DOKUZ EYLÜL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

DESIGN AND ANALYSIS OF A QUADCOPTER
ACTUATED BY BLADELESS FAN

by
Doğukan AKGÖL

July, 2022
İZMİR
DESIGN AND ANALYSIS OF A QUADCOPTER
ACTUATED BY BLADELESS FAN

A Thesis Submitted to the
Graduate School of Natural and Applied Sciences of Dokuz Eylül University
In Partial Fulfillment of the Requirements for the Degree of Master of Science
in Mechanical Engineering, Machine Theory and Dynamics Program

by
Doğukan AKGÖL

July, 2022
İZMİR
M.Sc THESIS EXAMINATION RESULT FORM

We have read the thesis entitled “DESIGN AND ANALYSIS OF A QUADCOPTER ACTUATED BY BLADELESS FAN” completed by DOĞUKAN AKGÖL under supervision of ASSOC. PROF. DR. ŞAHİN YAVUZ and we certify that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Assoc. Prof. Dr. Şahin YAVUZ
Supervisor

Prof. Dr. Zeki KIRAL
(Jury Member)

Assoc. Prof. Dr. Sercan ACARER
(Jury Member)

Prof. Dr. Okan FISTIKOĞLU
Director
Graduate School of Natural and Applied Sciences
ACKNOWLEDGMENTS

I would like to express my respect and gratitude to my mentor, Assoc. Prof. Dr. Şahin YAVUZ, who was by my side throughout my research process and did not spare me his support and experience and added meaning to my work.

I owe thanks to Assoc. Prof. Dr. Sercan ACARER for his valuable help for this study.

Doğukan AKGÖL
DESIGN AND ANALYSIS OF A QUADCOPTER ACTUATED BY
BLADELESS FAN

ABSTRACT

Unmanned Aerial Systems (UASs) have a variety of applications in our daily life that have attracted the attention of many researchers around the world. There are variety of innovations in the flight mechanisms that UASs are applying for flight. To this end, new concepts for different environments are being developed. Each of these UASs exhibits certain advantages and disadvantages for deployment in particular missions.

The main goal of this research is to design and manufacture a new concept that has bladeless thrusters in place of conventional propellers. The motivation behind this design is to eliminate the harmful effects of conventional multirotor UAVs with exposed propellers for people and other living environments. This new design idea with bladeless propulsion system was inspired by the commercial bladeless fan concepts. This research aims to concentrate on optimal design of the bladeless thruster cross-section geometry and airflow properties to get sufficient thrust for flight. Final thruster design is manufactured with a 3D printer.

Keywords: Bladeless thruster, bladeless fan, safe drone design, quadcopter design
PERVANESİZ FAN TAHRİKLİ BİR QUADCOPTERİN TASARIM VE ANALİZİ

ÖZ

İnsansız Hava Sistemleri (İHS), günlük hayatımızda; dünya çapında birçok araştırmacının dikkatini çeken çeşitli uygulamaları sahiptir. İnsansız Hava Sistemlerinde kullanılan uçuş mekanizmalarında çeşitli yenilikler bulunmaktadır. Bu amaçla, farklı ortamlar için yeni konseptler geliştirilmektedir. Bu İHS’lerin her biri, belirli görevler için avantajlar ve dezavantajlar sergiler. Bu araştırmanın temel amacı, geleneksel pervaneler yerine pervanesiz iticilere sahip yeni bir konsept tasarlamak ve üretmektir.

Bu tasarımın arkasındaki motivasyon, insanlar ve diğer yaşam ortamları için açık pervaneli, geleneksel çok rotorlu insansız hava araçlarının zararlı etkilerini ortadan kaldırmaktır. Kanatsız tahrik sistemine sahip bu yeni tasarım fikri, ticari pervanesiz fan konseptlerinden ilham alınmıştır. Bu araştırma, uçus için yeterli itme kuvveti elde etmek için kanatsız fan kesit geometrisi ve hava akışı özelliklerinin optimal tasarımına odaklanmayı amaçlamaktadır. Nihai itici tasarımını 3D yazıcı ile üretmiştir.

Anahtar kelimeler: Pervanesiz itici, pervanesiz fan, güvenli drone tasarım, quadcopter tasarım
# CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>M.Sc TESIS EXAMINATION RESULT FORM</td>
<td>ii</td>
</tr>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td>iii</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>iv</td>
</tr>
<tr>
<td>ÖZ</td>
<td>v</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>viii</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>x</td>
</tr>
</tbody>
</table>

## CHAPTER ONE - INTRODUCTION ........................................ 1

1.1 Introduction ........................................................................ 1
1.2 Literature Review .......................................................... 1
1.3 Scope of the Thesis .......................................................... 4
1.4 Organisation of the Thesis ................................................ 5

## CHAPTER TWO - BLADELESS THRUSTER MODEL DESIGN .................. 6

2.1 Introduction ........................................................................ 6
2.2 Bladeless Fan Model Design ............................................... 7
  2.2.1 Bernoulli’s Equation for Steady Flow ............................. 7
  2.2.2 The Coanda Effect ........................................................ 9
  2.2.3 Determination of Design Parameters ............................... 11

## CHAPTER THREE - THREE-DIMENSIONAL FLOW ANALYSES ............ 16

3.1 Introduction ........................................................................ 16
3.2 Creating Flow Models for Bladeless Thruster ........................ 16
3.3 The Coanda Surface Curvature Effect ................................... 21
  3.3.1 Boundary Conditions .................................................... 22
  3.3.2 Mesh Independence ....................................................... 23
3.4 Three-Dimensional Flow Simulation Results ........................ 25
3.5 Conclusion ........................................................................... 31
CHAPTER FOUR - TWO-DIMENSIONAL FLOW ANALYSES.................... 33

4.1 Introduction .................................................................................. 33
3.2 Design of the Fluid Domain and the Thruster Model ................... 33
4.3 Mesh Domain Creation ................................................................... 35
   4.3.1 Examination of the Mesh Quality ........................................... 36
4.4 Determination of Boundary Name Selections ................................. 40
4.5 The CFD Model Construction ....................................................... 40
   4.5.1 Boundary Conditions .......................................................... 43
4.6 Simulation Results ......................................................................... 44
4.6 Conclusion .................................................................................... 52

