ANALYSIS OF FAN BLADE ATTACHMENT

Master Degree Project in Applied Mechanics
A-Level, 30 ECTS
Spring term 2014

Patrick Shingu
Miguel García Cabrera

Supervisor: Dr. Karl Mauritsson
Examiner: Dr. Thomas Carlberger
ABSTRACT

This thesis work was based on the analysis of a fan blade attachment whereby a complete 3D model was presented by a partner company. The acceptability of a new design regarding to the mechanical loads and consisting of dividing the hub into two parts instead of using a solid hub is studied. From the model some critical parameters for the attachment of the blade with respect to the stresses were chosen such as the rotational speed, fillet of the blade and the neck of the blade. Parametric studies of these parameters were carried out in order to suggest the new design. Bearing in mind that a safety factor of 2 was the prerequisite, based on the analysis performed on ANSYS Workbench, it was suggested from the preliminary design that the axial fan can operate in two specific scenarios consisting of a rotational speed of 1771 rpm and a rotational speed of 1594 rpm. Using this set of parameters, a suggestion was drawn up on the blade fillet which will give the lower stress. Blade fillet size of 30 to 35mm was recommended while a size of 45mm was recommended on the neck of the blade. A modal analysis was performed in order to find at what frequency will the model be vibrating and a lowest and critical frequency of 16.8 Hz was obtained. Finally, a fatigue analysis of some interesting areas was performed in order to determine the numbers of cycles before fatigue failure occur. It was recommended to use the rotational speed mention previously since these speeds have offered a High Cycle Fatigue results.
# TABLE OF CONTENTS

1. **INTRODUCTION** .................................................................................................................. 1

2. **BACKGROUND** .................................................................................................................. 1

3. **PROBLEM** ............................................................................................................................ 4

4. **GOAL AND PURPOSE** ........................................................................................................ 6

5. **LITERATURE STUDY OF THE LOADS** ............................................................................ 6

   5.1. **AERODYNAMICAL LOADS** .......................................................................................... 6

      5.1.1. **LIFT** ....................................................................................................................... 7

      5.1.2. **DRAG** ..................................................................................................................... 7

      5.1.3. **SKIN FRICTION FORCE** ......................................................................................... 8

   5.2. **MECHANICAL LOADS** .................................................................................................... 9

   5.3. **STRESS CONCENTRATION** .......................................................................................... 10

6. **METHODS** .......................................................................................................................... 10

   6.1. **PARAMETRIC STUDY** ................................................................................................. 10

   6.2. **THE FINITE ELEMENT METHOD** ................................................................................. 11

   6.3. **MODAL ANALYSIS** ...................................................................................................... 13

   6.4. **CONVERGENCE ANALYSIS** ....................................................................................... 13

   6.5. **FATIGUE ANALYSIS** .................................................................................................... 15

7. **DELIMITATIONS** .................................................................................................................. 18
8. **FINITE ELEMENT MODEL** ................................................................. 19

8.1. **LOADS** ...................................................................................... 19

8.2. **BOUNDARY CONDITIONS** ........................................................... 21

8.3. **MESH** ......................................................................................... 22

8.4. **CONTACT COUPLING BETWEEN PARTS** ........................................ 23

9. **IMPLEMENTATION** ........................................................................... 25

9.1 **SOFTWARE ANALYSIS** ............................................................... 25

9.1.1. **ENGINEERING CONSIDERATIONS** ........................................ 25

9.1.2. **PARAMETRIC STUDY** ............................................................. 26

9.1.3. **MODAL ANALYSIS** ................................................................. 30

9.1.4. **FATIGUE ANALYSIS** ............................................................... 31

9.1 **ANALYTICAL ANALYSIS** ............................................................ 25

9.2.1. **CONTACT PRESSURE** ............................................................ 32

9.2.2. **BENDING STRESS IN THE BLADE** .......................................... 37

9.2.3. **CENTRIFUGAL STRESS** ........................................................... 38

9.2.4. **STRESS CONCENTRATION AROUND THE WHOLE** ..................... 40

10. **RESULTS** ....................................................................................... 41

10.1. **FILLET SIZE OF THE BLADE** ...................................................... 41

10.2. **DIAMETER OF THE NECK OF THE BLADE** .................................. 48

10.3. **CONVERGENCE** ....................................................................... 51
# TABLE OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Sketch of Impeller PFJ1 and components</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Sketch of the blade and components</td>
<td>3</td>
</tr>
<tr>
<td>3</td>
<td>Old design using a solid hub</td>
<td>5</td>
</tr>
<tr>
<td>4</td>
<td>New design using a split hub</td>
<td>5</td>
</tr>
<tr>
<td>5</td>
<td>Airfoil shape (Zare, 2013)</td>
<td>7</td>
</tr>
<tr>
<td>6</td>
<td>The direction of the air force</td>
<td>8</td>
</tr>
<tr>
<td>7</td>
<td>Area affected by the skin friction force</td>
<td>8</td>
</tr>
<tr>
<td>8</td>
<td>A solid of domain $\Omega$ (Dixit, 2009)</td>
<td>12</td>
</tr>
<tr>
<td>9</td>
<td>Support conditions (Dixit, 2009)</td>
<td>12</td>
</tr>
<tr>
<td>10</td>
<td>Straight line showing that error in $\varphi$ is proportional to $hq$ (Cook et al, 2002)</td>
<td>15</td>
</tr>
<tr>
<td>11</td>
<td>S-N Curves for Aluminium Alloy and Structural Steel (Fatemi, 2013)</td>
<td>16</td>
</tr>
<tr>
<td>12</td>
<td>Constant Amplitudes (Fatemi, 2013)</td>
<td>17</td>
</tr>
<tr>
<td>13</td>
<td>Fictive centrifugal load applied to get equilibrium</td>
<td>19</td>
</tr>
<tr>
<td>14</td>
<td>Centrifugal force in terms of rotational speed</td>
<td>20</td>
</tr>
<tr>
<td>15</td>
<td>In red color, pressure load applied in a face</td>
<td>21</td>
</tr>
<tr>
<td>16</td>
<td>In yellow, boundary conditions applied to the model</td>
<td>22</td>
</tr>
<tr>
<td>17</td>
<td>The quadratic (Ten node) tetrahedron using element with planar faces and side nodes located at side midpoints</td>
<td>23</td>
</tr>
<tr>
<td>18</td>
<td>Mesh used in the analysis</td>
<td>23</td>
</tr>
<tr>
<td>19</td>
<td>Detail of the fillet on the blade in the initial design (mm)</td>
<td>27</td>
</tr>
<tr>
<td>20</td>
<td>Detail of the blade fillet on the whole model in the initial design (mm)</td>
<td>28</td>
</tr>
<tr>
<td>21</td>
<td>Magnification of the neck of the blade</td>
<td>29</td>
</tr>
<tr>
<td>22</td>
<td>Detail of the neck of the blade on the whole model</td>
<td>30</td>
</tr>
<tr>
<td>23</td>
<td>Sketch of the assembly between hub and boss</td>
<td>32</td>
</tr>
<tr>
<td>24</td>
<td>Boundary condition for $\sigma_{1a}$</td>
<td>35</td>
</tr>
<tr>
<td>25</td>
<td>Boundary Condition for $\sigma_{2a}$</td>
<td>35</td>
</tr>
<tr>
<td>26</td>
<td>Sketch of kinematical condition</td>
<td>36</td>
</tr>
<tr>
<td>27</td>
<td>Sketch of the cantilever beam used to calculate the moment</td>
<td>38</td>
</tr>
<tr>
<td>28</td>
<td>Sketch to calculate the centrifugal stress</td>
<td>39</td>
</tr>
<tr>
<td>29</td>
<td>Stress analysis in the whole blade</td>
<td>41</td>
</tr>
<tr>
<td>30</td>
<td>Maximum stress against blade fillet size for each rotational speed</td>
<td>45</td>
</tr>
<tr>
<td>31</td>
<td>Stress in the fillet against blade fillet size for each rotational speed</td>
<td>46</td>
</tr>
</tbody>
</table>
Figure 32. Stress in the neck of the blade against blade fillet size for each rotational speed. 46
Figure 33. Maximum stress against rotational speed for each blade fillet size...................... 47
Figure 34. Stress in the fillet against rotational speed for each blade fillet size.................. 47
Figure 35. Stress in the neck of the blade against rotational speed for each blade fillet size. 48
Figure 36. Maximum stress against rotational speed............................................................... 49
Figure 37. Stress at the blade fillet against rotational speed .................................................. 50
Figure 38. Stress at the neck of the blade against rotational speed ........................................ 50
Figure 39. Convergence on the fillet of the blade................................................................. 51
Figure 40. Convergence on the neck of the blade..................................................................... 52
Figure 41. Mode number against natural frequencies and coarse mesh against fine mesh... 53
Figure 42. Picture of the fine mesh on the whole blade......................................................... 54
Figure 43. Detail of the fine mesh on the blade ................................................................. 54
Figure 44. Picture of the coarse mesh of the blade ............................................................... 55
Figure 45. Constant amplitude load obtained from ANSYS analyses (Fatemi 2013)........... 56
Figure 46. Rotational speed against Number of cycles, where N stands for the number of cycle................................................................................................................... 57
Figure 47. Fatigue Analysis on the Fillet of blade at 1594 rpm............................................ 57
Figure 48. Fatigue Analysis on the neck of the blade at 1594 rpm ...................................... 58
Figure 49. Maximum stress on the fillet of the boss............................................................. 59
Figure 50. Detail of maximum Stress on the fillet of the boss ............................................ 59
Figure 51. Maximum stress in the fillet of the blade............................................................. 59
Figure 52. Approximation of FE solution of the bending stress (Pa)................................. 63
Figure 53. Bending stress analytical solution (Pa)............................................................... 63
LIST OF SYMBOLS

$\rho$ = Density

$\mu$ = Dynamic Viscosity

$w$ = Weight

$A$ = Area

$C_L$ = Lift Coefficient

$C_D$ = Drag Coefficient

$C_f$ = Skin friction coefficient

$V$ = Velocity

$F_D$ = Force due to drag

$F_s$ = Force due to skin friction

$Re$ = Reynolds Number

$K_I$ = Stress intensity factor

$\alpha$ = Angle of attack

$LE$ = Leading edge

$TE$ = Trailing edge

$\sigma_{cen}$ = Centrifugal stress

$W$ = Rotational speed

$[K]$ = Stiffness matrix

$[M]$ = Mass matrix

$w$ = Natural frequency

$h$ = Element size

$P$ = Pressure

$C, M$ = Fatigue Parameters (materials constants)

$\frac{da}{dN}$ = Crack growth rate

$\Delta K$ = range of the stress intensity factor

$a$ = crack length

$\Delta$ = Diametric grip
\[ \nu = \text{Poisson’s ration} \]

\[ I = \text{Moment of inertia} \]
LIST OF SPECIAL TERMS AND ABBREVIATIONS

FEA- Finite Element Analysis
FEM- Finite Element Method
UTS- Ultimate tensile strength
SF- Safety Factor
HCF- High Cycle Fatigue
LCF- Low Cycle Fatigue

Leading Edge (LE): first part of solid surface in contact with fluid. Alternatively upstream or front

Trailing Edge (TE): last part of solid surface in contact with fluid. Alternatively downstream side or back

Angle of attack: angle of attack is the angle between the body's reference line and the oncoming flow

Neck of the blade: the neck of the blade is the small piece of shaft which support the fan blade on to the hub

Blade fillet: the blade fillet is the frontal fillet on the blade

Convergence: convergence is a way of confirming if the results obtained are corrected

Stress concentration: stress concentration is also known as stress raiser which occurs due to sharp changes of geometry

Contact Pressure: contact pressure occur when there is contact between two bodies
1. INTRODUCTION

For all the students registered for the Applied Mechanics program at the University of Skövde, it is required to do a project in order to complete the program. The project runs over a period of six months. The project also requires students to work in pair. This project was conducted in partnership with Akron. Akron is a company which specialized in the design and manufacturing of grain handling, fans and bioenergy components. This report covers the analysis of a fan blade attachment. The company was planning to change the design in order to reduce the cost and the fabrication of a component called the axial fan hub.

