system.

Measuring and comprehending wall pressure distribution along duct walls can provide insights into airflow properties, flow patterns, and potential aerodynamic influences that can influence fin device behaviour. The force per unit area exerted by fluid flow on the surfaces of walls or boundaries encountered within a ducting system is referred to as wall pressure. Ducting angle, on the other hand, represents the ducting path's orientation or inclination with respect to the fluid flow direction. Understanding these parameters and how they interact is critical for optimising the design, operation, and maintenance of fluid transport systems such as ventilation ducts, pipelines, and wind tunnels. (Balantrapu et al., 2023)

The effects of wall pressure and ducting angle are particularly significant in situations where the flow of fluid is subject to constraints or alterations due to structural elements. For example, in ventilation systems, the angle at which ducts turn or bend can influence the pressure distribution along the walls, potentially leading to variations in airflow velocity, turbulence, and energy losses. These factors have implications for system efficiency, occupant comfort, and energy consumption. (Pasha et al., 2021)

2.7 Summary

Based on the previous studies, information consist of a ducting shows that ducting is a method of air management that use a series of metal or plastic pipes to transport hot or cooled air from one location to another. Ductwork is another term for a duct system. It is used to shield cables and pipes from earth settlement damage. A PRISMA systematic review can be conducted to gather and analyze relevant studies that have investigated different aspects of ducting design, including airflow characteristics, pressure losses, and thermal performance. Besides that, studies about meshing and turbulence also showed that the review process is rigorous, transparent, and reproducible. It involves clear documentation of search strategies, study selection criteria, data extraction methods, and quality assessment of the included studies.

In summary, this systematic approach helps to minimize biases, enhance the reliability of the review findings, and provide a comprehensive summary of the historical research in the field. By synthesizing the findings from multiple studies, researchers can identify common trends, knowledge gaps, and areas for further research.

CHAPTER 3

METHODOLOGY

3.1 Introduction

The methodology explores deeper into project planning and process flow. It also refers to the appropriate analysis of the methods used in the field of this study. It is also a methodical approach to most tasks, and it is defined as a set of processes, techniques, equipment, procedures, and documentation for data collection in the study. In this section, we will look at the investigative methods used in this experiment. The general concept behind this chapter covers in detail the procedure and tactics used to achieve the project's stated goal. The processes below are used to confirm that the project's outputs match expectations and that the study's goal is met. Furthermore, throughout this chapter, this research gives a flowchart and methodology that emphasises the overall process and strategy of the project.

3.2 Research Design

The study focuses on how wall pressure and ducting angle collectively influence the effectiveness of fin devices used in ducting systems. The essence of the approach used in this project is centered on the concept of 3D Modelling. This research design is illustrated in a flowchart based on figure 3, which arranges the project in a flow chart to make it more methodical and intelligible. Below are the flowcharts of the project concept, that was divided into two which are Computational Modelling process and Physical Validation process.

In general, the flowchart below shows how the project should be carried out in order to achieve the purpose of the research study.

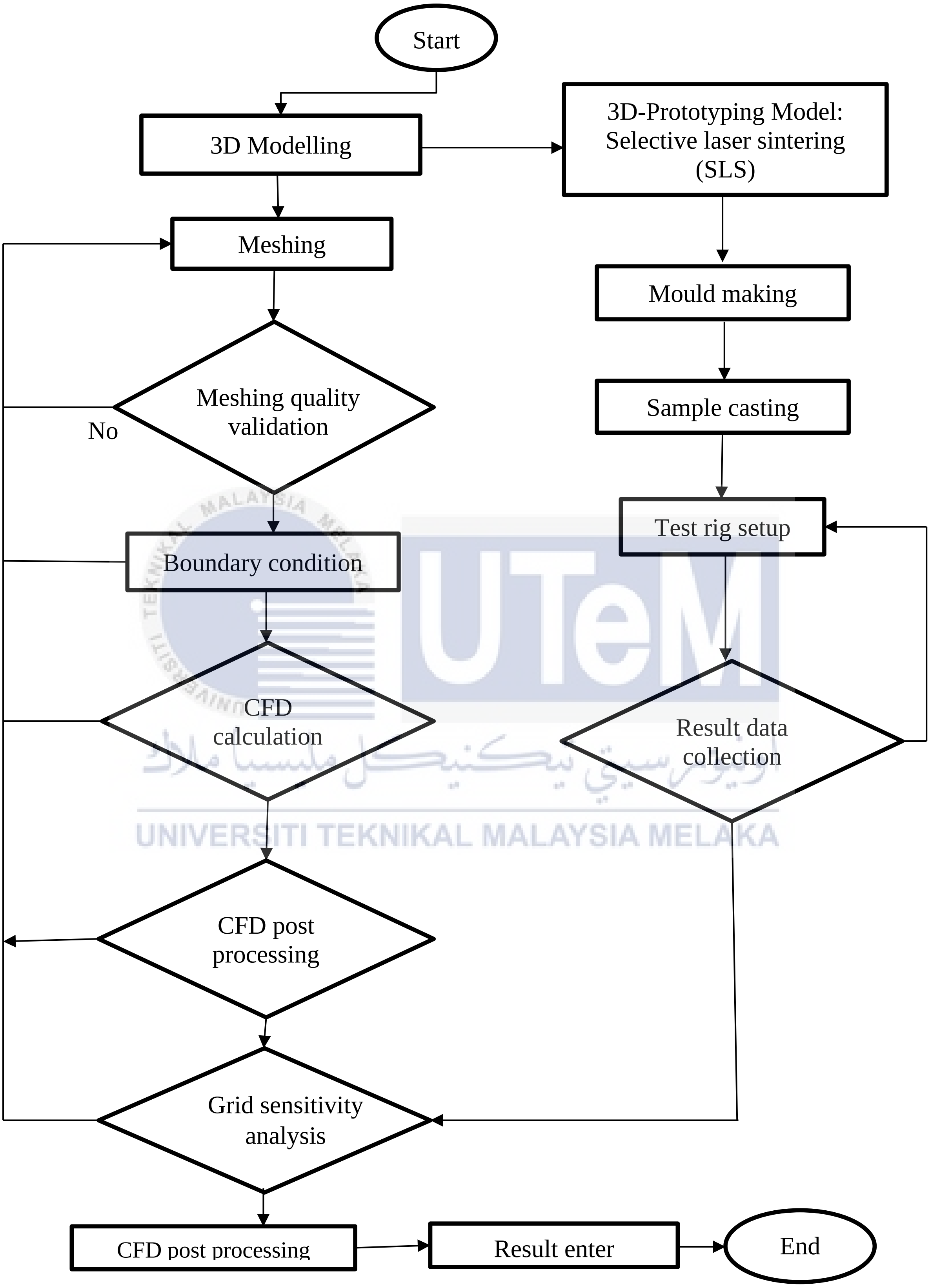

Figure 3 Flow diagram of physical validation and CFD simulation

3.3 Flowchart

Flowcharts are visual representations of a process or methodology's step-by-step advancement just like what was shown at figure 8 They represent the sequence of activities, decision points, and the flow of information or resources using standardised symbols and arrows. A study by Jithin S Kuruvila mentioned about flowcharts are widely used in a variety of industries such as software development, business process management, project management, and quality control. (Kuruvila et al., 2017)

For this simulation project, a flow diagram of physical validation and CFD simulation is combined into one flowchart. It shows step by step on how the project started and what will the outcome shows after performing a test rig. The flowchart also stated a few methods on how to design the model, how to perform meshing validation, CFD post processing, testing, data collecting and grid sensitivity analysis.

3.4 Computational Modelling

3.4.1 3D Modelling

Based on both figures, it shows that the process of producing a three-dimensional virtual representations of things, locations, or characters using specialised software. There is a study mentioning on creating digital models that may be modified, animated, or displayed for a range of applications, including video games, movies, architectural visualisation, product design, and more. (Zhang et al., 2023) Artists, designers, and engineers can use 3D modelling to bring their ideas to life in a realistic and engaging manner. For figure 4.1, it is a modelling of a benchmark with fin while for figure 4.2 it has a fin inside it.