CHAPTER FIVE - PROTOTYPE PRODUCTION ....................................... 54

5.1 Production of Test Structure ......................................................... 54

CHAPTER SIX - CONCLUSION .......................................................... 56

REFERENCES .................................................................................. 58
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>First ever UAV attack in Venice, 1849</td>
<td>2</td>
</tr>
<tr>
<td>2.1</td>
<td>The fluid particle moving along a streamline</td>
<td>8</td>
</tr>
<tr>
<td>2.2</td>
<td>The flow over a convex surface</td>
<td>10</td>
</tr>
<tr>
<td>2.3</td>
<td>Air flow inducement of the bladeless fan</td>
<td>11</td>
</tr>
<tr>
<td>2.4</td>
<td>Bladeless fan CAD model</td>
<td>12</td>
</tr>
<tr>
<td>2.5</td>
<td>Cross-sectional parameters of bladeless fan considered in design</td>
<td>13</td>
</tr>
<tr>
<td>2.6</td>
<td>Air flow distribution for the first model designed with single inlet region</td>
<td>14</td>
</tr>
<tr>
<td>2.7</td>
<td>Ideal thrust force direction and bladeless fan axis position</td>
<td>15</td>
</tr>
<tr>
<td>3.1</td>
<td>Chaotic motion of the fluid named turbulent flow</td>
<td>17</td>
</tr>
<tr>
<td>3.2</td>
<td>Turbulence models provided by ANSYS-Fluent</td>
<td>18</td>
</tr>
<tr>
<td>3.3</td>
<td>The turbulence model used on first bladeless fan CFD simulation</td>
<td>19</td>
</tr>
<tr>
<td>3.4</td>
<td>Airflow streamline view for the second model which has two symmetrical</td>
<td>20</td>
</tr>
<tr>
<td></td>
<td>inlets</td>
<td></td>
</tr>
<tr>
<td>3.5</td>
<td>Four main thruster models to observe Coanda surface curvature effect on</td>
<td>22</td>
</tr>
<tr>
<td></td>
<td>average outlet velocity. (a) Model 1, (b) Model 2, (c) Model 3 (d) Model 4</td>
<td></td>
</tr>
<tr>
<td>3.6</td>
<td>Mesh independence study results for eight different mesh</td>
<td>24</td>
</tr>
<tr>
<td>3.7</td>
<td>(a) Velocity contour view inside the thruster ring. (b) Airflow streamlines</td>
<td>25</td>
</tr>
<tr>
<td></td>
<td>for Model 1</td>
<td></td>
</tr>
<tr>
<td>3.8</td>
<td>Area-weighted average of the $y^+$ values for Model 1</td>
<td>26</td>
</tr>
<tr>
<td>3.9</td>
<td>The $y^+$ value distribution for Model 1 wall regions</td>
<td>27</td>
</tr>
<tr>
<td>3.10</td>
<td>Wall functions for the k-ε turbulence model</td>
<td>27</td>
</tr>
<tr>
<td>3.11</td>
<td>(a) Viscous sublayer resolving approach to decide boundary layer which</td>
<td>28</td>
</tr>
<tr>
<td></td>
<td>is shown in red line. (b) Wall functions to resolve the boundary layer</td>
<td></td>
</tr>
<tr>
<td>3.12</td>
<td>(a) Velocity distribution inside the thruster in contour view. (b)</td>
<td>29</td>
</tr>
<tr>
<td></td>
<td>Streamline view of the air inside the thruster</td>
<td></td>
</tr>
<tr>
<td>3.13</td>
<td>(a) Simulation results for Model 3. (b) Simulation results for Model 4</td>
<td>30</td>
</tr>
<tr>
<td>4.1</td>
<td>Computational domain and 2D thruster model that designed for the flow</td>
<td>33</td>
</tr>
<tr>
<td></td>
<td>simulations</td>
<td></td>
</tr>
</tbody>
</table>
Figure 4.2 Three-dimensional model view of the thruster model which is created to use in two dimensional analyses ................................................................. 35
Figure 4.3 Mesh view of the thruster model ............................................................... 35
Figure 4.4 Inflation layer parameters ..................................................................... 36
Figure 4.5 Skewness of the mesh element .............................................................. 37
Figure 4.6 Mesh elements with close to 1 and large aspect ratios ............................ 38
Figure 4.7 Calculation of the orthogonal quality of a mesh cell ............................. 39
Figure 4.8 Boundary name selections to specify boundary conditions on CFD simulations ................................................................................................... 40
Figure 4.9 Solver settings for the 2D flow simulations ............................................ 41
Figure 4.10 The selected turbulence model SST $k$-$w$ settings .............................. 42
Figure 4.11 The inlet boundary condition specifications ....................................... 43
Figure 4.12 The frictionless wall boundary specifications ...................................... 44
Figure 4.13 The volumetric flow rates for inlet (black line) and vfr_back (green line) surfaces ........................................................................................................ 45
Figure 4.14 The residuals graph for the simulation ................................................... 45
Figure 4.15 The $y^+$ distribution for the Model 5 ................................................... 46
Figure 4.16 Near-wall velocity profiles .................................................................. 46
Figure 4.17 The static pressure contour of the bladeless thruster ............................ 47
Figure 4.18 The velocity contour of the bladeless thruster ..................................... 48
Figure 4.19 The Coanda surface curvature difference between (a) Model 1 and (b) Model 5 ........................................................................................................ 48
Figure 4.20 The change in the volumetric flow rate at the vfr_back surface by the Coanda surface curvature ........................................................................... 50
Figure 4.21 The static pressure (a) and velocity distribution (b) for the Model 5 ..... 51
Figure 4.22 The compressible, ideal gas model simulation result ............................ 52
Figure 5.1 The first thruster model (a) Perspective view (b) Side view (c) Top view ................................................................................................................. 54
Figure 5.2 The test structure with four identical bladeless thrusters ....................... 55
LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 3.1</td>
<td>Cross-sectional dimensions table of all four models</td>
<td>22</td>
</tr>
<tr>
<td>Table 3.2</td>
<td>The results obtained from mesh independence analysis of Model 2</td>
<td>23</td>
</tr>
<tr>
<td>Table 4.1</td>
<td>Minimum, maximum and average values of mesh metrics considered</td>
<td>39</td>
</tr>
<tr>
<td>Table 4.2</td>
<td>The parametric analysis result of the Coanda surface curvature</td>
<td>49</td>
</tr>
<tr>
<td>Table 4.3</td>
<td>The parametric analysis result of the Coanda surface curvature</td>
<td>52</td>
</tr>
</tbody>
</table>
CHAPTER ONE
INTRODUCTION

1.1 Introduction

Unmanned Aerial Systems (UASs) have a wide range of uses in our daily lives that have attracted the interest of many experts worldwide. The flying mechanisms that UASs use for flight have seen several advancements. There is also considerable interest in the creation of new types of drones that can fly independently in a variety of environments, including cities, oceans, and space, and execute a variety of missions. In order to expand the areas where unmanned aerial vehicles will be used and to increase their qualifications, studies are carried out in many areas. In order to diversify the usage areas of unmanned aerial vehicles, areas such as battery life, ensuring safe flight and efficiency are prioritised, and various researchers are working on improvements in these areas.

Increasing efficiency and safety for UAVs is the main purpose of this thesis. In order to ensure these goals, a new concept of thruster system is designed and analysed in this research. The bladeless thruster concept is replaced with traditional propellers of drones to obtain more thrust with the same power consumption. Furthermore, this bladeless concept allows the air to be increased thanks to its special design. By using the Coanda effect, it also induces the stagnant air in its outer part to create a higher air flow than traditional propeller systems, thus increasing the thrust force. In this way, it increases the efficiency of the propulsion systems of the UAVs.

1.2 Literature Review

After successful trials in Vienna by Franz Uchatius, the designer of unmanned balloon bombs, the first unmanned air vehicle strike took place on July 2nd, 1849 (see Figure 1.1). Unmanned aerial vehicles have since begun to take on essential jobs in a variety of fields, solidifying their role in our lives.
Julius Gustav Neubranner invented and built a breast-mounted aerial camera for photography that was carried by pigeons in 1903 (Rambat, 2012). These technologies were later replaced by aircraft for aerial photography due to advancements in the aviation industry.

Following the deployment of the first military satellites into orbit in 1960 (see Figure 1.2), and the development of more sensitive sensors and high-resolution cameras, the demand for satellites for aerial surveillance grew (Jafari et al., 2016).

UASs can be outfitted with a variety of sensors and cameras, allowing them to carry out missions both outdoors and indoors in extreme conditions. The tasks, flight zones and habitats of these autonomous systems can all be classified (Darvishpoor et al., 2020). Drones have been employed in operations such as search and rescue, delivery, environmental protection, and undersea and planetary research in a variety of areas.

As the usage areas of drones expand, our expectations from them also increase. For this reason, many researchers are working on improved new designs. The increase in
interaction with the environment creates dangers due to the exposed drone propellers. Rones & Hoff (2017) created and prototyped a quadrotor with a ducted propeller configuration. In this configuration four propellers of the quadrotor are niched inside a cage to avoid any dangerous interaction with the environment. At any time of contact, the drone's propellers may be damaged, resulting in the termination of the drone's function and harming the other object or living thing. Valdenegro et al. (2018) proposed a safer design of an autonomous multicopter with four bladeless thrusters with four Electric Ducted Fans (EDF) attached to draw air to the bladeless thruster ring. Valdenegro et al. (2018) observed some inefficiencies due to pressure losses which are observed through the generated back-flows at the wall and tip of the EDF. These back-flows created resistance to the electric motor and caused overload on the motor.

In order to make bladeless fans more efficient many studies are done by researchers. Li et al. (2014) examined the effect of the Coanda surface on the performance of the commercial bladeless fans. This research uses numerical studies to get an understanding of the effect of curvature on the performance of bladeless fans. ANSYS-Fluent software is used to do three-dimensional numerical simulations. Two-dimensional numerical simulation is also used to gain precise information on the flow field around the Coanda surface. Li et al. (2014) suggested that there is an optimal curvature interval between 0.026 - 0.065 in Y direction which is the most suitable curvature range for the flow over the Coanda surface. Li et al. (2014) also proposed that with the increasing curvature, several low-pressure regions emerge. This large area of low-pressure regions may cause the detaching of the flow over the Coanda surface and make the flow over the wall unstable. The greater curvature on the Coanda surface creates greater angle between air velocity vector at the nozzle outlet and the bladeless thruster angle and this may cause detaching from the fan wall.