An axial fan is a type of compressor which moves air or gas parallel to the axis of rotation unlike the centrifugal or radial flow fans which moves air or gas perpendicular to the axis of rotation (Anon, 2002).

This report focuses on analyzing the attachment of the fan blade, according to mechanical loads. The stresses acting on both the blades and the hub are analyzed, a modal analysis is performed and finally, a fatigue analysis is carried out using the stress life approach.

2. BACKGROUND

Throughout history, the use of fans and ventilation systems has been increasing parallel to the industrial developments as well as the requirements of the situations. Although Leonardo da Vinci in the end of the 15th century described fans in his designs, the spreading of them became more important with the appearance and exploitation of the mining industry. Thus, along the 16th century, due to the extraction of coal, metal ores and other substances the ventilation became basic in order to avoid flow gas which might either asphyxiate the miners or explode with disastrous results. Indeed, the first mine ventilation system was published by Georgious Agricola (1912). Many of the sophisticated airflow systems used nowadays were embodied in his book. However, due to the lack of knowledge of the air properties, safety and health measures, non-important advances appeared until the end of 17th century thanks to the investigation of some important scientists like Galileo, Torricelli, Pascal, Boyle and Newton. Along the 18th and the beginning of the 19th century, until the First World War, the main researches were developed in United Kingdom, both in the seat of the British Government...
and the mine industry. The present day applications of fans are far too numerous to list owe to the improvement achieved, highlighting the aviation industry (Cory, 2005).

An axial flow fan is a machine which creates flow within a fluid (gas) in such a way that air flows linearly along the axis of it. It is composed of blades that force air to move parallel to the shaft about the axis at which the blades rotate, which are rigidly secured. Industrial axial are used in extremely conditions such as high temperature, large vibrations amplitudes or high corrosion, which cause large strains. Due to this, both the hub and the fan blades are made out of casting aluminum, which is capable to resist aggressive environments, have light weight and be easily malleable to the conditions required. However, nowadays, Akron is using a solid model, which means an increase of the material used and consequently, the cost of production is higher. A sketch of an impeller, its components and a detailed sketch of the fan blade are shown in Figure 1 and Figure 2.

![Figure 1. Sketch of Impeller PFJ1 and components](image)

Figure 1. Sketch of Impeller PFJ1 and components
Due to the importance of the fans and ventilation systems in the industry, as explaining above, several researches have been carried out using different analysis and investigations in order to know the behavior both the stresses and the fracture mechanics. Most of them have been performed by using of the Finite Element Methods (Cory, 2005).

With respect to the analysis of the stresses, Abdullah and Schlattmann (2012) performed an analysis taking into account the effect of centrifugal and aero-dynamical loads and varying geometrical parameters, concluding that in case of solid hub, maximum stresses are induced at the root of the blade, in such a way that if the thickness increases reducing the stresses at the root, natural frequency decreases. Furthermore, they suggested that the increment in the disk radius implies higher stresses and deformations. The above statements were supported by Arewar and Bhope (2013). According to Zare and his analysis of axial fan blade (2013), the relation between the safety factor, chord length and pressure load is such that increasing the constant pressure load, remaining the chord length constant, von Mises stress increases.
With respect to the fracture mechanics analysis, based on Tsai (2004) and his study about the rotating vibration behavior of the turbine blades, dynamic behavior and resonant frequencies varies widely regarding to the position of the fan blades. He also suggested that highest stresses are located at the first neck of the blade root. According to Amoo (2013), vibration loads acting on the fan blades can lead to fatigue especially if the blades are not designed properly. There are two main types of fatigue which affect the fan blades: low cycle fatigue (LCF), which is less than 100000 cycles or within the range of 0.1-5 Hz, and high cycle fatigue (HCF) which has values over 100000 cycles or within the range of 17-50 Hz (Totten and MacKenzie, 2003). The HCF is known for having a high frequency which is subjected to cyclic bending and the LCF is known for higher amplitudes and a low frequency (Amoo, 2013).

Barlow and Chandra (2005) carried out a fatigue crack propagation simulation under centrifugal and aero-dynamical loads, suggesting that stress intensity factors indicate a strong Mode I ($K_I$) and Mode III ($K_{III}$) at the edge of contact, while only crack opening condition ($K_I$) is observed in free surface. Finally, Poursaeidi and Salavation (2007) performed an analysis of the failure of a generator rotor fan blades, stating that failure was caused by aero-dynamical disturbances that created a resonant condition of vibration which induced to stress intensity factors superior to the critical stress intensity factor. However, despite all the analysis mentioned in the previous paragraphs there is no work done on an axial flow fan with a split hub.

3. PROBLEM

As it has been explained in the background, a new innovative concept design has been modeled by Akron in order to overcome the material wasted and consequently the costs when manufacturing the fans using solid hubs (Figure 3). The design consists of dividing the hub into two different parts, in such a way that both parts are joined to the base of the fan blades by bolting each other (Figure 4). Therefore, stress concentrations are expected in the attachment points. A thorough analysis of the mechanical loads is required in order to predict the behavior of the model and the mechanical viability of the new design.
Figure 3. Old design using a solid hub

Figure 4. New design using a split hub
4. GOAL AND PURPOSE

The goal of the project is a suggestion of an acceptable design according to the stresses acting on the fan blade attachment points. That is, the stresses both on the fillet of the blade and the neck of the blade are analyzed in such a way that a safety factor ≥ 2 is required. The suggestion will be based on the lowest stresses considering the prerequisite of the safety factor varying the rotational speed.

The purpose of obtaining this goal is to reduced production costs due to a reduction of the material used.

5. LITERATURE STUDY OF THE LOADS

As it has been explained in previous sections, an axial fan is loaded by both aerodynamical and mechanical loads. A literature study was performed about these loads in order to gather information.

5.1. AERODYNAMICAL LOADS

In order to find the aerodynamical load, research was based on the design load and the profile of the fan blade. It was noted that the blade is profiled in a streamline shape of cross section called airfoil (Zare, 2013). This shape plays a vital role on the performance of the blade (Figure 5). The airfoil can increase the efficiency of the blade and can also reduce the flow turbulence (Zare, 2013). A higher efficiency is achieved if the rotor blades are profiled in an appropriate way and have been twisted as well (Schildhauer and Minges, 2010). The airfoil section is being used in various areas such as airplane wings, wind turbine, gas turbine, fan blade, compressor, etc... (Zare, 2013)
As seen in Figure 5, the chord length is the distance between the leading edge (LE) and the trailing edge (TE). The angle of attack ($\alpha$) is between the chord length and the relative air velocity and where $L$ stands for the lift.

5.1.1. LIFT

Lift can be defined as the part of the aerodynamic force which is perpendicular to the relative airflow and $F$ is the resultant force. The lift force is given by equation (1):

$$F_L = 0.5 \cdot C_L \cdot \rho \cdot A \cdot v^2$$  \hspace{1cm} (1)

Where $F_L$ is the lifting force, $\rho$ is the density of air, $v$ is the relative velocity of the airflow, $A$ is the area of the airfoil as viewed from an overhead perspective and $C_L$ is the lift coefficient (Sullivan, 2006).

Further research was done in order to have a better understanding about the lift theory. The theory used is Bernouilli’s principle, which states that a gas or liquid, which has a high relative velocity, will create a lower pressure and the area with a higher pressure will have a lower relative velocity (Sullivan, 2006). This means that the airfoil is shaped in such a way that the upper surface is larger than the lower surface. Therefore, there is a separation as the air separates from the leading edge (Sullivan, 2006). The air which is going through the upper section has a greater path to travel.

5.1.2. DRAG

The drag force is the resistance created by a medium flowing over a body. This force acts in the direction of the object as it is shown in Figure 6 (Sullivan, 2006).
The drag force helps to choose what strength of driving system is required to propel the fan blades. The drag force equation is given by equation (2):

\[ F_D = 0.5 \cdot C_D \cdot p \cdot A \cdot v^2 \]  \hspace{1cm} (2)

The formula is very similar to the lift force equation; the \( C_L \) is replaced by the \( C_D \) which is the coefficient of drag which can be obtained by the use of a graph or various formulas depending on magnitude (Sullivan, 2006).

### 5.1.3. SKIN FRICTION FORCE

The skin friction force is the force created by the actual parallel perimeter of the object (Figure 7). The skin friction depends on what type of material is used as a skin of the fan blades; the rougher the skin, the more the drag. The skin friction can also be created by the smooth surface but the result is by far less compare to the rough surface.

The Reynolds number in this case is characterized by the formula \( Re = \frac{p \cdot v \cdot x}{u} \). Here \( x \) is the horizontal surface length of the above cross section (distance from the leading edge to the trailing edge). The larger \( x \) is the more surface length there is to create skin friction drag. The skin friction force is given by (3)

\[ F_S = 0.5 \cdot C_s \cdot p \cdot A \cdot v^2 \]  \hspace{1cm} (3)
where $C_s$ is the skin friction coefficient which is calculated in various ways depending on the scenario. If the roughness of the surface is specified then one would calculate the relative roughness ($L/e$) where $e$ represents the roughness ratio, the Reynolds number and get the specific skin friction coefficient of a graph (Sullivan, 2006).

In general most of the airfoil sections are obtained from the catalogue mostly provided by NACA. These sections are mainly used in the turbine industry, aviation industry and fans industry (Zare, 2013). Note that NACA stands for National Advisory Committee for Aeronautics and has been producing the airfoil section since 1930 (Zare, 2013). The first step to take when designing an axial fan is to obtain the tip diameter and the hub diameter, after that select an airfoil profile, then using NACA catalogue, the blade chord will be provided.

### 5.2. MECHANICAL LOADS

As part of the project brief, the mechanical load acting on the fan blades and the hub had to be determined. Research was done in order to find these loads. It was found that the fan blades are subjected to continuous steady and vibratory loads during operation (Amoo, 2013). The vibratory loads will be discussed at a later stage. The steady loads consist of centrifugal pull load and the torque (Amoo, 2013). Theses loads can be analysed, assessed and determined. It was also noted that the centrifugal pull loads are dependent on the span-wise mass distribution of the blade while the torque load is dependent on the span-wise mass distribution, chord mass distribution and the twist of the blade (Amoo, 2013).

