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

MOIRA PATERUS

B092010156

BACHELOR OF MECHANICAL ENGINEERING TECHNOLOGY (REFRIGERATION AND AIR-CONDITIONING SYSTEMS) WITH HONOURS

2024

Faculty of Mechanical Technology and Engineering

TKE and Ducting Fin: A Study of the Joint Effect on the Efficiency of Fin Devices in Ducting

Moira Paterus

Bachelor of Mechanical Engineering Technology (Refrigeration and Air-conditioning Systems) with Honours

2024

TKE and Ducting Fin: A Study of the Joint Effect on the Efficiency of Fin Devices in Ducting

MOIRA PATERUS

A thesis submitted
in fulfilment of the requirements for the degree of
Bachelor of Mechanical Engineering Technology (Refrigeration and Air-Conditioning Systems with Honours

Faculty of Mechanical Technology and Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2024

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: TKE and Ducting Fin: A Study of the Joint Effect on the Efficiency of Fin Devices in Ducting

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

Saya **MOIRA PATERUS**

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:

Kampung Long Tuan, Trusan
98850 Lawas, Sarawak

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 "TKE 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	:	Moira Paterus
Date	:	6 February 20242024

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 Mechanical Engineering Technology (Refrigeration and Air-Conditioning Systems) with Honours.

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

Supervisor Name : Ts. Mohd Faruq Bin Abdul Latif

Date : 6 February 20242024

DEDICATION

To my parents an unending gratitude for your unwavering support. Throughout my academic journey, you were my pillars of strength, offering words of encouragement when self-doubt crept in and providing guidance when challenges seemed insurmountable. Your sacrifices, both seen and unseen, have shaped me into the person I am today.

ABSTRACT

The efficiency of fin devices within ducting systems is examined in this study in relation to the combined effects of turbulence kinetic energy (TKE) and ducting fin. This study has three main objectives in mind. To begin with, a scale model of square ducting is created as a platform for the validation of computational fluid dynamics (CFD) simulations. The airflow within the physical test rig is further tested to guarantee appropriate reproduction of real-world conditions. Third, to acquire insight into the intricate flow patterns and phenomena, the flow zone of square ducting is examined using CFD techniques. To understand how the angle of the duct affects the effectiveness of fin devices, the relationship between the two is lastly assessed. In order to accomplish these goals, a scale model of square ducting that closely resembles the geometry of actual ducting systems is built. This model is used as a standard to verify the precision and dependability of CFD simulations. The precision and validity of the simulation methods are confirmed by comparing the experimental data received from the physical test rig with the outcomes from CFD simulations of the grid sensitivity analysis.

Furthermore, CFD techniques are used to analyse the flow zone inside the square ducting. The flow patterns, velocity distribution, and pressure gradients within the ducting system are well understood thanks to the CFD simulations. This information assists in enhancing the functionality and design of fin devices used in ducting applications. The study also explores the connection between TKE and the fin device. The impact on TKE levels is systematically changed, allowing for analysis of the efficiency and effectiveness of fin devices in various ducting configurations.

The results of this study shows that by performing grid sensitivity analysis and validate it by physical validation. From the simulation, the different between the TKE levels in the ducting with no fin added and ducting with fin shows that reduction in TKE levels. It indicates a lower intensity of turbulent motion within the flow. Fin devices advance our knowledge of how TKE affect the functionality of fin devices in ducting systems. The CFD simulations provide us a clearer knowledge of the flow dynamics inside the square ducting. The performance and energy efficiency of ducting systems in numerous industrial and technical applications can be improved with the use of these insights.

ABSTRAK

Kecekapan peranti-fin dalam sistem saliran udara (ducting) diperiksa dalam kajian ini berkaitan dengan kesan bersama Tenaga Kinetik Turbulen (TKE) dan fin ducting. Kajian ini mempunyai tiga objektif utama. Pertama, model berskala ducting segi empat dicipta sebagai platform untuk pengesahan simulasi dinamik bendalir berkomputer (CFD). Aliran udara dalam rangka ujian fizikal diuji lebih lanjut untuk memastikan reproduksi yang sesuai dengan keadaan dunia nyata. Ketiga, untuk mendapatkan wawasan mengenai corak aliran dan fenomena yang rumit, zon aliran ducting segi empat diperiksa menggunakan teknik CFD. Bagi memahami bagaimana sudut ducting mempengaruhi keberkesanan peranti-fin, hubungan di antara keduanya diakhirnya dinilai. Bagi mencapai objektif ini, model berskala ducting segi empat yang hampir sama dengan geometri sistem ducting sebenar dibina. Model ini digunakan sebagai piawai untuk mengesahkan ketepatan dan kebolehpercayaan simulasi CFD. Ketepatan dan kebolehpercayaan kaedah simulasi disahkan dengan membandingkan data eksperimen yang diperoleh dari rangka ujian fizikal dengan hasil dari simulasi CFD analisis sensitiviti grid.

Selanjutnya, teknik CFD digunakan untuk menganalisis zon aliran dalam ducting segi empat. Corak aliran, taburan kelajuan, dan gradien tekanan dalam sistem ducting difahami dengan baik berkat simulasi CFD. Maklumat ini membantu meningkatkan fungsi dan reka bentuk peranti-fin yang digunakan dalam aplikasi ducting. Kajian juga meneroka hubungan antara TKE dan peranti-fin. Impak ke atas tahap TKE diubah secara sistematik, membolehkan analisis keberkesanan dan keberkesanan peranti-fin dalam konfigurasi ducting yang pelbagai.

Hasil kajian ini menunjukkan bahawa dengan menjalankan analisis sensitiviti grid dan mengesahkannya melalui pengesahan fizikal, perbezaan antara tahap TKE dalam ducting tanpa fin tambahan dan ducting dengan fin menunjukkan pengurangan dalam tahap TKE. Ini menunjukkan intensiti yang lebih rendah dalam gerakan turbulen dalam aliran. Peranti-fin memajukan pengetahuan kita tentang bagaimana TKE mempengaruhi fungsi peranti-fin dalam sistem ducting. Simulasi CFD memberikan pemahaman yang lebih jelas tentang dinamika aliran dalam ducting segi empat. Prestasi dan kecekapan tenaga dalam sistem ducting dalam pelbagai aplikasi industri dan teknikal dapat ditingkatkan dengan menggunakan pandangan ini.

ACKNOWLEDGEMENTS

First and foremost, I would like to thank and praise God, for everything I received since the beginning of my life and letting me experience Your guidance through all the difficulties. I would like to extend my appreciation to UTeM for providing me this platform to do my research. My utmost appreciation for my Supervisor Ts. Mohd Faruq Bin Abdul Latif for his continuous guidance and support throughout this journey. His constant patience for guiding and providing priceless insight that I receive. Finally, for my beloved family that has morally supported me and always praying for my success. My friend and classmate that has been on my side, I would like to extend my gratitude for your support.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS AND ABBREVIATIONS

CFD	-	Computational Fluid Dynamics
TKE	-	Turbulence Kinetic Energy
ω	-	Omega
HVAC	-	Heating Ventilation Air Conditioning
PRISMA	-	Preferred Reporting Items for Systematic Reviews and Meta-Analyses
RANS	-	Reynolds-Averaged Navier-Stokes
ε	-	epsilon
	-	

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Background

Various engineering applications, particularly in the fields of fluid dynamics and heat transport, depend heavily on turbulence and how it is managed. For the transfer of fluids, gases, or air, ducting systems are frequently used in the HVAC (Heating, Ventilation, and Air Conditioning), aerospace, and automotive sectors. For optimal system performance and energy usage, these ducts must be capable of efficiently transferring heat or mass.

Turbulence kinetic energy (TKE) is a critical quantity associated with the behaviour of fluid flow in ducts. TKE is the energy associated with the random fluctuation of velocity within the flow, which reflects the intensity of the turbulent motion. Understanding and managing TKE is critical for improving ducting system efficiency and performance.

There has been an increase in interest in employing fin devices to improve fluid flow qualities in ducts in recent years. Fins, which are typically in the form of expanded surfaces, are purposefully placed within ducts to disrupt flow and enable heat transfer through increased surface area. These fin devices have the ability to improve fluid mixing and promote convective heat transfer when properly made and positioned.

The existence of such geometrical elements complicates the flow patterns, turbulence characteristics, and heat transfer performance within the ducts. Investigating the combined influence of turbulence and fin devices in ducts can provide useful insights into optimising the efficiency and design of such systems.

As a result, the main objective of this thesis is to investigate the interaction of turbulence kinetic energy (TKE) and fin device efficiency in ducting. This study attempts to assess the effect of fin and flow conditions on TKE distribution within angle ducts using computational methodologies.

The findings of this work will add to the existing body of information on turbulence and fin device interaction, providing insights into the optimisation of ducting systems in terms of energy efficiency, heat transfer enhancement, and enhanced fluid dynamics. The findings of this study will have practical consequences for sectors that rely on ducting systems and may help drive the design and deployment of more efficient and sustainable thermal management solutions.

1.2 Problem Statement

Due to the high ambient temperatures and humidity, Malaysia's tropical environment presents special difficulties for HVAC (Heating, Ventilation, and Air Conditioning) systems. Maintaining comfortable indoor temperatures and reducing consumption of energy require effective heat transfer and optimal airflow dispersion. These systems frequently employ fin devices, such as heat exchangers and cooling coils, to aid in the transfer of heat from the air to a heat source or sink.

Understanding the relationship between TKE and ducting fin is essential for optimizing the design and performance of fin devices. The efficiency of fin devices is typically evaluated based on parameters such as heat transfer coefficient, pressure drop, and overall heat transfer effectiveness. However, the specific impact of TKE and ducting on these parameters and the resulting efficiency of the fin devices is not well-documented.

Therefore, the aim of this study is to investigate the impact of TKE and ducting fin on the efficiency of fin devices in ducting systems. The research will involve experimental and numerical simulations, focusing on analysing the convective heat transfer coefficients, pressure drop, and thermal performance of various fin designs under different TKE levels.

By addressing this problem, we can provide valuable insights into the optimal configuration of fin devices for specific TKE levels. The outcomes of this study will enable engineers and designers to enhance the efficiency of heat dissipation in ducting systems, leading to improved system performance, energy savings, and enhanced reliability.

1.3 Research Objective

The main aim of this research is to study the joint effect of TKE and angle duct on the efficiency of fin devices in ducting. Specifically, the objectives are as follows:

a. To develop scale model of square ducting as validation for CFD.

b. To investigate the flow region of square ducting using CFD.

c. To evaluate the relationship between fin ducting and TKE.

1.4 Scope of Research

The scope of this research are as follows:

a. This study solely focusing on square duct only.

b. The duct is angled to 90° angle.

c. Study conducted on a reduced or scaled-down version of a larger model.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The literature review is an essential part of any research study because it offers a thorough overview of the body of knowledge and body of research pertinent to the subject being studied. In this chapter, we examine the body of material that has already been written about the interaction between turbulent kinetic energy (TKE) and ducting, which affects the effectiveness of fin devices in ducting systems. We seek to pinpoint knowledge gaps, highlight the present state of research, and lay the groundwork for our study by reviewing earlier studies and research findings.

2.2 PRISMA

This section discusses the technique used to find articles about TKE And Ducting fin: A Study of The Joint Effect on The Efficiency of Fin Devices in Ducting. The reviewers employed the PRISMA methodology, which comprises data abstraction and analysis, inclusion and exclusion criteria, phases of the review process (identification, screening, and eligibility), and resources (Scopus and Web of Science) utilised to conduct the systematic review.

The PRISMA Statement, also known as Preferred Reporting Items for Systematic Reviews and Meta-Analyses, served as a guide for the review to follow. Within the realm of engineering, PRISMA is utilised quite frequently. Among the many guidelines for performing and reporting meta-analyses and systematic reviews, the PRISMA (Preferred Reporting Items for Systematic Reviews and Meta-Analyses) statement stands out as a top pick. Originally released in 2009, the statement has since been revised and reissued as the

PRISMA 2020 statement. The purpose of the PRISMA statement is to increase openness, precision, and completeness in systematic review and meta-analysis paper reports by providing authors with a detailed checklist to follow.

In their assessment of the PRISMA statement, (Moher et al., 2010) concluded that it had a positive impact on the quality of reporting for systematic reviews and meta-analyses. Studies reporting using the PRISMA statement were found to have much greater quality reporting than those not using the statement, according to the review's analysis of 178 studies. The PRISMA declaration was found to be an important resource for authors of systematic reviews and meta-analyses, and its use was strongly encouraged. The review also discovered that using the PRISMA statement and its expansions was related to improved reporting quality, as studies using the PRISMA statement were more likely to present key study features and bias assessments than studies that did not utilize the statement (Page & Moher, 2017).

2.2.1 Resources

The main journal databases used for the scoping review were Scopus and Web of Science, which is one of the study's strengths. Both databases offer thorough coverage of peer-reviewed literature across a wide range of disciplines and are widely used in academic research. A more thorough investigation of the impact and implementation of the PRISMA statement and its extensions through time is possible thanks to Web of Science's enormous back file and citation data. Furthermore, Clarivate Analytics' ranking system offers a helpful indicator of the significance and influence of the journals included in the database. On the other hand, Scopus provides a better selection of publications and a wider range of subject areas, making it a handy resource for finding relevant research across many different fields. For a scoping assessment that seeks to evaluate the influence of the PRISMA declaration and its extensions across various fields of research, the inclusion of varied topic areas. Overall, the scoping review's usage of Web of Science and Scopus ensures a thorough and thorough investigation of the adoption and impact of the PRISMA statement and its extensions across many fields. This method improves the validity and generalizability of the results and offers a solid framework for ongoing study in this field (Page & Moher, 2017).

2.2.2 Systematic review process

2.2.2.1 Identification

There are four stages involved in the systematics review process. The first process is Identification. Based on the title of the research, identifying keywords used for the search purposed. Using thesaurus and any similar keywords that are related to the TKE, Ducting and fin devices. In Table 2.1 is the developed keywords from the title of the study for Web of Science and Scopus.

Table 2-1 Search query for Web of Science and Scopus

Databases	Keyword
Web of Science	TITLE-ABS-KEY (Duct$ or pipe* or funnel* or well* or canal*) AND (Fin* or Flipper* or aerofoil* air foil* or appendage* or vane*) AND ("Turbulent kinetic energy") AND (CFD or "computational fluid dynamics" or modelling or simulation)
Scopus	TS= ((Duct$ or pipe* or funnel* or well* or canal*) AND (Fin* or Flipper* or aerofoil* air foil* or appendage* or vane*) AND ("Turbulent kinetic energy") AND (CFD or "computational fluid dynamics" or modelling or simulation))

2.2.2.2 Screening

The PRISMA statement's screening procedure is intended to ensure that every relevant research study is included in the systematic review or meta-analysis while reducing the possibility of bias and mistake. The systematic approach to screening offers a transparent and repeatable technique for choosing articles and guarantees the quality and rigour of the meta-analysis or systematic review (Liberati, 2009). The eligibility and exclusion factors in this section are important for establishing the parameters of the literature search and making sure that the review only includes relevant studies. For this study, literature type is only article journal excluded book, book series, chapters in book and conferences proceeding. To avoid misinterpretation and difficulties in translation, the search efforts eliminated non-English publications and focuses solely on items published in English. Then, the timeline for the related studies, are a period of 5 years (between 2019 and 2023) was selected as an ideal amount of time for analysing the evolution of research and related publications.

2.2.2.3 Eligibility

A crucial phase in the systematic review process is the PRISMA eligibility process, which ensures that only studies that meet the predefined inclusion criteria are selected for analysis (Moher et al., 2010).The eligibility requirements are established at the beginning of the review and are designed to ensure that the selected studies are relevant to the research topic and meet a set of quality standards.

