Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 08, August-2022

CFD Analysis and Design of Ceiling Fan Blade

Lagudu Bhargav[1], Megha Pathak[2], S Yugandhar[3], Jagadeesh T[4]
Dr. Pankaj Kumar Gupta [5] [MS(R) & Ph.D. (IIT Delhi)]
[1,2,3,4,] Department Of Mechanical Engineering, Guru Ghasidas Vishwavidyalaya
(A Central University),Bilaspur,Chattisgarh
[5] Associate Professor, Department of Mechanical Engineering, Guru Ghasidas Vishwavidyalaya
(A Central University),Bilaspur, Chhattisgarh, India.

***Abstract*:- Ceiling fans are widely used as a method of cooling in India. In recent years, there is an increase in the usage of air conditioning systems for achieving much cooling than the later and to reduce the amount of sweat too. For the comfort of humans, the usage of these systems increased progressively even in most of the public places like malls, theatres, etc. Even though there is comfort but there are a lot of emissions in terms of CFCs and Carbon emissions. This consumes more energy and that makes usage costs high. On behalf of that ceiling fans come into picture in terms of energy savings and environmental causes. It is always an option for most of the people. This project leads to the design and analysis of ceiling fan blades.**

In such context the other parameters like fan speed, radius, angle of attack and design of blades plays a crucial role in air flow. For that, CFD analysis make the product more effective. Fan blade is simulated by giving suitable requirements. The results make clear about the better one among the compared ones. In this project there is a study of air foils and fan blades at different angle of attacks and different velocities in 2D and 3D. This helps to understand the design of fan blade which performs better when compared regular fan blades.

Keywords: **FEM,CFD,LIFT AND DRAG,ANGLE OF ATTACK,FAN BLADE,AIR FOIL,Ansys**

INTRODUCTION

People don’t feel comfortable when they sweat. Therefore, sometimes they try to create air breeze around them either naturally or mechanically to make sweat evaporate and it brings a feeling of comfort to the body. As per economic condition of people in India, it is very difficult for many people to acquire an air conditioner or cooler for their comfort. It is one of the causes that makes large number of people depend on ceiling fans in offices and/ or in houses for cooling off. These are very affordable and easy to install and there is no need of regular maintenance.

Most air conditioners have low-efficiency and use high number of refrigerants. Air conditioning provides uniform thermal environmental conditions. It has high energy consumption and has significantly high CO_2 emissions leads to severe negative implications for the global climate. On the other hand, ceiling fans generate non uniform velocity profiles, but there is comfort and have low energy consumptions. The aim of this project is to perform design and analysis of fan blades which can help later to understand the working and improvement of the performance of a ceiling fan.

In this project CFD is used to study the flow of air across different NACA airfoils which helps in validating air foils and to learn about the change in aerodynamic properties like lift coefficient, drag coefficient, boundary layer, region of wake, etc. with the change in speed of air around it and the angle of attack. Along with that various research papers were studied and come up with unique and creative designs of fan blades that are capable of delivering a good performance as compared to regular fans that are used in the present day.

LITERATURE REVIEW

These were the primary research papers that we referred to for this project:

1. **Wenhua Chen and Shichao Liu** [1] investigated the indoor air movement distribution with an office ceiling fan and found that increasing rotational speed of a ceiling fan enhances average air speeds in the occupied zone, especially in the region below the fan blades.
2. **Yang Zhang, Zhou Zhou, Kelei Wang and Xu Li** [2] studied the aerodynamic characteristics of different airfoils under varied turbulence intensities at low Reynolds numbers and stated that under low Reynolds number conditions, the level of the turbulence intensity determines the flow characteristics also at small angle of attack and higher turbulence intensity levels, airfoil encounters a minor decrease in lift, whereas at pre-stall angle of attack, lift increases and stall is delayed.
3. **Ramadan Bassiouny, Nader S. Korah** [3] studied the features of air flow induced by a room ceiling-fan and showed that increasing the fan rotational speed results in increasing the local downward velocity distribution inside the space.
4. **Francesco Babich and Malcolm Cook** [4] studied transient three-dimensional CFD modelling of ceiling fans and the results showed that their turbulence model is more accurate in all considered locations, namely below the centre of the fan, on the perimeter, and on a radius at growing distance for any given height and therefore air speed.
5. **Pushpesh Singh, Dr. Gajendra Vasantrao Patil** [5] investigated the design and CFD analysis of ceiling fan for regular room size and found that larger air columns can be observed near the fan and along the floor and side walls also increasing the air velocity in recirculation zone which results in better air quantity from the fan.
6. **Ehsan Adeeb, Adnan Maqsood and Ammar Mushtaq** [6] studied the effect of number of blades on performance of ceiling fans and observed that if energy efficiency coupled with volumetric flow rate is taken under consideration, a three or four blade configuration is more desirable.
7. **P. D. M. P. Rajapakshe, M. H. H. G. Mapa, R. Thanushan and R. A. C. P. Ranasinghe** [7] investigated aerodynamic performance of a ceiling fan and identified the flow patterns around the ceiling fan and as the speed increases, centre of the flow circulations due to near wall