In general it is considered that the centrifugal load is by far the greatest contributor taking into account both the mechanical and the aerodynamical loads acting on the fan blades and hub. It was also found that the blades are also subjected to bending stress but usually it is low so that most of the time it is being neglected (Poursaeidi & Salavatia, 2007).

The centrifugal stress equation is given by (4): 

$$ \sigma_{cen} = \frac{M \cdot V^2}{A \cdot r_c} = \frac{M \cdot r_c \cdot \omega^2}{A} $$

(4)

where $M$ is the mass of the airfoil, $V$ the surface Velocity, $r_c$ the radius of the mass from the rotation axes, $A$ the area of the cross section, and $\omega$ is the rotor speed in rpm.
5.3. STRESS CONCENTRATION

The literature on the stress concentration was done based on the geometry of the model. Stress concentration is also called stress raiser, where the stress is concentrated due to a geometry changes (Budynas et al, 2006). This may be caused by:

- The geometry discontinuities such as holes, notches, sharp corners and fillets
- The material not being homogeneous during the molding and casting.
- The irregularities in the surface such cracks and marks created during the machining operations.

It can be summarized that the stress concentration is affected by both the geometry and the loading. Different methods can estimate the stress concentration factor and each of them offer advantages and disadvantages (Budynas et al, 2006). The stress concentration may be estimated using the catalogue, FEM and theoretical values calculated.

6. METHODS

Various text books, journals articles and internet websites were consulted in order to gather enough information about the different possible methods used to perform the analysis in order to achieve the goal and the purpose of this dissertation. The methods were being split up into five parts, which are parametric study, the Finite Element Analysis, modal analysis, convergence analysis and fatigue analysis.

6.1. PARAMETRIC STUDY

A parametric study is a method that consists of a description, evaluation and examination of relationships between different parameters (Caicedo, 2007). The design step of an axial fan is an iterative process, since product engineers use to modify this process in such a way that acceptance criteria defined by safety, cost, performance, convenience and shape are found out (Singh, et al., 2011). In order to optimize product development, a parametric study will provide enough data to automate these iterations and find the best design. The study compares the results of different scenarios in such a way that the best alternative may be found easily (Singh, et al., 2011).
The goal of this thesis will be reached by performing a parametric study in order to suggest an acceptable design (considering stresses). In this case, the method is applied in the following parameters since all of them are expected to affect the stresses: the fillet of the blade, the neck of the blade and the rotational speed.

6.2. THE FINITE ELEMENT METHOD

The Finite Element Methods (FEM) is a method to get an approximated solution to differential equations (Zare, 2013). The FEM is a practical method for analysing structure with many degrees of freedoms (Zare, 2013). It is always recommended to use some engineering software in order to carry out the analyses. The software is more useful when the geometry, the loadings and material properties are complicated and the analytical approach would not be suitable in order to obtain solutions. The tools are not only used for saving time but also it prevents specialists from solving problems manually (Zare, 2013).

Before performing any finite element method, some basics terms used needs to be known. Since the FEM method is an approximation solution to differential equation, the differential equation will be subjected to boundary conditions (Dixit, 2009). The boundary condition can be defined as the value of the field variables where the field variables are the variables of interest used by the differential equation (Dixit, 2009). A node can be defined as a point in the finite element where the field variable is required to be calculated explicitly (Dixit, 2009). The shape functions can be defined as the interpolation functions of the nodal values. This can be explained by the following equation:

\[ \varphi(x,y) = N_1(x,y) \cdot \varphi_1 + N_2(x,y) \cdot \varphi_2 + N_3(x,y) \cdot \varphi_3 \]  

(5)

where \( \varphi_1, \varphi_2 \) and \( \varphi_3 \) are the field variable and \( N_1, N_2 \) and \( N_3 \) are known as the shape functions.

A formulation of FEM for a linear differential equation can also be shown as

\[ L \cdot u + q = 0 \]  

(6)

where \( u \) can be considered as the functions of the coordinates, \( L \) is the differential operator and \( q \) is the vector of known functions (Dixit, 2009). From the above equation \( u \) can be considered to be the boundary condition.
STRESS FORMULATION

The formulation of the stress is important for this thesis. Consider the figure below where there is a uniform thickness bounded by two parallel planes and any closed boundary $\Gamma$, as shown in the Figure 8. The significance of the boundary condition is shown in Figure 9.

In the instance of the FEM formulation for the stress, the constitutive matrices are added. Therefore, the stress formula becomes (7):

\[
\frac{\partial \sigma_x}{\partial x} + \frac{\partial \sigma_{xy}}{\partial y} + f_x = \rho \cdot \frac{\partial^2 u}{\partial t^2}
\]

\[
\frac{\partial \sigma_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + f_y = \rho \cdot \frac{\partial^2 v}{\partial t^2}
\]

where $f_x$ and $f_y$ are the body force per unit volume on the x and y directions, $\rho$ is the density of the material, $\sigma_x$ and $\sigma_y$ are the normal stresses, $u$ and $v$ are the displacement in both the x and y axis respectively and $\sigma_{xy}$ is the shear stress in the $xz$ and $yz$ planes (Dixit, 2009).

Figure 8. A solid of domain $\Omega$ (Dixit, 2009)

Figure 9. Support conditions (Dixit, 2009)
6.3. MODAL ANALYSIS

A modal analysis is a study of the dynamic properties of structures under vibrational excitation (Cook et al, 2002). A modal analysis is performed in structural mechanics in order to determine the natural mode shapes and frequencies of an object or structure during free vibration. The vibratory loads are loads which may include aerodynamics and installation dynamics (include inlet flow distortion, inlet vortex, altitudes effects and thrust reverser) (Amoo, 2013). FEM is used to determine the magnitudes of both the mechanical and vibratory loads. This is done by using the concept of modal amplitudes (eigenvectors) and frequencies (eigenvalues) of the blades (Amoo, 2013). The equation used by most engineering software to find the eigenvalues and eigenvectors is given by equation (8):

\[(K) - \omega^2 \cdot (M) \cdot (D)=0\]  

(8)

where \([K]\) is the stiffness matrix, \([M]\) is the mass matrix, \(\omega^2\) is an eigenvalue, \(\omega\) is the natural frequency and \(D\) is the eigenvector. From this equation, it can be said that a lower mass and / or a stiffer beam increases the natural frequency where else the higher the mass and / or softer beam will decrease the natural frequency (Cook et al, 2002).

A natural frequency can be defined as a frequency at which a free object vibrates once it is in motion (Anon, 2014). It was found that the stress induced through vibrations is critical when showing resonant in a structure, as this can lead to a catastrophic failure of the blades (Amoo, 2013).

According to Cook et al. (2002), the use of FEM in order to carry out the modal analysis is profitable since the object to be analyzed may have arbitrary shape and the results obtained are acceptable. Throughout the Experimental of the Modal Analysis, once the frequencies and the shape modes have been determined, the physical test can calibrate if the assumptions taken into account in the finite element model are correct (Ohman and Singhal, 1993).

6.4. CONVERGENCE ANALYSIS

In order to achieve trustworthy results, it is very important to check whether the solution has converged. As the mesh is refined, an acceptable FE formulation has to converge to the exact solution of the mathematical model. Furthermore a satisfactory rate of convergence is also
important, so that acceptable accuracy can be reached. The rate of convergence of a particular
type of element can be obtained by analysis, or by study of results provided by a sequence of
successively refined meshes (Cook et al, 2002).

H-method was chosen in order to perform the convergence in the software. This method uses
simple shape functions and many small elements (Cook et al, 2002). The method consists of
running the analysis using a coarse mesh. The results obtained from this method are compared
with a second running analysis in which the mesh used is finer than in the previous analysis
(Cook et al, 2002). The output from the two runs is compared and so on and so forth until the
percentage of change between solutions is less than 2%. This percentage represents the
discretization error, that is, the difference between the mathematical model and its discretized
(finite element) model (Cook et al, 2002). Therefore, the error is calculated according to (9).

\[
\text{Error} = \frac{\sigma_{i+1} - \sigma_i}{\sigma_i} \cdot 100
\]  

(9)

where \(i\) represents the number of analysis.

Then, convergence may be represented by means of Richard´s Extrapolation, according to
Equation 10 (Cook et al, 2002).

\[
\varphi_\infty = \frac{\varphi_1 \cdot h_2^q - \varphi_2 \cdot h_1^q}{h_2^q - h_1^q}
\]  

(10)

where \(\varphi_\infty\) corresponds to element size \(h=0\), \(h^q\) is the order of error of \(\varphi\), which is a quantity
of interest calculated at some location in a FE mesh.

According to Equation 10, convergence is represented by a straight line in a plot of \(\varphi\) versus
\(h^q\), as it is shown in Figure 10.
6.5. FATIGUE ANALYSIS

Many parts on the axial fan may work well initially. In any point these parts may fail due to fatigue failure caused by repeating the cyclic loading. The fatigue analysis is a study which is capable of finding if material can survive the many cyclic components. There are at least three ways of performing a fatigue analysis which are the strain life, the stress life and the fracture mechanics.

The strain life has an advantage in measuring quantity which has a low cycle fatigue. This method is suitable when there is also crack initiation, since the fatigue failure begins most of the time in the notch, crack or the area where higher stress concentration are located (Budynas, 2006). Therefore, when the stress exceeds the elastic limit, automatically plastic strain would take over. If fatigue failure would occur, there is a strong possibility of being plastic strain (Budynas, 2006). The strain life equation is shown in the equation (11)

\[
\frac{\Delta \varepsilon}{2} = \frac{\sigma_f}{E} \cdot (2 \cdot N_f)^b + \varepsilon_f \cdot (2 \cdot N_f)^c
\]  

(11)

where \(\frac{\Delta \varepsilon}{2}\) is the total strain amplitude, \(E\) is the Young’s Modulus, \(N_f\) is the number of cycles to failure, \(\sigma_f\) is the fatigue strength coefficient, \(b\) is the fatigue strength exponent, \(c\) is the fatigue ductility exponent and \(\varepsilon_f\) is the fatigue ductility coefficient.
With respect to the stress life method, in order to find the materials strength when it undergoes the action of fatigue loads, the materials undergo repeated or varying forces of certain magnitudes under while the cycle are counted to failure (Budynas, 2006). This method is based on S-N curves (Stress- Cycle curves) or Wöhler curve. The graphical representation of S-N curves is shown below for Aluminium Alloy and Structural Steel.

![S-N Curve](image)

**Figure 11. S- N Curves for Aluminium Alloy and Structural Steel (Fatemi, 2013)**

The stress life method is applied to the total life and does not differ from the initiation and the propagation of the crack (ANSYS, 2004). The method is mainly used for a high number of cycles which is usually more than $10^5$, this method deals mainly for HCF.

Regarding the stress method, it is important to describe the type of loading. There are two different types of loading: constant amplitudes with proportional loading and constant amplitudes with non-proportional loading.