During the initial screening phase, the search results are assessed for eligibility based on their titles and abstracts. This screening helps to identify studies that clearly do not meet the criteria for inclusion or violate the predefined exclusion criteria. Such studies are excluded from further consideration at this stage, reducing the number of irrelevant or unsuitable studies in the review. Following the initial screening, the remaining studies undergo a more comprehensive evaluation by reading their full-text versions. This step involves a thorough assessment of the selected studies to determine if they meet the specified inclusion criteria. During this process, researchers carefully examine the content of each study to ascertain its relevance to the research question and its alignment with the predefined eligibility requirements.

In addition to the initial screening and full-text assessment, researchers may employ manual searching and citation tracking techniques to identify additional studies that meet the inclusion criteria. Manual searching involves exploring relevant journals, conference proceedings, and grey literature to identify potentially relevant studies that may not have been captured in the initial search. Citation tracking, on the other hand, involves examining the reference lists of included studies to identify other relevant studies that could be included in the review.

By rigorously following the PRISMA eligibility process, researchers ensure that only studies that meet the predefined criteria are included in the systematic review. This approach helps to enhance the validity and reliability of the review findings by focusing on studies that are directly relevant to the research question and adhere to a set of quality requirements.

2.2.2.4 Included

Only high-quality papers are incorporated into the systematic review due to the rigorous process employed by PRISMA (Preferred Reporting Items for Systematic Reviews and Meta-Analyses) (Page & Moher, 2017). The PRISMA guidelines emphasize the importance of data extraction and analysis using standardized techniques, which minimize the potential for bias and enhance the transparency of the review process (Liberati, 2009). By following a consistent methodology, the systematic review ensures that the methodology and findings can be easily compared to other studies that have utilized the same inclusion criteria, enabling the replication of the study (Page & Moher, 2017).

The inclusion of studies in the systematic review is crucial to ensuring its comprehensiveness and accuracy in representing the available evidence (Higgins et al., 2019). During the inclusion process, the research question is addressed by extracting, synthesizing, and analysing the data from eligible studies (Liberati, 2009). This process involves a critical evaluation of the design, methodology, and findings of each study to determine its suitability for inclusion (Higgins et al., 2019). Only studies that meet the predefined eligibility criteria and demonstrate robustness in their design and methodology are included in the systematic review (Moher et al., 2010).

The systematic review's adherence to the inclusion process ensures that the outcomes are precise, trustworthy, and significant (Krappel et al., 2014). By including high-quality studies, the review minimizes the risk of biased results and enhances the overall reliability of the findings. The systematic and transparent approach to including studies in the review helps to maintain the integrity of the review process and ensures that the evidence presented is valid and representative of the research question (Liberati, 2009).

In summary, the inclusion of studies in a systematic review is a critical step that ensures the review's thoroughness, accuracy, and validity. PRISMA guidelines promote the incorporation of high-quality papers, and the use of standardized data extraction and analysis techniques helps reduce bias and enhance transparency. Through the inclusion process, the research question is addressed, and the data from eligible studies are critically evaluated. This rigorous approach ensures that the systematic review produces precise, trustworthy, and significant outcomes.

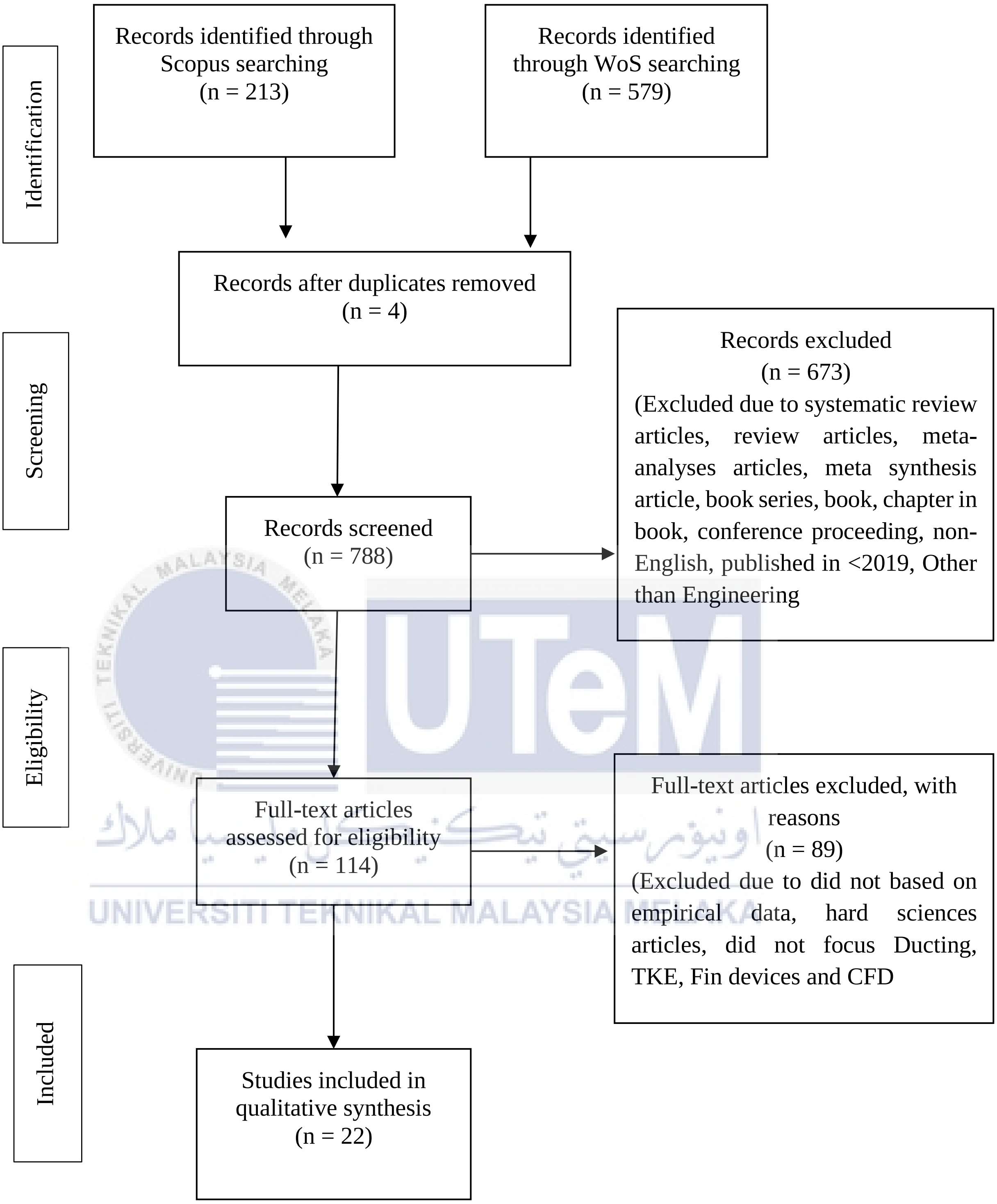


Figure 2-1 Flow of information through the different phase of systematic review

2.3 History of finding

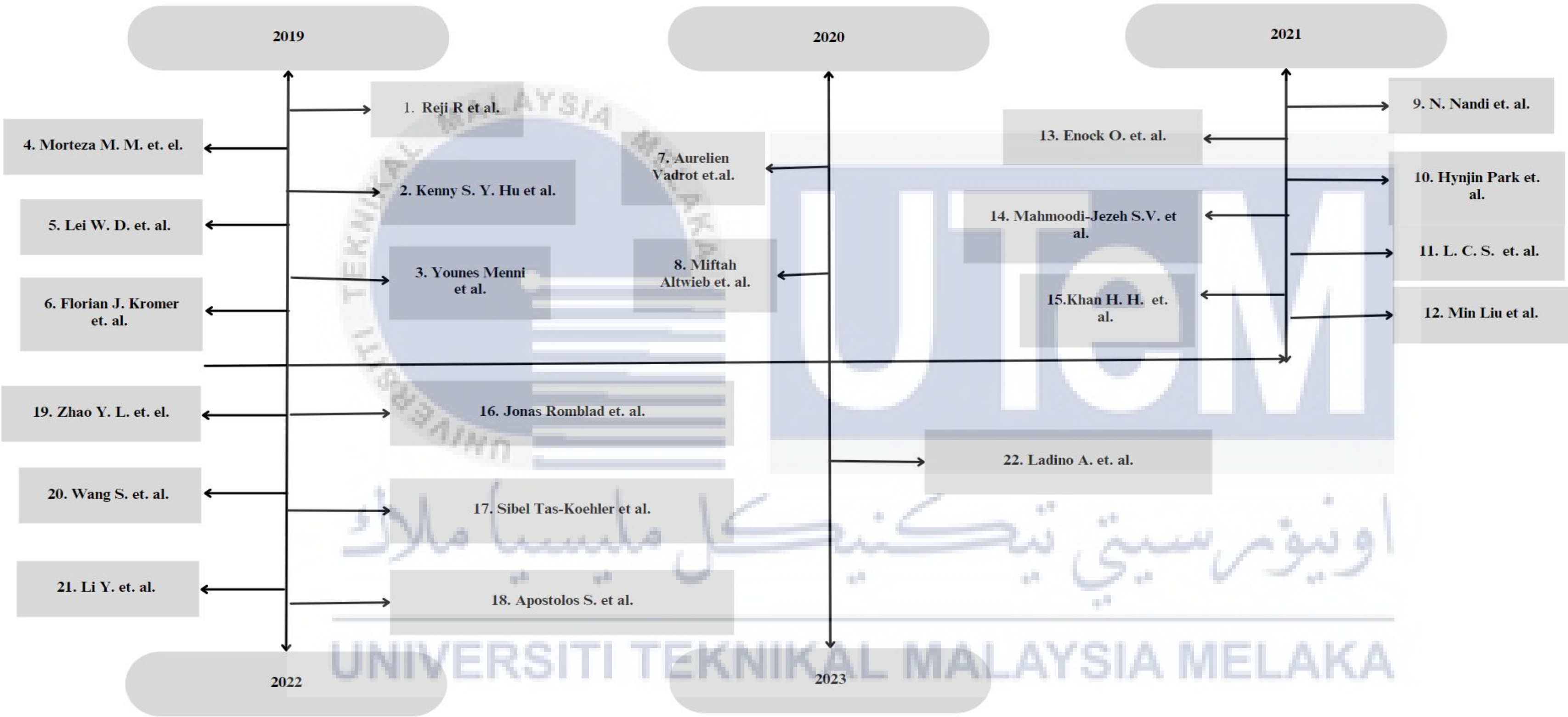


Figure 2-2 Name of authors and year their study is published.

2.3.1 Table of finding

Table 2-2 Table of finding

No	Author	Finding
2019		
1.	Reji Reghunathan Valsala, S. W. Son, Abhilash Suryan, Heuy Dong Kim	guide vanes for reducing energy consumption and improving the performance of fluid transportation systems and minimize pressure losses and enhance energy efficiency.
2.	Kenny S.Y. Hu, Xingkai Chi, Shih Tom I.P, Minking Chyu, Michael Crawford	pin fins in the duct affects the flow patterns and heat transfer performance.
3.	Younes Menni, Ahmed Azzi, Ali J Chamkha, Souad Harman,	staggered baffles significantly affect the flow patterns and heat transfer within the duct.
4.	Morteza Moosa, Mohamad Hamed Hekmat	multi-hole flowmeter design can reduce the upstream disturbance of the flow, leading to more accurate measurements of the flow rate.
5.	Lei Wei Du, Songtao Luo, Xinghong Zhang Wang	dimple/protrusion depth has a remarkable influence on the flow structure and heat transfer.
6.	Florian J Krömer, Stéphane Moreau, Stefan Becker	fan's design, specifically the skew angle, has a significant impact on both the sound emission and the flow patterns.
2020		
7.	Aurélien Vadrot, Alexis Giauque, Christophe Corre	compressibility effects significantly impact the turbulence, altering the vortical structures and their interactions.
8.	Miftah Altwieb, Krzysztof J. Kubiak, Aliyu M. Aliyu, Rakesh Mishra	indicate that the model can effectively identify the optimal design and operating conditions to maximize the heat transfers efficiency.
2021		
9.	N. Nandi, S. kumar Saha	presence of a guide vane has a significant effect on the turbulence characteristics within the pipe bend.

10.	Hyunjin Park, Christian K. Bach	resulting in improved temperature uniformity throughout the duct system.
11.	Chien Shing Lee, Tom I.Ping Shih	Higher heat loads result in changes in flow patterns appropriate balance between heat loads and flow conditions is essential
12.	Min Liu, Jun Yao, Yanlin Zhao	particle dispersion in sedimentary duct flows is influenced by the interplay between turbulent eddies.
13.	Enock Omweri Obwogi, Hai-long Shen a, Yu-min Su	rudder-bulb-fin device can effectively reduce flow resistance and improve energy efficiency.
14.	Mahmoodi-Jezeh, S. V. Bing Chen Wang	presence of transverse ribs significantly affects the flow field and heat transfer characteristics within the duct.
15.	Hamid Hassan Khan, Syed Fahad Anwer, Nadeem Hasan, Sanjeev Sanghi	Shorter duct lengths tend to favour an earlier transition to turbulence, while longer ducts provide more stability to the laminar flow.
	2022	
16.	Jonas Romblad Michael, Greiner Amandine Guissart, Werner Würz	The findings of the study reveal that different grid configurations produce varying levels of turbulence.
17.	Sibel Tas-Koehler, Martin Neumann-Kipping, Yixiang Liao, André Bieberle, Uwe Hampel	Constriction in the pipe leads to changes in the liquid velocity profiles, generating regions of higher turbulence intensity.
18.	Apostolos Spanelis, Alastair Duncan Walker	multi-objective factorial design approach leads to improved performance compared to traditional methods.
19.	Zhao Yanlin, Min Liu, Li Jinzhui, Yan Yudong, Yao Jun	charged particles can significantly alter the turbulence characteristics in pipe flow.
20.	Shengye Wang, Xiang Fu, Xiaogang Deng	higher-order numerical simulations with the inverse-ω scale variable exhibit improved accuracy and stability, particularly in regions with high flow gradients and shock waves.

21.	Li Yanjun, Li Yalin, Shouqi Yuan, Xikun Wang, Soon Keat Tan	the curvature of the duct has a significant influence on the flow structure and turbulence intensity
2023		
22.	Alexander Ladino, Carlos Alberto Duque-Daza, Santiago Lain	particle deposition efficiency in 90-degree square bend configurations is greatly impacted by large-scale roughness.

In table 2-2, in 2019 there are six related studies found and included in the table are their main finding. Based on their research on reduction of pressure losses in pipe bend using guide vanes they found out using guide vanes improves the performance of fluid transportation systems and that the optimal configuration of guide vanes varied depending on the specific bend geometry and flow condition (Reghunathan Valsala et al., 2019). The study done by (Hu et al., 2019) shows that the presence of pin fin in the duct significantly affects the flow patterns and heat transfer performance and The pin fins promote the generation of vortices, resulting in enhanced heat transfer from the duct walls. Additionally, the aspect ratio and Reynolds number influence the flow behaviour, with higher aspect ratios leading to increased pressure drop and heat transfer. The research done on staggered baffles on duct resulting in the staggered baffles significantly affect the flow patterns and heat transfer within the duct. This is caused by the baffles formation of vortices which enhance the mixing and improves heat transfer rates (Menni et al., 2019). The finding for research on flow structure and heat transfer characteristics in a 90-deg turned pin fined duct with different dimple/protrusion depths is that it has a remarkable influence on the flow structure and heat transfer (Du et al., 2019). Based on the experimental investigation by (Krömer et al., 2019), their findings of the study suggest that the fan's design, specifically the skew angle, has a significant impact on both the sound emission and the flow patterns. It is observed that increasing the skew angle leads to changes in the flow field, resulting in modifications in the sound spectrum and sound pressure level generated by the fan.