Geometrical parameters have a great influence on the performance and efficiency of the bladeless fans. Jafari et al. (2016) used an Eppler 473 airfoil profile in order to design a bladeless fan model. Five different cross-sectional parameters are determined such as thickness of the nozzle outlet, hydraulic diameter of the bladeless fan, aspect ratio for cross-sections, height of the fan from floor and angle between outlet flow and
fan axis. Jafari et al. indicated that for a height of 3 cm, the discharge ratio of the fan had the highest value. The numerical results of outlet thickness change showed that this is one of the most important criteria for a bladeless fan's aerodynamic performance. The discharge ratio increased dramatically as the outlet thickness decreased for outlet thicknesses of 1, 2, and 3 mm, according to the results.

Drăgan (2012) used a series of mathematical equations to precisely represent the velocity field around a convex surface which can be used to examine velocity profile on the Coanda surface. It may be possible to predict the velocity distribution around the Coanda surface using this set of equations. The Coanda effect is directly proportional to the velocity of the fluid jet and the curvature of the convex wall. Mehmood et al. (2022) conducted a study using a bladeless annular ceiling fan that was computationally investigated in a standard room (4m x 4m x 4m). Average velocities were calculated under the influence of different design features such as fan radius, height from floor, fan jet distance and mass flow rate, and the impact of these parameters on perceived comfort in terms of velocity dispersion were studied parametrically. This study looked at the health consequences of air flow characteristics in commercial bladeless fan devices that were utilised for ventilation in domestic spaces such as the home or office.

1.3 Scope of the Thesis

Today, the use of drones is getting more and more common. It helps people in many fields, especially in the logistics sector. In addition to this, individual use for hobby purposes has also become very common. Therefore, the safety of drones has started to be discussed due to these intense interactions with humans and living environments. Rotating propellers in the open have the potential to harm living things and cause the drone to fail to fulfill its function at any time of contact. In addition, due to the short stay in the air and the limitations in the battery life, it causes the drones to not show their full potential yet.

The main purpose of this thesis is to design a new propulsion system that will eliminate the disadvantages of unmanned aerial vehicle systems such as battery life
and security, and make the propulsion units of these systems more efficient and powerful. For this purpose, flow analyzes were carried out on a new fan system designed without a fan. Parametric designs were created by examining the effect of the determined parameters on the fan thrust power. An optimum fan design was obtained under the guidance of the findings, and this bladeless fan was produced using 3D printing technology and a prototype drone system was created.

1.4 Organisation of the Thesis

This thesis consists of four chapters to indicate the design steps of a bladeless thruster to use in the place of exposed propeller systems. Firstly, Chapter 1 presents a literature survey of Unmanned Aerial Systems (UASs) and bladeless fan technology. Then in Chapter 2 a parametrical 3D design of the bladeless thruster is performed. In Chapter 3 and Chapter 4, three dimensional and two-dimensional flow analyses carried out to observe the effect of some design parameters such as the Coanda surface and nozzle outlet gap on the thrust performance of the bladeless thruster. Finally, in Chapter 5, the conclusions for the bladeless thruster design are presented.
CHAPTER TWO
BLADELESS THRUSTER MODEL DESIGN

2.1 Introduction

After the increasing importance and demand of Unmanned Aerial Vehicle (UAV) systems, design considerations and expectations from these systems have also increased. Conventional UAV systems gained popularity at every part of our daily life. The main weaknesses of these aerial systems are low battery life and limited load carrying capabilities. Recent application areas of UAV systems are mainly imaging and delivery operations. Flight time and carrying capacity of an UAV are important parameters for these types of tasks. Besides that, conventional UAV systems have hazards to surrounding such as human beings or animals created by exposed propellers. These exposed propellers on conventional type UAVs rotate at high speed and sharp edges of these propellers may cause injuries at contact with the living environment. These interactions also may cause damage on UAVs and become unable to perform their quest and functions unexpectedly.

Nowadays, it is of great importance to review and revise the propulsion systems as well as the development of other components such as battery systems or flight controllers to make UAVs more efficient and safer. Increasing efficiency comes with better flight times and more carrying capacity. These developments will allow UAV systems to take their place in various areas of our lives and make some processes easier and fast.

The main purpose of this study is to eliminate disadvantages of conventional UAV systems and make them more efficient and safer. New design idea behind this research is replacing exposed propulsion systems especially propellers with bladeless propulsion design. Concept of bladeless thruster system minimising the contact between UAV propeller systems with the external environment and besides, providing more efficient thrust under favour of special geometry of bladeless fan.

In recent years, bladeless fan technologies are widely used in domestic areas such as home or office for cooling and ventilation. Bladeless fan designs gaining popularity
because of their safer designs by means of no exposed propellers to the environment and blowing performance on ventilation processes. These fans are also easy to clean and thanks to the insulated propellers from surroundings are safer for the people compared to traditional fans. By the reason of special geometry of the bladeless fan inlet air increases at the outlet in some way of the nozzle geometry. The air sucked in with the help of an impeller that is insulated from the surrounding area leaves the fan at high speed after circulating in the specially designed cross-section. At the outlet air leaves the fan from a very small slit and creates high speed air flow. Therefore, bladeless fans are also called as air multipliers or air accelerators.

Compared to traditional radial or axial fans, bladeless fans stated in other words air multipliers have some advantages especially such as aerodynamic efficiency and security concerns. Depending on the design parameters these fans can accelerate the air flow at its inlet up to 15 times (Li et al., 2014). High velocity flow at the outlet section of the fan creates pressure difference between stagnated air and the fan centre and this pressure difference induces a passive air flow from surrounding to the back of the fan. Since there is no propeller in the middle of the fan, this induced air flow can be directly transferred to the back of the fan, thereby increasing the total air flow.

The key factors that cause people to prefer bladeless fans are doubling intake air flow and lack of an exposed impeller or propeller. Bladeless fans are gaining popularity and led to an increase in the related studies. Increase in popularity of the bladeless fans also causes innovations and scientific studies on bladeless fans to rise.

2.2 Bladeless Fan Model Design

2.2.1 Bernoulli’s Equation for Steady Flow

Bernoulli’s principle states that an increase in the speed of the fluid particle change in concert with the decrease in the static pressure. The potential energy of the fluid decreases with the decrease in the static pressure and this energy converts into kinetic energy and the velocity of the fluid increases. This principle is applicable for only isentropic flows. The effects of irreversibility (such as turbulence) and non-adiabatic processes are small and can be neglected for isentropic flows. Bernoulli’s equation for fluid flow for any point along a streamline is given in Equation 2.1.
\[ \frac{V^2}{2} + \frac{p}{\rho} + g\zeta = \text{constant} \]  

Equation 2.1

The terms of Equation 2.1 are velocity head \( \frac{V^2}{2} \), pressure head \( \frac{p}{\rho} \) and elevation head \( g\zeta \), respectively. In this equation dynamic pressure of the flow can be written as the half of the square of the stream velocity of the particle. The static pressure of the fluid particle represented in pressure head, \( \frac{p}{\rho} \). The last term of the Bernoulli equation is the pressure of the particle that arises due to the elevation difference in the flow stream, \( g\zeta \). Sum of all terms gives the total head of the flow which is constant for all points at the same streamline. The pressure variation along a single streamline can be studied thanks to the consistency of the total head for inviscid flow along a streamline. A fluid particle moving along a streamline, at time \( t \), the particle is at point \( P \). Distance, \( s \), is measured along the streamline and is positive in the streamwise direction, as shown in Figure 2.1

![Figure 2.1](image)

Figure 2.1 The fluid particle moving along a streamline (Katopodes, 2019)

The natural coordinate system \((s, n)\) is oriented along and normal to the flow stream in two space dimensions. If viscous stresses are ignored, the only forces operating on an elementary stream-tube with a cross-sectional area equal to \( dA \) are pressure differences and the fluid weight component along the streamline. We can write, in \( s \) coordinate direction
\[- \frac{\partial P}{\partial s} - \gamma \frac{\partial \xi}{\partial s} = \rho V_s \frac{\partial V_s}{\partial s} \]  

(2.2)

where $V_s$ is the velocity component in the direction of the $s$ coordinate. Since the velocity magnitude in normal direction $n$ is zero, $V_s$ is equal to the velocity vector magnitude $|V|$. Therefore, dividing the Equation 2.2 by $\gamma$, we obtain,

\[- \frac{\partial}{\partial s} \left( \frac{P}{\gamma} + \xi + \frac{|V|^2}{2g} \right) = 0 \]  

(2.3)

It asserts that throughout the streamline, the total energy per unit weight or head remains constant. Similarly, if we add the forces normal to the streamline, we get

\[- \frac{\partial P}{\partial n} - \gamma \frac{\partial \xi}{\partial n} = \rho \frac{|V|^2}{r} \]  

(2.4)

This is an important finding with far-reaching implications for Bernoulli’s equation application. Because the total head remains constant not just along a single streamline, but across the whole flow segment, assuming the streamlines have no significant curvature.