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

boundaries moves downward direction producing more downward flow.

8. **Yongchao Zhai, Yufeng Zhang and Hui Zhang [8]** studied human comfort and perceived air quality in warm and humid environments with ceiling fans and concluded that the use of ceiling fan improved the subjects' thermal comfort and PAQ significantly, and increased acceptance of environmental air movement and humidity without causing dry-eye discomfort.

METHODOLOGY

The method used to determine the performance of various airfoil wings and fan blade is CFD (Computational Fluid Dynamics). The fan blade profiles were first drawn in CAD software. These profiles are then enclosed inside an enclosure that is much bigger in size than the size of the blade.

The enclosure is assumed to have properties of fluid (air here), and the fan blade profile is treated as a wall. Therefore, when the simulation runs, the software (Ansys Fluent) lets air flow inside the enclosure and treats the fan blade like an obstacle, that resists the flow of air from inlet to outlet, thus disturbing its path. This simulation is similar to real life observation, the only change is the frame of reference. Normally, the fan blade (or any other wing) moves through air, disturbing the air's random flow. But in CFD, the fan blade is fixed and the air flows around it. The volume through which the airflow is observed is fixed, i.e., the volume of the enclosure.

The pressure, density, velocity, turbulence, etc. properties are recorded near the fan blade wall. The total lift coefficient and the drag coefficient of the body are also recorded. These properties are then used determine the type of air flow across the wall.

The CFD analysis was first carried out in 2D cross section of fan blades and then extended to the 3D analysis of a full fan blade.

Properties of lift and drag coefficient were noted and compared depending on two factors-

- Velocity of air
- Angle of attack

Higher value of lift coefficient means that the performance of the blade is good. Similarly, less value of drag coefficient means that the blade consumes less energy to run efficiently.

DETERMINING THE INLET VELOCITY:

When a fan rotates, its blade moves with the same angular velocity. But due to the rotational motion, the linear velocities of a fan blade increase linearly as a function of the blade length. So, at the root of a fan blade, the linear velocity of the blade is very less, and it is the maximum at the tip.

The speed setting on a fan also determines the velocity of a particular point in blade. Therefore, 4 standard speeds in RPM were taken and converted into m/s. The analysis was done by keeping these 4 speeds as inlet velocity in m/s.

This was done to capture the basic properties of the fan blade with simple calculations, under the influence of a constant inlet velocity. In a fan, however, the variation of the velocity is a linear function of the radius of the fan blade.

$$V = f(R) = R\omega$$

Where V is the linear velocity of the fan blade in m/s,
R is the radius of the fan or the length of the fan blade
ω is the angular velocity in radians.

The four speeds selected were 50 RPM, 100 RPM, 200 RPM and 300 RPM.

The radius of the fan was taken as 0.5 m.

Putting these values, the velocity in meters per second was obtained:

- $50\ RPM = \frac{2\times\pi\times0.5\times50}{60}$

$50\ RPM = 2.61\ m/s$

- $100\ RPM = \frac{2\times\pi\times0.5\times100}{60}$

$100\ RPM = 5.23\ m/s$

- $200\ RPM = \frac{2\times\pi\times0.5\times200}{60}$

$200\ RPM = 10.46\ m/s$

- $300\ RPM = \frac{2\times\pi\times0.5\times300}{60}$

$300\ RPM = 15.47\ m/s$

In the validation part, however, some other velocities were also used to conform to research data.