In 2020, two article are found. The finding of the article is the various aspect of turbulence The findings of the study shed light on various aspects of the turbulence characteristics. It reveals that the compressibility effects significantly impact the turbulence, altering the vortical structures and their interactions. The analysis also highlights the importance of density fluctuations in the mixing layer, which play a key role in turbulent mixing processes. Furthermore, the study provides insights into the evolution of kinetic energy, density variance, and turbulent transport phenomena within the compressible mixing layer. It presents detailed information about the velocity and density fluctuations, turbulent dissipation rates, and other turbulence-related quantities (Vadrot et al., 2020). The next study discusses the implementation and validation of the proposed CFD model using experimental data. It demonstrates the capability of the model to accurately predict the temperature distribution, pressure drop, and heat transfer coefficient within the multi-fin heat exchanger. This resulting indicate that the model can effectively identify the optimal design and operating conditions to maximize the heat transfers efficiency (Altwieb et al., 2020).

In 2021 there were seven articles found. The first article indicate that the presence of a guide vane has a significant effect on the turbulence characteristics within the pipe bend. The guide vane alters the flow patterns and modifies the turbulence intensity, particularly in the vicinity of the bend region. The vane-induced flow alterations lead to changes in the turbulence levels and affect the flow distribution within the bend (Nandi, 2021). The next was by (Park & Bach, 2021), The findings of the study indicate that the air mixing devices have a significant impact on the airflow patterns and temperature distribution within the square ducts. These devices effectively enhance air mixing, resulting in improved temperature uniformity throughout the duct system. However, the introduction of these devices also leads to a slight increase in pressure drop, which needs to be considered in system design. The finding in the study of the effects of heat loads on flow and heat transfer in the entrance region of a cooling duct with a staggered array of pin fins shows a significant influence on the flow. Higher heat loads result in changes in flow patterns, such as the formation of secondary flows and flow separation regions. These alterations affect the heat transfer performance, leading to variations in heat transfer rates and temperature distributions. Furthermore, the study identifies the optimal range of heat loads for achieving enhanced heat transfer performance in the entrance region. It demonstrates that an appropriate balance between heat loads and flow conditions is essential to maximize the heat transfer efficiency of the cooling duct with pin fins (Lee & Shih, 2021). The other article published in that year has their findings of the study reveal several key insights. It is observed that particle dispersion in sedimentary duct flows is influenced by the interplay between turbulent eddies and the sedimentation process. The particles tend to cluster and settle due to gravitational forces, leading to variations in particle concentration along the flow direction. Additionally, the study demonstrates that particle size plays a crucial role in determining the settling behaviour and dispersion characteristics. Smaller particles tend to disperse more readily and travel farther downstream, while larger particles settle closer to the sedimentary boundary (Liu et al., 2021). The finding of the thirteenth article in **table 2-2** are of the study demonstrate that the rudder-bulb-fin device can effectively reduce flow resistance and improve energy efficiency. The CFD simulations provide insights into the flow patterns, pressure distributions, and hydrodynamic forces acting on the device. The model tests validate the CFD results and confirm the energy-saving benefits of the device. Moreover, the article presents a comprehensive analysis of the optimal design parameters for the rudder-bulb-fin device, including the bulb shape, fin configuration, and positioning. By optimizing these parameters, the device's energy-saving performance can be further improved (Obwogi et al., 2021). the findings of the study (Mahmoodi-Jezeh & Wang, 2021) indicate that the presence of transverse ribs significantly affects the flow field and heat transfer characteristics within the duct. The ribs induce turbulence and promote heat transfer by enhancing the mixing of the fluid. The authors explain the mechanisms underlying the increased heat transmission that they have found and stress the significance of choosing the right rib arrangement to maximise heat transfer effectiveness. The last article indicates that the length of the duct influencing the transition behaviour. Shorter duct lengths tend to favour an earlier transition to turbulence, while longer ducts provide more stability to the laminar flow (Khan et al., 2021).

The study conducted by (Greiner & Würz, 2022) reveal that different grid configurations produce varying levels of turbulence. They observed that smaller mesh sizes and higher solidity ratios tend to generate higher turbulence levels. They also found that the blockage ratio, which represents the obstruction caused by the grid in the flow path, influences the turbulence intensity. According to (Tas-Koehler et al., 2022) the results of the study conducted provide insights into the flow patterns, pressure drops, and turbulence characteristics for both single-phase and two-phase flow regimes. They observed that the presence of a constriction in the pipe significantly affects the flow dynamics. It leads to changes in the liquid velocity profiles, generating regions of higher turbulence intensity. The experimental measurements and numerical simulations agree reasonably well, demonstrating the validity of the CFD models used in predicting the flow characteristics. (Spanelis & Walker, 2022) the results show that the multi-objective factorial design approach leads to improved performance compared to traditional methods, as it allows for a more comprehensive exploration of the design space and the consideration of multiple conflicting objectives. Next, the findings of the study reveal that the presence of charged particles can significantly alter the turbulence characteristics in pipe flow. The charged particles exert forces on the fluid, leading to modifications in the flow patterns and turbulence intensity. The researchers observe changes in the distribution of turbulent structures and fluctuations, indicating that the particles act as a modulation mechanism for the turbulence .(Zhao et al., 2022). In the same year, the article by (S. Wang et al., 2022) their results indicate that the higher-order numerical simulations with the inverse-ω scale variable exhibit improved accuracy and stability, particularly in regions with high flow gradients and shock waves. The simulations also achieve a higher resolution of turbulent structures and provide better predictions of important aerodynamic quantities. Another study was also conducted on turbulent flow in curved ducts with variable curvature convex wall (Li et al., 2022). The findings of the study provide insights into the complex nature of turbulent flow in curved ducts with variable curvature convex walls. The researchers observed that the curvature of the duct has a significant influence on the flow structure and turbulence intensity. As the radius of curvature decreases, the flow becomes more turbulent and exhibits stronger secondary flow patterns.

Lastly, according to the study's findings, particle deposition efficiency in 90-degree square bend configurations is greatly impacted by large-scale roughness. By creating separation and recirculation zones, the roughness modifies the flow patterns and affects particle behaviour (Ladino et al., 2023).

2.4 Duct Design

In general, a duct is a conduit or tube that is used to carry a variety of materials, including air, fluids, gases, or even cables. HVAC systems, ventilation systems, plumbing, electrical wiring, and other systems and applications all frequently use ducts. The previous studies included a various design of ducting.

2.4.1 Duct sizing

The process of calculating the proper dimensions or cross-sectional area of ductwork in an HVAC system to meet the necessary airflow rates is known as duct sizing.

To ensure efficient and effective airflow distribution throughout the building, proper duct sizing is essential. Based on variables such the airflow volume, velocity, and pressure drop, the ducts' size is chosen. The major objective is to maintain desired comfort levels while minimising energy usage and supplying enough airflow to meet the heating or cooling demands of each room or zone. Airflow restriction, an increase in pressure drops, and decreased system performance might result from inadequate duct size. Inadequate heating or cooling, pain, and energy waste may arise from this.

On the other hand, oversized ducts may result in excessive airflow rates, noise production, and greater beginning expenses (Besant & Asiedu, 2000). Calculations for duct sizing consider elements including the design of the duct system, the length of duct runs, the number of bends and fittings, and the kind of duct material. For estimating the optimum duct size based on the necessary airflow rates and other design variables, many guidelines and industry standards, such as the ACCA Manual D or ASHRAE standards, give detailed techniques and charts. Table 2.4.1 is duct input parameter in accordance to ASHRAE standards (Besant & Asiedu, 2000).

Table 2-3 Duct input parameter (Besant & Asiedu, 2000)

Duct section	Length (m)	Flow rate (M^3/s)	Type	Dimension (m)	Extra pressure Losses (Pa)	Dynamic Loss Coefficient†
1	14	0.7	Rectangular	0.252*	25	0.8
2	12	0.22	Round		37.5	0.65
3	8	0.92	Round	0.33**	0	0.18
4	16	0.5	Round		0	0.65
5	19.81	1.42	Round		27.5	1.5

*Presized height pf 253 mm **Presized 330 mm diameter duct †Fitting loss coefficient

2.4.2 Air distribution

The effective and efficient transportation of conditioned air throughout a structure or space is referred to as air distribution in duct design. It entails planning the duct system such that proper ventilation rates, balanced airflow, and even air dispersion is all guaranteed. For maintaining constant comfort levels, temperature management, and indoor air quality, proper air dispersion is essential. The architecture and dimensions of supply and return ducts, the number and location of supply outlets, and the choice of diffusers or grilles are all factors in duct design that affect air dispersion. Pressure losses need to be kept to a minimum, airflow patterns are to be optimised, and temperature stratification is to be avoided.

Efficient air distribution requires careful balancing of airflow rates to ensure uniform air delivery to different zones or rooms. Balancing dampers may be installed in the duct system to adjust airflow in different branches or to regulate individual room conditions. Air distribution design also considers factors such as noise control, air velocity, and directionality of supply air. Proper diffuser selection and placement help promote effective mixing of supply air with room air, preventing drafts and ensuring proper ventilation. Overall, air distribution in duct design focuses on achieving uniform air delivery, comfort, and indoor air quality while minimizing energy consumption and maintaining system efficiency. It is a critical aspect of HVAC system design to create a comfortable and healthy indoor environment.

2.4.3 Duct Material and Type

A form of ductwork known as rigid duct is made of rigid and unyielding materials, usually fibreglass duct board or sheet metal. It is referred to as "rigid" because it holds its shape and is difficult to flex or bend. In HVAC (heating, ventilation, and air conditioning) systems, rigid duct is frequently used to move air from the air handling unit to different areas within a structure.

Although a variety of materials can be used to create rigid ducting, galvanised steel and aluminium are the most popular options. For commercial and industrial uses, galvanised steel ducts are reliable, fire-resistant, and acceptable. Lightweight and resistant to corrosion, aluminium ducts are frequently utilised in residential and light commercial settings. The material of choice is commonly flat sheets that are used to construct rigid ducting. To build a continuous duct system, the sheets are cut, shaped, and attached to one another by screws, rivets, or sealants. It's important to adequately seal the joints to reduce air leakage. There are several different shapes for rigid ducts, including rectangles, squares, and circles. The design specifications and airflow calculations establish the precise size and shape of the duct. The duct's dimensions should be chosen to allow for the necessary airflow volume while minimising pressure drop

2.4.4 Pressure static

The pressure drop that happens while air or fluid passes through a duct system is referred to as a pressure drop in ducting design. When constructing and analysing HVAC (Heating, Ventilation, and Air Conditioning) systems or any other system involving fluid flow through ducts, it is an important element to consider.

2.4.4.1 Losses due to friction

The pressure drop brought on by the resistance that air or fluid travelling through a duct encounter is referred to as friction loss in ducting. Frictional forces between the moving fluid and the duct's inner surface are mostly to blame for this resistance. Friction loss must be considered when designing duct systems since high losses can limit airflow, increase energy use, and degrade system effectiveness. In order to reduce friction losses in ducting systems, proper dimensions, careful design, and material selection are essential.

As was already established, frictional losses happen as a result of the resistance the fluid faces as it moves over the inner surface of the duct. There is a pressure drop as a result of the frictional forces between the fluid and the duct walls. Several variables, including duct size, shape, length, roughness, and fluid velocity, affect this pressure drop.

2.5 Turbulence Model

Modelling in computational fluid dynamics (CFD) is the process of using mathematical equations to approximate or reflect the physical phenomena that occur in fluid flows. Different modelling approaches are employed to simplify and make the simulations more manageable because it is frequently computationally expensive or difficult to solve the entire set of governing equations for fluid flow.

In computational fluid dynamics (CFD), a turbulence model is used to simulate and model the effects of turbulence in fluid flow. Turbulence refers to the chaotic and irregular motion of fluid that occurs at high Reynolds numbers, resulting in fluctuations in velocity, pressure, and other flow properties.

Turbulence models are mathematical equations or closures that approximate the behaviour of turbulence based on various assumptions and simplifications. These models help predict the turbulent flow characteristics within a given fluid domain, considering factors such as turbulent eddies, energy dissipation, and the interaction between different scales of turbulence. There are different types of turbulence models, each with its own assumptions and complexity.

Reynolds-Averaged Navier-Stokes (RANS) Models are the most widely used turbulence models (Ghai et al., 2022) (Zhang et al., 2008). They assume that the flow properties can be decomposed into a time-averaged mean component and a fluctuating turbulent component. Examples of RANS models include the k-epsilon model, the k-omega model, and the Reynolds Stress Model (RSM). The study, in flow structure and heat transfer on the endwall surface, selected k−ω (SST) among others turbulent model in which it has better performance at low Re number to reveal Nusselt number and predicted friction factor more accurately (Du et al., 2019). The turbulent flow characteristics through 90° pipe bend are carried out using RANS equation using the k−ω (SST) turbulence model (Nandi, 2021). Similar study are conducted with on pipe bends using guide vanes (Reghunathan Valsala et al., 2019).

Large Eddy Simulation (LES) is a hybrid approach that captures the large-scale turbulent structures explicitly while modelling the small-scale turbulence (Liu et al., 2021). It resolves the large-scale eddies and models the effects of smaller-scale turbulent motions (Zhang et al., 2008). LES is computationally more expensive than RANS models but provides more accurate results for certain applications.

Direct Numerical Simulation (DNS) is a highly accurate approach that directly solves the Navier-Stokes equations without any turbulence modelling. It provides a detailed and accurate representation of turbulent flow but is computationally expensive and limited to relatively low Reynolds number flows (Ghai et al., 2022). The simulations were carried out using 1280 x 148 x 152 body-fitted grid points in the x-, y-, and z-directions. The mesh is non-uniform in all three directions, and refined near all solid surfaces.

2.5.1 Grid and Meshing Strategy

Grid and mesh strategies are methods for discretizing the domain of interest into a grid or mesh of smaller elements in computer simulations, particularly in computational fluid dynamics (CFD) and finite element analysis (FEA). These methods are crucial for resolving numerically described partial differential equations for physical occurrence.

A structured grid technique divides the computational domain into a network of cells that have predictable shapes, such as squares or rectangles in 2D or cubes in 3D. The connectivity pattern of this grid is clearly specified, with a predetermined number of neighbouring cells for each cell. Simple geometries benefit from structured grids because they offer strong numerical precision and stability. Unstructured grids are more adaptable and work well with complex geometries. They are made up of cells that can have any shape, such as tetrahedra or triangles in 2D or 3D, respectively. A mesh connectivity table is often used to define the connectivity between cells. Unstructured grids provide greater flexibility, although they can occasionally be less accurate and computationally demanding.

According to the studies carried out. (Moosa & Hekmat, 2019), the hybrid method which are a combination of structured and unstructured grids indicates that the advantageous of utilizing hybrid grid to significantly reduce the number of grids and computational cost. To simulate the incompressible steady fluid flow and heat transfer a quadrilateral-type structured grid. The grid system with the number of nodes equal to 245 × 95 (in X and Y directions respectively) performs around 0.330 %, and 0.372 % deviation for the Nu and f, respectively, compared with the grid of size 370 × 145. Therefore, the grid cell of 245 × 95 is selected for the rest of the study (Menni et al., 2019).

A Cartesian mesh, also known as a structured mesh, is generated by dividing the computational domain into a regular grid aligned with the coordinate axes. Each cell in the mesh represents a small volume or element for analysis. Cartesian meshes simplify grid generation and are particularly useful for problems with complex geometries. A body-fitted mesh, also called a non-orthogonal or curvilinear mesh, is constructed by conforming the mesh to the boundaries of the physical geometry. This type of mesh follows the shape of the object being analysed, allowing for accurate representation of curved or irregular surfaces. Body-fitted meshes are suitable for simulations that require high geometric fidelity.