2.2.2 The Coanda Effect

The pressure falls as the radial distance from the centre of curvature increases in the presence of streamline curvature. The Coanda effect, named after Henri Marie Coanda (1885–1972), a Romanian aeronautical engineer who made substantial contributions to aircraft technology, is a powerful manifestation of this in various forms of environmental flow.

As shown in Figure 2.2, when a fluid jet goes out from a small slit, ambient air is entrained into the jet from all sides as it emerges to open air. When a curvilinear wall is placed next to one side of the jet, however, entrainment is reduced on that side, pressure is reduced due to the narrowing of the route, and the jet is diverted towards the wall. When fluid flows over a sloping roof and adheres to the surface of a
moderately sloping gutter, the Coanda Effect happens naturally. When a fluid is poured out of a bottle or other container, the impact can be seen.

![Diagram of Curved Wall and Jet](image)

Figure 2.2 The flow over a convex surface (Katopodes, 2019)

The Coanda effect is widely used in aerodynamics, where a jet of air blown across the upper surface of an airfoil, for example, can have a significant impact on lift, particularly at high angles of attack.

Coanda Effect is one of the key factors behind the design of the bladeless fan. The Coanda phenomenon describes the deflection of a fluid jet on a convex surface. Since a fluid jet changes direction before detaching from the convex surface which it flows through, it creates a low-pressure zone on the convex surface, resulting in a force nearly equivalent to twice the jet momentum at the nozzle outlet (Han et al., 2005). The blowing wind effect has been proved to be the most efficient in terms of bladeless fans, based on the crucial factor of the Coanda surface. The Coanda effect, which is named also as wall attachment effect, is the main theoretical foundation for flow multiplier. The air multiplication of bladeless fans based on the Coanda effect phenomenon is shown in Figure 2.3.
Several studies in several fields of research have focused on the Coanda effect. Drăgan (2012) described the velocity field accurately near a convex surface by using a set of mathematical equations. By using this set of equations, it can be possible to estimate the velocity distribution near the Coanda surface. Coanda effect directly related to fluid jet velocity and convex wall curvature.

2.2.3 Determination of Design Parameters

To date various analyses have been performed and introduced to reveal the performance of bladeless fans. Mehmood et al. (2022) proposed a study with a bladeless ceiling fan of the annular kind and computationally examined in a conventional empty room (4m x 4m x 4m) environment. Computed average velocities under the influence of different design features such as fan radius, height from the ceiling and floor, fan jet breadth, mass flow rate, and orientations analysed
parametrically, and the impact of these parameters on perceived comfort in terms of velocity dispersion. This study related to commercial bladeless fan products which were used for ventilation purposes in domestic areas such as home or office and examined the health effects of airflow characteristics.

Some studies performed to investigate the noise levels and aerodynamic evaluation of a bladeless fan. Jafari et al. (2016) carried out numerical investigation with Finite Volume Method for commercial bladeless fan model. As a cross-section profile Eppler 473 airfoil was used for the fan. By solving momentum, continuity and noise equations, performance and noise level of the bladeless fan were studied.

In this study, several design parameters were determined and the fan geometry was tried to be optimised to use as a thruster for bladeless quadcopter. In Figure 2.4. The bladeless fan was designed to obtain a significant thrust level which is appropriate for flight of a quadcopter with four identical bladeless fans. The aims of the designs were to obtain maximum average nozzle outlet velocity and to reduce inefficiencies due to turbulent flow inside the fan ring.
Fan model cross-sectional area designed with several parameters (see Figure 2.5) such as Coanda surface curvature, cross-section thickness and nozzle outlet gap. These parameters are the main parameters that affect average nozzle outlet velocity which is the main objective of the design process to maximise produced thrust. Other cross-sectional design parameters such as height of the fan, high flow passageway and inner leading edge are important in terms of transmitting the flow inside the fan to the outlet as without turbulence as possible.

![Cross-sectional parameters of bladeless fan considered in design](image)

**Figure 2.5** Cross-sectional parameters of bladeless fan considered in design

Firstly, bladeless fan models designed with a single inlet region where the air sucked from outside is transferred to the fan, is of great importance in the homogeneity of the outlet velocity. Laminarity and homogeneity of the air at the outlet of the fan are key properties in order to obtain sufficient thrust.

In the first analysis carried out to observe the internal flow behaviour, it has observed that the single inlet zone was insufficient to distribute the flow homogeneously inside the fan. For this reason, two separate inlets have been designed to give the air sucked from both sides of the fan at the same time (see Figure 2.6).
As can be clearly seen from the Figure 2.6 the air flow is not circulating homogeneously inside the fan and additionally, too many vortexes emerging inside which affects the flow efficiency and decreases output air velocity. Furthermore, it can also be observed that not enough air flow is reaching the distant part of the fan farthest from the inlet region.

However, the second model designed after the initial simulations has two inlet parts that have more homogeneous air flow inside the fan. Moreover, vortexes inside the bladeless fan decreased significantly and more powerful air flow produced at the outlet nozzle which is indispensable to obtain thrust. Homogenous and laminar flow at the outlet of the bladeless fan is necessary for balanced flight of the quadcopter. Also, necessary to obtain stable thrust force which is always parallel to the fan axis, as seen in Figure 2.7.
Figure 2.7 Ideal thrust force direction and bladeless fan axis position
CHAPTER THREE
THREE-DIMENSIONAL FLOW ANALYSES

3.1 Introduction

Fluctuating velocity fields describes turbulent flows. These changes mix transported quantities such as velocity, energy, and species concentration, causing them to fluctuate as well. These oscillations are too computationally expensive to mimic directly in actual engineering calculations due to their small scale and high frequency. Instead, the exact governing equations can be time-averaged, ensemble-averaged, or otherwise adjusted to reduce small scale resolution, resulting in a modified set of equations that are computationally less expensive to solve. The amended equations, however, contain extra unknown variables, which must be determined using turbulence models in terms of known quantities.

In this chapter, the flow behaviour of the bladeless fan at the inside region and at the outside flow field have been examined and design evaluation steps have been explained. For the purpose of modelling this turbulent regime of the bladeless fan ANSYS-Fluent has been used and CFD simulations carried out.

3.2 Creating Flow Models for Bladeless Thruster

Initially, turbulence was described as an uneven motion in fluids (see Figure 3.1). The flowing water in rivers and the creation of clouds in the atmosphere demonstrate this idea. From the blood that flows through our veins and arteries to the motion of air within our lungs and around us, from the flow of water in streams to atmospheric and oceanic currents, the great majority of flows in nature and engineering applications are somehow turbulent.
In turbulent flow modelling various turbulence models are used for CFD simulations, as seen in Figure 3.2. Regrettably, no single turbulence model is universally regarded as superior for all types of turbulence problems. The physics of the flow, the established practice for a given type of problem, the level of precision required, the computational resources available and the amount of time available for the simulation will all influence the turbulence model selection. Understanding capabilities and the limitations is necessary to select the most appropriate turbulence model.
Primarily, a bladeless fan model with a single inlet hole, which is the inlet region of the surrounding air sucked by an impeller, was designed. The inlet air flow with the velocity of 40 m/s was filled into the bladeless fan ring cross-section from this single hole. The Realizable $k$-$\varepsilon$ turbulence model was used to model the turbulent flow inside the bladeless fan (see Figure 3.3). The Realizable $k$- model proposed by Shih et al. (1995) with a new realizable eddy viscosity formulation. This turbulence model differs from the Standard $k$-$\varepsilon$ in two substantial ways: A different formulation for turbulent viscosity is included in the Realizable $k$-$\varepsilon$ model and from an accurate equation for the transport of the mean-square vorticity fluctuation, a modified transport equation for the dissipation rate, $\varepsilon$, has been developed. The term "realizable" refers to the model's
ability to satisfy specific mathematical limits on Reynolds stresses that are consistent with turbulent flow physics. The *Realizable k-ε* model has the immediate benefit of more precisely predicting the spreading rate of both planar and round jets. It's also likely to outperform other flow types, such as rotation, boundary layers under high pressure gradients, separation, and recirculation.