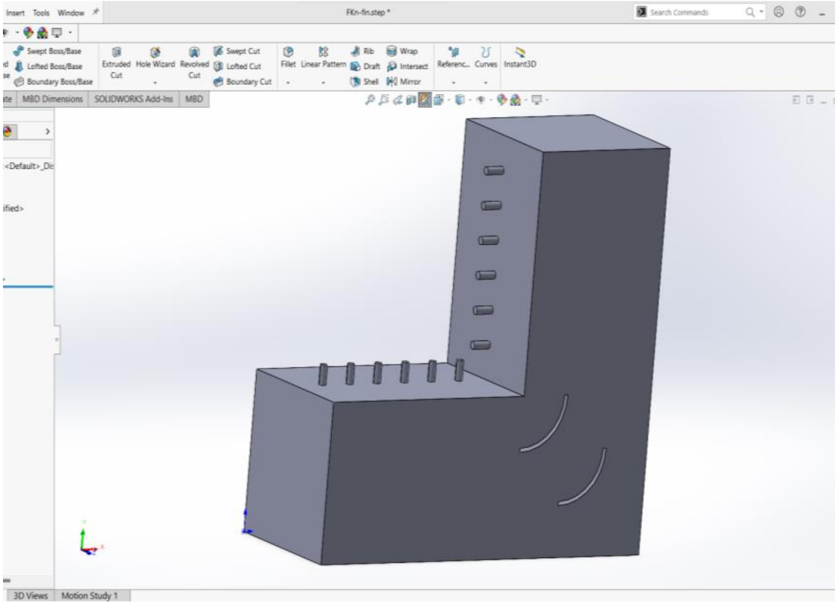

Figure 4.1 3D-Modelling with fin

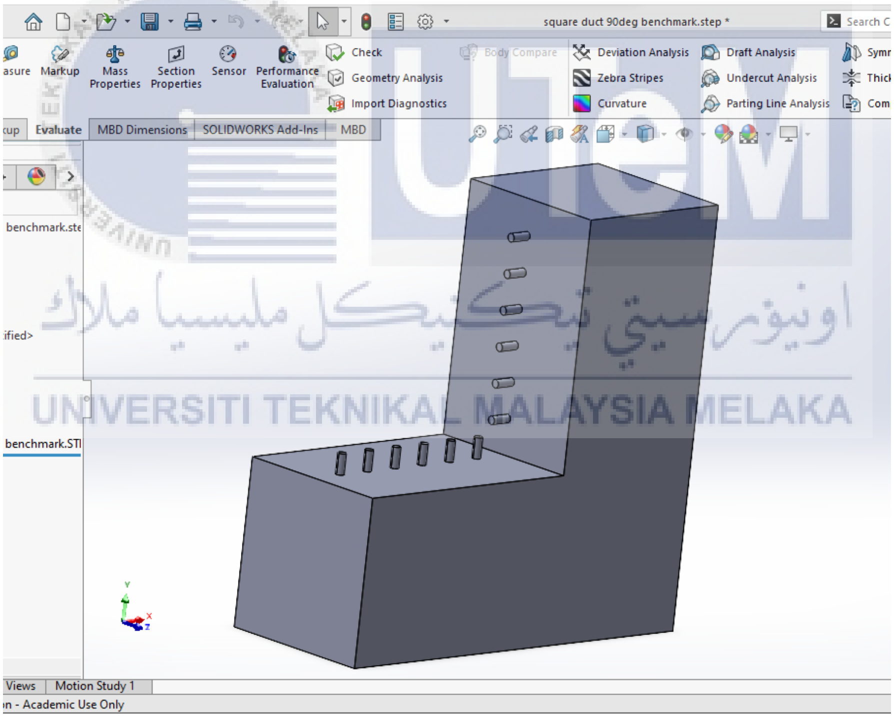

Figure 4.2 3D-Modelling without fin

The use of 3D modeling has revolutionized many industries by providing a powerful tool for visualizing and prototyping. The 3D modelling in figure 9 above is a ducting of 90°C with twelve pressure points, designed in Solidworks. To sketch the duct in 2D profile, sketch tools is important to define the shape and size of the duct. The use of the extrusion features will give the 2D sketch depth and create a 3D model of the duct. After adding the features and creating assembles finally, the 3D model of duct is created to communicate the design intent and provide manufacturing information. One of Roi Otero's study says, in architecture, 3D models help visualize and communicate designs to clients, while in the entertainment industry, they are used to create lifelike characters and stunning visual effects. Additionally, 3D modeling finds applications in medical and scientific fields, where it aids in anatomical study, simulations, and virtual reality experiences. (Otero et al., 2023)

3.4.2 Meshing

Meshing, in the context of computational modeling and simulation, refers to the process of dividing a complex geometry or domain into a collection of smaller, interconnected elements called meshes (Pei & Rim, 2021). Each mesh element is a representation of a section of the original geometry and serves as the foundation for numerical computations and simulations. Many fields, including finite element analysis (FEA), computational fluid dynamics (CFD), and computer-aided design (CAD), rely on meshing.

For this study, HyperMesh software is applied due to its main function which is to divide the computational domain into distinct cells or elements to simulate fluid flow. This abstract presents a high-level overview of meshing techniques such as structured and unstructured meshes, adaptive mesh refinement (AMR), and hybrid meshing approaches (Li et al., 2017). It explains how method meshing optimises the meshing process by using alternative meshing methods based on the flow region characteristics, resulting in accurate and efficient simulations.

The meshing process in this study started after designing the ducting in Solidworks. There were five benchmarks starting from 800k elements until 3 million elements. By running HyperMesh on each benchmark, it can show the comparison of several models performance and optimise the design to reach the desired results. This can show the result in increasing the efficiency, lower resource use, and better overall performance. Figure 5 below shows the example of meshing process of a 3 million elements.

Figure 5 Meshing process of 3 million elements

3.5 Boundary Condition in Ansys

After performing meshing process, all of the meshing geometry of benchmarks will be imported to another software called Ansys. Inside Ansys, it requires to find the value of pressure point for each of them. Both fin and without fin.. But before that, in Ansys there will be a boundary condition that is critical to follow, which is setting the parameters before running the process. They can be categorized into three main things; inlet velocity, outlet, types of model used (k-omega SST) and references value such as height, length and area.

Table 3 Boundary condition set up

Turbulence model	Area	Length	Inlet velocity
k-omega SST	$0.0016m^2$	0.1m	6.92m/s

3.5.1 Turbulence Model

Turbulence models are used in CFD to simulate the effects of turbulence on fluid flow. Turbulence is defined as unpredictability of fluid motion, which is characterised by whirling eddies and changes in velocity, pressure, and other flow parameters. Directly resolving all small-scale turbulent motions would be computationally expensive and frequently impractical, especially for engineering applications. Turbulence models, which approximate the effects of turbulence on mean flow parameters, give a realistic approach to simulating and understanding turbulent flows.

The RANS equations are a collection of averaged Navier-Stokes equations that are used to simulate turbulent flows. They are calculated by dividing the flow variables into mean and fluctuation components using Reynolds averaging. The RANS equations are useful for capturing mean flow behaviour because they provide a time-averaged picture of fluid flow.

Turbulence models that was used in this study is the SST k-omega model. It is also frequently employed in tandem with RANS equations. They give closures for the flow's unresolved turbulent fluctuations, which are critical in computing turbulent stresses and incorporating them into RANS equations. By solving the time-averaged equations alongside the turbulence model equations, it is possible to simulate the total flow behaviour, including the influence of turbulence. The first of the sets encompasses the equation of continuity and the RANS equations, which is shown in figure 6.

Continuity	$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0$
Momentum	$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$
Energy	$\frac{\partial}{\partial t}(\rho h_{tot}) + \frac{\partial}{\partial x_j}(\rho h_{tot} u_j) = \frac{\partial P}{\partial t} + \frac{\partial}{\partial x_j}(u_i \tau_{ij} + \lambda \frac{\partial T}{\partial x_j})$

where $\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} + \frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_j}\right)$ $\qquad h_{tot} = h + \frac{1}{2}u_i^2$

Figure 6 RANS Continuity Formula

The k-omega turbulence model is a popular two-equation model used to forecast and analyse turbulent flows in computational fluid dynamics (CFD) simulations. It is classified as a two-equation turbulence model since it solves two additional transport equations for two turbulence-related variables: turbulent kinetic energy (k) and particular turbulence dissipation rate (omega). The k-omega model seeks to provide a more accurate description of turbulence properties such as intensity, eddy viscosity, and boundary layer behaviour. Refer to figure 7 that shows the formula of SST k-omega.

$$F_4 = \frac{1}{1 + C_{RC} R_i}$$

$$R_i = \frac{W}{S}\left(\frac{W}{S} - 1\right)$$

$$S = \sqrt{2 S_{ij} S_{ij}} \qquad S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

$$W = \sqrt{2 W_{ij} W_{ij}} \qquad W_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right)$$

$$C_{RC} = 1.4$$

Figure 7 Specific Dissipation Rate k-omega equation

3.6 CFD Calculation & Post Processing

After initialization process, another method called CFD calculation will be performed. On this process, the simulation will be running around four to six hours, depending on how long does it take for one benchmark to run simulation, For 800k elements, it took five hours while the last meshing element which is 3 million, took twenty hours or more. In CFD calculation method, after making sure the boundary condition is in good shape and initialized, before the file is saved, in Ansys, there will be a parameter that set up to 1000 of interation. Finally, we managed to save the file in Ansys.