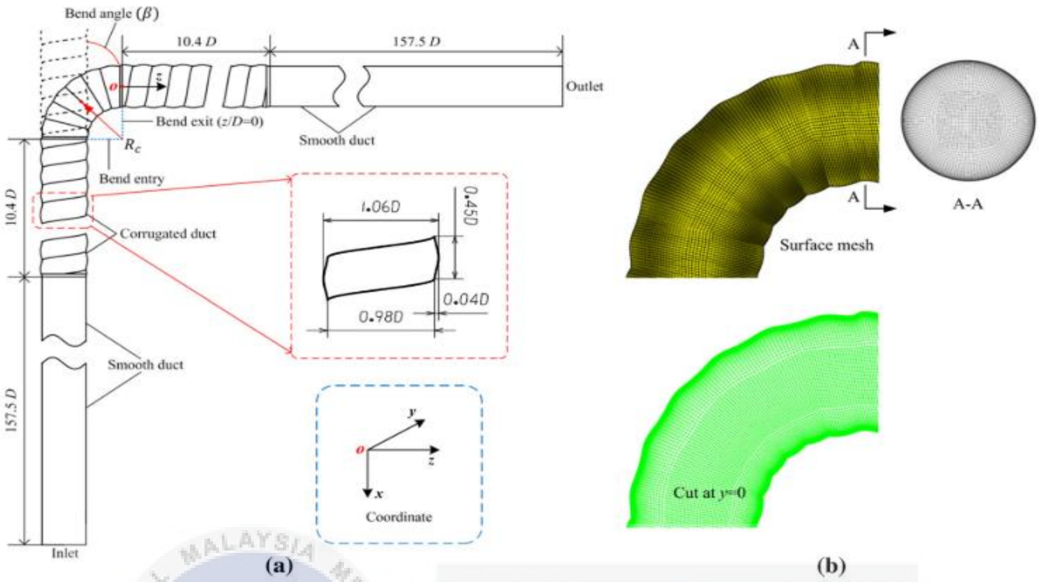


Figure 2-3 (a) Schematic of computational domain and geometric parameters of the corrugated duct; (b) grid distribution within the bend region (Du et al., 2019)

Figure 2-3a represents the case for β = 90◦, and (D = 1in.) and bend curvature ratio (γ = R/Rc = 0.46, where R is the effective radius of corrugated duct hexahedral mesh is generated within the computational domain. In order to achieve an accurate prediction of the origin of coordinates is located at the centre of the bend exit (i.e., z/D = 0). In this study, the high-quality flow behaviour near the duct wall, the first hexahedral layer is constructed at 0.02 mm together with a 10% increase in grid size from the wall, which renders the wall y+ to be less than 1 (Figure 2-3b). The corrugated duct part, i.e., the region of interest, is set as the grid refinement region (Figure 2-3b), which generally consists of denser hexahedral meshes Du *et al.*, 2019).

Grid modelling plays a crucial role in CFD simulations as it directly affects the accuracy, efficiency, and reliability of the numerical solution. The grid must capture the geometric details and flow features of the domain, while also satisfying certain criteria, such as grid quality and resolution, to ensure accurate and stable simulations.

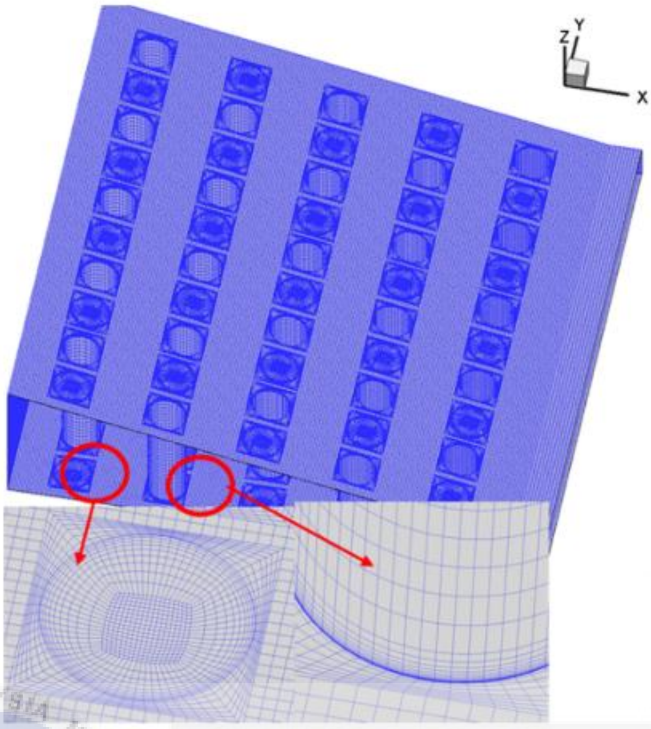


Figure 2-4 Mesh Details (Du et al., 2019)

In (Du et al., 2019) study, the an internal code built on the MATLAB platform creates the multi-block structured grid. When the mesh number is increased from 4.0 million to 5.0 million, it is discovered that the Nusselt number and friction factor remain almost constant. As a result, 4.0 million is chosen as the mesh number for all calculations. Figure 2-6 are the details of the mesh.

2.6 Turbulence Kinetic Energy (TKE)

Turbulent kinetic energy (TKE) is a measure of the energy associated with the turbulent motion of fluid in a flow field. It represents the fluctuating component of kinetic energy resulting from the chaotic and irregular motion of fluid particles in turbulent flow. TKE is an important parameter in understanding and characterizing turbulence in fluid dynamics. Mathematically, TKE is defined as the average of the square of the velocity fluctuations.

TKE provides information about the intensity and magnitude of turbulence in a flow field. It quantifies the energy available for the production and dissipation of turbulent eddies (G. Wang et al., 2021). Higher values of TKE indicate more energetic turbulence, while lower values indicate less turbulent flow. Figure 2-5 is a turbulent kinetic energy fields for S-upstream and S-downstream cases as a function of Reynolds number. The contour plots of turbulent kinetic energy fields obtained under turbulent flow regime for both S-baffle configurations with five various values of Reynolds number, Re = 12,000, 17,000, 22,000, 27,000, and 32,000. The trend for turbulent kinetic energy is similar. The plots show the largest value in the region opposite the right S-baffle and the smallest value in the region around the left S-baffle for both the S-upstream and S-downstream simulated. The turbulent kinetic energy augments with the augmentation of the Reynolds number (Re), and consequently, the Re = 32,000 provides the maximum turbulent kinetic energy in both cases studied. The use of S-upstream baffle shows better turbulent kinetic energy distribution values over the S-downstream baffle at almost stations.

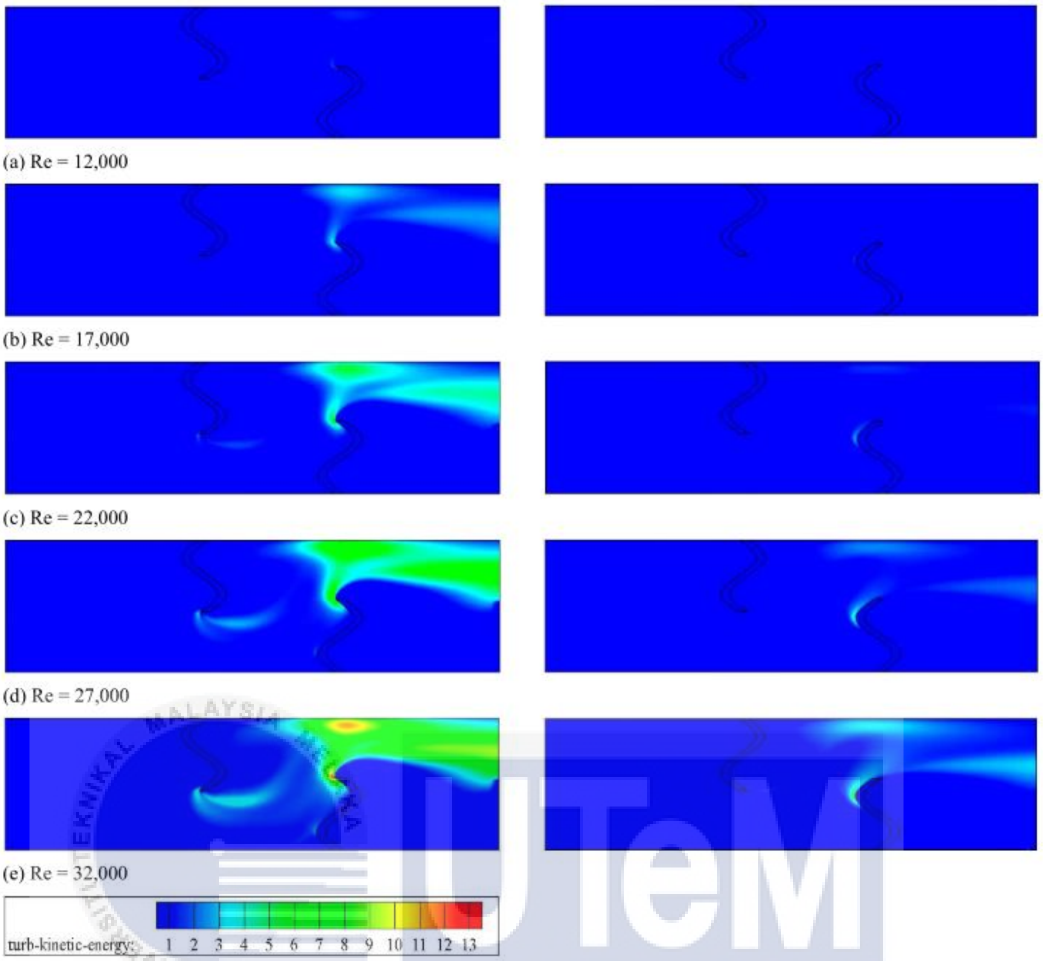


Figure 2-5 Turbulent kinetic energy fields for S-upstream and S-downstream cases as a function of Reynolds number. Dynamic pressure value is m^2/s^2. (Menni et al., 2019)

Turbulent kinetic energy is dependent on only the normal turbulent stress components. The production of turbulent kinetic energy results from the dissipation of mean kinetic energy by turbulent stresses. Contours of turbulent kinetic energy and the production of turbulent kinetic energy at the 45° and 90° cross sections are shown in **Figure 2-8** and **figure 2-9**, respectively. The turbulent kinetic energy contours are presented dimensionally, (m^2/s^2). The production of turbulent kinetic energy contours represents the summation of the nine individual terms. Production levels have been nondimensionalize by the INLET dynamic pressure, INLET centreline velocity and duct width.

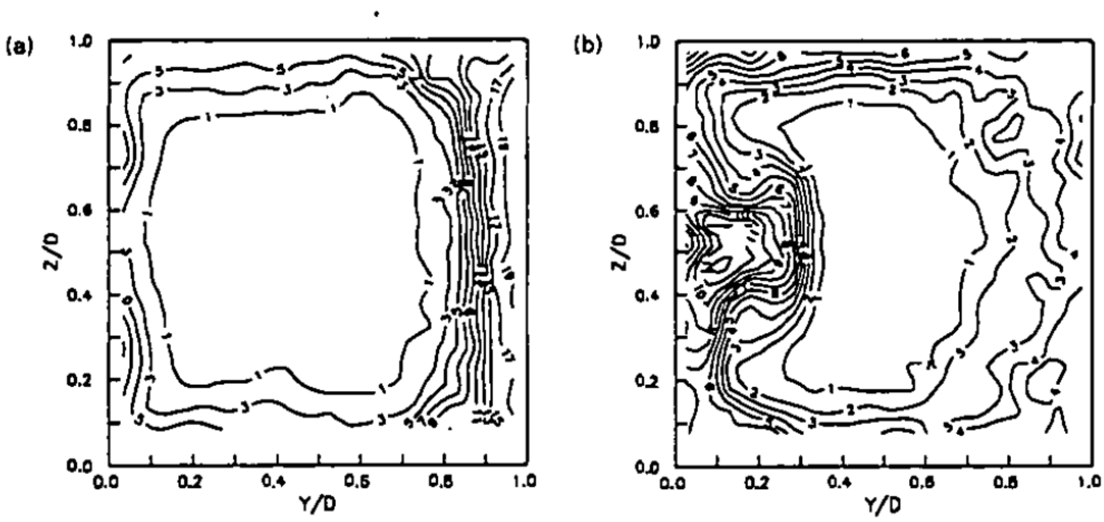


Figure 2-6 Turbulent kinetic energy at the (a) 45 0 and b) 90° cross sections presented dimensionally.

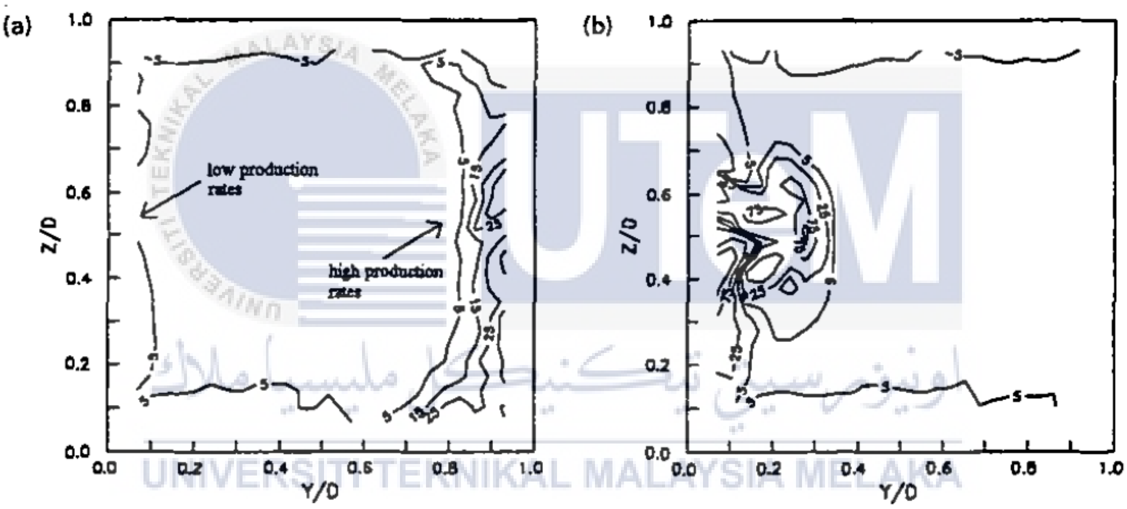


Figure 2-7 Production of turbulent kinetic energy at the (a) 45°, and (b) 90° cross sections, nondimensionalized by the INLET dynamic head, INLET velocity and duct width.

2.7 Summary

The effectiveness of fin devices is critical in many engineering applications, including heat exchangers and cooling systems, in the fields of heat transfer and fluid dynamics. Researchers have investigated several aspects and elements impacting these devices' efficiency in order to improve their performance. Turbulence kinetic energy (TKE) and ducting fin gained significant attention among these parameters because of their potential to affect the heat transfer process. Turbulence kinetic energy refers to the energy associated with the turbulent motion of fluid particles.

While TKE and ducting angle have been studied individually in connection to heat transfer performance, there is a lack of comprehensive understanding of their combined effect on the efficiency of fin devices in ducting. The purpose of this literature review is to bridge this knowledge gap by investigating existing research on the combined influence of TKE on the efficiency of fin devices in ducting. This study attempts to identify the present state of knowledge, evaluate the approaches used, and analyse the findings linked to the joint influence of TKE and ducting angle on the efficiency of fin devices in this study by reading relevant literature.

Furthermore, the aim of this review will be to highlight any inconsistencies, gaps, or limitations in existing research, paving the way for future research in this area. Understanding the joint effect of TKE on the efficiency of fin devices in ducting has significant practical implications for optimising heat transfer processes in a variety of engineering applications. This leads to enhanced performance, energy efficiency, and cost-effectiveness. This literature review provides a detailed summary of the studies conducted to date, summarise the methodology used, and analyse the important findings in the sections that follow. This information's synthesis will contribute to a better understanding.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this section, will delve into the detailed methodology adopted for in this research, outlining the research design, data collection methods, and analytical techniques used. By explaining the methodology, this aims to provide transparency and enable for this study to follow through and the reliability and validity of this research. Figure 3-1 is flowchart in fulfilling this research. The flow chart shows each of the process and methodology in this research. This section will explain in detailed each one. The flow chart has two-part, computational modelling and physical validation.