Figure 3.3 The turbulence model used on first bladeless fan CFD simulation

However, as a result of the very first simulations carried out to observe flow behaviour of the air, it was seen that the air flow is creating vortexes inside the fan and not circulating inside homogeneously. This behaviour causes the air to leave the cross-section unevenly as mentioned in the previous chapter. As a result of the analysis
carried out for the first fan model with a single inlet, 143.039 m/s average velocity obtained at the nozzle outlet. It was determined that, in order to obtain a more balanced distribution, bladeless fan models need to have one more inlet placed symmetrically with the first inlet after the first CFD simulation. Inlet air velocity condition separated into two identical inlets with the air speed of 20 m/s for each. As shown in Figure 3.4, the air flow is distributed homogeneously through the cross-section of the bladeless fan.

![Airflow streamline view for the second model which has two symmetrical inlets](image)

All CFD simulations carried out with the second model, the Realizable \( k-\varepsilon \) turbulence model with an enhanced wall treatment method near the wall regions. Enhanced wall treatment for the \( \varepsilon \) - equation is a near-wall modelling technique that combines a two-layer model with enhanced wall functions. The improved wall treatment will be identical to the classic two-layer zonal model if the near wall mesh (inflation layer mesh) is fine enough to resolve the viscous sublayer (usually first layer element node located at \( y^+ \approx 1 \)). This wall treatment method allows the application of the model independent of the wall \( y^+ \) resolution. Hence, the Realizable \( k-\varepsilon \) model can be suitably used in 3D CFD simulations of the bladeless fan. Since the number of the elements is very high in 3D simulations, it is an appropriate approach to use this
method. In that enhanced wall treatment method does not require fine meshing or thin inflation layer near wall regions, this method has used all 3D simulations of the bladeless thruster.

The SIMPLE algorithm is used for pressure-velocity coupling with second-order upwind scheme for momentum, turbulent kinetic energy ($k$) and turbulent dissipation rate ($\varepsilon$).

### 3.3 The Coanda Surface Curvature Effect

Designed four bladeless fan models to observe the effect of Coanda surface curvature parameter on average outlet velocity magnitude shown in Figure 3.5. These models have different curvatures from each other, shown in Table 3.1 below. The designed flow simulation models are representing the extracted volume of air inside the bladeless fan cross-section. All four models have the same nozzle outlet gap at the same angular orientation. These models also have a flat surface parallel to the fan thrust axis on the trailing edge part, where the air leaves the bladeless fan geometry laminarly. Velocity vectors at the fan exit have a direction which is parallel to the thrust axis and this makes the thruster more efficient as the entire flow is directed along the thrust axis without any vortex or turbulence.
Figure 3.5 Four main thruster models to observe Coanda surface curvature effect on average outlet velocity. (a) Model 1, (b) Model 2, (c) Model 3 (d) Model 4

Table 3.1 Cross-sectional dimensions table of all four models

<table>
<thead>
<tr>
<th>Model</th>
<th>Cross-Section Thickness (mm)</th>
<th>Coanda Curvature Radius (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model 1</td>
<td>$t_1 = 31$</td>
<td>$R_1 = 35$</td>
</tr>
<tr>
<td>Model 2</td>
<td>$t_2 = 26.25$</td>
<td>$R_2 = 25$</td>
</tr>
<tr>
<td>Model 3</td>
<td>$t_3 = 23.06$</td>
<td>$R_3 = 20$</td>
</tr>
<tr>
<td>Model 4</td>
<td>$t_4 = 22$</td>
<td>$R_4 = 15$</td>
</tr>
</tbody>
</table>

3.3.1 Boundary Conditions

The main purpose of this section is to observe the effect of Coanda surface curvature on performance of the bladeless thruster. Therefore, the impeller part that draws in air
to the thruster part was not considered in CFD analyses. Instead, the air flow was set to two inlet parts of every model together as an inlet velocity boundary condition. Air flow considered as coming from the impeller, enters through the 60 mm diameter cylindrical inlet line from both sides of the thruster symmetrically as shown in Figure 3.5. The air flow circulates the entire cross-section of the thruster and travels through a small nozzle gap to leave the bladeless fan.

All of the designed models in Figure 3.5 have the same nozzle outlet gap which is 5 mm. All CFD simulations carried out for thruster models have the same inlet boundary conditions which are velocity inlet boundary conditions for INLET 1 and INLET 2 faces. For the outlet boundary condition pressure outlet is used for OUTLET surfaces of thruster models. All other fluid domain faces are treated as a wall fluid boundary condition.

3.3.2 Mesh Independence

The mesh independence study is an investigation method, whether the simulation results are independent of the number of mesh. A series of CFD simulations were performed for one model to measure the mesh independence of the CFD model and appropriate mesh number obtained. Before simulating the various case studies to observe Coanda surface curvature effects, mesh independence of the first simulation is performed to obtain an appropriate number of mesh.

Model 2 was used for the mesh independence study. Eight different mesh sizes from 12 mm element height to 0.8 mm were created for the CFD simulations of the Model 2. The mesh independence study results for the second model, which have an outer diameter of 101 mm and 5 mm nozzle outlet gap, shown in Table 3.2 below.

Table 3.2 The results obtained from mesh independence analysis of Model 2

<table>
<thead>
<tr>
<th>Analysis Number</th>
<th>Mesh Element Size (mm)</th>
<th>Number of cells</th>
<th>Nozzle Outlet Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12</td>
<td>503132</td>
<td>229.8884</td>
</tr>
</tbody>
</table>
Eight simulations were carried out with different element sizes. It can be seen from the table that the nozzle outlet velocities vary between about 230 m/s to 239 m/s from Table 3.2. The relation between the mesh sizes and the nozzle outlet velocity magnitudes is shown in Figure 3.6 below.

<table>
<thead>
<tr>
<th>2</th>
<th>8</th>
<th>505508</th>
<th>231.9662</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>7</td>
<td>505542</td>
<td>233.4112</td>
</tr>
<tr>
<td>4</td>
<td>6</td>
<td>509768</td>
<td>234.6435</td>
</tr>
<tr>
<td>5</td>
<td>4</td>
<td>529440</td>
<td>236.892</td>
</tr>
<tr>
<td>6</td>
<td>2</td>
<td>736811</td>
<td>238.2792</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>1999827</td>
<td>239.0117</td>
</tr>
<tr>
<td>8</td>
<td>0.8</td>
<td>3188485</td>
<td>238.7405</td>
</tr>
</tbody>
</table>

Figure 3.6 Mesh independence study results for eight different mesh

It can be seen from Figure 3.6 and Table 3.2 that the 736811 elements or more is the appropriate number of mesh for these simulations. Since the model geometry did
not change significantly, this determined element size and number was used for all models.

### 3.4 Three-Dimensional Flow Simulation Results

A series of flow analyses were carried out for the four designed models, given in Figure 3.5, and the effect of the curvature of the Coanda surface on the performance of the bladeless thruster was examined over the nozzle output velocity parameter. First simulation was performed for the first model, which has the greatest curvature radius, $R = 35$ mm, with the previously determined mesh size. Simulation results are given in Figure 3.7.

![Simulation Results](image)

Figure 3.7 (a) Velocity contour view inside the thruster ring. (b) Airflow streamlines for Model 1
The maximum velocity at the nozzle exit is almost 168 m/s, as shown in Figure 3.7 (a). The Model 1’s average outflow velocity was also determined to be 154.07 m/s. The magnification factor between the intake and output airflow velocity can be determined to be around 3.8517. The CFD data in Figure 3.7 shows that the developed model increases the airflow velocity by almost four times. It should be noted that the Model 1’s average $y^+$ value is calculated as 63.281 shown in Figure 3.8 and the $y^+$ distribution inside the bladeless thruster is given in Figure 3.9 below.

![Figure 3.8 Area-weighted average of the $y^+$ values for Model 1](image)
This $y^+$ value is a critical parameter to describe how the air velocity behaves from the near wall region or far away from the wall. Some turbulence models such as $k$-$\varepsilon$ are only valid in the fully developed turbulence area. Therefore, the $k$-$\varepsilon$ model does not perform well in the area close to the wall. However, wall functions can be used to deal with that problem (see Figure 3.10).

Wall functions are equations that have been obtained empirically and are used to fulfil the physics in the near-wall area. To ensure the correctness of the results, the first cell centre must be located in the log-law region. Wall functions are utilised to connect the inner part of the wall to the fully developed turbulent region, see Figure 3.11. There
is no need to resolve the boundary layer when employing the wall functions approach, resulting in a significant reduction in mesh size and computational domain.