CFD post-processing is a step in CFD simulations that involves analyzing and interpreting the results obtained from the simulation. It focuses on extracting valuable information from the computed data, visualizing the flow behavior, and drawing meaningful conclusions to gain insights into the studied phenomena. The outcome of CFD post-processing plays a significant role in understanding fluid flow, optimizing designs, and making informed engineering decisions.

According to an article (Berger & Cristie, 2015), the challenges caused by computational time and uneven grids on the data side, as well as visual clutter and occlusion while visualising in 3D, are highlighted. In this section, the main purpose of this method is to find pressure point for each benchmark and perform the procedure with the duct that has fin. Based on figure 8, it shows the data of the very first post processing progress for a benchmark without fin.

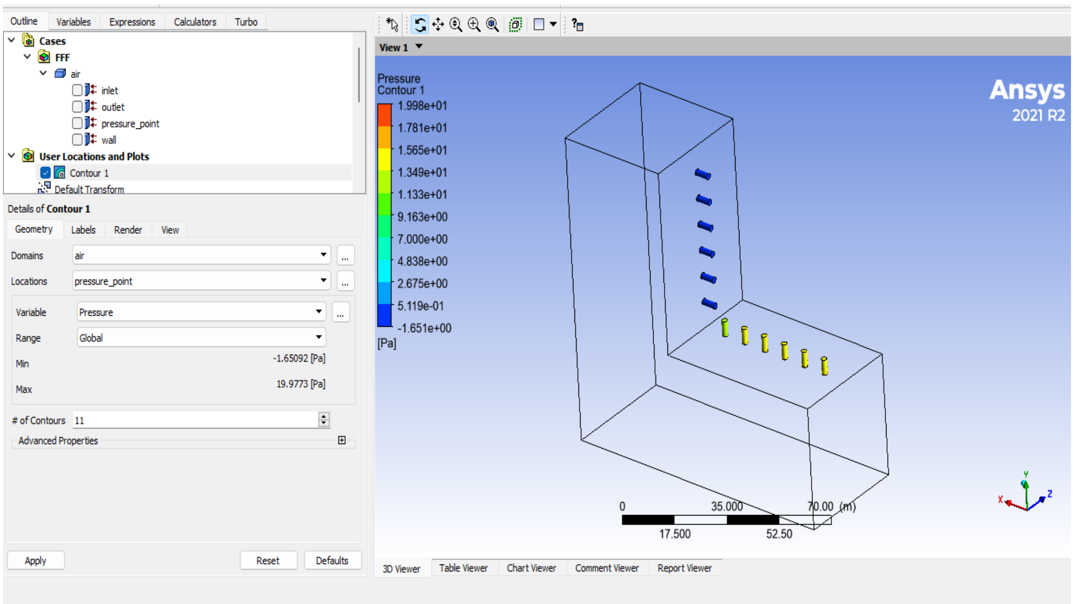

Figure 8 CFD post processing of benchmark without fin

The evaluation of performance measures and validation against experimental or analytical data is another key part of CFD post-processing. In this study, it is an important process where the aim of this method is to find the pressure point for all benchmark. After collecting data and sketching the line graph, it stated that the 3 million elements are the most accurate since it is the only line that is overlapping with other benchmarks. Hence, the 3 million element is choosen. From the benchmark of 3 million, a ducting with fin was designed with the same method and parameters. The only difference is the other duct will be added a fin inside it.

3.7 Grid Sensitivity Analysis

Grid sensitivity analysis is a technique used in computational fluid dynamics (CFD) to examine the sensitivity of numerical simulations to changes in the computational grid. It is also known as grid convergence study or grid refinement research. In CFD, the domain is discretized into a grid or mesh, and the resolution of this grid determines the accuracy of the simulation results. (Yang et al., 2001). In this sub topic, another CFD post processing will be shown as the complete benchmarks had gone through the Ansys simulation process.

3.7.1 CFD Post Processing

In this second CFD Post Processing, the main purpose of doing this practice was to focus on the main keyword of the study, which is pressure. Basically, the second process is about finding the contour of pressure by creating a plane inside the duct. Each of the plane that was placed will create a volume. There, we can find the contour of pressure and velocity. All the data will be placed in the table viewer to make it easier to plot the graph and show the comparison wall pressure between benchmark fin and without fin. Below are figure 9 and figure 10 where it shows the result of pressure and velocity contour without fin.

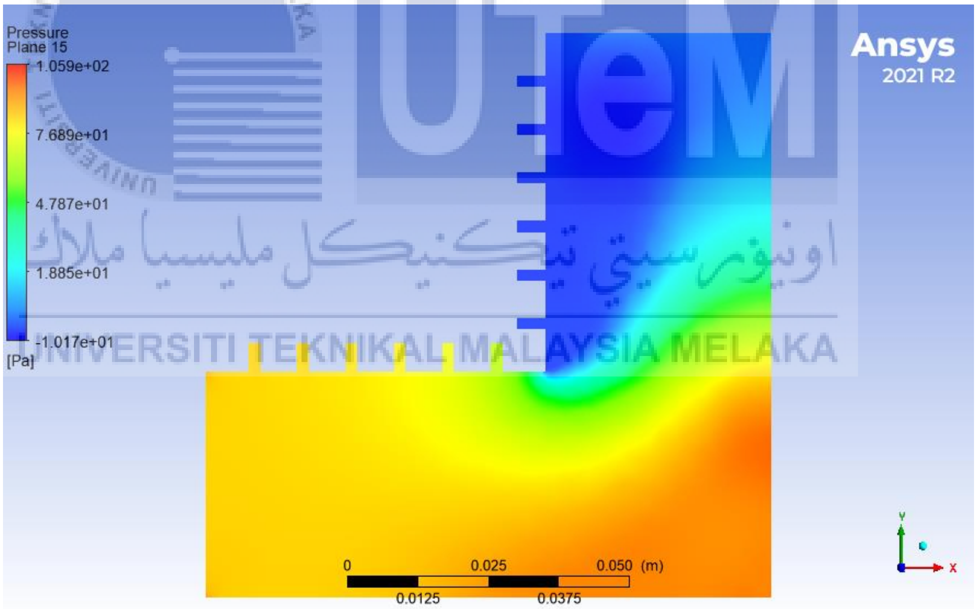

Figure 9 Contour Pressure without fin

Above appeared figure 9 that shows the result pressure of contour without fin. While on figure 10 is the result velocity of contour without fin.

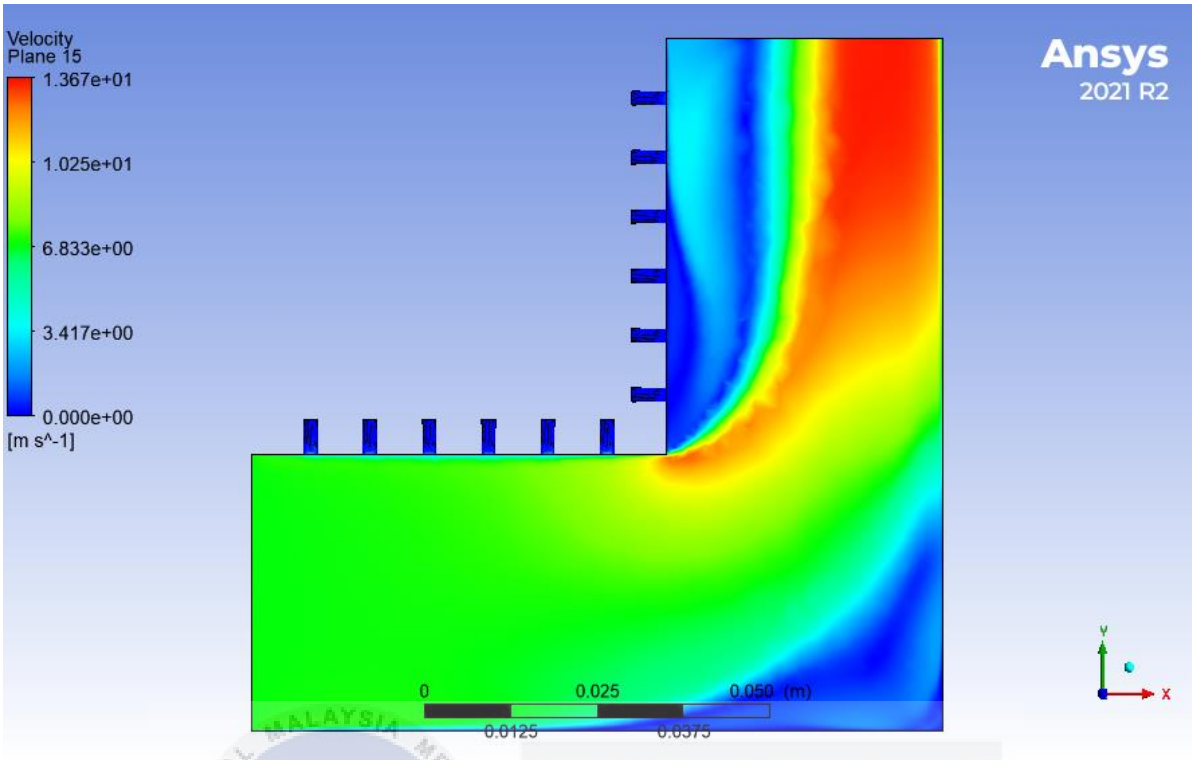

Figure 10 Contour Velocity without fin

Same goes to benchmark with fin, the value of both contour are different since pressure contour without fin has larger scale compare to the pressure contour with fin. Below are figure 11 and figure 12 that shows the result contour of pressure and velocity with fin.