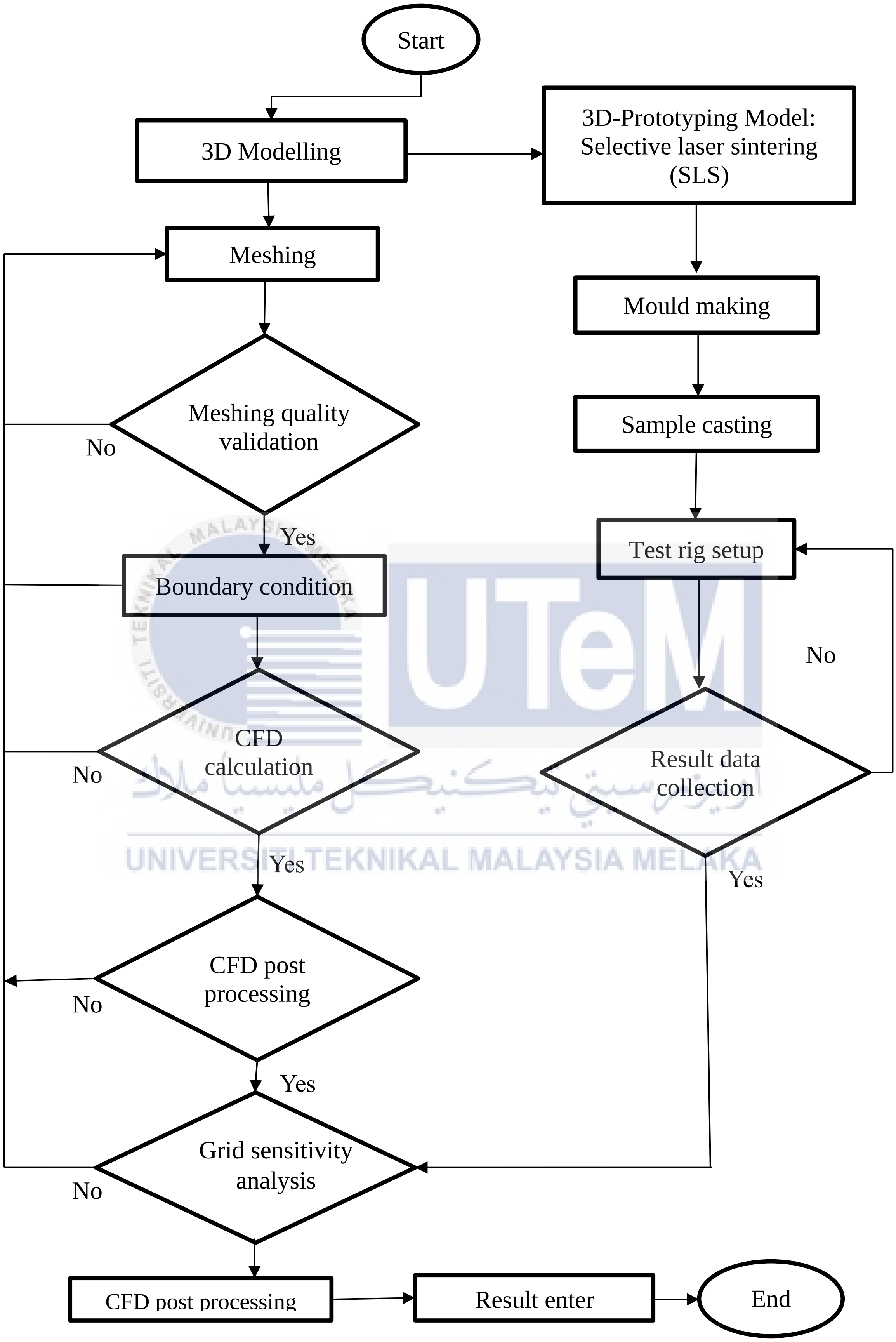


Figure 3-1 Computational modelling and Physical Validation Flow Chart

3.2 3D Modelling

The use of three-dimensional (3D) modelling tools in TKE allows for the simulation and analysis of fluid and airflow behaviour inside of ducting systems. A metric called TKE, or turbulent kinetic energy, is used to describe how turbulent a fluid flow is. This can help better understand the effects of flow patterns and turbulence on the performance of fin devices by introducing 3D modelling into TKE.

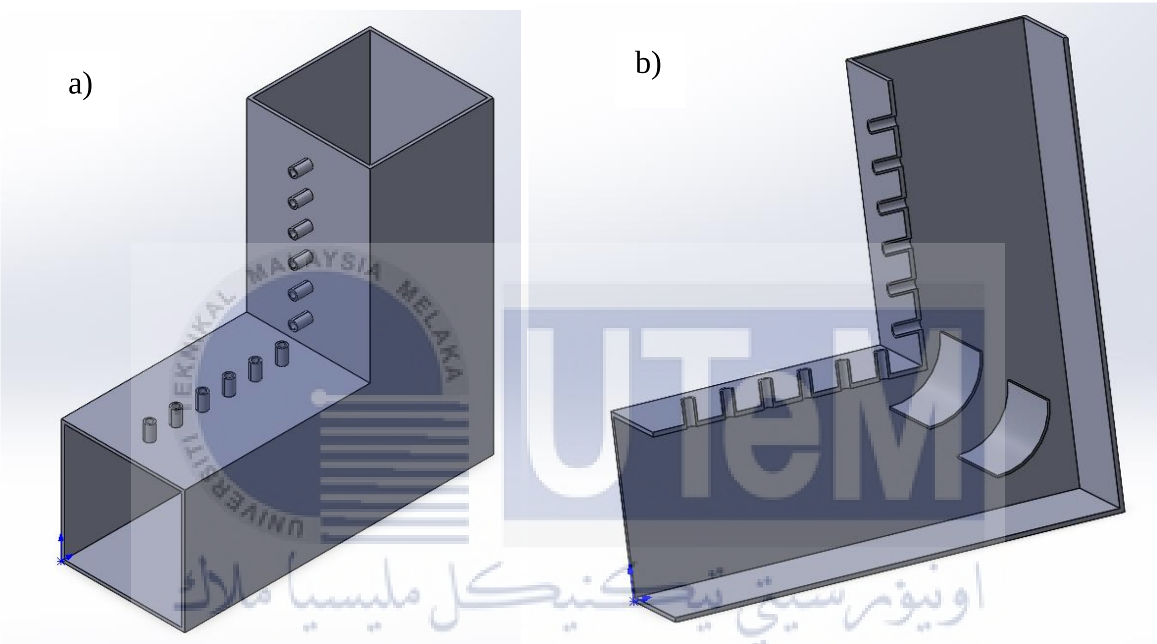


Figure 3-2 a) Geometry 90° square duct. b) Section view of the square ducting with added fins.

3.3 Meshing

In CFD simulations, meshing refers to the process of dividing the computational domain into smaller finite elements or cells to represent the physical geometry. Meshing is important in CFD simulations because it discretizes the domain, allowing the equations governing fluid flow and heat transfer to be solved numerically. The accuracy and efficiency of the simulation results depend on the quality and resolution of the mesh. There are few types of meshing that are used to generate mesh. In this research, triangular element are used for 2D geometry and tetrahedral for 3D.

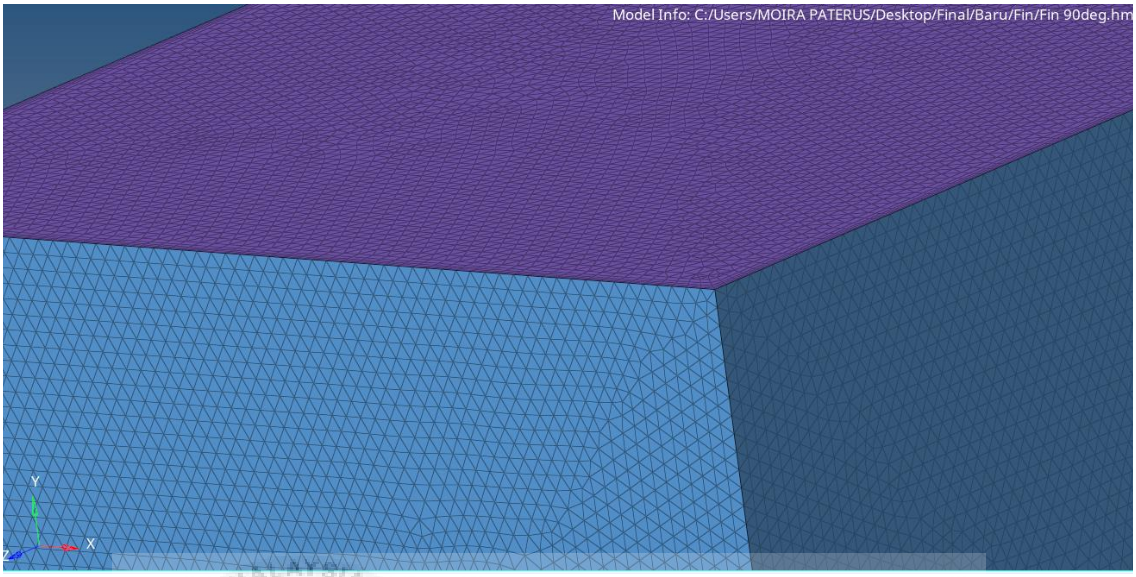


Figure 3-3 Mesh

3.4 Meshing Quality Validation

Mesh quality validation is an essential step in computational simulations to ensure the accuracy and reliability of results. It involves assessing the quality of the mesh generated for a given computational domain. Mesh quality metrics quantify the geometric and connectivity aspects of the mesh, such as element shape, size, skewness, and aspect ratio.

This statistic evaluates the distortion and shape of each mesh element. Aspect ratio, skewness, and Jacobian determinants are examples of typical measurements. For precise and reliable simulations, well-shaped parts are preferred, but distorted elements might cause numerical mistakes. The mesh should be sufficiently accurate to ensure that further mesh refining has no discernible impact on the simulation results. Mesh independence studies compare the outcomes of various mesh resolutions in order to assess convergence.

The mesh must faithfully depict the simulated geometry's bounds. For a realistic representation of flow mechanics, boundary conformance and the ability to capture complex forms or curves are crucial.

Grid Convergence Index (GCI) analysis is a method for calculating the inaccuracy linked to a specific mesh resolution. In order to do this, results from various mesh resolutions must be compared, and an index must be calculated that reveals information about the convergence behaviour and anticipated error.

The purpose of meshing quality validation is to make sure that the computational mesh accurately captures significant flow features, accurately represents the geometry, and produces reliable simulation results. It facilitates the early identification of possible problems throughout the simulation process and directs the mesh refining process to improve accuracy and convergence.

3.5 Boundary Condition

When solving mathematical models or running computational simulations, boundary conditions in ducting relate to the specifications that are applied at the boundaries of the duct system. The behaviour of the fluid or air flow at the entrance and outflow points of the duct or other surfaces of interest is defined by these boundary conditions. The boundary conditions used depend on the simulation type and the issue being studied. Inlet, outlet, model, reference value are considering boundary setup.

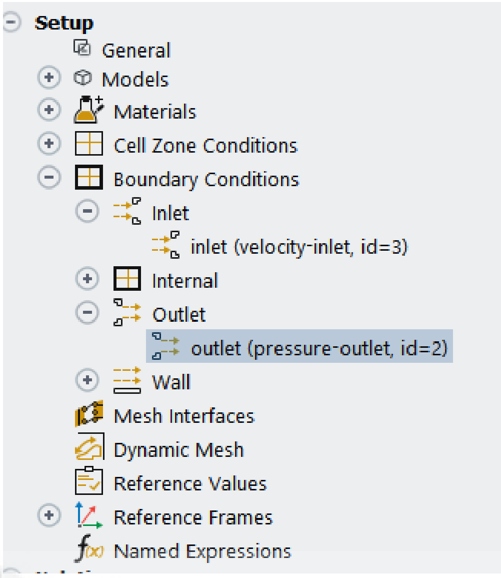


Figure 3-4 Setup for boundary condition in ANSYS

3.5.1 Turbulence model

In computational fluid dynamics (CFD), a turbulence model is used to simulate and model the effects of turbulence in fluid flow. Turbulence refers to the chaotic and irregular motion of fluid that occurs at high Reynolds numbers, resulting in fluctuations in velocity, pressure, and other flow properties. Turbulence models are mathematical equations or closures that approximate the behaviour of turbulence based on various assumptions and simplifications. These models help predict the turbulent flow characteristics within a given fluid domain, considering factors such as turbulent eddies, energy dissipation, and the interaction between different scales of turbulence. Figure 3-5 is the models setup for this research, turbulence model used are k-ω SST.

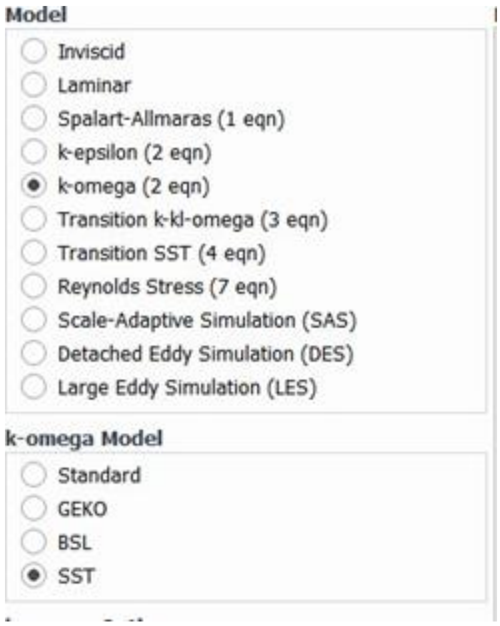


Figure 3-5 models setting

3.5.1.1 k-ω SST (Shear Stress Transport)

Among the plethora of turbulence models available, the k-ω SST (Shear Stress Transport) model has gained considerable popularity due to its ability to handle a wide range of flow conditions and its improved accuracy compared to traditional models.

To provide reliable predictions for both attached and separated turbulent flows, the k- ω SST model incorporates components of the k-ε and k-ω models. It excels at capturing flow separation, boundary layer transition, and reattachment processes. The capacity of the k-ω SST model to fluidly shift between the k-epsilon and k-omega regimes based on flow conditions is a crucial feature. To account for the impacts of wall proximity in the near-wall region, the model employs the k-omega formulation. It switches to the k-epsilon formulation away from the wall to accurately capture free shear layers and recirculation zones.

From the previous study that has use turbulence model, k-ω SST turbulent model has better performance at low Reynolds number. It is found that the k-ω SST can predict the friction factor more accurately compared to other turbulent models. The two transported variables are turbulent kinetic energy (k), which determines the energy in turbulence, and specific turbulent dissipation rate (ω), which determines the rate of dissipation per unit turbulent kinetic energy. ω is also referred to as the scale of turbulence. The standard k−ω model is a low Re model it can be used for flows with low Reynolds number where the boundary layer is relatively thick and the viscous sublayer can be resolved (Menter, 1994).

$$\tau = \overline{-\rho u' v'}$$

By inclusion of the term:

$$\frac{Dr}{Dr} =: \frac{\partial r}{\partial t} + u_k \frac{\partial r}{\partial u_k} \qquad \text{(Menter, 1994)}$$

$$F1 = tanh\{\{min\ [\ max\ (\frac{\sqrt{k}}{\beta * \omega y'} \frac{500v}{y^2 \omega}), \frac{4\sigma_{\omega 2} k}{CD_{k\omega} y^2}]\}^4$$

3-1 Transport of the principal turbulence transport of the principal turbulent shear stress

The equation above that defines the turbulent shear stress, denoted by the symbol tau (τ). The equation is written as τ = -ρu'v', where ρ is the density of the fluid and u' and v' are the fluctuating velocities in the x and y directions, respectively.

The ducting flows often involve complex near-wall phenomena, including boundary layer development, flow separation, and reattachment. The k-ω SST model incorporates a specific formulation for the near-wall region, known as the wall function approach. This allows it to accurately capture the turbulent behaviour near walls and provide reliable predictions of flow characteristics in ducts.