The constant amplitudes with proportional loading is used since the loading step is constant and only one set of FE stress results is required to calculate the alternating and mean values (ANSYS, 2004). Figure 12 shows how to deal with constant amplitudes.
From the above figure the following equation can be derived (Equation 12, 13, 14 and 15):

\[ S_a = \frac{\Delta S}{2} = \frac{S_{\text{max}} - S_{\text{min}}}{2} \]  
\[ S_m = \frac{S_{\text{max}} + S_{\text{min}}}{2} \]  
\[ S_{\text{max}} = S_m + S_a \]  
\[ S_{\text{min}} = S_m - S_a \]  

In order to obtain the fatigue properties, there are two common reference test condition used (Fatemi, 2013). The first one is \( R = -1 \) which is called fully reversed condition. This implies that \( S_{\text{min}} = -S_{\text{max}} \). The second one is \( R = 0 \), which means that \( S_{\text{min}} = 0 \), that is, pulsating tension. As it can be noticed from the above diagram with constant amplitudes loading type, once cycle is equal to two reverses (Fatemi, 2013). From Figure 12, a cycle can be defined as the smallest segment of the stress versus time history which is repeated periodically (Fatemi, 2013).
However, the constant amplitude with non-proportional loading is mostly suitable for nonlinear contact, compression only or bolt load (ANSYS, 2004). Therefore, it is not relevant for this dissertation.

The fracture mechanics approach is done by initially assuming the initial crack in order to determine the crack’s growth. This method is called “crack life”. It is used to determine inspection intervals. From the initial crack, the critical crack can be obtained from the crack growth. According to the ANSYS manual (2004), the crack initiation is determined by the strain life method while the fracture mechanics approach determines the crack life.

\[ Total \ Life = Crack \ initiation + Crack \ Life \]  

(16)

7. DELIMITATIONS

As part of the thesis, some limitations have to be considered in order to restrict the analysis carried out. Since computational fluid dynamics (CFD) is not our area of expertise and due to the lack of information, a CFD analysis will not be performed. Therefore, the aerodynamical loads which include lift force and drag force are not included. Furthermore, according to analytical calculations, the values of those forces are minimum with respect to the centrifugal load and the pressure. Thus, they are neglected along the thesis.

The model provided by the company consisted of two split hub, cast out of aluminum. An analysis of an alternative material is not carried out.

Due to the time constraint, vibrations coming from waves spread to the fan due to proximity of other fans are not performed.

The environment impact is another area to look at. As it has been explained in the background, the axial fan is subjected to extreme conditions of corrosion and temperature. The materials used in the design of the model are Aluminum Alloy and Structural Steel. Both materials have been design in room temperature of 22 °C. This means that running the fan at a higher and lower temperature might affect the performance of the axial fan. When manufacturing the model, in order to protect the axial fan against corrosion, the axial fan is painted. Although these factors can affect the analysis, it is assumed that the axial fan is
protected against these environmental conditions. Thus, they have not been taken into account when running the analysis.

Finally despite the parametric study, the general shape of the model (split hub joined together by bolting them) did not change, since it was supposed to be the expected design.

8. FINITE ELEMENT MODEL

In order to perform the analysis, all loads, boundary conditions, mesh and contact relations have to be defined. Along this chapter an explanation of how the model is defined in ANSYS Workbench with respect to these terms is presented.

8.1. LOADS

Two types of load were considered when performing the analysis. The first load was the centrifugal force. The centrifugal force is applied as a fictive force in order to achieve the equilibrium. Although it is a virtual force, it is defined like a volume; otherwise the results could not be trusted since the model would be rotating (Figure 13).

Figure 13. Fictive centrifugal load applied to get equilibrium
This load was applied in terms of the rotational speed. The software automatically generates the centrifugal force once the rotational speed is applied. Since the rotational speed is constant that means that it can be applied throughout the blade, as it is shown in Figure 14.

![Figure 14. Centrifugal force in terms of rotational speed](image)

Pressure load is the second type of load that the axial fan is subjected to. This is the pressure which goes on to the inner and outer of the fan. The load is applied in a face in the form of pressure load, as it is represented in Figure 15.
8.2. BOUNDARY CONDITIONS

Based on the concepts of strength of materials and taking into account that the model has been defined as a solid, there are three degrees of freedom. These degrees of freedom are the displacement in each axis and they have been fixed (Table 1). Furthermore, according to Dynamics, the rotations may not be considered. Otherwise, if a single point is considered, as the displacements are fixed, the model would rotate indefinitely without equilibrium. The boundary conditions applied to the model are represented in Figure 16.

<table>
<thead>
<tr>
<th>CONDITION</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement X</td>
<td>Fixed</td>
</tr>
<tr>
<td>Displacement Y</td>
<td>Fixed</td>
</tr>
<tr>
<td>Displacement Z</td>
<td>Fixed</td>
</tr>
</tbody>
</table>

Table 1. Boundary conditions applied in the boss
An important aspect when performing an FE analysis is to carry out the mesh. The axial fan was modelled using a 10 node tetrahedral element. This element was part of the software package used and the software uses the automatic meshing in order to create the mesh. The element is known in performing better when it comes to stress analysis in structures and solid mechanics (ANSYS, 2004). The element is also known for having four corner nodes, three degrees of freedom at each node (ANSYS, 2004). Figure 17 shows a typical 10 Node element.

Another concept pops up when performing the mesh is the difference between coarse and fine mesh. The literature shows that the coarse mesh cannot be trusted. In order to get accurate results, the mesh has to be finer when certain geometry changes such as fillets, holes, notches and sharp edges appear (Figure 18)(Cook et al, 2002). Differences between coarse and fine mesh are compared in future chapters.
**8.4. CONTACT COUPLING BETWEEN PARTS**

Another important concept when performing analysis is to define the contact conditions. There are five types of contact: bonded, no separation, rough, frictionless and frictional. ANSYS Workbench treats the last three type of contact as Non Linear (Rough, Frictionless and Frictional) (Save, 2013).
**BONDED**

The bonded surfaces can be defined as surfaces which are rigidly fixed or glued together. The surfaces are not allowed to separate or to slide. This means that the surface will be mated without taking into account the penetration, gap, loading and behavior of other parts and contact. ANSYS uses this setting as a default and it is suitable for linear type of contact (Save, 2013).

**NO SEPARATION**

No separation type of contact is almost similar to the bonded contact with the only difference that the parts are allowed to slide slightly. This setting is used when knowing that the parts will not be separated and the sliding will be always frictionless. However the sliding is very limited (Save, 2013).

**ROUGH**

In the case of the rough contact type, the parts are not allowed to slide. Nevertheless, the part can be separated depending on the loading (Save, 2013).

**FRICTIONLESS**

The parts can slide freely and the contact can be opened and close depending on the loading. This type of loading can give rigid body error under constraint. This effect can happen on any nonlinear contacts which are allowed to separate (Save, 2013).

**FRICTIONAL**

The parts can be slide if the user specifies the coefficient of friction and the parts can open and close as well. This case can also be explained by having two parts sliding against one another (Save, 2013).

When performing the analysis the bonded type of contact was used. This type of contact was most suitable due to the fact that there was not sliding and that the contact type was under the linear type of contact. This approach was used in order to simplify the problem.
9. IMPLEMENTATION

In this chapter, the procedure and the main considerations used in order to perform the analysis are explained. It is divided into two main parts, according to the type of analysis performed. The first part is called Software Analysis, while the second one is named Analytical Analysis.

9.1. SOFTWARE ANALYSIS

Different software has been used to perform the software analysis as it follows. The company provided the model in SolidWorks format. Using this software, the pertinent modifications (parametric study) has been done in order to get the final required model. Then, the model was saved as a step file and finally imported to ANSYS Workbench in order to carry out the respective finite element analysis.

The software analysis consists of the five following main parts: engineering considerations, a parametric study, a modal analysis, convergence analysis and fatigue analysis.

9.1.1. ENGINEERING CONSIDERATIONS

To carry out the software analysis, finite element analysis was performed in order to obtain the stresses using ANSYS Workbench software. Some considerations had to be taken into account before running the analysis.

- MATERIAL PROPERTIES

It is necessary in order to perform the analysis in Finite Element Software to specify the material properties of the model. The whole axial fan was modeled using Aluminum Alloy except for the fasteners (screw and washers), which were modeled using Structural Steel. A linearly elastic material model was used to perform the analysis. In Table 3, the main material properties of both elements are shown.
### Table 3. Material properties of Aluminum Alloy and Structural Steel

<table>
<thead>
<tr>
<th>Property</th>
<th>Aluminum Alloy</th>
<th>Structural Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>2770</td>
<td>7780</td>
</tr>
<tr>
<td>Young’s Modulus (Pa)</td>
<td>$7 \cdot 10^{10}$</td>
<td>$2.06 \cdot 10^{11}$</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
<td>0.3</td>
</tr>
<tr>
<td>Shear Modulus (Pa)</td>
<td>$2.7 \cdot 10^{10}$</td>
<td>$7.7 \cdot 10^{10}$</td>
</tr>
<tr>
<td>Tensile Yield Strength (Pa)</td>
<td>$2.8 \cdot 10^{8}$</td>
<td>$2.5 \cdot 10^{8}$</td>
</tr>
<tr>
<td>Tensile Ultimate Strength (Pa)</td>
<td>$3.1 \cdot 10^{8}$</td>
<td>$4.6 \cdot 10^{8}$</td>
</tr>
</tbody>
</table>

**9.1.2. PARAMETRIC STUDY**

As it was explained in Chapter 6.1, the parametric study can be defined as an iterative method developed in order to obtain acceptance criteria both for safety, cost, performance, shape and convenience. This iteration describes and provides certain relationships between different parameters. By means of the study and the comparison of the results obtained, different scenarios are created, suggesting an idea of an acceptable design for the model. To perform the parametric study, ANSYS Workbench was used. Three different parameters, whose modifications are expected to affect the concentration of the stress in the attachment point between the fan blades and the hub, are studied. These parameters are: rotational speed, fillet size of the blade and diameter of the blade neck.

**ROTATIONAL SPEED**

The literature shows that the rotational speed and the pressure load are dependent of one another. Therefore the purpose of this section was to establish the required relationship. It is expected that the increase in one of them, will entail an increase of the stresses. However, both the rotational speed and the pressure can be studied at the same time, since fans operate under a predictable set of laws concerning them. According to Sullivan (2006), a change in rotational speed of any fan will predictably change the pressure rise and power necessary in order to operate the fan at these new values.

The relation between the rotational speed and the pressure can be stated according to Equation (17)

$$\frac{p_1}{p_2} = \left(\frac{N_1}{N_2}\right)^2$$  \hspace{1cm} (17)
where \( P \) is referred to the pressure and \( N \) to the rotational speed.

In order to perform the parametric analysis, seven different values of the rotational speed and the pressure had been taking into account (Table 4). Both parameters starts at the maximum values allowed, and it is decreasing as it has been explained above.