Figure 3.11 (a) Viscous sublayer resolving approach to decide boundary layer which is shown in red line. (b) Wall functions to resolve the boundary layer (Jousef, 2020)

Enhanced Wall Treatment (see Figure 3.10) wall function is used to resolve boundary layer to obtain the velocity profile accurately near wall regions in flow.

For the Model 2, which is designed to have lesser curvature on the Coanda surface than the Model 1, flow simulations are carried out by using the same mesh size and boundary conditions of the Model 1. Second model has a curvature radius of $R = 25 \text{ mm}$ on the Coanda surface.

The second prototype's CFD simulation results, illustrated in Figure 3.12, reveal that circulation at the inner leading section decreased and the outlet velocity increased
dramatically. The maximum outlet velocity is 277.8 m/s, while the average output velocity is 254.334 m/s. The magnification factor between the input and exhaust airflow is around 6.35. The airflow can be accelerated up to 6.35 times in the second prototype.

Figure 3.12 (a) Velocity distribution inside the thruster in contour view. (b) Streamline view of the air inside the thruster

The same conditions were used to simulate the other two models, Model 3 and Model 4 respectively. Figure 3.13 (a) and (b) shows the simulation results for the
Model 3 and Model 4 respectively. The Coanda curvature and cross-sectional thickness were changed to 20 mm and 23.06 mm, respectively, in the third prototype. As shown in Figure 3.13 (a), the maximum nozzle output velocity is reduced to 157 m/s as a result of this design adjustment. This can be caused by a decrease in the fan ring's cross-sectional air capacity.

Figure 3.13 (a) Simulation results for Model 3. (b) Simulation results for Model 4
Finally, for the fourth design, the Coanda surface with the lowest curvature is used. The Coanda effect in airflow is increased when a surface has a curvature, according to theory (Rašuo & Mirkov, 2014). Fluid flows more freely down the surface due to the Coanda effect than it does on a flat surface. As a result of the low curvature at the Coanda surface, the Coanda effect is reduced. The average nozzle outlet velocity of the Model 4 is increased to $199.47 \text{ m/s}$.

All simulations for four models repeated with the same boundary conditions but with different nozzle outlet gap sizes to observe the effect of this parameter on the outlet velocity of the bladeless thruster. As expected, narrowing the nozzle outlet gap provided an increase in outlet velocity. Increase in the outlet velocity creates greater pressure difference between stagnated air in atmospheric pressure at the outside of the thruster and the inside of the bladeless thruster. This pressure difference creates induced air flow from outside to the inside region of the bladeless thruster. The inducement effect of bladeless fan design is the key point to increase thrust and this makes bladeless thrusters more efficient compared to the traditional propeller systems.

### 3.5 Conclusion

The goal of this chapter was to see how geometrical parameters affected the outlet velocity of small-scale bladeless fans such as the Coanda surface curvature and nozzle outlet gap size. The findings of this chapter show that geometrical parameters of bladeless fans have a relationship with each other. The first model has the largest cross-sectional thickness and a considerable curvature at the Coanda surface, whereas the second model has a higher average outflow velocity than the first. As a result, there is an ideal design that maximises the outlet velocity between Model 1 and Model 4. Bladeless fans not only speed up inlet air, but also induce exterior stationary air, resulting in up to $15 \text{ times}$ greater airflow depending on the geometry of the thruster.

It should be noted that this study solely depicts internal flow effects; however, these simulations can be integrated with outward flow simulations in next chapters. Since the computational sources are limited, in the next chapter, bladeless thruster models will be designed as 2D for flow analyses to be carried out together with the enclosure domain, which contains stagnated air at atmospheric pressure located in the outer
volume of the thruster, and the effects of geometric parameters in particular the Coanda surface curvature on the performance of the thruster will be examined.
CHAPTER FOUR
TWO-DIMENSIONAL FLOW ANALYSES

4.1 Introduction

In this chapter, flow simulations carried out to evaluate the performance of a two-dimensional model is designed in ANSYS-design modeller software based on 3D simulation results and observations shown in the previous chapter and a simplified 2D model was created. The computational domain and the first 2D model designed is given in Figure 4.1.

![Figure 4.1 Computational domain and 2D thruster model that designed for the flow simulations](image)

3.2 Design of the Fluid Domain and the Thruster Model

The computational domain is symmetrically designed to reduce the computational load. Symmetry axis is the x-axis of the model. In this way, the flow problem is designed as an axis-symmetric problem. Model cross-section is representing the shell of the thruster thus, a sketch of the thruster cross-section model cut out from the air domain in that it is solid. The thruster model is placed in an enclosure domain with 500 mm width and 100 mm height. The fluid domain is separated into two different regions. By applying different mesh sizes to these regions, as shown in Figure 4.1, it is possible to make more precise calculations around the thruster section. Mesh sizing is applied to these two domains with different element sizes. Fine mesh domain has
smaller mesh elements than coarse mesh domain to make it possible to observe flow changes around the thruster.

Divided mesh domains formed as zero thickness surfaces for each and the thruster wall thickness was determined as 2 mm. Thruster cross-section dimensions scaled down with the same wall thickness, due to manufacturing restrictions. The thruster models, which are shown in Figure 4.2 are printed with a resin 3D printer to use in experimental setup, which will be explained in later chapters. Since the 3D printer manufacturing capacity has certain dimensional limitations, the thruster models were downsized to comply with these restrictions.

The essential dimensions of the thruster such as total thruster height, nozzle outlet gap, inlet diameter and Coanda surface radius are constructed as 56 mm, 2.8 mm, 3.5 mm and 86 mm, respectively. The Coanda surface radius is measured from its centre at a distance of 100 mm from the fan axis. By changing this radius, the model was created in such a way that the Coanda curvature would be increased or decreased. The trailing edge of the thruster is constructed as a flat surface to provide parallel flow to the thruster axis to obtain balanced thrust force. General form of the thruster model created like an airfoil as in the previous 3D analyses. This form allows the fan to direct the inlet air flow 90 degrees down and create vacuum pressure in the middle of the fan thanks to the Coanda surface of the air flow induced from outside.
Figure 4.2 Three-dimensional model view of the thruster model which is created to use in two dimensional analyses

4.3 Mesh Domain Creation

The mesh domain is created with Quadrilateral elements with two different mesh sizes for fine and coarse mesh regions, as shown in Figure 4.3. In order to allow more precise calculations, 0.4 mm and 2 mm element sizes are applied for the fine mesh region and the coarse mesh region respectively.

Figure 4.3 Mesh view of the thruster model
In order to accurately generate the velocity profile in the areas close to the wall, inflation layer meshes are applied to the wall areas. Wall faces are selected (26 Edges) for inflation layers and the first layer thickness method is used. This method allows creating an inflation layer by determining parameters shown in Figure 4.4, such as first layer thickness, which is the height of the nearest inflation element to the wall, the maximum number of layers and the growth rate. The first layer thickness is specified as 0.01 mm after the \( y^+ \) analysis for the 2D model. The maximum number of layers parameter means that the maximum allowable number of inflation layers, which depends on the model geometry and mesh structure. The growth rate parameter describes how many times the size of the element in the next layer can be increased after each successful layer. The growth rate of the inflation layers specified as 1.3 for the first model.

![Figure 4.4 Inflation layer parameters](image)

4.3.1 Examination of the Mesh Quality

Mesh quality significantly affects the consistency and accuracy of the simulations. In order to determine the mesh quality, some mesh metrics such as skewness, aspect ratio and orthogonal quality are used. The skewness metric is defined as the difference between the shape of the cell and the shape of the equilateral cell of equivalent volume.
The difference between the form of the cell and the shape of an equilateral cell of equivalent volume is defined as *skewness*, as shown in Figure 4.5. Cells that are highly skewed can reduce accuracy and destabilise the solution. In most flow problems, the maximum skewness for a triangular/tetrahedral mesh should be kept below 0.95, with an average value of less than 0.33, according to a general rule. A maximum value greater than 0.95 may cause problems with convergence, necessitating changes to the solver controls, such as lowering under-relaxation factors or switching to the pressure-based coupled solver. The range of values distributed in 0 and 0.5 of maximum skewness and for the average skewness value of approximately 0.1 is acceptable for 2D simulations.