3.5.2 TKE

Turbulent kinetic energy (TKE) is a measure of the energy associated with the turbulent motion of fluid in a flow field. It represents the fluctuating component of kinetic energy resulting from the chaotic and irregular motion of fluid particles in turbulent flow. TKE is an important parameter in understanding and characterizing turbulence in fluid dynamics. Mathematically, TKE is defined as the average of the square of the velocity fluctuations:

TKE provides information about the intensity and magnitude of turbulence in a flow field. It quantifies the energy available for the production and dissipation of turbulent eddies (G. Wang et al., 2021). Higher values of TKE indicate more energetic turbulence, while lower values indicate less turbulent flow.

Turbulence kinetic energy refers to the energy associated with the turbulent motion of fluid particles in a flow.

$$TKE = \frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)$$

3-2 TKE formula

Where $\overline{u'^2}$, $\overline{v'^2}$ are $\overline{w'^2}$ the Reynolds-averaged velocity fluctuations in the three coordinate directions.

3.6 CFD Calculation

The term "CFD calculation" refers to the process of running simulations and analysis using computational fluid dynamics (CFD) In the field of fluid mechanics known as computational fluid dynamics (CFD), problems involving fluid flow and are analyses and solved using numerical methods and algorithms.

After setting up the parameter and initializing, the number iteration is set up to1000 before the mesh is calculated as shown in Figure 3-6. The result will be updated once the calculation is done running. Then, the result can be analysed for post-processing.

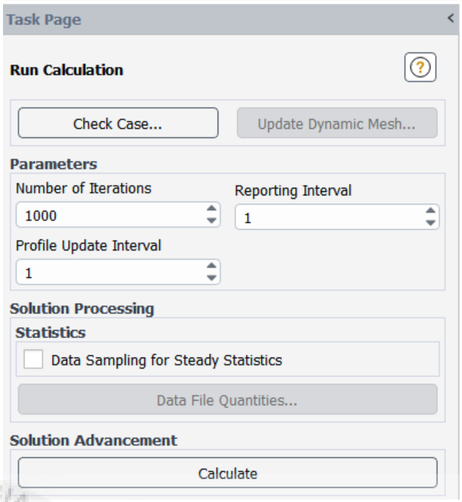


Figure 3-6 Task page for CFD calculation in ANSYS

3.7 Grid sensitivity analysis

Grid Sensitivity Analysis can reveal how sensitive the solution is to various mesh configurations by changing the mesh resolution and analysing the effect on the simulation results. This research aids in locating key regions that demand increased mesh resolution.

For this research, five mesh resolution are process in ANSYS FLUENT to see the Grid sensitivity analysis is essential for ensuring the credibility of numerical simulations. It helps in understanding the impact of mesh resolution on the accuracy and stability of the results, to choose an appropriate level of mesh refinement for their specific simulation requirements. Performing such analyses is to achieve reliable and robust results.

3.8 CFD Post Processing

After the CFD simulation is completed, the results can be analysed using post-processing techniques. TKE values should be extracted from simulation data to better understand the distribution and behaviour of turbulence inside the ducting system. Using contours, streamlines, or velocity vectors to visualise flow patterns and identify locations of high turbulence intensity or flow recirculation.

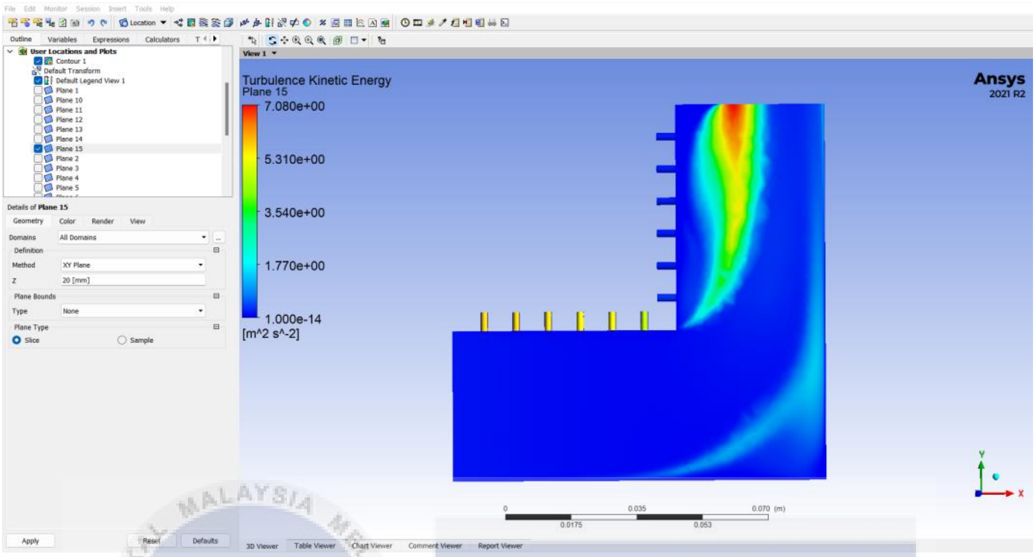


Figure 3-7 Contour for pressure point

In order to see the value for a variable such as TKE, choose the contour and set up the plane. After setting up and click apply it will show the result as shown in Figure 3-7. Figure 3-8 planes are the created along the modelling to find the value of TKE production on each of the plane.

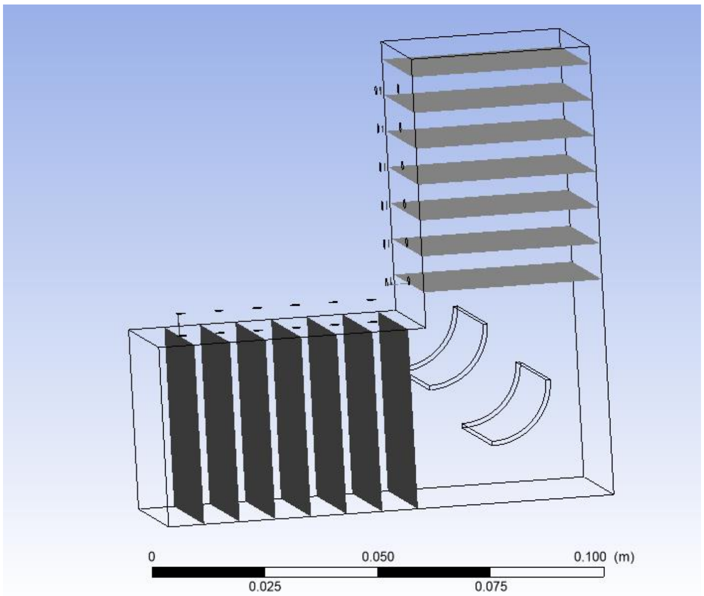


Figure 3-8 plane

3.9 Physical Validation

Computational Fluid Dynamics (CFD) is a field that uses numerical methods and algorithms to simulate and analyses fluid flow phenomena. However, the accuracy and reliability of CFD simulations heavily depend on the validation and verification of the results. Physical validation refers to the process of comparing the CFD results with experimental or real-world data to ensure that the simulations accurately represent the physical reality.

The combination of grid sensitivity analysis and physical validation enhances the robustness of CFD simulations by refining the mesh and confirming the reliability of the numerical model. These processes ensure a balance between accuracy and computational efficiency in the simulation.

3.9.1 Test rig setup

In Figure 3-9 and Figure 3-10, shows the whole test rig setup. After setting up the test rig, before conducting the actual test, calibrate and verify the sensors and instrumentation to ensure accurate measurements. Create the necessary conditions for the test environment such as the temperature are set to 25° Celsius. The parameter that needs to be taken are the pressure of the air flow inside the duct. This result will be compared with benchmark model of CFD.

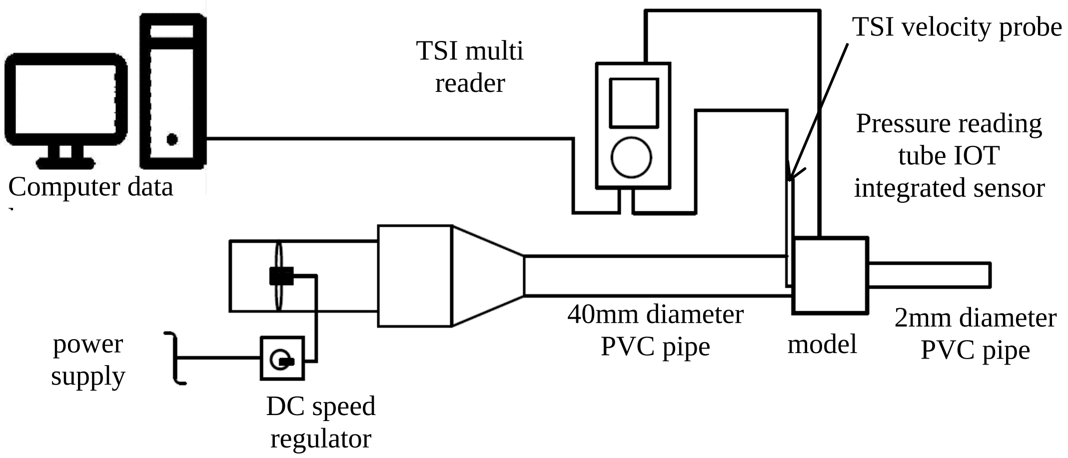


Figure 3-9 Setup of the test rig

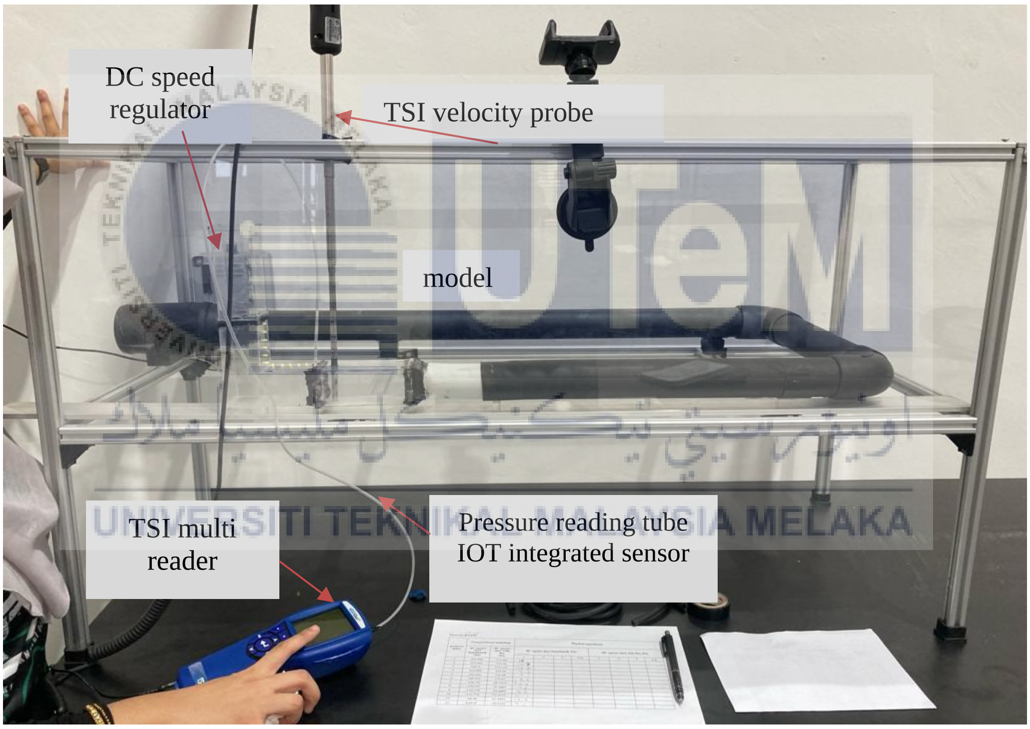


Figure 3-10 actual test rig setup.

After comparing the data between Grid sensitivity analysis and the physical test, the best mesh size is further analysed together with the new model ducting with added fin.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

In this chapter, the finding of this study is analysed and discussed thoroughly the insights obtained from numerical and physical investigations offers a foundation for in-depth interpretations of the result that have been observed. This study focuses on the effectiveness of fin devices in a 90° degree square duct. The simulations are on benchmark model which is ducting with no fin and duct with fins added to it. The results are compared to see the different between the benchmark model and the finned duct design on how it varies in different variable like the pressure, TKE production and velocity.

4.2 Results for grid sensitivity analysis

The grid sensitivity analysis is conducted to see impact of mesh resolution on the accuracy and stability of the results, to choose an appropriate level of mesh refinement for their specific simulation requirements. Figure 4-1 shows the pressure for 5 benchmark models. The benchmarks are of the same geometry with different mesh element count. The benchmark velocity inlet is setup to 6.92 m/s with the ducting area of 40mm x 40mm that from that will get the CFM.

As the mesh element count increases, pressure values at some points appear to converge or stabilize, while at others, the pressure may vary. Negative pressure values are observed at some points, indicating potential issues or challenges in the simulation. Evaluate how pressure values change at each point concerning the mesh element count. The change of pressure value at each point are evaluated to identify the points where pressure values are sensitive to mesh changes and points and they stabilize with increased mesh resolution.

Figure 4-2 is a graph with grid sensitivity analysis together with the result of physical validation. From this grid sensitivity analysis, it becomes clear that the 3 million element count mesh resolution offers the best optimal balance between computational efficiency and accuracy, exhibiting numerical solution convergence and producing outcomes that closely match experimental result.

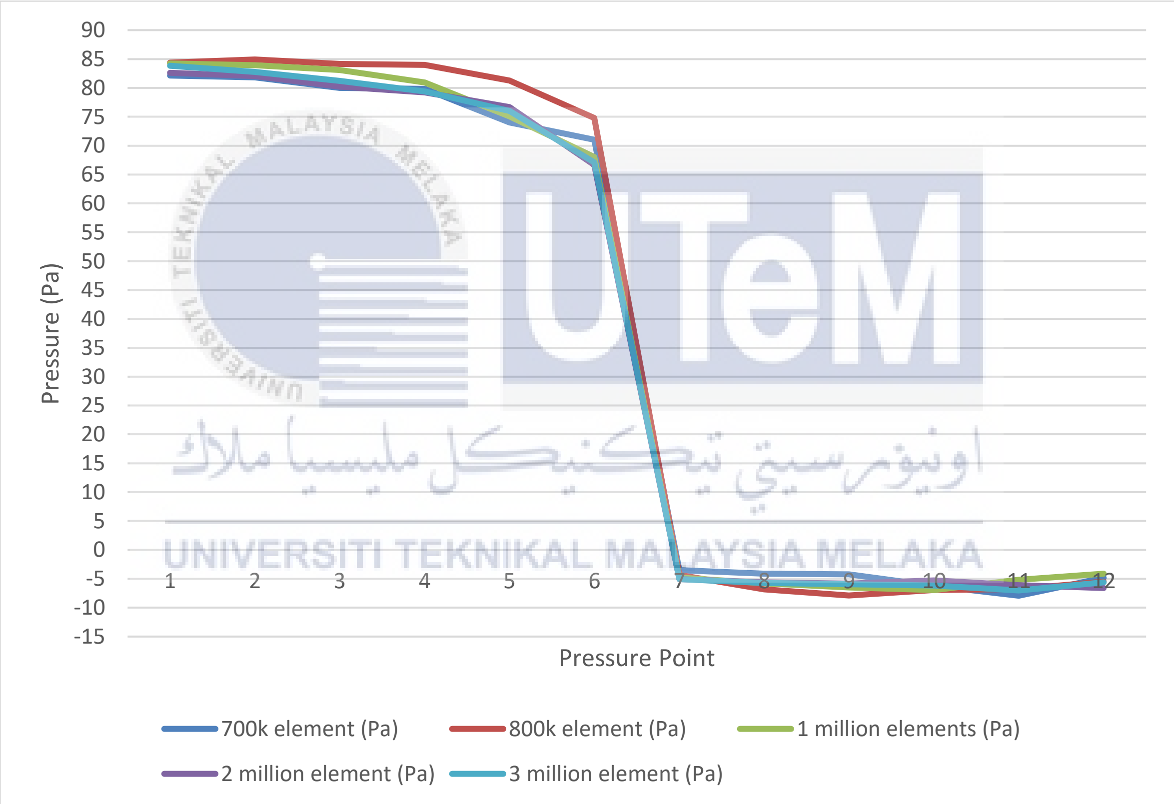


Figure 4-1 Pressure at each of the pressure point for five different elements for benchmark square duct.