<table>
<thead>
<tr>
<th>ROTATION SPEED (rpm)</th>
<th>PRESSURE (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>9000</td>
</tr>
<tr>
<td>2700</td>
<td>7290</td>
</tr>
<tr>
<td>2430</td>
<td>5905</td>
</tr>
<tr>
<td>2187</td>
<td>4783</td>
</tr>
<tr>
<td>1968</td>
<td>3874</td>
</tr>
<tr>
<td>1771</td>
<td>3138</td>
</tr>
<tr>
<td>1594</td>
<td>2542</td>
</tr>
</tbody>
</table>

Table 4. Values of rotational speed and static pressure used

**FILLET SIZE OF THE BLADE**

In Figure 19 and Figure 20, the green color represents the fillet of the blade which would need to be modified. It is expected that, owing to both the air pressure and the centrifugal load, the increase of the fillet size will result in the decrease of the stress.

Figure 19. Detail of the fillet on the blade in the initial design (mm)
The fillet size has been modified in such a way that it has been taken three different oversized values with respect to the original model provided by the company, and three different more undersized, apart from the original one. Each of them has been analyzed for each value of the rotational speed. Table 5 is represented in order to show combination used to perform the analysis.

<table>
<thead>
<tr>
<th></th>
<th>10</th>
<th>15</th>
<th>20</th>
<th>26.5</th>
<th>30</th>
<th>35</th>
<th>40</th>
<th>Fillet Size (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3000</td>
<td>2700</td>
<td>2430</td>
<td>2187</td>
<td>1968</td>
<td>1771</td>
<td>1594</td>
<td>Rotational speed (rpm)</td>
</tr>
</tbody>
</table>

Table 5: Values to proceed with the fillet size analysis (All combinations were used to perform the analysis)

**DIAMETER OF THE NECK OF THE BLADE**

A parametric study of the neck of the blade was performed (Figure 21 and Figure 22). This is a crucial part due to the geometry, since the neck of the blade is the support of the blade, that is, the base of the blade. Then, it is expected to give high stress concentration. Crack would start propagating either from the top of the neck of the blade or from the bottom. For this reason, a parametric study was required in order to examine what will be the outcome if the
neck of the blade had increased while rotating at the maximum speed. Therefore, three parameters were chosen and analyzed at different rotational speed. A neck diameter of 45 mm had been used on the preliminary design but this value was increased in such a way that the stress around the blade would decrease or increase when it would be subjected to certain rotational speed parameters. The values chosen were 47, 49 and 51 mm. The preliminary design was the design provided by the company with no changes on any of the geometrical parameters.

The first couple of analyses were performed using the preliminary design subjected into different rotational speed and pressure load. The remaining analyses were performed using the new parameters. The new parameters were analyzed using the worst condition and the lowest condition. That means a diameter of 51 mm was analyzed using the rotational speed of 3000 rpm. By contrast, an analysis was performed using the lowest rotational speed of 1594 rpm.

![Figure 21. Magnification of the neck of the blade](image)
9.1.3. MODAL ANALYSIS

The modal analysis was performed using the modal toolbox on ANSYS Workbench in order to determine the undamped free vibration and the modes shapes of the model. The modal toolbox is very similar to the structural analysis toolbox, with the only difference that it is not possible to apply any type of loading when performing the modal analysis. This is realistic since the undamped free vibration only depends on the mass and the stiffness of the structure.

Usually there are four steps when performing the modal analysis on ANSYS Workbench. The first is to build the model, then choose analysis type and option, after that apply boundary condition and solve and finally, review results.

In this case the model was already built which meant that choose the analysis type and options were the second task to perform. In the analysis type, the material of the structure was assigned, the model was meshed using a 10-node element and the software chose the mode extraction methods. For this case, Block Lanczos setting was chosen as a default setting since the Block Lanczos setting is recommended for most applications. This method consists of substituting block algorithms for matrix block multiplies and block solvers for matrix-vector products and simple solvers in unblocked algorithms. This setting was suitable since it has the
possibility of extracting larger number of modes in models, works well in rigid body, and it can be used when the model is complex with a mixture of shell, solids and beam (ANSYS, 2004).

To apply the boundary condition was the third step. The boundary condition was applied as it has been explained in section 8.2. The boundary condition was applied over the whole model and not using the symmetry is because a symmetry boundary condition would result in the shaped modes being symmetrical and consequently, some modes would be missing. After the boundary conditions, the next step was to solve the analysis and review the results.

9.1.4. FATIGUE ANALYSIS

The fatigue life gives an indication of the remaining life for a specific fatigue analysis. Since the loading was of constant amplitudes, the results would represent the remaining cycle until the parts fail due to fatigue. Fatigue loading can be defined as the type of loading which results in the cyclic variations in the applied stress on a component (Gopinath and Mayuram, 2014). The fatigue load will have different frequency depending on the rotational speed. For instance running the fan at 3000 rpm, a frequency of 50 Hz will be obtained, for 2187 rpm will give a frequency of 36.5 Hz, 1771 rpm will give a frequency of 29.5 Hz and finally 1594 gives a frequency of 26.6 Hz. When performing the analysis, the stress life method was used in order to estimate the remaining life. Several analyses were performed using different rotational speed (Fatemi, 2013). The life was estimated at certain point of interested which include the whole blade, the neck of the blade and the fillet in front of the blade. When performing the analysis, the stress life approach was chosen in order to estimate the remaining life, obtaining the stresses which are required to use the S-N curves.

The analysis was performed in the same way that the previous analyses were done in order to find the von Mises stress. However, in this case, there was an extra tool provided on the software package called Fatigue tool. From the tool the life estimation of the model could be estimated using the von Mises stresses obtained in the analysis. After that finding out the loading type was the next step in order to perform the life estimation.

In this case, constant amplitude with proportional loading is the most suitable due to the fact that this kind of loading has constant amplitude. Therefore, the loading helps to find out if the load has a constant maximum value or values which change continually with time (ANSYS,
2004). The loading is said to be proportional also because of the fact that only one set of FE results are needed.

9.2. ANALITICAL ANALYSIS

The analytical analysis was performed in order to get an estimated value of the different analysis through the theory. Different theory models were used and some simplifications were taken into account in order to get the results. Due to this, it is important to mention that the results were an approximation. Moreover, some of the parameters and material properties required were taken from the software SolidWorks and material catalogue.

This step is divided into five main parts: contact pressure, bending stress in the blade, centrifugal stress and stress concentration around the hole.

9.2.1. CONTACT PRESSURE

In this case, the boss and the shaft are considered as cylinders which were rotating, with inner pressure and massive axles with outer pressure. It was necessary to consider equations of strength of material to get the results since they were influenced by multiaxial state of stress and strain. That is, equilibrium, compatibility and the constitutive law equations were applied to calculate the contact pressure. Figure 17 shows a sketch with a simplification of the model.

Figure 23. Sketch of the assembly between hub and boss
It was important to take into account that two different parts were required in order to get the complete solution of each cylinder (Stigh, 2012). The first part was the solution of the homogeneous differential equation, while the second one was the solution of the particular solution. Finally, throughout the application of boundary conditions on $r=a$ and $r = b$, the four constants of integration could be determined (Stigh, 2012).

In Table 6 the material properties of the hub and the aluminum are shown. Here are some of the material properties:

<table>
<thead>
<tr>
<th>ALUMINUM HUB</th>
<th>STEEL SHAFT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus</td>
<td>$E_1 = 70 \text{ GPa}$</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>$\nu_1 = 0.3$</td>
</tr>
<tr>
<td>Density</td>
<td>$\rho_1 = 2770 \text{ kg/m}^3$</td>
</tr>
<tr>
<td>Radius</td>
<td>$a = 55 \text{ mm}$</td>
</tr>
</tbody>
</table>

Table 6. Material properties to take into account in the analysis

Regarding to the maximum and minimum diametric grip, it was calculated according to the tolerances in order to obtain the value for the worst scenarios. The tolerance represents the permissible limit or limits of variation in a measured value when the fan is manufactured. The tolerances of the hub are class G7, which means minimum tolerance of +0.01 mm and maximum tolerance of +0.04 mm. The tolerances of the shaft are class M6, which means a minimum tolerance of +0.011 and a maximum tolerance of +0.03 mm. Therefore, based on these tolerances, the maximum diametric grip is 3 µm, while the minimum is 1.1 µm.

**ALUMINIUM HUB**

On one hand, the radial displacement of the hub was determined by Equation (18) (Stigh, 2012):

$$u_1 = u_{h1} + u_{p1} = A_1 \cdot \frac{r}{a} + A_2 \cdot \frac{r}{a} - (1 - \nu_1^2) \cdot \frac{\rho_1 \cdot \omega^2}{8 \cdot E_1} \cdot r^3$$  (18)

On the other hand, the radial stress was defined by Equation (19) (Stigh, 2012).

$$\sigma_{r1} = \sigma_{hr1} + \sigma_{pr1} =$$
$$= \frac{E_1}{1 - \nu_1^2} \cdot \left[(1 + \nu_1) \cdot \frac{A_1}{a} - (1 - \nu_1) \cdot \frac{A_2 \cdot a}{r^2}\right] - (3 + \nu_1) \cdot \frac{\rho_1 \cdot \omega^2}{8} \cdot r^2$$  (19)
Where \( w \) is the rotation speed, and the rest of parameters are defined in Table 6.

**STEEL SHAFT**

In a similar way as in the previous case, the displacement of the shaft is determined by equation (20) (Stigh, 2012).

\[
\begin{align*}
  u_2 &= u_{h2} + u_{p2} = A_1 \cdot \frac{r}{a} + A_2 \cdot \frac{r}{a} - (1 - v_2^2) \cdot \frac{\rho_2 \cdot \omega^2}{8 \cdot E_2} \cdot r^3 \\
  (20)
\end{align*}
\]

And the radial stress by Equation (21) (Stigh, 2012).

\[
\begin{align*}
  \sigma_{r2} &= \sigma_{hr2} + \sigma_{pr2} = \\
  &= \frac{E_2}{1 - v_2^2} \cdot \left[ (1 + v_2) \cdot \frac{B_1}{a} - (1 - v_2) \cdot \frac{B_2 \cdot a}{r^2} \right] - (3 + v_2) \cdot \frac{\rho_2 \cdot \omega^2}{8} \cdot r^2 \\
  (21)
\end{align*}
\]

**BOUNDARY CONDITIONS**

1. \( r = 0 \to u_2(0) = 0 \)

Inserting this boundary condition in equation (18) it is obtained the following term:

\[ A_2 \cdot \frac{a}{r} = 0 \text{ since it tends to } \infty \text{ when } r \text{ tends to } 0. \text{ Therefore:} \]

\[ A_2 = 0 \]

2. \( r = b \to \sigma_{r2}(b) = 0 \) (Free surface)

Replacing in equation (21).

\[
\begin{align*}
  \frac{E_2}{1 - v_2^2} \cdot \left[ (1 + v_2) \cdot \frac{A_1}{a} - (1 - v_2) \cdot \frac{A_2 \cdot a}{r^2} \right] - (3 + v_2) \cdot \frac{\rho_2 \cdot \omega^2}{8} \cdot r^2 &= 0 \\
  (22)
\end{align*}
\]

3. \( r = a \)