![Figure 4.5 Skewness of the mesh element](image)

Another quality metric for mesh considered in simulation of the first 2D model is the *aspect ratio* (see Figure 4.6). This metric defines the stretching of the mesh element. Generally, aspect ratios in excess of 5:1 in the bulk flow (flow region that away from walls). On the other hand, the quadrilateral cells inside the boundary layer can be stretched up to 10:1 aspect ratio in most cases.
Figure 4.6 Mesh elements with close to 1 and large aspect ratios (Özgün, 2022)

The orthogonal quality refers to how near the angles between adjacent element faces (or adjacent element edges) are to a predetermined ideal angle (depending on the relevant topology), as shown in Figure 4.7. The minimum value for the orthogonal quality of 0.14 is acceptable for 2D simulation. The values of all mesh metrics are listed below in Table 4.1
Figure 4.7 Calculation of the orthogonal quality of a mesh cell (Avraham, 2020)

Table 4.1 Minimum, maximum and average values of mesh metrics considered

<table>
<thead>
<tr>
<th>Metric</th>
<th>Minimum</th>
<th>Maximum</th>
<th>Average</th>
</tr>
</thead>
<tbody>
<tr>
<td>Skewness</td>
<td>$1.3331e-10$</td>
<td>0.78407</td>
<td>$6.6952e-2$</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>1</td>
<td>62.587</td>
<td>2.1866</td>
</tr>
<tr>
<td>Orthogonal Quality</td>
<td>0.2001</td>
<td>1</td>
<td>0.9813</td>
</tr>
</tbody>
</table>

As can be seen from Table 4.1, average values of all metrics are in acceptable range. It is observed that average values of the skewness and the orthogonal quality of the mesh elements are appropriate for the 2D flow simulations. Average aspect ratio value is higher than 1 however, considerably close to 1 and in acceptable range.
4.4 Determination of Boundary Name Selections

Before the transition to CFD simulations, boundary conditions regions were determined on the model surfaces. Name selections for the boundary surfaces are specified as inlet, axis, Patm_back, Patm_front and frictionlesswall as shown in Figure 4.8. The detailed qualifications of these boundary conditions specified in ANSYS-Fluent software.

The inlet surface is used to determine the inlet pressure of the airflow to be directed into the fan. The axis surface name is used for the symmetry axis of the problem designed as axis symmetric. Patm_back and Patm_front surfaces are used to define the pressure values at the right and left boundaries of the enclosure domain. Finally, by using the frictionlesswall boundary condition for the surface located far above the fan model, it is desired to ensure that the airflow that will occur here can proceed parallel to that surface.

![Figure 4.8 Boundary name selections to specify boundary conditions on CFD simulations](image)

4.5 The CFD Model Construction

After the creation of the mesh elements, the model is transferred to the ANSYS-Fluent. The solver settings are specified as shown in Figure 4.9. The flow simulation
is designed as an *axisymmetric* flow problem in 2D space and *steady* flow. The Axisymmetric option indicates that the flow domain is axisymmetric about the $X$-axis. When the Axisymmetric option is enabled, the governing equations are solved in 2D axisymmetric form rather than 2D Cartesian version. Solver type of the simulation is specified as *pressure-based*. The pressure-based solver enables the pressure-based Navier-Stokes coupled solution algorithm to be used. The velocity formulation to be used in the simulation specified as *absolute* velocity formulation.

![Figure 4.9 Solver settings for the 2D flow simulations](image)

The flow model is specified as the $k$-$\omega$ viscous model. This model contains four different types such as *standard*, *Generalised k-$\omega$ (GEKO)*, *Baseline (BSL) k-$\omega$* and *Shear-Stress Transport (SST) k-$\omega$* models. These models adjust the viscosity of the flow for every individual point and solves the laminar flow with these local viscosities. SST $k$-$\omega$ turbulence model is selected for this problem, as shown in Figure 4.10.

In the previous section, the $k$-$\epsilon$ turbulence model was used in the 3D simulations of the thruster. This model requires near-wall treatments for wall-bounded turbulent flows to constitute the velocity profile of the flow at near-wall areas. In contrast, the $k$-$\omega$ model does not require any near-wall treatment because this model defines it automatically.
When enhanced wall treatments are employed with the $k-w$ models, the wall boundary conditions for the $k$ equation are addressed in the same way as the $k$ equation is treated. This means that all boundary conditions for wall-function meshes will follow the wall function approach, whereas boundary conditions for fine meshes will follow the Low-Reynolds number approach.

The excessive creation of turbulence energy near stagnation points is a drawback of traditional two-equation turbulence models. Two formulations are available to limit the production term in the turbulence kinetic energy equations in order to avoid the growth of turbulent kinetic energy in the stagnation regions: *Production Limiter* and *Production Kato-Launder*.

![Figure 4.10 The selected turbulence model SST $k-w$ settings](image)

Figure 4.10 The selected turbulence model SST $k-w$ settings
4.5.1 Boundary Conditions

The boundary conditions of the flow model contain two pressure inlets, one pressure outlet, a frictionless wall and an axis. The inlet air boundary conditions specified as shown in Figure 4.11. The total pressure of 5 bar is applied to the inlet region with %10 turbulence intensity. The $P_{\text{atm_front}}$ and $P_{\text{atm_back}}$ faces are specified as pressure inlet and pressure outlet respectively, with 0 bar gauge pressure which means that these regions are at atmospheric pressure.

![Pressure Inlet](image)

Figure 4.11 The inlet boundary condition specifications

The frictionless wall face is specified as a wall region with 0 Pa shear-stress which describes the surface as frictionless, as shown in Figure 4.12.
4.6 Simulation Results

The coupled scheme is used for the Pressure - Velocity coupling and model initialised with Hybrid Initialisation before running calculation. 1000 iterations are carried out to observe velocity and pressure changes inside the fan and the outside air.

In order to observe the convergence of the analysis, the volumetric flow rates in the two determined locations such as inlet and vfr_back faces (see Figure 4.8), which are shown in Figure 4.13, were followed. The graph showing the course of these physical parameters is given below. It can be observed from Figure 4.13 that volumetric flow rates at the inlet and vfr_front faces remain constant after approximately 320 iterations. This gives an idea about convergence, besides that, the residuals graph which is given in Figure 4.14 shows the convergence of the differential equation solutions for CFD simulation. It can be seen from Figure 4.14 that the residual value is set below $10^{-8}$ as the convergence criteria of the governing equation.
The inflation layers for the 2D thruster model applied to obtain the velocity profile at the near-wall regions. Thus, the first layer thickness method is used to create inflation layers for the wall surfaces. The height of the first inflation layer is specified as 0.00167 mm to reduce the $y^+$ value below 5. The $y^+$ value distribution is given below for the thruster wall regions.
Figure 4.15 The $\gamma^*$ distribution for the Model 5 (see Table 4.2)

The velocity profile at the near-wall regions of the Model 5 constructed as shown in Figure 4.16 below.

Figure 4.16 Near-wall velocity profiles
The velocity and static pressure distribution contours obtained from the CFD simulations are given below in Figure 4.17 and Figure 4.18. It can be seen from Figure 4.17; some low-pressure regions are created inside the bladeless thruster. The reason for these pressure drops is the high-velocity airflow created at the nozzle outlet region bladeless thruster. This pressure drop at the centre of the bladeless thruster creates vacuum and induces airflow from outside to inside of the thruster.

![Figure 4.17 The static pressure contour of the bladeless thruster](image)

The drawn air joins the high flow that comes from the nozzle outlet and total air flow leaves the thruster with a high velocity of 500 m/s approximately, as shown in Figure 4.18.
The effect of the Coanda surface on the bladeless thruster outlet volumetric flow rate is examined by a parametric analysis on five different model (see Table 4.2). Increase in the central distance of the Coanda surface from the thruster axis makes it flatter. Therefore, Model 5 has a more curved surface at the Coanda region and Model 1 has the flattest surface on the Coanda surface (see Figure 4.19).
Five design points are determined for this analysis to observe the effect of the Coanda curvature radius on the volumetric flow rate of the bladeless thruster, at the $vfr_{\,back}$ surface as shown in Table 4.2 below. As the Coanda surface curvature increases, the average volumetric flow rate at the $vfr_{\,back}$ surface is also increased (see Figure 4.20).