TYPES OF FAN BLADES USED:

After referring to various research papers listed in the literature review. The following designs were selected:

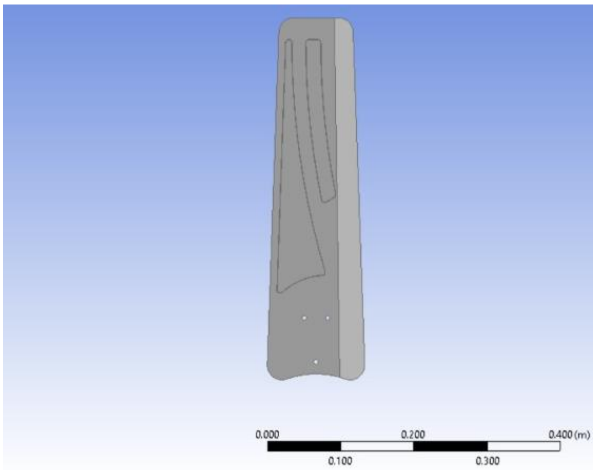
- **Plain fan blade**: This fan blade was the extended version of the 2D fan blade cross section. This was a simple fan blade and it did not have any particular aerodynamic properties. This fan was taken as a reference to compare other fan blades.

- **Grooved fan blade**: This fan blade was a commercially used fan blade, that was taken from the manufacturer. The shape of the plain fan blade was similar to this grooved fan blade. The difference was in the cross section of the blade and the presence of grooved like design. The plain fan blade and the grooved fan blade were compared in order to see the effect of the grooved design.

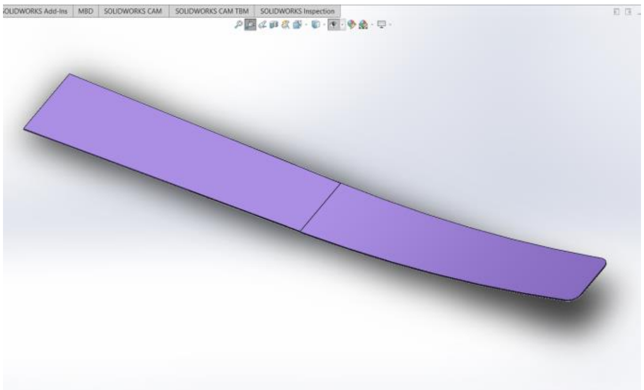
- **Curved Fan blade:** This fan blade was unique and created by the team. The fan was plain on its first half but it was curved upwards on its second half.

- **Airfoil Cross-section Fan blade:** As the name suggests, airfoil was used as the cross section of the fan blade, because of its aerodynamic properties. This blade was also lofted and was inspired from Haiku fan research paper.[1]

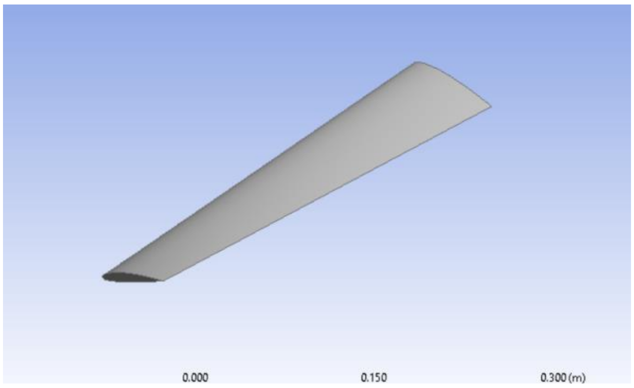
(This work is licensed under a Creative Commons Attribution 4.0 International License.)



Grooved Fan blade



Curved Fan blade



Air foil cross section

EFFECT OF DIMPLE IN AERODYNAMIC BODIES:

The shape of a body contributes very intensively to its aerodynamic properties. Slight changes in its design can greatly affect its lift and drag coefficient. In the present day, research on the effect of dimple in bodies has been going on.

A big successful example of improving the aerodynamic properties (here, range) of a body using dimples is a golf ball. A golf ball can go two times as farther away than if the ball was smooth.

The theory behind why this affect works is that:

Normally when a smooth (say, spherical) body passes through air. The air around its outline gets spread out tangential to its outermost boundary. This creates a wake region, that is a region of backflow, vortex and very slow-moving air. This is what contributes to the drag of the body.