A representation of this boundary condition is shown in Figure 24. According to it:

\[ \sigma_{r1}(a) = -P \]

Replacing in terms of rotational speed and pressure (20):
\[-P = \frac{E_1}{1 - v_1^2} \cdot \left[ (1 + v_1) \cdot \frac{A_1}{a} - (1 - v_1) \cdot \frac{A_2}{a^2} \right] - (3 + v_1) \cdot \frac{\rho_1 \cdot \omega^2}{8} \cdot a^2 \]

\[A_1 = \frac{-P + 3.46 \cdot \omega^2}{1.8 \cdot 10^{12}}\]

Figure 24. Boundary condition for $\sigma_{r1}(a)$

4. $r = a$

Similarly and based on the sketch of Figure 25:

$\sigma_{r2}(a) = -P$

Figure 25. Boundary Condition for $\sigma_{r2}(a)$

Replacing in terms of rotational speed and pressure (21):
\[-P = \frac{E_2}{1 - \nu_2^2} \left[ \left(1 + \nu_2\right) \cdot \frac{B_1}{a} - \left(1 - \nu_2\right) \cdot \frac{B_2 \cdot a}{a^2} \right] - \left(3 + \nu_2\right) \cdot \frac{\rho_2 \cdot \omega^2}{8} \cdot a^2 \]

\[-P = 5.3506459 \cdot 10^{12} \cdot B_1 - 2.8811 \cdot 10^{12} \cdot B_2 - 9.707981 \cdot \omega^2 \quad (23)\]

- Operating Equation (22) = Equation (23)

\[B_2 = \frac{P + 29.1 \cdot \omega^2}{2.2 \cdot 10^{12}}\]

- Operating \(b^2\) = Equation (22) = \(a^2\) = Equation (23):

\[B_1 = \frac{0.003 \cdot P + 0.4 \cdot \omega^2}{48.6 \cdot 10^9}\]

**KINEMATICAL CONDITION**

Finally, applying kinematical condition at \(r = a\) (Figure 26):

\[
\frac{\Delta}{2} = u_2(a) - u_1(a) \quad (24)
\]

![Figure 26. Sketch of kinematical condition](image)

The results of the contact pressure analysis for each rotational speed and worst scenarios are shown in Table 7.
The bending stress in the blade was calculated using Equation 25:

\[
\sigma = \frac{M}{W_b} = \frac{M}{I} \cdot |y|_{\text{max}}
\]  
(25)

where \( I \) is the moment of inertia, \( y \) is the distance from the centroid to the outer part of the blade and \( M \) is the moment.

Some estimation was done to simplify the model and to calculate the moment. The moment of inertia is taken from SolidWorks. Furthermore, the bending stress was calculated using the pressure load. Although the pressure load was given in Pa, the load was converted into Newton by using the following formula (Equation 26):

\[
P = \frac{F}{A}
\]  
(26)

where \( F \) was the applied load and \( A \) the cross section area.

Finally, knowing that the force was applied as a uniformly distributed load in the blade (Figure 27) and simplifying the blade as a cantilever beam, the moment could be calculated by applying equilibrium. Therefore, the moment was given by Equation (27):

\[
M = F \cdot \frac{a^2}{2}
\]  
(27)

<table>
<thead>
<tr>
<th>rpm</th>
<th>( \Delta = 3 \mu m )</th>
<th>( \Delta = 1.1 \mu m )</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>12 MPa</td>
<td>3.2 MPa</td>
</tr>
<tr>
<td>2700</td>
<td>12.4 MPa</td>
<td>3.5 MPa</td>
</tr>
<tr>
<td>2430</td>
<td>12.7 MPa</td>
<td>3.8 MPa</td>
</tr>
<tr>
<td>2187</td>
<td>12.9 MPa</td>
<td>4.1 MPa</td>
</tr>
<tr>
<td>1968</td>
<td>13.1 MPa</td>
<td>4.3 MPa</td>
</tr>
<tr>
<td>1771</td>
<td>13.2 MPa</td>
<td>4.4 MPa</td>
</tr>
<tr>
<td>1594</td>
<td>13.4 MPa</td>
<td>4.6 MPa</td>
</tr>
</tbody>
</table>

Table 7. Table of results of contact analysis

9.2.2. BENDING STRESS IN THE BLADE
Both the parameters required to calculate the bending stress in the lower of the blade and the final result are shown in Table 8.

<table>
<thead>
<tr>
<th>PARAMETERS</th>
<th>VALUES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Force (N)</td>
<td>966.7</td>
</tr>
<tr>
<td>Moment (N · m)</td>
<td>138.03</td>
</tr>
<tr>
<td>Moment of Inertia (m⁴)</td>
<td>6.4 · 10⁻⁶</td>
</tr>
<tr>
<td>Bending Stress Result (MPa)</td>
<td>5.8</td>
</tr>
</tbody>
</table>

Table 8. Table of results of bending stress analysis in the blade

### 9.2.3. CENTRIFUGAL STRESS

The centrifugal stress at the lower of the blade was calculated in order to compare the analytical results with the FE solutions. The equation of the centrifugal stress was derived from the basic stress formula shown in Equation (28).

\[
\sigma = \frac{F}{A} \tag{28}
\]

where \( F \) is the reaction force and \( A \) is the cross section area at the bottom of the blade.

When expanding \( F \) using Newton’s Second Law (Equation 29) which is in the normal direction

\[
R_A = m \times a_n \tag{29}
\]
Then, the linear acceleration could be converted into rotational acceleration by using Equation (30)

\[ a_n = r \cdot \omega^2 \]

(30)

where \( \omega \) is the rotational speed.

Therefore, replacing, the final centrifugal stress could be determined by Equation (31)

\[ \sigma = \frac{M \cdot r_c \cdot \omega^2}{A} \]

(31)

where \( M \) is the mass of the blade, \( r_c \) is the distance from the center of mass to the outer part of the blade, \( \omega \) is the rotational speed and \( A \) is the cross section area. Figure 28 shows a sketch of the fan blade.

![Figure 28. Sketch to calculate the centrifugal stress](image)

In a similar way than in previous chapter, the parameters to calculate the centrifugal stress as well as the centrifugal stress result are represented in Table 9.
### 9.2.4. STRESS CONCENTRATION AROUND THE WHOLE

Linear Elastic Fracture Mechanics was used in order to evaluate where the crack start to propagate around the hole. The maximum stress around the hole was equal to 94 MPa and knowing that the stress intensity was given by Equation 32:

$$K_I = \sigma_0 \cdot \sqrt{\pi \cdot a \cdot f_3(s)}$$

(32)

where $s = \frac{a}{r+a}$

From the above equation, the literature suggested an initial crack size value of 20 µm (Chattopdhay, 2008). Some basic assumption was made in which a 2D case was chosen over 3D. By using this assumption, automatically the shear stress was not neglected.

From this, $s$ was calculated to be equal to 0.003 which made it valid since the value is between the ranges proposed by Tada Data provided by Anderson (2005). From the theory, based on Anderson (2005), the crack will start propagating when the stress intensity is equal to the fracture toughness, which means that $K_I=K_{IC}$.

In Table 10, the parameters to calculate where the crack start to propagate are represented.

<table>
<thead>
<tr>
<th>PARAMETER</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial Crack (µm)</td>
<td>20</td>
</tr>
<tr>
<td>$f_3(s)$</td>
<td>0.003</td>
</tr>
<tr>
<td>Fracture toughness (MPa ∙ √m)</td>
<td>36</td>
</tr>
<tr>
<td><strong>Crack Start Propagating at (GPa)</strong></td>
<td><strong>1.4</strong></td>
</tr>
</tbody>
</table>

Table 10. Table of results of crack size around the hole analysis
10. RESULTS

The results obtained throughout the software analyses are shown in the next chapters.

10.1. FILLET SIZE OF THE BLADE

From Table 11 to Table 17, the results with tabulated format of the different variation of the blade fillet size are shown according to Abdullah and Schlattmann (2012). For each rotational speed and pressure, the parameters to take into account are: the maximum stress, the stress in the fillet of the blade and the stress in the neck of the blade. Also, in Figure 29 an analysis of the whole blade is shown. All the stresses obtained in the analyses were von Mises stresses.

![Figure 29. Stress analysis in the whole blade](image-url)
<table>
<thead>
<tr>
<th>Rotational speed (rpm)</th>
<th>Max. Stress (MPa) (Inside the boss)</th>
<th>Blade Fillet Stress (MPa)</th>
<th>Neck of the Blade Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>356</td>
<td>31.9</td>
<td>27.0</td>
</tr>
<tr>
<td>2700</td>
<td>302</td>
<td>27.6</td>
<td>23.3</td>
</tr>
<tr>
<td>2430</td>
<td>240</td>
<td>21.3</td>
<td>17.6</td>
</tr>
<tr>
<td>2187</td>
<td>197</td>
<td>15.4</td>
<td>14.3</td>
</tr>
<tr>
<td>1968</td>
<td>157</td>
<td>10.9</td>
<td>11.1</td>
</tr>
<tr>
<td>1771</td>
<td>121</td>
<td>9.6</td>
<td>9.8</td>
</tr>
<tr>
<td>1594</td>
<td>110</td>
<td>8.1</td>
<td>7.6</td>
</tr>
</tbody>
</table>

Table 11. Table of results for a blade fillet size of 40 mm

<table>
<thead>
<tr>
<th>Rotational speed (rpm)</th>
<th>Max. Stress (MPa) (Inside the boss)</th>
<th>Blade Fillet Stress (MPa)</th>
<th>Neck of the Blade Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>368</td>
<td>31.8</td>
<td>26.4</td>
</tr>
<tr>
<td>2700</td>
<td>301</td>
<td>27.3</td>
<td>22.5</td>
</tr>
<tr>
<td>2430</td>
<td>241</td>
<td>26.3</td>
<td>17.0</td>
</tr>
<tr>
<td>2187</td>
<td>192</td>
<td>15</td>
<td>12.2</td>
</tr>
<tr>
<td>1968</td>
<td>158</td>
<td>11.1</td>
<td>10.2</td>
</tr>
<tr>
<td>1771</td>
<td>121</td>
<td>10.6</td>
<td>8.6</td>
</tr>
<tr>
<td>1594</td>
<td>104</td>
<td>7.2</td>
<td>7.4</td>
</tr>
</tbody>
</table>

Table 12. Table of results for a blade fillet size of 35 mm
### Table 13. Table of results for a blade fillet size of 30 mm

<table>
<thead>
<tr>
<th>Rotational speed (rpm)</th>
<th>Max. Stress (MPa) (Inside the boss)</th>
<th>Blade Fillet Stress (MPa)</th>
<th>Neck of the Blade Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>351</td>
<td>27.6</td>
<td>27.2</td>
</tr>
<tr>
<td>2700</td>
<td>296</td>
<td>21.1</td>
<td>22.6</td>
</tr>
<tr>
<td>2430</td>
<td>230</td>
<td>18.6</td>
<td>17.12</td>
</tr>
<tr>
<td>2187</td>
<td>191</td>
<td>14.1</td>
<td>12.8</td>
</tr>
<tr>
<td>1968</td>
<td>151</td>
<td>11.5</td>
<td>8.9</td>
</tr>
<tr>
<td>1771</td>
<td>116</td>
<td>7.8</td>
<td>8.8</td>
</tr>
<tr>
<td>1594</td>
<td>99.1</td>
<td>7.1</td>
<td>7.4</td>
</tr>
</tbody>
</table>