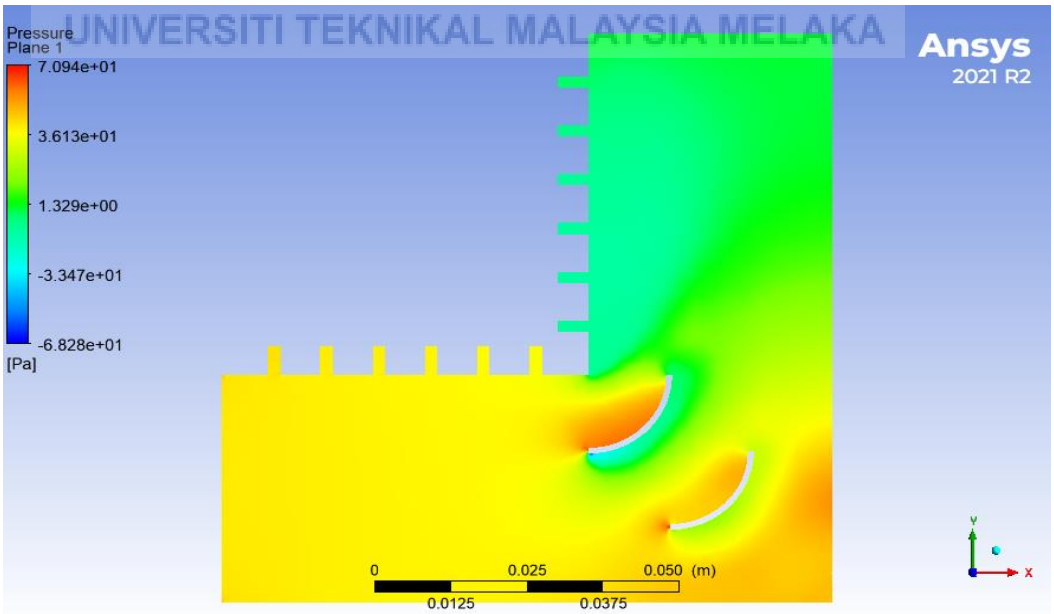

Figure 11 Contour Pressure with fin

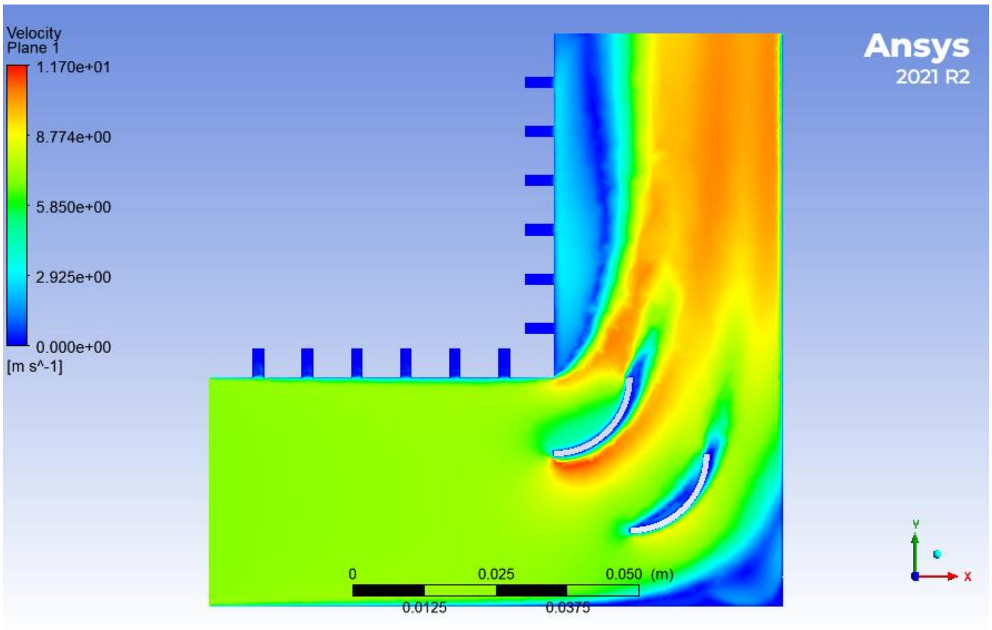

Figure 12 Contour Velocity with fin

3.8 Test Rig setup

The arrangement of tools, instruments, and components used to recreate and evaluate the operation of a certain system or device is referred to as a test rig setup. It is widely utilised in a variety of industries, including engineering, manufacturing, and research and development. Figure 13 shows the example scematic diagram of laboratory training equipment that was used to conduct the simulation.

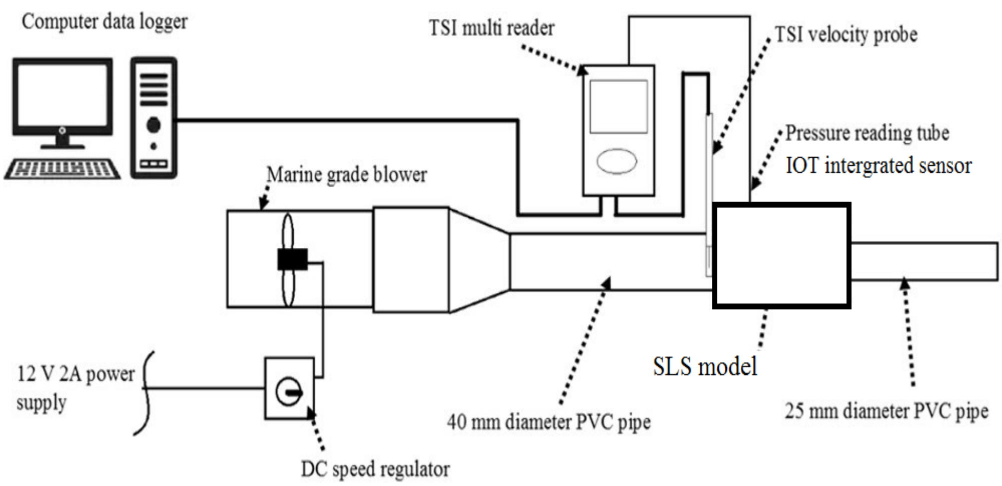

Figure 13 Scematic Diagram of Laboratory Training Equipment

For this section, the set up of physical validation consist of several equipment such as physical components, mounting structures, fixtures, and support systems. A TSI Velocity Meter was used to measure the pressure flow. Figure 14 shows the full image of the test rig set up. While in figure 15 shows the image of TSI Velocity Meter.