The combination of grid sensitivity analysis and physical validation enhances the robustness of CFD simulations by refining the mesh and confirming the reliability of the numerical model. These processes ensure a balance between accuracy and computational efficiency in the simulation.

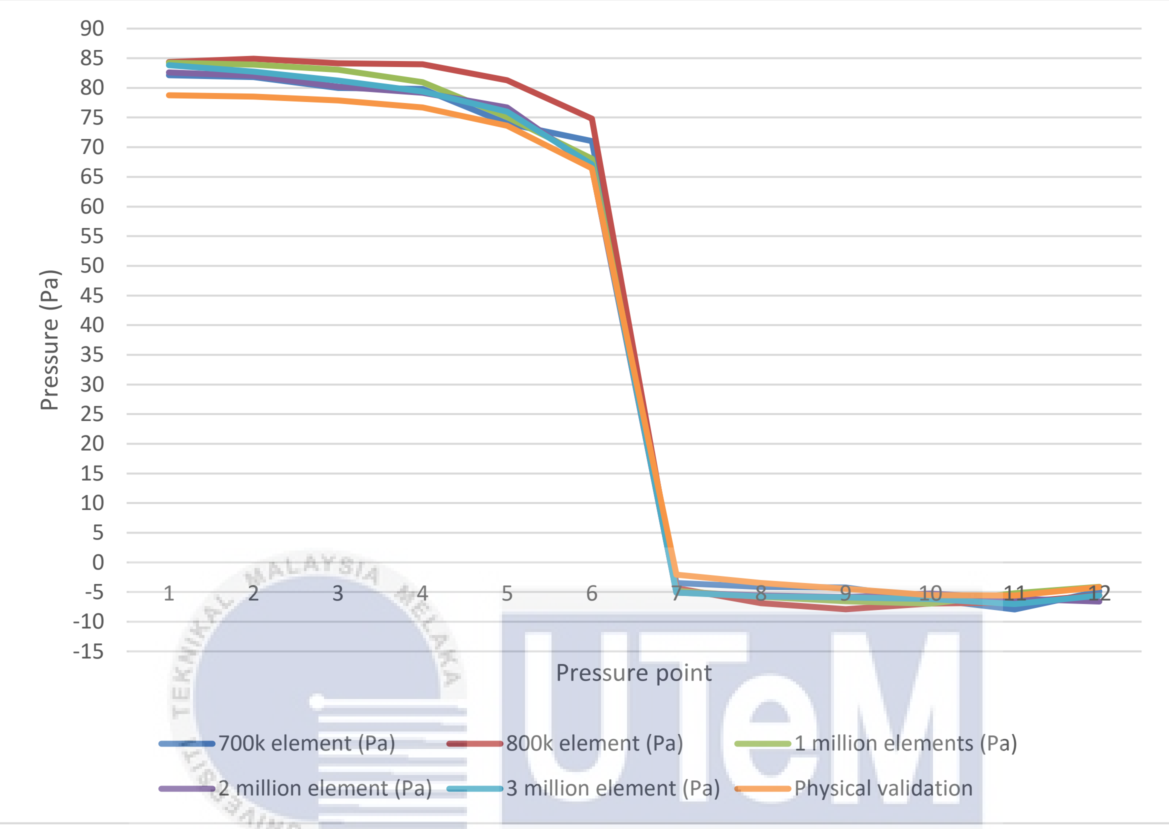


Figure 4-2 Pressure at each of the pressure point for five different elements for benchmark square duct with pressure from physical validation

4.3 Result between benchmark model and finned square duct

The following part will discuss and elaborate the result of flow region in ducting with TKE, the contour for velocity and pressure in both the models.

4.3.1 Turbulence Kinetic Energy (TKE)

The presence of a fin in ducting system can have a significant impact on the TKE within the flow. Fin is typically designed to influence the fluid flow by redirecting and controlling the direction and intensity of the flow. Table 4-1 show the value of TKE in the ducting of benchmark model and model with fin. The presence of a fin in a square ducting affects TKE by reducing the TKE value. This is because the fin helps to turn the air flow, reducing the secondary flow and the separation zone in the elbow, which are the main sources of turbulence and energy loss. Figure 4-3 showing that the TKE value in ducting with a fin is lower than in ducting without a fin. The fin helps to reduce pressure loss and energy dissipation, contributing to a more efficient flow.

Table 4-1 TKE for benchmark and finned duct

Benchmark ($J\ kg^{-1}$)	**Finned square duct ($J\ kg^{-1}$)**
1.83E+03	1.13E+03
1.46E+03	8.98E+02
1.33E+03	7.55E+02
1.28E+03	6.54E+02
1.29E+03	5.98E+02
1.47E+03	5.66E+02
1.91E+03	6.10E+02
5.13E+03	3.19E+03
5.81E+03	3.47E+03
6.65E+03	4.48E+03
7.20E+03	4.34E+03
8.68E+03	5.23E+03
1.01E+04	6.00E+03
1.50E+04	7.48E+03

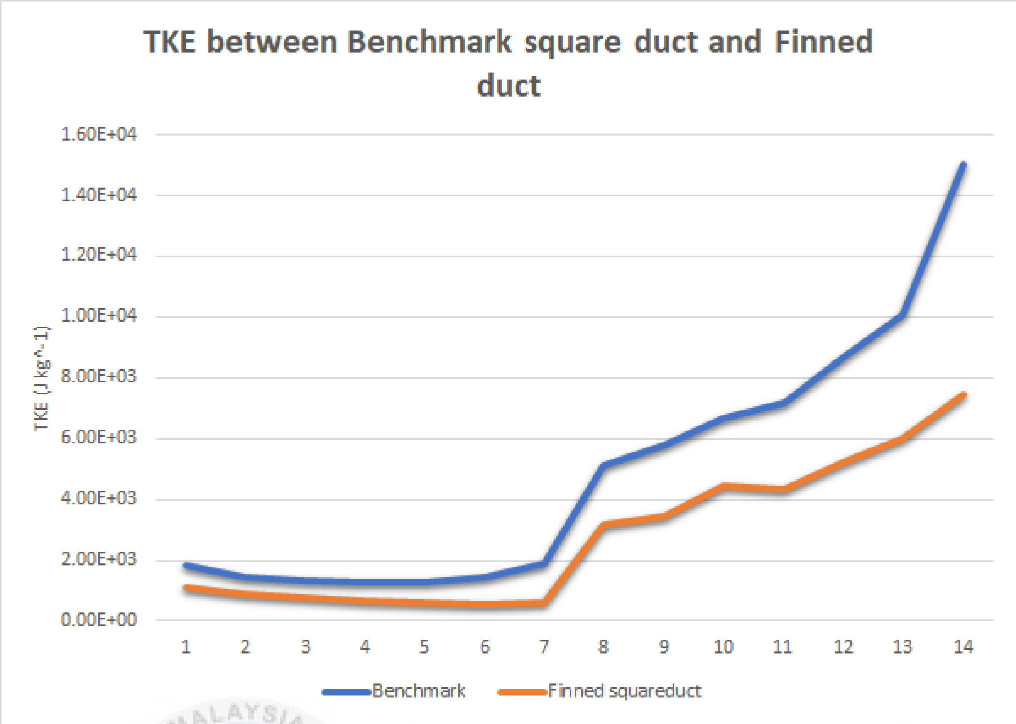


Figure 4-3 TKE between benchmark and finned square duct

Figure 4-4 is a contour of TKE values in the duct with no fin and finned square duct. Table 4-1 shows the value of TKE are consistently lower compared to the benchmark. Across the board, the addition of fins to the square duct results in a reduction in TKE values. This suggests that the presence of fins has a dampening effect on turbulence levels.

Figure 4-4 presents a contour illustrating the Turbulence Kinetic Energy (TKE) values within the duct, comparing configurations with and without fins. Table 4-1 provides a comprehensive overview of the TKE values, revealing a consistent pattern of lower TKE levels in ducting with fin compared to the benchmark. Notably, the incorporation of fins into the square duct uniformly leads to a reduction in TKE values across various sections.

The systematic decrease in TKE levels with the introduction of fins implies a discernible dampening effect on turbulence within the duct. This observation suggests that the presence of fins acts as a mitigating factor, influencing and suppressing turbulence levels. The fins, by their nature, seem to contribute to a more controlled and stabilized flow environment within the duct, resulting in a diminished intensity of turbulent motion.

This reduction in TKE values with the addition of fins holds implications for the overall flow dynamics and efficiency within the ducting system. The dampening effect on turbulence is indicative of a more controlled and ordered flow regime facilitated by the fins.

The consistent trend across the various sections of the duct reinforces the reliability and reproducibility of the findings. This insight into the dampening effect of fins on turbulence levels adds a valuable dimension to the understanding of how specific design elements, like fins, can influence and modulate the turbulence characteristics within a ducted system.

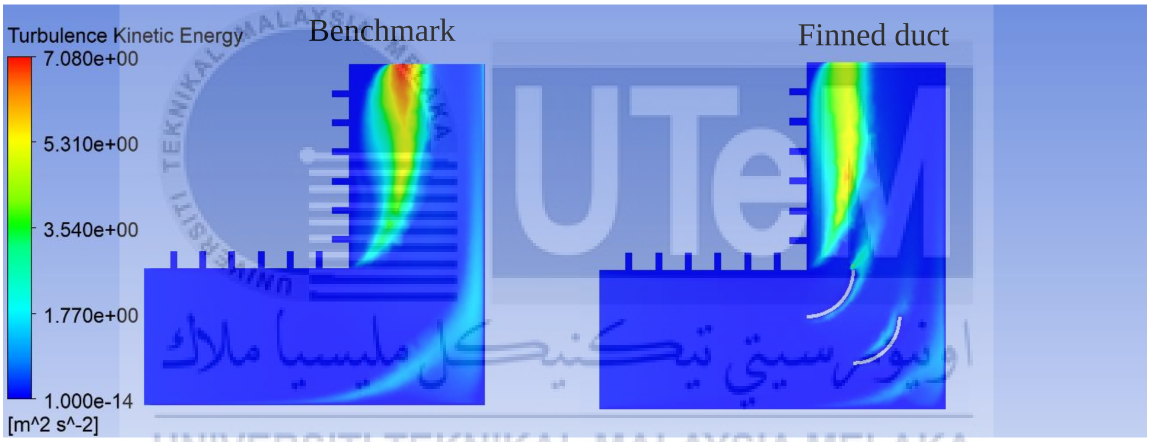


Figure 4-4 contour of TKE values in the duct with no fin and finned square duct

4.3.2 Contour of pressure and velocity

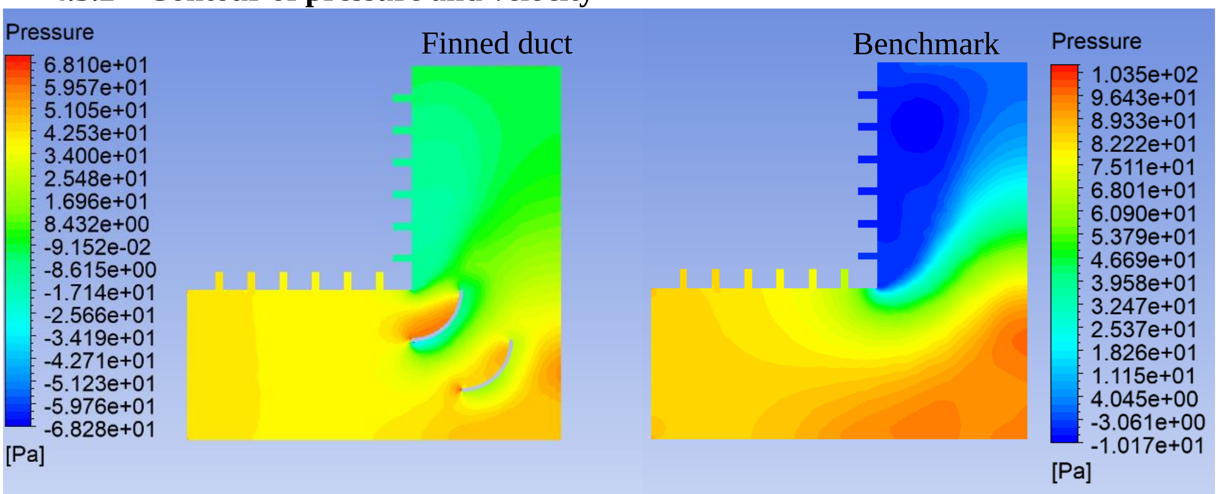


Figure 4-5 contour of pressure benchmark and fin duct

Figure 4-5 shows the pressure different between benchmark and finned duct. The pressure level in a ducting system can have significant effects on the flow behaviour and turbulence characteristics, which, in turn, relate to the production of Turbulence Kinetic Energy (TKE). Low pressure generally corresponds to reduced flow velocities within the duct. Slower flows are more likely to exhibit laminar characteristics, and the turbulence intensity may be lower compared to higher-pressure conditions. Low-pressure conditions can make the flow more sensitive to disturbances, leading to potential laminar-turbulent transition. Disturbances from obstacles or changes in geometry may have a more pronounced effect.

The production of TKE is typically lower in laminar flows. Lower pressure and reduced flow velocity contribute to a lack of energy in the flow, resulting in less intense turbulence and lower TKE production. High-pressure conditions often correspond to higher flow velocities. Increased flow velocities can promote turbulence, leading to higher turbulence intensity and TKE production, especially in regions with flow disturbances or geometric complexities. Higher pressure tends to favour the development and sustenance of turbulent flow patterns. Turbulence becomes more prominent, and the flow is characterized by the presence of eddies, vortices, and fluctuations, contributing to increased TKE. Pressure gradients within the duct can influence the distribution of TKE. High-pressure regions may coincide with higher TKE levels, especially in areas where the flow is subjected to sudden expansions, contractions, or bends.

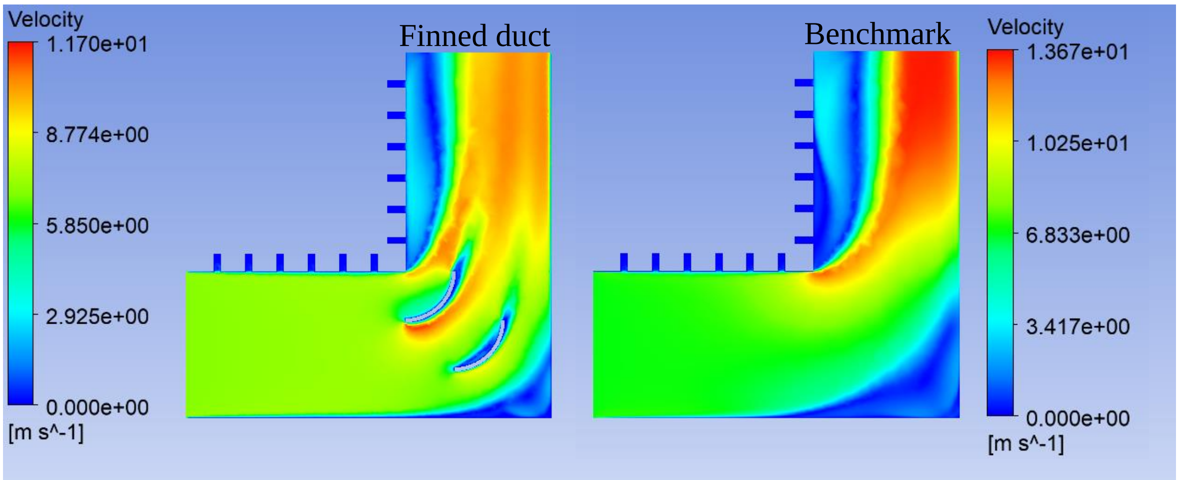


Figure 4-6 contour of velocity

The turbulence intensity, which is related to the pressure and flow velocity, influences TKE production. Higher turbulence intensity is associated with higher TKE levels, while lower turbulence intensity corresponds to reduced TKE. Pressure variations, such as those caused by obstacles or changes in duct geometry, can create flow disturbances that impact TKE production. Regions with flow separation or reattachment may exhibit variations in TKE levels. Changes in pressure conditions can influence the transition between laminar and turbulent flow regimes. Higher pressures are generally conducive to sustained turbulence, resulting in increased TKE, while lower pressures may promote laminar conditions with lower TKE production.