Now if a series of tiny dimples are introduced in the body's surface along the direction parallel to the flow of air, the air follows the curvature of that dimple and forms a very small region of circular backflow (vortex), the air that comes out of this vortex has some angular momentum and before escaping, it gets caught in the next dimple.

This process can continue upto a region that is situated much further than the outermost boundary of the body, which means that the air sticks to the surface for longer area before leaving the body. Thus, here the wake region created is significantly smaller, and the drag of the body is reduced. This effect was also used to check performance of other wings and positive results were obtained

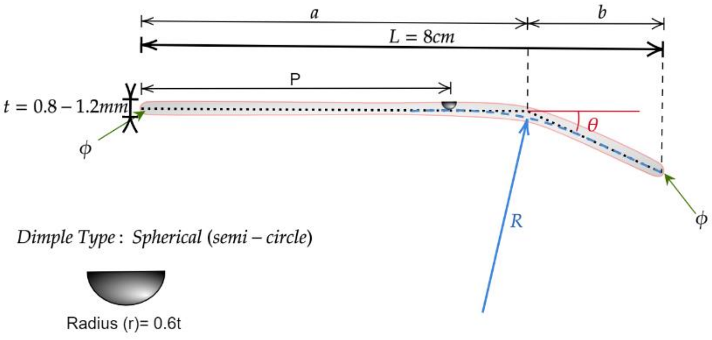
DESIGN PROCESOF CEILING FAN BLADE

2D FAN BLADE CROSS SECTION:

For the first analysis, the 2D cross section of a fan blade was used. The geometry planned for the 2D cross section was very simple. It was assumed during the 2D analysis that the fan blade does not have any twist or loft if extended in 3D. A blueprint was used to create the fan blade cross section.

Blueprint:

A blueprint for the creation of 2D fan blade was created:



Blueprint of 2D fan blade

There were two major sections the straight section and the bent section of lengths a and b respectively. When designed, these lengths were kept as follows:

$$a = 6cm \,, b = 2cm$$

Thickness was kept as 0.9 mm. The orientation of the 'b' part was varied by changing the angle theta.

A dimple was introduced at P distance (P=5 cm) and this dimple was the shape of a semi-circle in 2D. The radius of this dimple was taken as 0.54mm.

Therefore, the following variants of this 2D cross section were created:

- Dimpled: 6 degrees, 7.5 degrees, and 9 degrees
- Non- dimpled: 6 degrees, 7.5 degrees, and 9 degrees

Creation of enclosure:

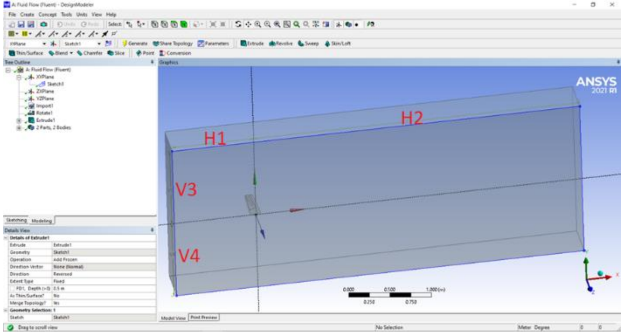
The enclosure creation for all fans and airfoils was done by keeping same features in mind, the enclosure must be large enough to contain the blade and its right side (towards outlet) must be bigger than the left side (towards inlet). Some space was also left before the walls, in case of fan blades, therefore the blades were completely situated inside the enclosure,

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

unlike the 3D airfoil that was extruded along the complete thickness of the enclosure.

The dimensions of the enclosure for a 50 cm long fan were chosen as:

H1=1m
H2=2m
V3=0.5m
V4=0.5m
T1(behind XY plane) = 0.15m
T2 (above XY plane; contains blade) = 0.6m



Details of Extrude1

Extrude	Extrude1
Geometry	Sketch1
Operation	Add Frozen
Direction Vector	None (Normal)
Direction	Reversed
Extent Type	Fixed
FD1, Depth (>0)	0.5 m
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch1

Enclosure for fan

For the validation of airfoils that were scaled down (Validation 3), the enclosure was scaled down as well and the ratios of the enclosure were same.