### Table 14. Table of results for a blade fillet size of 26.5 mm

<table>
<thead>
<tr>
<th>Rotational speed (rpm)</th>
<th>Max. Stress (MPa) (Inside the boss)</th>
<th>Blade Fillet Stress (MPa)</th>
<th>Neck of the Blade Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>367</td>
<td>25.7</td>
<td>26.5</td>
</tr>
<tr>
<td>2700</td>
<td>308</td>
<td>28.5</td>
<td>23.3</td>
</tr>
<tr>
<td>2430</td>
<td>241</td>
<td>24.7</td>
<td>17.4</td>
</tr>
<tr>
<td>2187</td>
<td>201</td>
<td>17.2</td>
<td>13.5</td>
</tr>
<tr>
<td>1968</td>
<td>158</td>
<td>12.9</td>
<td>11.8</td>
</tr>
<tr>
<td>1771</td>
<td>121</td>
<td>8.4</td>
<td>9.5</td>
</tr>
<tr>
<td>1594</td>
<td>104</td>
<td>7.2</td>
<td>7.5</td>
</tr>
<tr>
<td>Blade Fillet Size</td>
<td>Rotational speed (rpm)</td>
<td>Max. Stress (MPa) (Inside the boss)</td>
<td>Blade Fillet Stress (MPa)</td>
</tr>
<tr>
<td>------------------</td>
<td>------------------------</td>
<td>-------------------------------------</td>
<td>--------------------------</td>
</tr>
<tr>
<td>20 mm</td>
<td>3000</td>
<td>367</td>
<td>28.1</td>
</tr>
<tr>
<td></td>
<td>2700</td>
<td>316</td>
<td>27</td>
</tr>
<tr>
<td></td>
<td>2430</td>
<td>241</td>
<td>25.7</td>
</tr>
<tr>
<td></td>
<td>2187</td>
<td>206</td>
<td>20.0</td>
</tr>
<tr>
<td></td>
<td>1968</td>
<td>158</td>
<td>11.9</td>
</tr>
<tr>
<td></td>
<td>1771</td>
<td>121</td>
<td>8.7</td>
</tr>
<tr>
<td></td>
<td>1594</td>
<td>104</td>
<td>7.7</td>
</tr>
</tbody>
</table>

Table 15. Table of results for a blade fillet size of 20 mm

<table>
<thead>
<tr>
<th>Blade Fillet Size</th>
<th>Rotational speed (rpm)</th>
<th>Max. Stress (MPa) (Inside the boss)</th>
<th>Blade Fillet Stress (MPa)</th>
<th>Neck of the Blade Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>15 mm</td>
<td>3000</td>
<td>366</td>
<td>31</td>
<td>31</td>
</tr>
<tr>
<td></td>
<td>2700</td>
<td>319</td>
<td>28.8</td>
<td>26.1</td>
</tr>
<tr>
<td></td>
<td>2430</td>
<td>240</td>
<td>21.1</td>
<td>17.4</td>
</tr>
<tr>
<td></td>
<td>2187</td>
<td>242</td>
<td>18.9</td>
<td>15.7</td>
</tr>
<tr>
<td></td>
<td>1968</td>
<td>158</td>
<td>16.9</td>
<td>13.6</td>
</tr>
<tr>
<td></td>
<td>1771</td>
<td>121</td>
<td>9.4</td>
<td>8.8</td>
</tr>
<tr>
<td></td>
<td>1594</td>
<td>103</td>
<td>7.5</td>
<td>7.5</td>
</tr>
</tbody>
</table>

Table 16. Table of results for a blade fillet size of 15 mm
<table>
<thead>
<tr>
<th>Rotational speed (rpm)</th>
<th>Max. Stress (MPa) (Inside the boss)</th>
<th>Blade Fillet Stress (MPa)</th>
<th>Neck of the Blade Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>464</td>
<td>30</td>
<td>27.6</td>
</tr>
<tr>
<td>2700</td>
<td>360</td>
<td>29.6</td>
<td>24.5</td>
</tr>
<tr>
<td>2430</td>
<td>304</td>
<td>22.7</td>
<td>17.8</td>
</tr>
<tr>
<td>2187</td>
<td>258</td>
<td>20.5</td>
<td>14.9</td>
</tr>
<tr>
<td>1968</td>
<td>200</td>
<td>17.5</td>
<td>11.8</td>
</tr>
<tr>
<td>1771</td>
<td>124</td>
<td>14.6</td>
<td>9.5</td>
</tr>
<tr>
<td>1594</td>
<td>114</td>
<td>11.3</td>
<td>8.5</td>
</tr>
</tbody>
</table>

Table 17. Table of results for a blade fillet size of 10 mm

After these results, in order to have both a better understanding and easier way to compare the results, from Figure 30 to Figure 35 graphs comparing the four parameters against the blade fillet size and each rotational speed (horizontal axis) are plotted. It is important to notice that, from the results obtained in the analysis, the maximum stress occurred at the same point (the boss) while using different blade fillet size. Moreover, all the values are converged.

Figure 30. Maximum stress against blade fillet size for each rotational speed
Figure 31. Stress in the fillet against blade fillet size for each rotational speed

Figure 32. Stress in the neck of the blade against blade fillet size for each rotational speed
Figure 33. Maximum stress against rotational speed for each blade fillet size

Figure 34. Stress in the fillet against rotational speed for each blade fillet size
10.2. DIAMETER OF THE NECK OF THE BLADE

In Table 18, the results for each parameter of interest are given in a tabulated and graph format (Abdullah and Schlattmann, 2012).

<table>
<thead>
<tr>
<th>Analyses Number</th>
<th>Ω (rpm)</th>
<th>Max. Stress (MPa) (Inside the boss)</th>
<th>Stress Blade fillet (MPa)</th>
<th>Stress Neck of the Blade (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>3000</td>
<td>472</td>
<td>84</td>
<td>32.5</td>
</tr>
<tr>
<td>2</td>
<td>2700</td>
<td>383</td>
<td>68</td>
<td>26</td>
</tr>
<tr>
<td>3</td>
<td>2430</td>
<td>309</td>
<td>53</td>
<td>21</td>
</tr>
<tr>
<td>4</td>
<td>2187</td>
<td>251</td>
<td>44</td>
<td>19</td>
</tr>
<tr>
<td>5</td>
<td>1968</td>
<td>203</td>
<td>35.3</td>
<td>14</td>
</tr>
<tr>
<td>6</td>
<td>1771</td>
<td>166</td>
<td>29</td>
<td>10.8</td>
</tr>
<tr>
<td>7</td>
<td>1594</td>
<td>133</td>
<td>23</td>
<td>9.10</td>
</tr>
<tr>
<td>8 (49 mm)</td>
<td>2187</td>
<td>218</td>
<td>43.0</td>
<td>19</td>
</tr>
<tr>
<td>9 (51 mm)</td>
<td>3000</td>
<td>407</td>
<td>82</td>
<td>35</td>
</tr>
<tr>
<td>10 (47 mm)</td>
<td>1594</td>
<td>155</td>
<td>23</td>
<td>10.7</td>
</tr>
</tbody>
</table>

Table 18. Table of results of the parametric study in the neck of the blade
After presenting the result in a table, in a similar way as in the previous chapter, from Figure 36 to Figure 38 graphs are plotted for each scenario. When plotting the graph, the horizontal axis was labeled as rotational speed while the vertical axis was labeled as the von Mises stress in different points. The legend specifies that there are two types of graphs where the first one represents the stress as per the initial design, where initial design means the preliminary design which was provided by the company. The other line represents the change of the parameters. Table 18 shows all the tabulated values of each parameter used in the neck of the blade and the value of the preliminary design (26.5 mm) as per the company’s drawings.
Figure 37. Stress at the blade fillet against rotational speed

Figure 38. Stress at the neck of the blade against rotational speed
10.3. CONVERGENCE

Convergence both on the fillet of the blade and the neck of the blade is plotted in Figure 39 and Figure 40. Furthermore, in Table 19 and Table 20, either percentage of change, or number of elements, or number of nodes and element size taken into account are shown respectively.

![Figure 39. Convergence on the fillet of the blade](image)

<table>
<thead>
<tr>
<th>VON MISES STRESS (Pa)</th>
<th>CHANGE (%)</th>
<th>ELEMENT SIZE (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.00 2.48 \cdot 10^6</td>
<td>-</td>
<td>0.01018</td>
</tr>
<tr>
<td>2.00 2.96 \cdot 10^6</td>
<td>19.3558</td>
<td>0.001002</td>
</tr>
<tr>
<td>3.00 2.98 \cdot 10^6</td>
<td>0.6757</td>
<td>0.000157</td>
</tr>
</tbody>
</table>

Table 19. Table of results of the convergence analysis on the blade fillet
Figure 40. Convergence on the neck of the blade

<table>
<thead>
<tr>
<th>VON MISCE</th>
<th>CHANGE (%)</th>
<th>ELEMENT SIZE</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRESS (Pa)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>$5.31 \cdot 10^7$</td>
<td>-</td>
</tr>
<tr>
<td>2</td>
<td>$5.44 \cdot 10^7$</td>
<td>2.47</td>
</tr>
<tr>
<td>3</td>
<td>$5.5005 \cdot 10^7$</td>
<td>1.112</td>
</tr>
</tbody>
</table>

Table 20. Table of results of the convergence analysis on the neck of the blade
10.4. MODAL ANALYSIS RESULTS

The results of the natural frequencies obtained through the modal analysis are shown in Table 21. Furthermore, in Figure 41 is represented a graph plotting the mode numbers against the Eigen frequencies. Also, it is compared the coarse mesh against the fine mesh (Figure 42 to Figure 44).