In summary, the pressure level in ducting systems plays a crucial role in determining the flow characteristics and, consequently, the production of TKE. High-pressure conditions often lead to turbulent flow with increased TKE, while low-pressure conditions may result in laminar flow with lower TKE levels. The specific relationship between pressure and TKE depends on factors such as flow velocity, turbulence intensity, and the presence of flow disturbances.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Conclusion

In conclusion, this study focused on investigating the efficiency of fin devices within ducting systems, considering the combined effects of turbulence kinetic energy (TKE) and ducting fin. The research addressed three main objectives, firstly, the creation of a scale model of square ducting for validating computational fluid dynamics (CFD) simulations; secondly, testing airflow within a physical test rig to ensure the accurate reproduction of real-world conditions; and thirdly, employing CFD techniques to examine the flow patterns and phenomena within the square ducting, specifically assessing the relationship between duct angle and fin device effectiveness.

To achieve these objectives, a scale down model closely resembling the geometry of actual ducting was constructed. This model served as a standard for verifying the precision and reliability of CFD simulations, with the simulation methods validated through comparison with experimental data from the physical test rig, including outcomes from grid sensitivity analysis.

The CFD techniques employed provided a comprehensive understanding of flow zones inside the square ducting, elucidating flow patterns, velocity distribution, and pressure gradients. This knowledge proved instrumental in enhancing the functionality and design of fin devices utilized in ducting applications. The study further explored the connection between TKE and fin devices, systematically varying TKE levels to analyze the efficiency and effectiveness of fin devices across various ducting configurations.

The results highlighted a reduction in TKE levels when comparing ducting with no fin devices to those with fins, indicating a lower intensity of turbulent motion within the flow. The performance of fin devices in ducting systems was found to be influenced by TKE levels, advancing our understanding of their functionality. The clarity provided by CFD simulations regarding flow dynamics inside the square ducting contributes valuable insights that can be utilized to improve the performance and energy efficiency of ducting systems in and technical applications. The integration of grid sensitivity analysis and physical validation ensures the reliability of the study's findings, reinforcing the significance of the obtained insights for practical applications in the field of fluid dynamics.

5.2 Recommendations

Based on the findings and conclusions of this study, several recommendations for future research can be proposed to further advance the understanding of fin devices in ducting systems and optimise their performance.

a. Exploration of Different Fin Geometries

Future research can investigate the impact of various fin geometries on turbulence kinetic energy (TKE) and overall efficiency. Comparing different fin shapes, sizes, and configurations may provide insights into the most effective designs for specific applications.

b. Influence of Ducting Configurations

Consider studying the influence of different ducting configurations on the effectiveness of fin devices. Varying duct shapes, angles, and dimensions could reveal how these factors interact with fin devices and influence TKE levels.

c. Environmental Impact Assessment

Consider assessing the environmental impact of fin devices within ducting systems. Evaluate factors such as energy consumption, carbon footprint, and sustainability to guide the development of environmentally friendly designs.

By addressing these recommendations, future research endeavours can contribute to the continuous improvement of fin devices in ducting systems, offering practical solutions for enhanced energy efficiency and performance across various industrial and technical applications.

REFERENCES

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

Besant, R. W., & Asiedu, Y. (2000). Sizing and balancing air duct systems. *ASHRAE Journal*, *42*(12).

Du, W., Luo, L., Wang, S., & Zhang, X. (2019). Flow structure and heat transfer characteristics in a 90-deg turned pin fined duct with different dimple/protrusion depths. *Applied Thermal Engineering*, *146*(September 2018), 826–842. https://doi.org/10.1016/j.applthermaleng.2018.10.052

Ghai, S. K., Ahmed, U., Klein, M., & Chakraborty, N. (2022). Turbulent kinetic energy evolution in turbulent boundary layers during head-on interaction of premixed flames with inert walls for different thermal boundary conditions. *Proceedings of the Combustion Institute*, *000*, 1–10. https://doi.org/10.1016/j.proci.2022.08.055

Greiner, M., & Würz, W. (2022). In-flight measurement of free-stream turbulence in the convective boundary layer. *Experiments in Fluids*, *63*(10). https://doi.org/10.1007/s00348-022-03506-6

Hu, K. S. Y., Chi, X., Shih, T. I. P., Chyu, M., & Crawford, M. (2019). Steady RANS of Flow and Heat Transfer in a Smooth and Pin-Finned U-Duct With a Trapezoidal Cross Section. *Journal of Engineering for Gas Turbines and Power*, *141*(6). https://doi.org/10.1115/1.4042332

Khan, H. H., Anwer, S. F., Hasan, N., & Sanghi, S. (2021). Laminar to turbulent transition in a finite length square duct subjected to inlet disturbance. *Physics of Fluids*, *33*(6), 065128. https://doi.org/10.1063/5.0048876

Krappel, T., Ruprecht, A., Riedelbauch, S., Jester-Zuerker, R., & Jung, A. (2014). Investigation of Francis Turbine Part Load Instabilities using Flow Simulations with a Hybrid RANS-LES Turbulence Model. In N. Desy, C. Deschenes, F. Guibault, M. Page, M. Turgeon, & A. M. Giroux (Eds.), *27TH IAHR SYMPOSIUM ON HYDRAULIC MACHINERY AND SYSTEMS (IAHR 2014), PTS 1-7* (Vol. 22). https://doi.org/10.1088/1755-1315/22/3/032001

Krömer, F. J., Moreau, S., & Becker, S. (2019). Experimental investigation of the interplay between the sound field and the flow field in skewed low-pressure axial fans. *Journal of Sound and Vibration*, *442*, 220–236. https://doi.org/10.1016/j.jsv.2018.10.058

Ladino, A., Duque-Daza, C. A., & Lain, S. (2023). Effect of walls with large scale roughness in deposition efficiency for 90-degree square bend configurations. *Journal of Aerosol Science*, *167*(September 2022), 106093. https://doi.org/10.1016/j.jaerosci.2022.106093

Lee, C.-S. C. S., & Shih, T. I. P. T. I.-P. (2021). Effects of heat loads on flow and heat transfer in the entrance region of a cooling duct with a staggered array of pin fins. *International Journal of Heat and Mass Transfer*, *175*, 121302. https://doi.org/10.1016/j.ijheatmasstransfer.2021.121302

Li, Y., Li, Y., Yuan, S., Wang, X., & Tan, S. K. (2022). PIV measurement of turbulent flow in curved ducts with variable curvature convex wall. *International Journal of Heat and Fluid Flow*, *98*(November 2021), 109074. https://doi.org/10.1016/j.ijheatfluidflow.2022.109074

Liberati, A. (2009). The PRISMA Statement for Reporting Systematic Reviews and Meta-Analyses of Studies That Evaluate Health Care Interventions: Explanation and Elaboration. *Annals of Internal Medicine*, *151*(4), W. https://doi.org/10.7326/0003-4819-151-4-200908180-00136

Liu, M., Yao, J., & Zhao, Y. (2021). Particle dispersion in turbulent sedimentary duct flows. *Advanced Powder Technology*, *32*(11), 4245–4262. https://doi.org/10.1016/j.apt.2021.09.032

Mahmoodi-Jezeh, S. V., & Wang, B. C. (2021). Direct numerical simulation of turbulent heat transfer in a square duct with transverse ribs mounted on one wall. *International Journal of Heat and Fluid Flow*, *89*(May 2020), 108782. https://doi.org/10.1016/j.ijheatfluidflow.2021.108782

Menni, Y., Azzi, A., Chamkha, A. J., & Harman, S. (2019). Analysis of Fluid Dynamics and Heat Transfer in a Rectangular Duct with Staggered Baffles. *JOURNAL OF APPLIED AND COMPUTATIONAL MECHANICS*, *5*(2), 231–248. https://doi.org/10.22055/JACM.2018.26023.1305

Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, *32*(8), 1598–1605. https://doi.org/10.2514/3.12149

Moher, D., Liberati, A., Tetzlaff, J., & Altman, D. G. (2010). Preferred reporting items for systematic reviews and meta-analyses: The PRISMA statement. *International Journal of Surgery*, *8*(5), 336–341. https://doi.org/10.1016/j.ijsu.2010.02.007

Moosa, M., & Hekmat, M. H. (2019). Numerical investigation of turbulence characteristics and upstream disturbance of flow through standard and multi-hole orifice flowmeters. *Flow Measurement and Instrumentation*, *65*, 203–218. https://doi.org/10.1016/j.flowmeasinst.2019.01.002

Nandi, S. kumar S. and N. (2021). Effect of Guide Vane on Turbulence Characteristics for Single-Phase Flow through a 90-Degree Pipe Bend. *Journal of Applied Fluid Mechanics*, *14*(4), 1195–1208. https://doi.org/10.47176/jafm.14.04.32148

Obwogi, E. O., Shen, H. long, & Su, Y. min. (2021). The design and energy saving effect prediction of rudder-bulb-fin device based on CFD and model test. *Applied Ocean Research*, *114*(December 2020), 102814. https://doi.org/10.1016/j.apor.2021.102814

Page, M. J., & Moher, D. (2017). Evaluations of the uptake and impact of the Preferred Reporting Items for Systematic reviews and Meta-Analyses (PRISMA) Statement and extensions: A scoping review. *Systematic Reviews*, *6*(1), 1–14. https://doi.org/10.1186/s13643-017-0663-8

Park, H., & Bach, C. K. (2021). Performance characterization of air mixing devices for square ducts. *Applied Thermal Engineering*, *199*(August), 117495. https://doi.org/10.1016/j.applthermaleng.2021.117495

Reghunathan Valsala, R., Son, S. W., Suryan, A., & Kim, H. D. (2019). Study on reduction in pressure losses in pipe bends using guide vanes. *Journal of Visualization*, *22*(4), 795–807. https://doi.org/10.1007/s12650-019-00561-w

Spanelis, A., & Walker, A. D. (2022). A Multi-Objective Factorial Design Methodology for Aerodynamic Off-Takes and Ducts. *Aerospace*, *9*(3), 130. https://doi.org/10.3390/aerospace9030130

Tas-Koehler, S., Neumann-Kipping, M., Liao, Y., Bieberle, A., & Hampel, U. (2022). Experimental Investigations and Numerical Assessment of Liquid Velocity Profiles and Turbulence for Single- and Two-phase Flow in a Constricted Vertical Pipe. *International Journal of Multiphase Flow*, *157*, 104224. https://doi.org/10.1016/j.ijmultiphaseflow.2022.104224

Vadrot, A., Giauque, A., & Corre, C. (2020). Analysis of turbulence characteristics in a temporal dense gas compressible mixing layer using direct numerical simulation. *Journal of Fluid Mechanics*, *893*, A10. https://doi.org/10.1017/jfm.2020.218

Wang, G., Yang, F., Wu, K., Ma, Y., Peng, C., Liu, T., & Wang, L. P. (2021). Estimation of the dissipation rate of turbulent kinetic energy: A review. *Chemical Engineering*

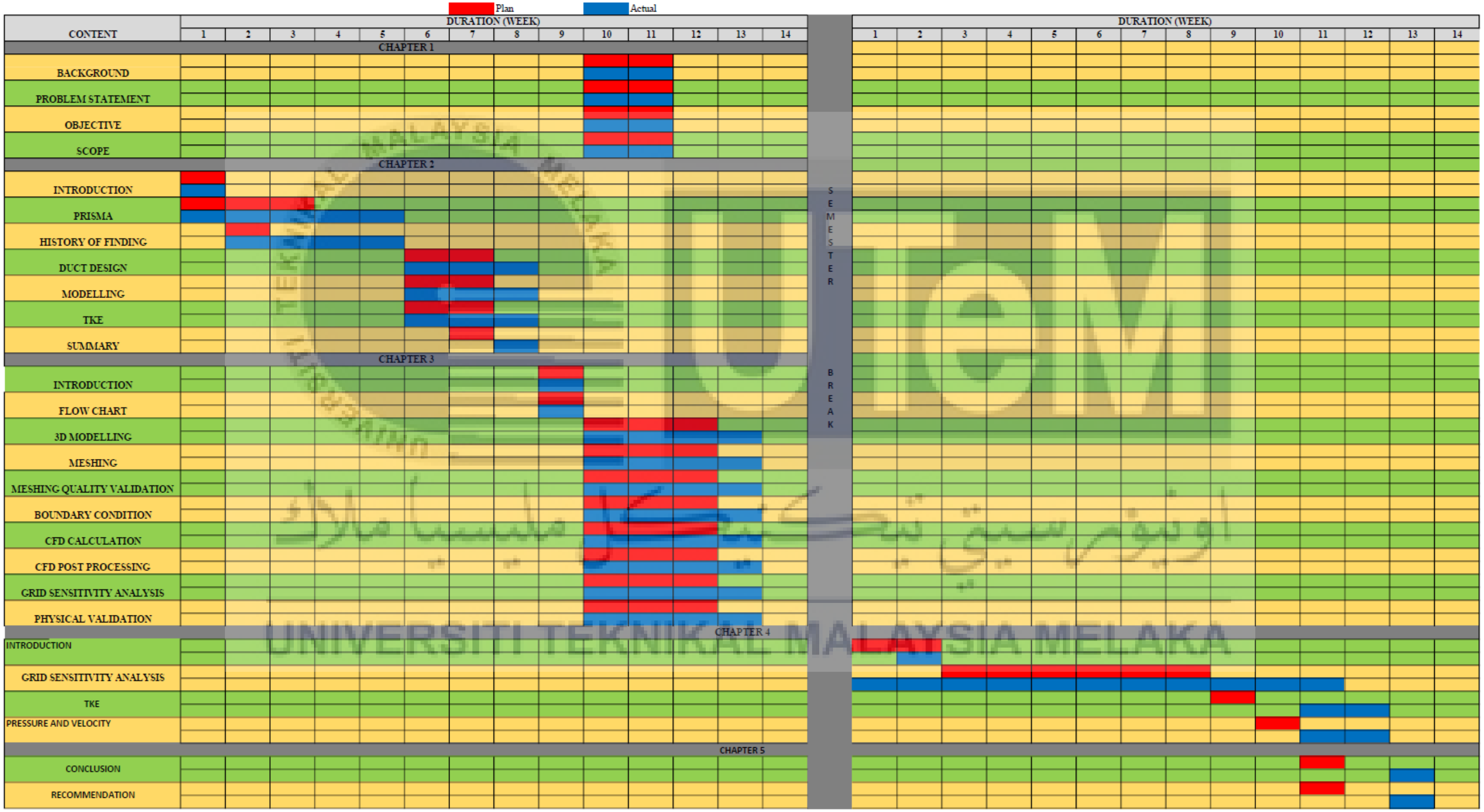
Science, *229*. https://doi.org/10.1016/j.ces.2020.116133

Wang, S., Fu, X., & Deng, X. (2022). Higher-order aerodynamic numerical simulations in compressible RANS framework with inverse-ω scale variable. *Aerospace Science and Technology*, *131*(A), 107971. https://doi.org/10.1016/j.ast.2022.107971

Zhang, J., Li, A., Altwieb, M., Kubiak, K. J., Aliyu, A. M., & Mishra, R. (2008). CFD simulation of particle deposition in a horizontal turbulent duct flow. *Chemical Engineering Research and Design*, *86*(1 A), 95–106. https://doi.org/10.1016/j.cherd.2007.10.014

Zhao, Y., Liu, M., Li, J., Yan, Y., & Yao, J. (2022). Modulation of turbulence by dispersed charged particles in pipe flow. *Physics of Fluids*, *34*(12), 123315. https://doi.org/10.1063/5.0130487

APPENDICES



Appendix 1 Gantt chart