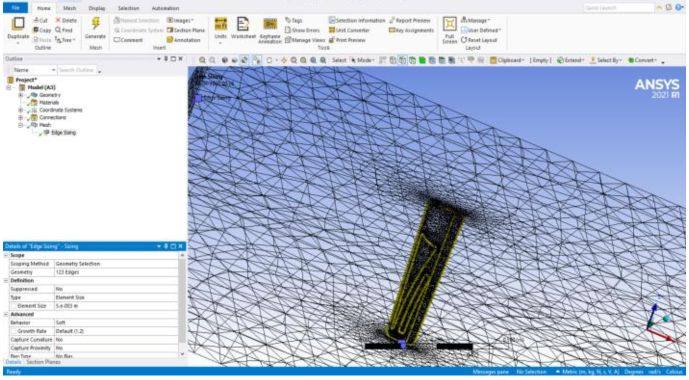
The geometry was created by performing a Boolean operation that performed subtraction of the fan geometry from the enclosure, thus, creating an empty space that behaved like wall in the enclosure which was the fluid domain.

Meshing process:

Meshing process consisted of following steps:

1. The mesh was generated without any conditions.
2. The element size was reduced to 0.05m.
3. Face sizing: All faces of the fan blade were selected and the element size was kept 0.005m.
4. Edge sizing: All edges of the fan blade were selected and the element size was kept 0.001m.
5. Inflation: The fan blade boundary was selected and an inflation was created in the enclosure with 10 layers and a growth rate of 1.2.
6. Named selections: The faces present in the geometry were named to be fed into setup-

Inlet – left
Outlet – right
Blade wall – blade faces all
Sym1 – Front-back
Sym2 – Up-down



Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	123 Edges
Definition	
Suppressed	No
Type	Element Size
Element Size	5.e-003 m
Advanced	
Behavior	Soft
Growth Rate	Default (1.2)
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Meshing of fan

2D ANALYSIS OF FAN BLADE CROSS SECTION

For the 2D analysis of a fan blade cross section, the blade with dimensions equal to 10 cm width and 1.2mm thickness was taken. It had a variable pitch ranging from 2 degrees to 10 degrees, around which the analysis was carried out.

Two variants of this cross section were compared. The overall design of the blade was the same, but the second variant had a small dimple situated right before the pitch. It was a preliminary attempt to find whether presence if a dimple makes the blade more aerodynamic.

Boundary conditions:

The left side was termed as inlet, while the right side was the outlet. The flow of air was horizontal from inlet to outlet.

The inlet velocity was kept 50 RPM. The method used in the analysis was K omega SST. The boundary condition for outlet was pressure outlet. The turbulent intensity was kept to 5% and the turbulent viscosity ratio was kept to 10%.

The upper and lower walls were defined as symmetry.

Here too, lift and drag coefficients were noted for different angles of attack in each variant.

Lift and drag coefficients for Non-Dimpled fan blade cross section are as follows:

Angle of Attack (degrees)	LIFT Coefficient (Non-Dimpled)	% Increase	DRAG Coefficient (Non-Dimpled)	% Increase
6	0.25131	-	0.02750	-
7.5	0.30707	22.18	0.02936	6.76
9	0.35880	16.84	0.03171	8.004

Lift and drag coefficients for Dimpled fan blade cross section were as follows:

Angle of Attack (degrees)	LIFT Coefficient (Dimpled)	% Increase	DRAG Coefficient (Dimpled)	% Increase
6	0.25244	-	0.02072	-
7.5	0.31439	24.54	0.02914	40.63
9	0.37382	18.90	0.03153	8.20

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

VALIDATION OF AIRFOILS

In order to first get familiar with the software and to verify if the data produced by it was true or not, a series of validation was carried out. In this, various airfoils were taken in different settings and then they were analysed. It also helped us to check ourselves the deviation between the results if a minor quantity is changed in the solver.