Table 4.2 The parametric analysis result of the Coanda surface curvature

<table>
<thead>
<tr>
<th>Model</th>
<th>Coanda Surface Radius Centre Dist. From Thruster Axis (mm)</th>
<th>Volumetric Flow Rate at $vfr_{,back}$ Surface ($m^3/s$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model 1</td>
<td>100</td>
<td>0.35204</td>
</tr>
<tr>
<td>Model 2</td>
<td>85</td>
<td>0.34653</td>
</tr>
<tr>
<td>Model 3</td>
<td>70</td>
<td>0.35176</td>
</tr>
<tr>
<td>Model 4</td>
<td>55</td>
<td>0.35925</td>
</tr>
<tr>
<td>Model 5</td>
<td>40</td>
<td>0.3592</td>
</tr>
</tbody>
</table>
The Model 5 has the most curved Coanda surface in five designed models. The simulation results of the model are given in Figure 4.21 below. Figure 4.21 (a) represents the static pressure and Figure 4.21 (b) is the velocity distribution of the thruster. It can be seen from Figure 4.19 (a), the average vacuum pressure inside the bladeless thruster is -0.433 bar approximately. Compared to the previous model, the new model provided more vacuum at the centre of the thruster. Likewise, the maximum velocity provided at the nozzle outlet gap of the thruster was 1010 m/s. Also, the volumetric flow rate at the vfr_back surface of the Model 5 increased compared to the Model 1. This difference between two models is caused by the Coanda surface curvature. Li et al. (2014) proposed that when the curvature of the Coanda surface exceeds the bounds of the 0.026 - 0.065 interval efficiency of the bladeless fans begins to decrease. It can also be seen from the Figure 4.20, through the increasing curvature direction (100 mm to 40 mm central distance) volumetric flow rate at the vfr_back surface firstly decreased until 85 mm central distance approximately. After that point volumetric flow rate begins to increase until the 40 mm radius.
When the curvature is further increased, it is seen that the efficiency decreases after a certain point. In addition, the linear velocity profile at the fan outlet, which is necessary to obtain maximum thrust, cannot be achieved and the flow regime begins to deteriorate. Thus, for every design there is an efficient interval of curvature.

In all analyses performed, the air density was taken as constant and the flow was modelled as incompressible. In the further analyses, a compressible flow model was established and air was modeled as an ideal gas. As can be seen from the Figure 4.22, turbulence regions are seen more clearly in the flow modeled as compressible. Also, the low-pressure areas can be seen clearly from Figure 4.22 which have pressure distribution below atmospheric pressure.
The air density has risen in wall regions. However, volume average of the air density calculated as $1.296 \text{ kg/m}^3$. The output parameters such as volume flow rate, the air flow speeds and the turbulent kinetic energy at the vfr_front and vfr_back surfaces and maximum nozzle output velocity are given in Table 4.3 below.

Table 4.3 The parametric analysis result of the Coanda surface curvature

<table>
<thead>
<tr>
<th>Surface/Region</th>
<th>Volume Flow Rate (m$^3$/s)</th>
<th>Air Flow Speed (m/s)</th>
<th>Turbulent Kinetic Energy (m$^2$/s$^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>vfr_back</td>
<td>0.07324345</td>
<td>52.392385</td>
<td>2.8271782</td>
</tr>
<tr>
<td>vfr_front</td>
<td>0.05118331</td>
<td>40.422047</td>
<td>0.35902054</td>
</tr>
<tr>
<td>Nozzle Outlet</td>
<td>-</td>
<td>259 (maximum)</td>
<td>-</td>
</tr>
</tbody>
</table>

4.6 Conclusion

In this section, 2-dimensional analyses of the bladeless thruster models are carried out. Due to calculation constraints, 2D analyses were preferred as they allow working with finer mesh elements. The model is symmetrically modelled in 2 dimensions and the flow problem is designed as axisymmetric.
By changing the curvature of the Coanda surface, five models were created and parametrically analysed to observe the effect of the Coanda surface curvature on the outlet volumetric flow rate.

According to the results obtained from the analyses made, as the Coanda surface curvature increases, the vacuum pressure formed inside the bladeless thruster also increases. This pressure difference accelerates the stagnated air outside the thruster and induces it through the thruster outlet.

However, too much curvature may cause the flow to not follow the surface. In addition, it is difficult for the flow passing over the Coanda surface to return to linear form and the linear output speed, which is of great importance for the thrust of the fan, cannot be obtained.
CHAPTER FIVE
PROTOTYPE PRODUCTION

5.1 Production of Test Structure

The bladeless thruster prototype was produced by using resin 3D printer. The resin 3D printers have better surface quality than PLA 3D printers and can print a layer precision of up to 25 microns. This sensitive printing ability of the resin printers not only ensures better surface quality, but also ensures more smooth flow inside the bladeless thruster. In addition, since this model does not contain any gaps, it also prevents the occurrence of air leaks. The first printed bladeless thruster model is shown in Figure 5.1.

![First Thruster Model](image)

Figure 5.1 The first thruster model (a) Perspective view (b) Side view (c) Top view

The thruster model has two symmetrical inlets at two sides. This structure creates a more homogeneous air distribution inside the bladeless thruster chamber to single inlet model.

An experimental setup was created by placing four fans with the same structure in a drone frame. By placing this experimental setup on a wooden chassis, it was desired
to measure how much thrust this structure produced (see Figure 5.2). As a result of the experiments carried out, a total thrust of $1.8 \, N$ was obtained when four fans were fed with separate compressor lines at a pressure of approximately 6 bar. However, since the pneumatic lines used in this test system have very small diameters, due to the manufacturing constraints, and 90-degree elbows are used to transmit the air flow symmetrically to the fan inlets, the flow rate losses increase a lot and this reduces the thrust considerably.

Figure 5.2 The test structure with four identical bladeless thrusters

In future studies, it is aimed to design suction impellers for each thruster in order to eliminate these disadvantages. Thus, the losses will be tried to be eliminated by feeding the fans with pneumatic lines with wider diameter.
CHAPTER SIX
CONCLUSION

In this study, design and analysis of a bladeless propulsion system for a quadcopter are performed. Firstly, a bladeless thruster model which has a cross-section like airfoil is designed. Several design parameters were established, and the fan geometry was optimized for usage as a thruster in a bladeless quadcopter. The bladeless fan was created to produce a substantial amount of thrust, which is suitable for quadcopter flight with four identical bladeless fans. The designs were created with the goal of achieving the highest average nozzle exit velocity and reducing inefficiencies caused by turbulent flow inside the fan ring.

Several characteristics, such as Coanda surface curvature, cross-section thickness, and nozzle output gap, were used to create the cross-sectional area of the fan model. These are the primary factors that influence average nozzle exit velocity, which is the primary goal of the design process in order to maximize produced thrust. Other cross-sectional design features, such as the fan's height, high flow tunnel, and inner leading edge, are critical in ensuring that the flow inside the fan reaches the exit with as little turbulence as possible.

The purpose of Chapter 3 was to explore how geometrical parameters like Coanda surface curvature and nozzle outlet gap size affected the outlet velocity of small-scale bladeless fans. The findings of Chapter 3 demonstrate that the geometrical characteristics of bladeless fans are related with each other. The first model has the largest cross-sectional thickness and the largest curvature at the Coanda surface, whereas the second model has a faster average outflow velocity. As a result, between Model 1 and Model 4, there is an optimal design that maximizes outlet nozzle velocity of the bladeless fan.

The bladeless thruster models are analyzed in two dimensions in Chapter 4. Due to time constraints, 2D analyses were chosen because they enable for finer mesh elements to be used. The flow problem is constructed as axisymmetric, and the model is symmetrically described in two dimensions.
Five models were developed and parametrically analyzed to observe the influence of the Coanda surface curvature on the outlet volumetric flow rate by modifying the curvature of the Coanda surface.

According to the findings of the analysis, the vacuum pressure created inside the bladeless thruster increases as the Coanda surface curvature increases. The stagnant air outside the thruster is accelerated by the pressure differential and induced through the thruster outlet. Too much curvature, on the other hand, may cause the flow to deviate from the surface. Furthermore, the flow that flows over the Coanda surface needs more time to return to laminar form again, and the linear output speed, critical for the fan’s thrust, cannot be reached.
REFERENCES

Anonymous, (2022, June 7). Mesh generation process. CFDyna
http://www.cfdyna.com/CFDHT/Meshing.html


Avraham T. (2022, June 8). Know thy mesh – mesh quality – part I. CFDisrael
https://cfdisrael.blog/2019/02/01/know-thy-mesh-mesh-quality-part-i/


Jousefm, (2022, June 8). What is Y+ (yplus)? SimScale CAE Forum.
https://www.simscale.com/forum/t/what-is-y-plus/82394


Özgün, Ö. (2022, June 1). *Is your mesh good enough?*. Mechead. https://www.mechead.com/mesh-good-enough/