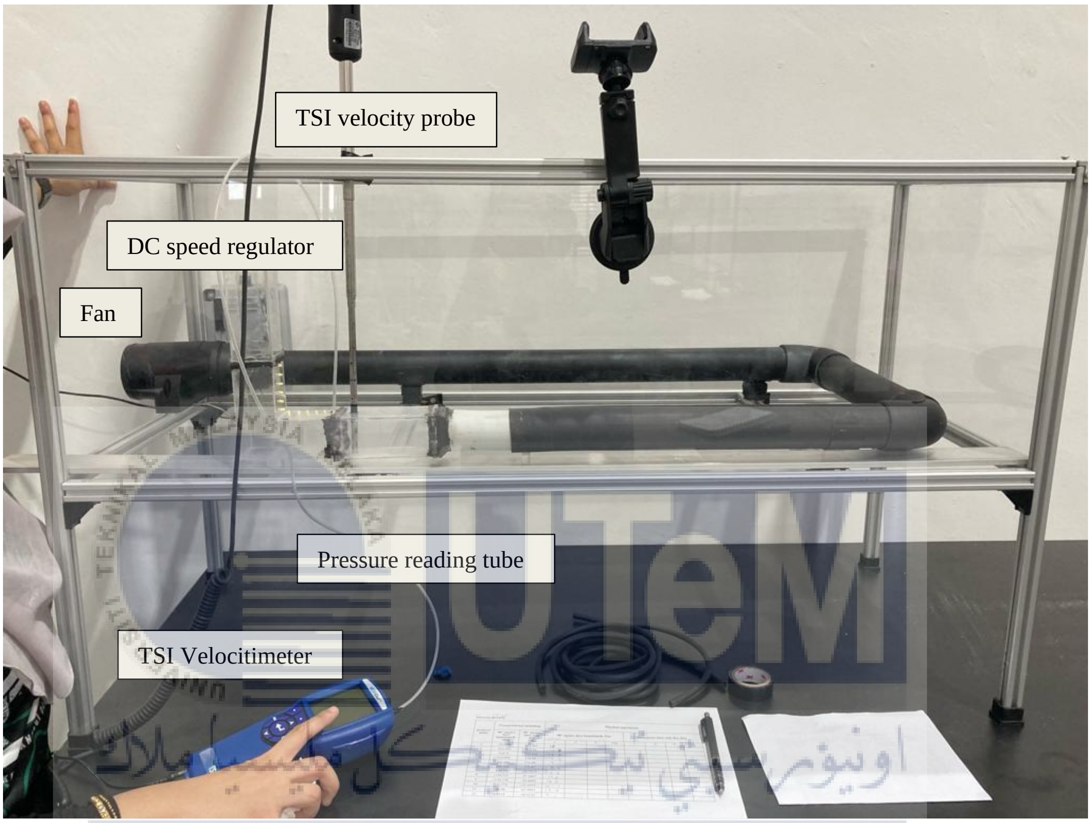

Figure 14 Test Rig set up

Figure 15 TSI Velocity Meter

Some of the connection that were used in this testing were motors, actuators, sensors, and control systems. Setting up the apparatus and devices in accordance with the appropriate test settings. To ensure precise and dependable data capture, a calibration process was conducted using the TSI velocity probe. After calibrate, proceed to validate the test rig configuration by running preliminary tests in each pressure point and comparing the results with the data from simulation.

By executing the tests on the test rig configuration, it also may subjecting the system to various operating circumstances, applying loads or forces, and monitoring various performance characteristics. After that, proceed with observation and and record pertinent measurements and data. Lastly, analyse the information to assess the system's effectiveness in relation to the stated goals.

3.9 Summary

In this section, there are some constraints to conduct the methodology after a research study and a portion of the experiment have been completed. Due to the testing to determine duct properties, the methodology's limitation is the time frame for experimentation. As a result, more samples are required to prepare for the experiment.

Furthermore, to resolve the problem presented in the problem statement, an engineer is required to approach the situation in a methodical manner. It is required to first identify the issue statement in order to begin the process of creating a 3D Modelling using the ANSYS software. In order to investigate and analyse the efficiency of fin devices in ducting using ANSYS, all the steps in the flowchart should be executed properly.

In order to cut down on the overall amount of time spent scoping, the parameter in question has been modified appropriately. Following that, the data are tallied in the paperwork, and a reference to the result analysis is also attached to the paperwork.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

This chapter will show the results and analysis based on the process step by step advancement in methodology. The data from the simulation and physical validation will be discussed in detail, while the results will be shown in a table and a form of a graph. Other than that, the discussion will show on how does it connect between benchmark and fin results that was created in the simulation. This will be a helpful guide of what to expect in the next phase to achieve the objectives of this research study.

4.2 Result of benchmark

Table 4 Result every benchmark

Pressure Point	700k element (Pa)	800k element (Pa)	1 million elements (Pa)	2 million elements (Pa)	3 million elements (Pa)
1	82.138	84.382	84.234	82.607	83.83
2	81.838	84.918	83.953	82.052	82.76
3	80.004	84.163	83.072	80.195	81.23
4	79.836	83.976	80.937	79.201	79.39
5	74.002	81.281	75.091	76.689	75.99
6	71.016	74.816	68.056	66.666	67.09
7	-3.4926	-4.3234	-4.8049	-5.0666	-5.08
8	-4.1019	-6.8468	-5.7926	-5.5904	-5.72
9	-4.2102	-7.8932	-6.5242	-5.8528	-5.92

10	-6.2186	-6.9232	-6.9508	-5.2515	-6.16
11	-7.9276	-6.7559	-5.1738	-6.1059	-7.02
12	-4.8552	-5.5222	-4.0938	-6.5977	-5.59

Table 4 above shows the result data of every benchmark that was collected on Ansys. For each pressure point has different value and based on the line graph below in figure 16, the graph that was created showed which of the benchmark is accurate. When running the simulation, each of the benchmark showed different images. Starting from 700k element until 2 million, the line graph does not show any stability. The duration of simulation for each element took six to seven hours. But starting at 3 million, the line graph showed positive image where it was started to become stable.

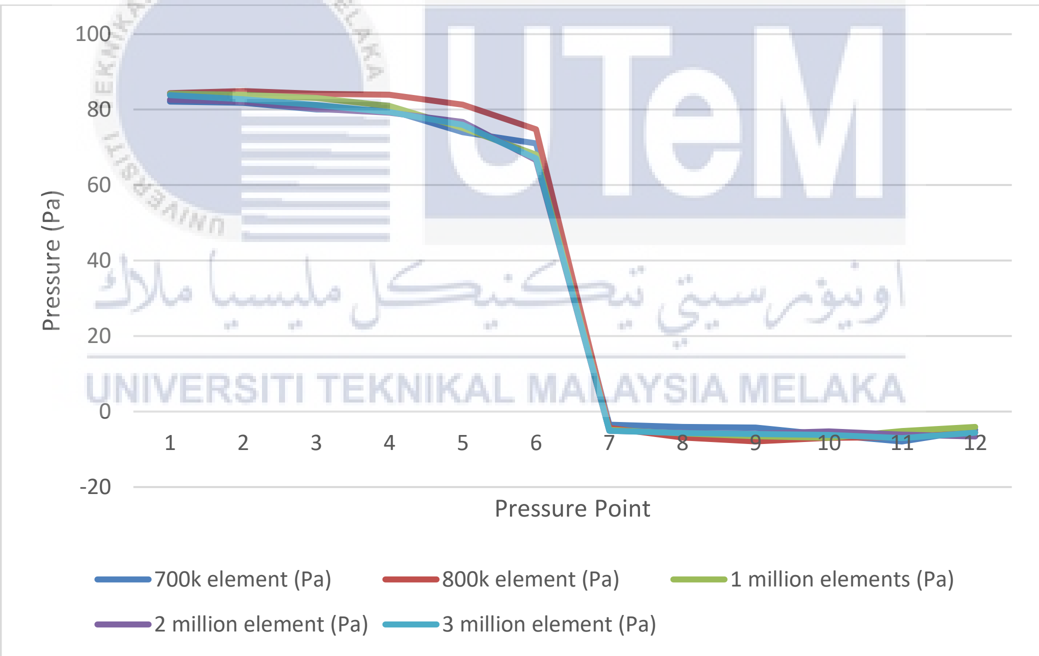

Figure 16 Line graph of every benchmark

4.3 Result Physical Validation

In physical validation, the experiment was conducted in a laboratory where the temperature and airflow was set up along other equipments. The aim for this procedure is to differentiate each of the pressure point between the benchmark of 3 million elements and the raw data that was collected. Based on the graph below, it shows how the benchmarks result data overlapped with the actual physical testing. It means that the airflow inside the physical module showed a turbulence is happening when the experiment is running.

The 3 million element was choosen to be the most accurate reading in the simulation because from all the benchmarks, the line graph started to stabilize when it reached 3 million. Pressure point plays an important role on this simulation because every point has different pressure value, eventhough the parameters had been set up. While doing the physical testing, the parameters such as airflow and temperature was set up in the TSI Velocitimeter. After calibrating and making sure the temperature stays around 24℃ until 26℃, then the data can be measured. Figure 17 shows the differences between all benchmarks and physical testing.

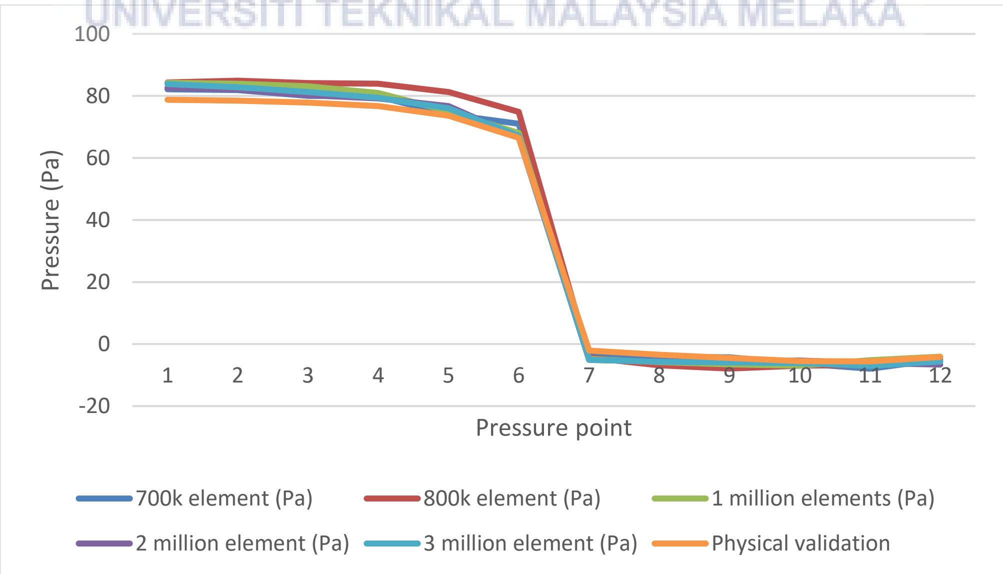

Figure 17 Line graph after added physical validation

4.4 Pressure Contour Results

The use of Ansys in the CFD post-processing is critical because the pressure distribution in the duct can be viewed and studied using the pressure contour. These contours can provide a pictorial representation of pressure change within the duct, allowing locations of high or low pressure to be detected. By examining the pressure contours, insights of airflow behaviour can be gained and can identified potential problems, such as pressure regions, which could suggest flow separation or recirculation. This information is useful for optimizing duct design and correcting any performance or efficiency issues associated with pressure distribution within the duct.