Validation 1:

For the first validation, airfoil NACA 2412 was taken. It was recreated in 2D and analysed. The data produced by the solver was compared with the Xfoil Data.[16]

The properties of NACA 2412 used in the analysis were:

- ➢ Cambered Airfoil
- ➢ Chord length = 1 m
- ➢ Reynold’s Number= 10,00,000
- ➢ Inlet Velocity = 14.6 m/s
- ➢ Studied at angle of attacks ranging from 0 degrees to 12 degrees

Validation 1 Observations:

The following coefficient of lift and coefficient of drag were observed at various angle of attacks:

Angle of Attack (degrees)	**LIFT Coefficient (CFD)**	**LIFT Coefficient (Experimental)**	**Error (%)**	**DRAG Coefficient (CFD)**	**DRAG Coefficient (Experimental)**
0	0.3074	0.2429	26.55	0.0176	0.0071
5	0.9803	0.796	23.15	0.0314	0.0091
7.5	1.2283	1.0619	15.67	0.0506	0.0125
9	1.3341	1.1807	12.99	0.0642	0.0143
10	1.3506	1.2709	6.27	0.0829	0.0157
12	1.4718	1.4252	3.26	0.1238	0.0193

The idea behind this validation was to apply the similarity condition using the paper. [reference]. For this, a low Reynolds number, equal to 5300 was taken. Now, the chord length and the inlet velocity were altered proportionally in two variants, such that the Reynolds number stays constant. Since, the fluid dynamics of two object with the same Reynolds number is the same, it was initially assumed that the lift and the drag coefficient of the airfoil at both levels would be the same. But the two values were highly different from each other and due to lack of proper experimental data at Rn 5300, the data was not considered as a validation of ANSYS software.[2]

Instead, some minor changes were made within the software, like:

- Different types of Enclosure
- Different methods of solving

Observations:

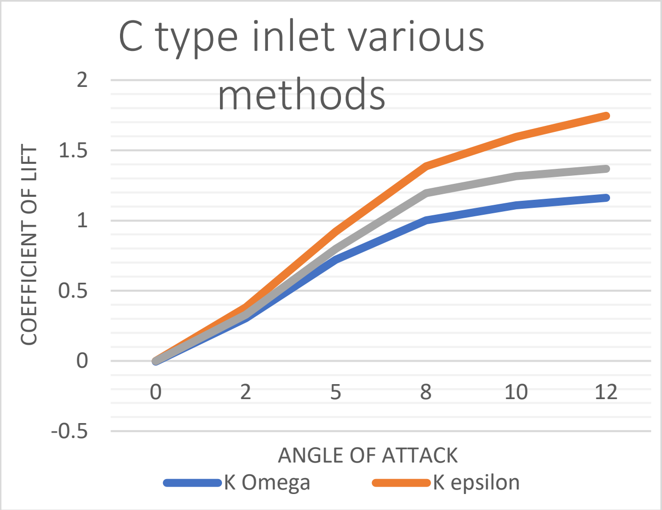
The following coefficient of lift and coefficient of drag were observed at various angle of attacks of 2D analysis.this values were obtained by changing the length of the scale to 2.5mm And 10mm respectively.inorder to compare with the values that were previously obtained from Research papers and tabulated below

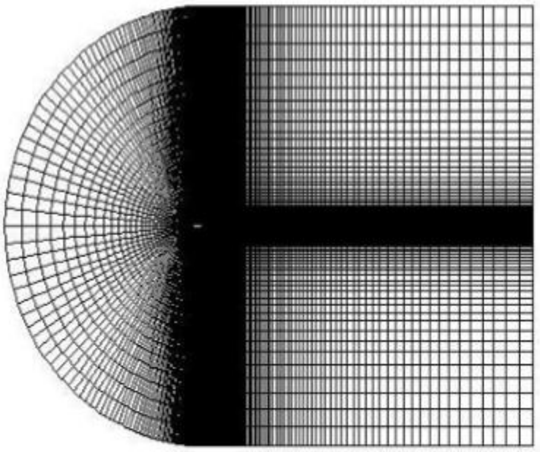
The following coefficient of lift and coefficient of drag were observed at various angle of attacks of 2D analysis:

Angle of Attack (degrees)	**LIFT (2.5 mm)**	**LIFT (10 mm)**	**LIFT (Paper)**	**DRAG (2.5 mm)**	**DRAG (10 mm)**
2	0.3299	0.0011	0.098	0.1370	0.0006
5	0.7993	0.0031	0.282	0.1550	0.0007
8	1.1956	0.0048	0.612	0.2087	0.0010
10	1.3157	0.0051	0.785	0.2785	0.0013
12	1.3683	0.0055	0.856	0.3651	0.0017