<table>
<thead>
<tr>
<th>MODE</th>
<th>FREQUENCY (Hz) (Fine mesh)</th>
<th>FREQUENCY (Hz) (Coarse mesh)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>16.8</td>
<td>17</td>
</tr>
<tr>
<td>2</td>
<td>63.2</td>
<td>64.9</td>
</tr>
<tr>
<td>3</td>
<td>63.3</td>
<td>65</td>
</tr>
<tr>
<td>4</td>
<td>67.3</td>
<td>68.8</td>
</tr>
<tr>
<td>5</td>
<td>67.5</td>
<td>69.6</td>
</tr>
<tr>
<td>6</td>
<td>67.9</td>
<td>69.6</td>
</tr>
</tbody>
</table>

Table 21. Table of results of modal analysis

![Mode number vs Eigenfrequency](image)

Figure 41. Mode number against natural frequencies and coarse mesh against fine mesh
Figure 42. Picture of the fine mesh on the whole blade

Figure 43. Detail of the fine mesh on the blade
10.5. FATIGUE ANALYSIS

In a similar way than in previous analyses, the constant amplitude load is represented in Figure 45. The fatigue loading cycle is caused by the variable stress at a point with respect to the time (Gopinath and Mayuram, 2014). The stress at a point is varied due to the rotation of the fan blades. The origin of the fatigue can be due to the geometry change where stress concentration may occur. This will result in a fatigue crack which will occur at the discontinuities in the material (Budynas, 2006). The other origin of fatigue may also be in the microstructure of the material which may contain voids (Findlay and Harrison, 2002). Furthermore, the relation between the rotational speeds against the number of cycles is represented in Figure 46 and finally pictures of the fillet of the blade and the neck of the blade are represented in Figure 47 and Figure 48 where it is shown the remaining cycle before the part fails due to fatigue. Table 22 shows the results of the fatigue analysis where N represents the number of cycles where cycle represents one complete turn of the axial fan.
<table>
<thead>
<tr>
<th>ROTATIONAL SPEED (rpm)</th>
<th>BLADE (N)</th>
<th>NECK OF THE BLADE (N)</th>
<th>FILLET OF THE BLADE (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>10000</td>
<td>4704</td>
<td>67972</td>
</tr>
<tr>
<td>2187</td>
<td>1.2 \cdot 10^6</td>
<td>2.9 \cdot 10^6</td>
<td>2.9 \cdot 10^7</td>
</tr>
<tr>
<td>1771</td>
<td>6.8 \cdot 10^7</td>
<td>5.6 \cdot 10^7</td>
<td>1 \cdot 10^8</td>
</tr>
<tr>
<td>1594</td>
<td>1 \cdot 10^9</td>
<td>1 \cdot 10^8</td>
<td>1 \cdot 10^8</td>
</tr>
</tbody>
</table>

Table 22. Table of results of fatigue analysis

Figure 45. Constant amplitude load obtained from ANSYS analyses (Fatemi 2013)
Figure 46. Rotational speed against Number of cycles, where N stands for the number of cycle

Figure 47. Fatigue Analysis on the Fillet of blade at 1594 rpm
Figure 48. Fatigue Analysis on the neck of the blade at 1594 rpm

11. ANALYSIS

MAXIMUM STRESS

When analyzing the results obtained in the FE analysis, the area where the maximum stress occurs was studied. This very small area was located in the fillet of the blade and the fillet of the boss (Figure 49, Figure 50 and Figure 51). Therefore, it can be stated that this area was affected by stress singularities. As mentioned in the literature review, these singularities are caused by the discretization error during the mesh generation.
Figure 49. Maximum stress on the fillet of the boss

Figure 50. Detail of maximum Stress on the fillet of the boss

Figure 51. Maximum stress in the fillet of the blade
As the previous researches suggest, these stresses can be ignored and deleted, changing the measure scale. This thesis took into account these considerations. The results of the maximum stresses were:

Evaluating the initial design (neck of the blade size equal to 45 mm and blade fillet size equal to 26.5 mm), using a rotational speed higher than 1771 rpm with its respective pressure loads, the maximum stress was found to be higher than 203 MPa. Therefore, by using the safety factor equation and reminding that the ultimate tensile strength is equal to 310 MPa, it can be checked that \( SF < 2 \). This safety factor did not take into consideration fatigue and the analysis was performed using a linearly elastic material. Thus, these scenarios will not be acceptable. Then, just the analysis 6, which corresponds to a rotational speed of 1771 rpm and Analysis 7, corresponding to a rotational speed of 1594 of pressure load, are acceptable, since the \( SF \geq 2 \).

On the neck of the blade, similar conclusions can be stated. Values for rotational speed higher than 1771 rpm are out of range based on the SF. Therefore, these analyses do not meet the requirements. Then, considering only acceptable analyses, and checking Figure 36, where a comparison between preliminary design and modifications are plotted, it may be stated that increment in the diameter of the neck of the blade will provoke an increment in the maximum stress. Therefore, it can be suggested that the initial design is the optimal design for a rotational speed of 1771 rpm or lower values according to maximum stress.

Similar conclusions are obtained from the parametric study of the blade fillet. Remaining constant the diameter of the neck of the blade (initial design), values higher than 1771 rpm are out of range with respect to the SF. Furthermore, comparing Figure 30, it can be noticed that a size of 30 mm in the blade fillet, will cause the lowest value for the maximum stress.

**STRESS IN THE FILLET OF THE BLADE**

Although all the analyses are acceptable since the SF requirement is fulfilled for the rotational speed and pressure load (Figure 34 and Figure 37), slightly higher stresses are found comparing to the initial design, increasing proportionally to the increase of the neck, as it is represented in Figure 37. Hence, initial design for the neck of the blade is the most appropriated design (45 mm).
Regarding the modifications in the fillet of the blade, according to Figure 31, lower stresses are caused when a fillet size of 30 mm is used, which suggests that optimal design appears, again, when this size is utilized.

**STRESS IN THE NECK OF THE BLADE**

Finally, the stresses obtained in the neck of the blade achieve the requirement of the SF for all the rotational speed and the pressure load (Figure 35 and Figure 38). Similarly than in previous cases, the increase of the diameter of the neck will cause an increase of the stresses. Therefore, the initial design with 45 mm is the appropriated according to Figure 38.

However, the variation of the size of the fillet blade is different than in previous cases. According to Figure 32, the size of 35 mm has the lower stresses, followed by the 30 mm size.

It may be noticed that the parameters of the neck of the blade chosen offer a slightly higher stresses compared to the preliminary design. This can be explained by the fact that a continuous increased of the diameter in the neck of the blade will generate a higher stress in the neck of the blade, since the mounting surface on the hub would decrease.

**CONVERGENCE ANALYSIS**

According to Figure 39 and Figure 40, convergence is achieved both in the fillet of the blade and in the neck of the blade, since the percentage of error is less than 2% (Table 20 and Table 21). That implies the results can be trusted.

**MODAL ANALYSIS**

From the results obtained the operating rotational speeds are 1500 to 2000 rpm which lies between the first and the second eigenfrequency. The operating rotational speeds are not close to any of the first two eigenfrequencies. However there may be problems during start up when passing the first natural frequency. Figure 41 shows all the natural frequency and their mode. This natural frequency may change if other factors are taken into account, for instance, aerodynamics, damping and a couple of axial fan operating next to one another.
FATIGUE ANALYSIS

From the results obtained, it could be noticed that the higher the rotational speed, the less is the number of cycle (N), as shown in Figure 46, since the rotational speed increases the stress and this will result in reducing the life of the fan. When plotting the graph it could be noticed that the graph were also following the trend of the S-N graph for aluminium.

Based on the results obtained from both the stress analysis and the life estimation, it is not advisable to run the fan at full speed of 3000 rpm. This will result in early fatigue failure of the components in the axial fan since the number of cycle fall under Low Cycle Fatigue. Although running the fan at 2187 rpm could also be ideal since the number of cycle falls under the high cycle fatigue, the rotational speed does not meet the safety factor requirement.

COMPARISON OF THE ANALYSTICAL AND THE EMPIRICAL SOLUTION

The first comparison is performed on the bending stress. The analytical calculations were performed in order to compare with the result obtained from the FE-solutions. The first comparison was performed in one quarter of the geometry where the structure was subjected only to pressure load of 9000 Pa. The maximum bending stress at the bottom of the blade was equal to 18 MPa where a stress of 5.8 MPa was obtained from the analytical calculation.

The second comparison was performed on the centrifugal force. The analytical calculation was performed in order to compare with the FE solutions. When performing the analytical calculation, a centrifugal stress of 79 MPa was found. An analysis was performed taking into account the centrifugal load. A stress of 90 MPa was found at the lower bottom of the blade.

The stress comparison was performed between the FEM analysis and the analytical solution. This comparison was performed in the stress distribution in the blades. From the analytical analysis the blade was treated like a rectangular box which had a thickness of 3 mm. After the analysis the stress distribution of both the analytical and the FE solution was plotted (Figure 52 and Figure 53). It could be noticed that from the analytical solution that the stress distribution was following a pattern where the maximum stresses were in the top edge and the in the FE solution maximum stress were only in one edge. This could be explained by the simplification performed and also due to the curvature on the airfoil and the twisting of the blade. From the results obtained in the stress distribution, it could be concluded the analytical solution results is quite close to the FE solution and it could also be noticed that the maximum
stress result from the approximation of the FE solutions is quite close to the analytical solution.

Figure 52. Approximation of FE solution of the bending stress (Pa)

Figure 53. Bending stress analytical solution (Pa)
12. CONCLUSIONS

As a general conclusion, from the above results, it could be noticed from the preliminary design that an increase in the rotational speed together with the pressure load will cause an increase in the maximum stress, stress in the blade fillet and stress in the neck of the blade. Bearing in mind that a safety factor of 2 was the pre-requisite, and taking into account that the safety factor is given as the ratio between the ultimate tensile strength over the design load or the maximum stress, the following conclusions can be stated:

With respect to the parametric study, it can be suggested that the required operating speed should be between 1500 rpm to 2000 rpm. The size of the diameter of the neck of the blade should be 45 mm, while the size of the fillet of the blade should be between 30 mm and 35 mm. Also, according to the parametric study, including the reduction of the material with respect to the old design, the new design is acceptable according to the mechanical requirements explained along this dissertation.

All these results can be trusted according to the values obtained throughout the convergence analysis, where the prerequisite of percentage of change less than 2% was fulfilled.

Regarding to the modal analysis, as mention in chapter 9, all the natural frequency maybe of interested. However when taking into account the suggested rotational speeds, it could be said that the first two natural frequency might be of interested and there may be problem at start up since the recommended rotational speeds are between the first and second natural frequencies.

With respect to the fatigue analysis, evaluating the results obtained in the interested area, it can be suggested that the design meets the requirements of use if it operates at 1771 and 1594 rpm since the number of cycle falls under the High Cycle Fatigue and both rotational speed fulfilled the requirement of the Safety Factor. It can be also stated that the constant amplitude graph obtained during the analysis implies full reverse condition, that is, $R=-1$. 
13. FUTURE WORKS

As suggestions, further works would need to be done in order to get more accurate results.

Computational fluid dynamics (CFD) analysis should be performed in order to get a better performance of the fan. A CFD analysis will also help in order to obtain the vibration loads acting on the fan blades. The aerodynamics load like the lift force or lift coefficient should also be provided in order to obtain results close to reality.

The use of composite material may also be suggested when designing the axial fan, since the literature also shows that some axial fan are being designed using composite materials. Obviously the pros and cons would need to be investigated in order to determine which material would be best suited for the current type of application.

If the prototype is built, it can also be recommended to put strain gauges on the fan blades. The results can be used in order to compare experimental results to the FE solutions.

It can also be suggested to the manufacturing section, the design should be as smooth as possible in order to avoid stress concentration.
14. REFERENCES


• Fatemi, A., 2013. *Fatigue tests and Stress-Life (S-N) approach*, Toledo: University of Toledo.


• Save, C., 2013. *Type of contact in ANSYS*. [Online] Available at: https://sites.google.com/site/mechanicalengineeringforlife/FEA/typesofcontactsinanys

[Accessed 20 March 2014].