Table 5 Result pressure contour (plane)

	Pressure	
Plane	Benchmark	Fin
Plane 1	1.09E+06	4.88E+05
Plane 2	9.44E+05	4.41E+05
Plane 3	8.97E+05	4.15E+05
Plane 4	8.70E+05	3.99E+05
Plane 5	8.50E+05	3.83E+05
Plane 6	8.67E+05	4.04E+05
Plane 7	8.43E+05	4.17E+05
Plane 8	3.01E+05	4.40E+04
Plane 9	1.69E+05	-4.02E+03
Plane 10	8.59E+04	-3.74E+04
Plane 11	2.11E+04	-3.65E+04
Plane 12	-2.31E+04	-4.42E+04
Plane 13	-4.37E+04	-4.09E+04
Plane 14	-3.75E+04	-2.57E+04

Table 5 shows the result data between benchmark and fin. For each plane has different value and based on the line graph below in figure 18, the graph showed the line pattern is almost the same. The only thing that can distinguish between both is the value and the fin that was placed inside the ducting. This is due to the presence of pressure points in duct system that can disrupt the intended airflow patterns, leading to inefficiencies and potential issues such as uneven air distribution or noise generation.

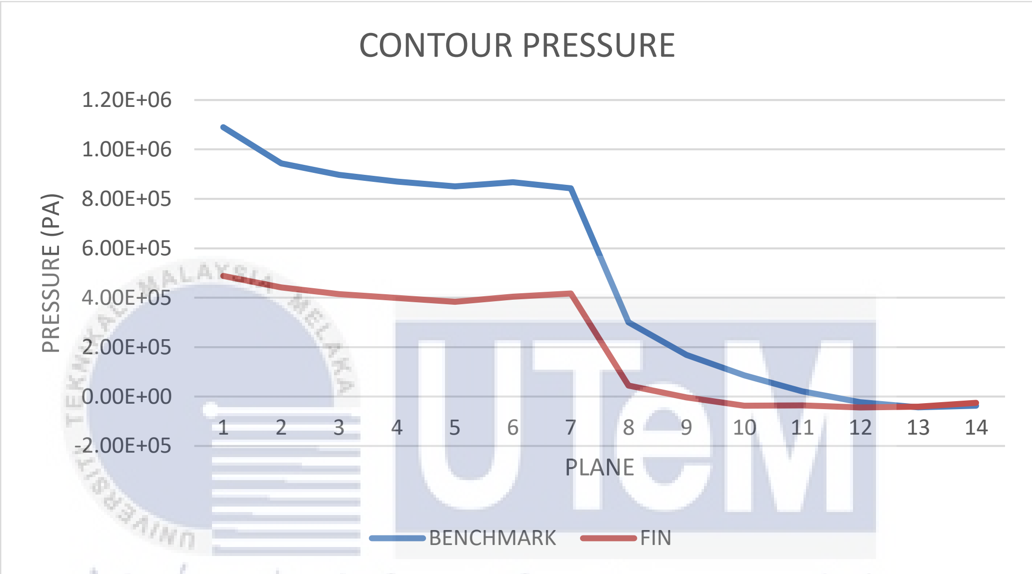

Figure 18 Line graph of contour pressure

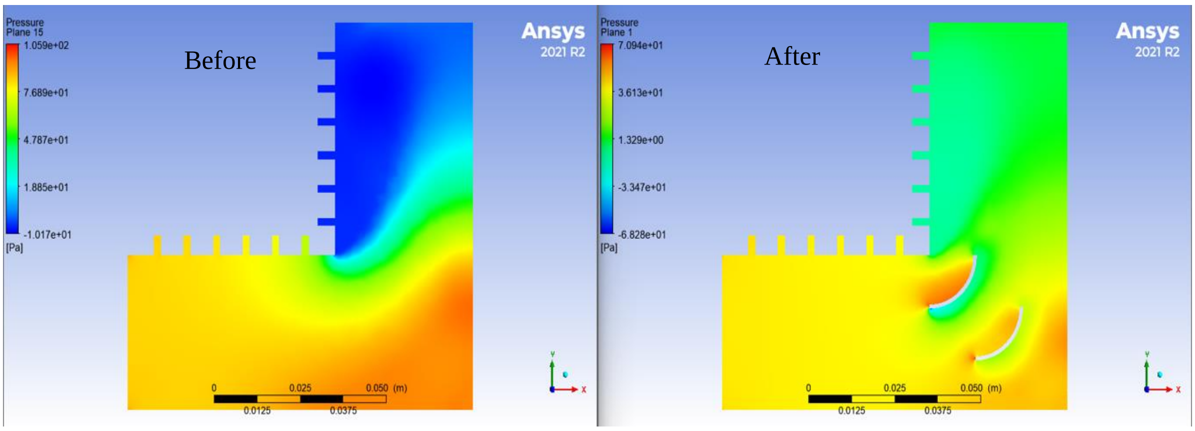

Figure 19 Pressure contours before fin and after fin

Figure 19 above shows the condition of pressure contours before fin and after fin. When the fin was attached to the duct, it demonstrated how it can reduce these impacts by optimizing airflow and reducing pressure losses. It also improved the overall efficiency of the duct system. A fin inside a duct can guide and control airflow, lessening the influence of pressure points and promoting more uniform air distribution, all of which can contribute to greater system efficiency and lower energy use.

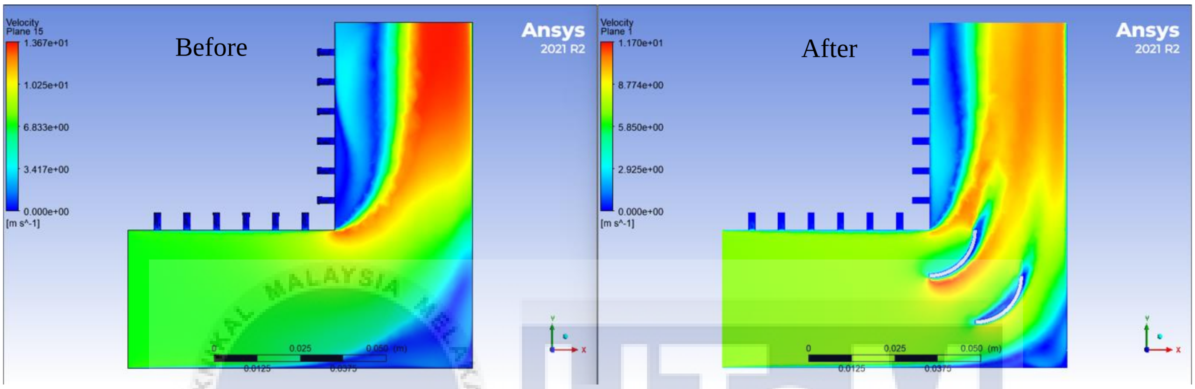

Figure 20 Velocity contours before fin and after fin

Figure 20 above shows the condition of velocity contours before fin and after fin. The purpose of finding velocity contours is to show how does velocity contours affect the efficiency of fin. The velocity distribution influences the convective heat transfer from the fin surface to the surrounding fluid. Higher velocities can enhance heat transfer, potentially improving fin efficiency. Areas of flow separation or recirculation, indicated by pressure and velocity contours, can negatively impact fin efficiency by reducing the effectiveness of heat transfer from the fin surface.

In summary, the flow regime contour of pressure and velocity can impact fin efficiency by influencing the heat transfer characteristics around the fin surface. High pressure and appropriate velocity distributions can potentially enhance heat transfer and improve fin efficiency, while flow separation and recirculation can have a negative impact.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

In methodology, a process had mentioned about designing a benchmark without fin and with fin in 3-D modelling using Solidworks software. After performing a CFD simulation through Ansys, a physical validation was conducted. From there, a comparison results between the simulation of benchmark and physical testing was created. This proved to be that the researcher managed to achieved the first objective by developing a scale model of square ducting as validation for CFD.