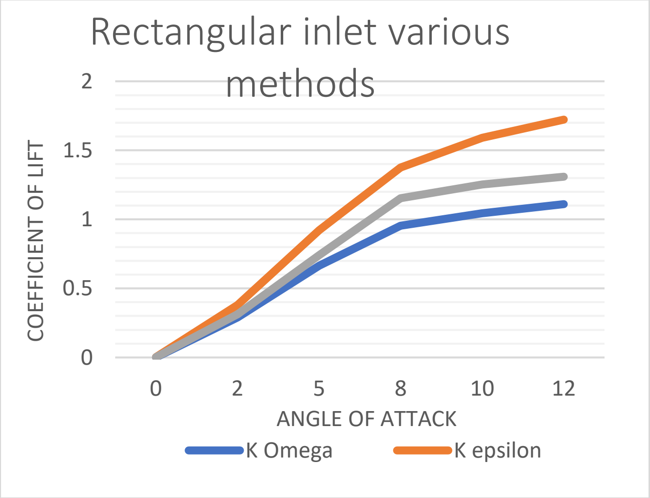
The following comparison shows values of lift and drag coefficients using different solver methods in C-Type enclosure:

Angle of Attack (degrees)	**LIFT (K omega)**	**LIFT (Spalart Allmaras)**	**LIFT (K epsilon)**	**DRAG (K omega)**	**DRAG (Spalart Allmaras)**	**DRAG (K epsilon)**
0	-0.00529	-0.00112	0.00009	0.13017	0.13452	0.17180
2	0.30343	0.32990	0.38206	0.13254	0.13700	0.17718
5	0.72042	0.79938	0.92240	0.15023	0.15508	0.20080
8	**1.00210**	**1.19560**	**1.38500**	**0.20450**	**0.20874**	**0.24987**
10	1.10770	1.31570	1.59630	0.26780	0.27852	0.29835
12	1.16160	1.36830	1.74600	0.34258	0.36517	0.35838

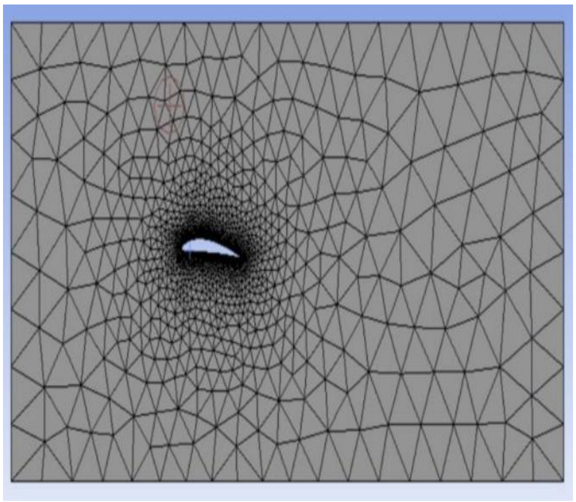




a Plot of lift and drag coefficient for C-type mesh



Plot of lift and drag coefficient for Rectangulat type mesh



Rectangular mesh

The following comparison shows values of lift and drag coefficients using different solver methods in Rectangular enclosure:

Angle of Attack (degrees)	LIFT (K omega)	LIFT (Spalart Allmaras)	LIFT (K epsilon)	DRAG (K omega)	DRAG (Spalart Allmaras)	DRAG (K epsilon)
0	0.00042	0.00177	0.00274	0.12943	0.13341	0.17041
2	0.28970	0.31285	0.38086	0.13185	0.13593	0.17402
5	0.66558	0.73997	0.92187	0.15299	0.15695	0..19895
8	0.95308	1.15250	1.37610	0.20971	0.21470	0.24991
10	1.04410	1.25250	1.59120	0.27070	0.28680	0.29800
12	1.11030	1.30910	1.72240	0.34390	0.37037	0.35753

Plot of lift and drag Coeff for rectangular mesh

3D ANALYSIS OF FAN BLADES

4.3.1 Boundary conditions:

The blade was enclosed in a rectangular 3D enclosure that was much greater than the size of the fan blade itself, this was done to ensure that no wall effect takes place. The left side was termed as inlet, while the right side was the outlet. The flow of air was horizontal from inlet to outlet.