Aside from that, the researcher was able to analyze the flow region of square ducting using CFD. Previously, CFD was used to investigate the impact of grid density, turbulence models, and interpolation approaches on pressure loss calculations in duct fittings. The study's goal is to present knowledge on improving ducting and fin devices in order to reduce pressure losses, increasing the energy efficiency and to improve overall HVAC system performance.

Lastly, researcher was able to evaluate the relationship between fin ducting and wall pressure where the study focuses on effects of fin devices on airflow distribution, noise reduction and heat transfer efficiency in HVAC system and by using CFD simulations, it also helped to analyze airflow behaviour and pressure distribution within duct system.

5.2 Recommendation

There are a few recommendations which are suggested throughout this research study for future improvements. This will help more in terms of simulation and validation. Among the ideas that can be recommended are :

i. Integrating CFD simulations with the purpose of analyzing the airflow and pressure distribution inside the duct system while accounting for impacts of fin devices, ducting angle, and wall pressure. This will assist in understanding the intricate interplay between these aspects and refining the design for increased efficiency and performance.

ii. Experimenting with CFD simulations to ensure their accuracy and dependability. This will aid in confirming the numerical results and ensuring that the proposed design modifications are effective in increasing the efficiency of fin devices in ducting systems.

iii. Use optimization techniques, such as genetic algorithms or particle swarm optimization, to determine the best design parameters for fin devices and ducting angles. This will assist in determining the ideal combination of settings that reduce pressure losses and enhance the efficiency of the HVAC system.

REFERENCES

A. Bhatia, I. (2012). *HVAC Ducting - Principles and Fundamentals*. *246*, 53. www.PDHonline.org

Altwieb, M., Kubiak, K. J., Aliyu, A. M., & Mishra, R. (2020). A new three-dimensional CFD model for efficiency optimisation of fluid-to-air multi-fin heat exchanger. *Thermal Science and Engineering Progress*, *19*(July), 100658. https://doi.org/10.1016/j.tsep.2020.100658

Andservices.com. (2022). *Types of Ductwork*. https://www.andservices.com/blog/types-of-ductwork/

Appelbaum, J., Ohno, D., Rist, U., & Wenzel, C. (2021). DNS of a turbulent boundary layer using inflow conditions derived from 4D - PTV data. *Experiments in Fluids*, *62*(9), 1–21. https://doi.org/10.1007/s00348-021-03287-4

Balantrapu, N. A., Alexander, W. N., & Devenport, W. (2023). *Wall-pressure fluctuations in an axisymmetric boundary layer under strong adverse pressure gradient*. https://doi.org/10.1017/jfm.2023.225

Banu, P. S. A., Lohith, D. N. S. R., Kalyan, M. P., Vempati, D. S., & Sai, B. H. (2022). Materials Today : Proceedings Simulation of fin and tube heat exchanger and validation with CFD analysis. *Materials Today: Proceedings*, *66*, 1471–1476. https://doi.org/10.1016/j.matpr.2022.05.552

Berger, M., & Cristie, V. (2015). CFD post-processing in Unity3D. *Procedia Computer Science*, *51*, 2913–2922. https://doi.org/10.1016/j.procs.2015.05.476

Boonloi, A., & Jedsadaratanachai, W. (2022). CFD analysis on heat transfer characteristics and fluid flow structure in a square duct with modified wavy baffles. *Case Studies in Thermal Engineering*, *29*(December 2021), 101660. https://doi.org/10.1016/j.csite.2021.101660

Cao, G., Su, H., Xu, J., & Xu, K. (2019). Implicit high-order gas kinetic scheme for turbulence simulation. *Aerospace Science and Technology*, *92*, 958–971. https://doi.org/10.1016/j.ast.2019.07.020

Casartelli, E., Mangani, L., Launchbury, D. R., & Del Rio, A. (2022). Application of Advanced RANS Turbulence Models for the Prediction of Turbomachinery Flows. *JOURNAL OF TURBOMACHINERY-TRANSACTIONS OF THE ASME*, *144*(1). https://doi.org/10.1115/1.4051938

Coccia, M. (2018). The Fishbone Diagram to Identify, Systematize and Analyze the Sources of General Purpose Technologies. *Journal of Social and Administrative Sciences*, *4*(4), 291–303. https://ssrn.com/abstract=3100011Electroniccopyavailableat:https://ssrn.com/abstract=3100011Electroniccopyavailableat:https://ssrn.com/abstract=3100011

Du, J., Hew, K. F., & Liu, L. (2023). What can online traces tell us about students' self-regulated learning? A systematic review of online trace data analysis. *Computers and Education*, *201*(October 2022), 104828. https://doi.org/10.1016/j.compedu.2023.104828

Edition, T. T., & Manual, A. (2013). *Residential Duct Systems Third Edition , Version 2 . 00* (Issue January 2014).

Eggers, T., Kim, H. R., Bittner, S., Friedrichs, J., & Seume, J. R. (2020). *Aerodynamic and Aeroelastic Effects of Design-Based Geometry Variations on a Low-Pressure Compressor*. https://doi.org/10.3390/ijtpp5040026

Elahi, S. S., Lange, E. A., & Lynch, S. P. (2019a). International Journal of Heat and Mass Transfer Effect of Reynolds number on turbulent junction flow fluid dynamics and heat transfer. *International Journal of Heat and Mass Transfer*, *142*, 118328. https://doi.org/10.1016/j.ijheatmasstransfer.2019.06.084

Elahi, S. S., Lange, E. A., & Lynch, S. P. (2019b). Effect of Reynolds number on turbulent junction flow fluid dynamics and heat transfer. *INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER*, *142*. https://doi.org/10.1016/j.ijheatmasstransfer.2019.06.084

Elmore, M., Fernandez, E., & Kapat, J. (2019). International Journal of Heat and Mass Transfer Analysis of heat transfer on turbulence-generating ribs using dynamic mode decomposition. *International Journal of Heat and Mass Transfer*, *xxxx*, 118961. https://doi.org/10.1016/j.ijheatmasstransfer.2019.118961

Fang, G., Zhao, L., Chen, X., Cao, J., Cao, S., & Ge, Y. (2020). Normal and typhoon wind loadings on a large cooling tower: A comparative study. *JOURNAL OF FLUIDS AND STRUCTURES*, *95*(0). https://doi.org/10.1016/j.jfluidstructs.2020.102938

Fukami, K., Nabae, Y., Kawai, K., & Fukagata, K. (2019). *PHYSICAL REVIEW FLUIDS 4 , 064603 (2019) Synthetic turbulent inflow generator using machine learning*. *064603*(June), 1–18. https://doi.org/10.1103/PhysRevFluids.4.064603

Grimshaw, J. M., Hróbjartsson, A., Lalu, M. M., Li, T., Loder, E. W., Mayo-Wilson, E., McGuinness, L., McDonald, S., Stewart, L. A., Thomas, J., Tricco, A. C., Welch, V. A., Whiting, P., Moher, D., Glanville, J., Chou, R., Brennan, S. E., Boutron, I., Akl, E., … Tetzlaff, J. M. (2021). Pravila PRISMA 2020. *Medicina Fluminensis*, *57*(4), 444–465. https://doi.org/10.21860/medflum2021_264903

Guo, T., Lu, D., Lu, Z., Zhou, D., & Lyu, B. (2020). CFD / CSD-based flutter prediction method for experimental models in a transonic wind tunnel with porous wall. *Chinese Journal of Aeronautics*, *June*. https://doi.org/10.1016/j.cja.2020.05.014

Higgins, J. P. T., Green, S., & Ben Van Den, A. (2020). Cochrane Handbook for Systematic Reviews of Interventions. In *International Coaching Psychology Review* (Vol. 15, Issue 2). https://doi.org/10.53841/bpsicpr.2020.15.2.123

Hu, X., Hayat, I., & Park, G. I. (2023). *Wall-modelled large-eddy simulation of three-dimensional turbulent boundary layer in a bent square duct*. 1–35. https://doi.org/10.1017/jfm.2023.143

Jain, S. K., Das Agrawal, G., Misra, R., Verma, P., Rathore, S., & Jamuwa, D. K. (2019). Performance Investigation of a Triangular Solar Air Heater Duct Having Broken Inclined Roughness Using Computational Fluid Dynamics. *JOURNAL OF SOLAR ENERGY ENGINEERING-TRANSACTIONS OF THE ASME*, *141*(6). https://doi.org/10.1115/1.4043751

Jerome, A., Helander, H., Ljunggren, M., & Janssen, M. (2022). Mapping and testing circular economy product-level indicators: A critical review. *Resources, Conservation and Recycling*, *178*(June 2021), 106080. https://doi.org/10.1016/j.resconrec.2021.106080

Kadivar, M., Tormey, D., & Mcgranaghan, G. (2021). International Journal of Thermo fl uids A review on turbulent fl ow over rough surfaces : Fundamentals and theories. *International Journal of Thermofluids*, *10*, 100077. https://doi.org/10.1016/j.ijft.2021.100077

Kamioka, H. (2019). Preferred reporting items for systematic review and meta-analysis protocols (prisma-p) 2015 statement. *Japanese Pharmacology and Therapeutics*, *47*(8), 1177–1185.

Kuruvila, J. S., Lal, M., Roy, R., Baby, T., Jamal, S., & Sherly, K. K. (2017). Flowchart Plagiarism Detection System: An Image Processing Approach. *Procedia Computer Science*, *115*, 533–540. https://doi.org/10.1016/j.procs.2017.09.111

Li, H., Rong, L., & Zhang, G. (2017). Reliability of turbulence models and mesh types for

CFD simulations of a mechanically ventilated pig house containing animals. *Biosystems Engineering*, *161*, 37–52. https://doi.org/10.1016/j.biosystemseng.2017.06.012

Lin, Y., & Li, X. (2020). *The Investigation of a Sliding Mesh Model for Hydrodynamic Analysis of a SUBOFF Model in Turbulent Flow Fields*.

Lintermann, A. (2021). *Computational Meshing for CFD Simulations*. https://doi.org/10.1007/978-981-15-6716-2_6

Malanichev, I., & Akhmadiev, F. (2020). Pressure loss reduction in ventilation ducts by shape optimization of the removable profiled components. *IOP Conference Series: Materials Science and Engineering*, *890*(1), 012154. https://doi.org/10.1088/1757-899X/890/1/012154

McGill AirFlow Corporation. (2003). *Duct System Design Guide*. 321.

Mi, B., & Zhan, H. A. O. (2019). *Numerical Simulation on Rigid Foreign Object Exclusion in the Turboprop Engine Intake System With a Bypass Duct*. *7*.

Misra, R., Singh, J., Jain, S. K., Faujdar, S., Agrawal, M., Mishra, A., & Goyal, P. K. (2020). Prediction of behavior of triangular solar air heater duct using V-down rib with multiple gaps and turbulence promoters as artificial roughness: A CFD analysis. *International Journal of Heat and Mass Transfer*, *162*, 120376. https://doi.org/10.1016/j.ijheatmasstransfer.2020.120376

Moher, D., Liberati, A., Tetzlaff, J., & Altman, D. G. (2009). Academia and Clinic Annals of Internal Medicine Preferred Reporting Items for Systematic Reviews and Meta-Analyses : *Annals of Internal Medicine*, *151*(4), 264–269.

NAIMA. (2003). A guide to insulated HVAC duct systems. *Publication No. AH121 3/04*, 1–45.

Nering, K. (2021). *Validation of Modified Algebraic Model during Transitional Flow in HVAC Duct*. 1–20.

Otero, R., Lagüela, S., Cabaleiro, M., Sousa, H. S., & Arias, P. (2023). Semi-automatic 3D frame modelling of wooden trusses using indoor point clouds. *Structures*, *47*(May 2022), 1743–1753. https://doi.org/10.1016/j.istruc.2022.11.122

Page, M. J., Shamseer, L., Altman, D. G., Tetzlaff, J., Sampson, M., Tricco, A. C., Catalá-López, F., Li, L., Reid, E. K., Sarkis-Onofre, R., & Moher, D. (2016). Epidemiology and Reporting Characteristics of Systematic Reviews of Biomedical Research: A Cross-Sectional Study. *PLOS Medicine*, *13*(5), 1–30.

https://doi.org/10.1371/journal.pmed.1002028
Pasha, A. A. L. I., Hameed, A. Z., Gani, A., Jameel, A., Reddy, V. M., & Juhany, K. A. (2021). Estimation of Chaotic Surface Pressure Characteristics of Ice Accreted Airfoils – A 0-1 Test Approach. *IEEE Access*, *9*, 114441–114456. https://doi.org/10.1109/ACCESS.2021.3103580
Pei, G., & Rim, D. (2021). Quality control of computational fluid dynamics (CFD) model of ozone reaction with human surface: Effects of mesh size and turbulence model. *Building and Environment*, *189*(December 2020), 107513. https://doi.org/10.1016/j.buildenv.2020.107513
Qiu, Q., Du, X., Zhu, X., & Shen, S. (2017). Study on flow and heat transfer in a finned internal cooling duct. *Applied Thermal Engineering*, *113*, 58–69. https://doi.org/10.1016/j.applthermaleng.2016.10.149
Şevik, S., & Abuşka, M. (2020). Enhancing the thermal performance of a solar air heater by using single-pass semi-flexible foil ducts. *Applied Thermal Engineering*, *179*(February), 115746. https://doi.org/10.1016/j.applthermaleng.2020.115746
Stark, C., Shi, W., & Atlar, M. (2021). A numerical investigation into the influence of bio-inspired leading-edge tubercles on the hydrodynamic performance of a benchmark ducted propeller. *Ocean Engineering*, *237*(July), 109593. https://doi.org/10.1016/j.oceaneng.2021.109593
Subrot, S., Singh, C. B., & Fielke, J. (2020). CFD modelling of physical velocity and anisotropic resistance components in a peaked stored grain with aeration ducting systems. *Computers and Electronics in Agriculture*, *179*(June), 105820. https://doi.org/10.1016/j.compag.2020.105820
Szulc, O., Doerffer, P., Flaszynski, P., & Suresh, T. (2020). *Numerical modelling of shock wave-boundary layer interaction control by passive wall ventilation*. *200*. https://doi.org/10.1016/j.compfluid.2020.104435
Tong, L., Gao, J., Luo, Z., Wu, L., Zeng, L., Liu, G., & Wang, Y. (2019). *Accepted Manuscript*. https://doi.org/10.1016/j.buildenv.2018.12.007
Tošić, M., Larsson, R., Jovanović, J., Lohner, T., Björling, M., & Stahl, K. (2019). A computational fluid dynamics study on shearing mechanisms in thermal elastohydrodynamic line contacts. *Lubricants*, *7*(8). https://doi.org/10.3390/lubricants7080069
Tosic, M., Larsson, R., Joyanovic, J., Lohner, T., Bjorling, M., & Stahl, K. (2019). A

Computational Fluid Dynamics Study on Shearing Mechanisms in Thermal Elastohydrodynamic Line Contacts. *LUBRICANTS*, *7*(8). https://doi.org/10.3390/lubricants7080069

Vasudevan, S., Etemad, S., Davidson, L., & Montero, G. (2021). International Journal of Heat and Mass Transfer Numerical model to estimate subcooled flow boiling heat flux and to indicate vapor bubble interaction. *International Journal of Heat and Mass Transfer*, *170*, 121038. https://doi.org/10.1016/j.ijheatmasstransfer.2021.121038

Yang, J. B., Jeon, B. H., & Oh, S. I. (2001). Design sensitivity analysis and optimization of the hydroforming process. *Journal of Materials Processing Technology*, *113*(1–3), 666–672. https://doi.org/10.1016/S0924-0136(01)00670-7

Zhang, J., Liu, S., Gao, R. X., & Wang, L. (2023). Neural rendering-enabled 3D modeling for rapid digitization of in-service products. *CIRP Annals*, *00*, 4–7. https://doi.org/10.1016/j.cirp.2023.04.013

Zhi, X., Zhang, Z., Cui, J., Zhai, X., Chen, X., & Su, J. (2017). Quality of meta-analyses in major leading orthopedics journals: A systematic review. *Orthopaedics & Traumatology: Surgery & Research*, *103*(8), 1141–1146. https://doi.org/https://doi.org/10.1016/j.otsr.2017.08.009

APPENDICES

APPENDIX A GANTT CHART FINAL YEAR PROJECT 1

							Plan			Actual				
CONTENT	DURATION (WEEK)													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14
CHAPTER 1														
BACKGROUND														
PROBLEM STATEMENT														
RESEARCH OBJECTIVE														
SCOPE OF RESEARCH														
CHAPTER 2														
INTRODUCTION														
PRISMA SYSTEMATIC REVIEW														
HISTORY OF FINDING														
DUCTING DESIGN														
MESHING & TURBULENCE														
WALL PRESSURE														
SUMMARY														
CHAPTER 3														
INTRODUCTION														
RESEARCH DESIGN														
FLOWCHART														
COMPUTATIONAL MODELLING														
BOUNDARY CONDITION														
CFD CALCULATION & POST PROCESSING														
GRID SENSITIVITY ANALYSIS														
TEST RIG SETUP														
SUMMARY														
TEST RIG SETUP														
SUMMARY														

GANTT CHART FINAL YEAR PROJECT 2