The inlet velocity was varied between: 50, 100 and 200 RPM. The method used in the analysis was K Epsilon. The boundary condition for outlet was pressure outlet. The turbulent intensity was kept to 5% and the turbulent viscosity ratio was kept to 10%.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

The upper and lower walls were defined as symmetry (sym1) and the front-back walls were also defined as symmetry (Sym2). Here too, lift and drag coefficients were noted for different angles of attack in each variant.

Analysis of Plain fan blade:

Inlet Velocity (RPM)	LIFT Coefficient	% Increase (Lift Coeff)	DRAG Coefficient	% Increase (Drag Coeff)
50	0.01235	-	0.00245	-
100	0.05205	321.45	0.00828	237.95
200	0.21830	319.40	0.02800	238.16

Analysis of Grooved fan blade

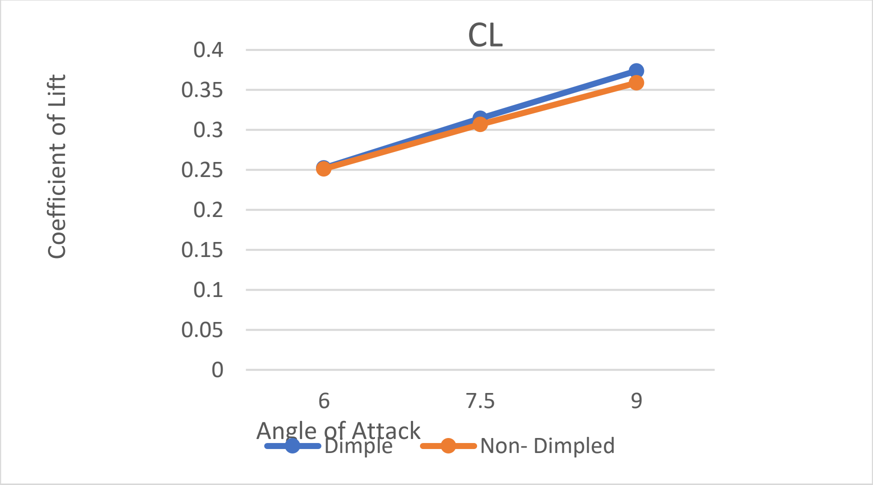
Inlet Velocity (RPM)	LIFT Coefficient	% Increase (Lift Coeff)	DRAG Coefficient	% Increase (Drag Coeff)
50	0.01397	-	0.00265	-
100	0.05814	316.17	0.00923	248.30
200	0.24080	314.17	0.03250	252.11

COMPARISON

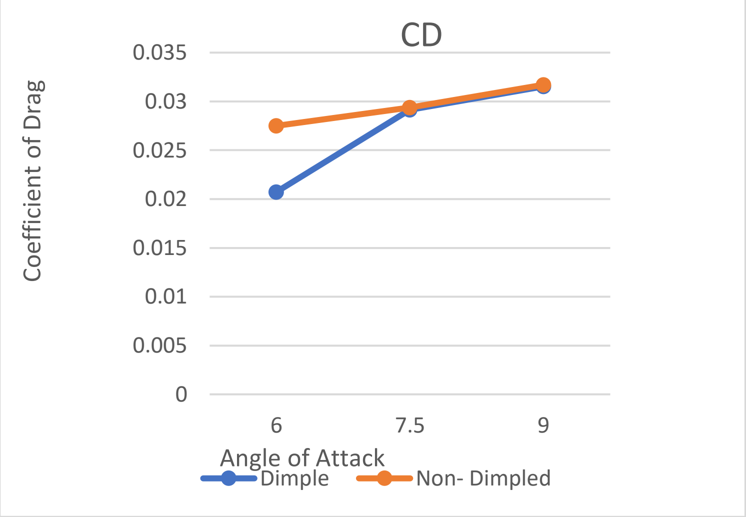
After performing the CFD analysis on various fan blades, the results were compared to draw conclusion in three ways:

Comparison of simple and dimpled fan blade cross section:

The lift coefficient and drag coefficients of dimpled and non-dimpled fan blade cross sections were studied at various angles of attacks. The speed of the inlet was kept as 50 RPM. The comparison is represented by the graph



Plot of lift coefficient for dimpled and non-dimpled fan blades



Plot of drag coefficient for dimpled and non-dimpled fan blades

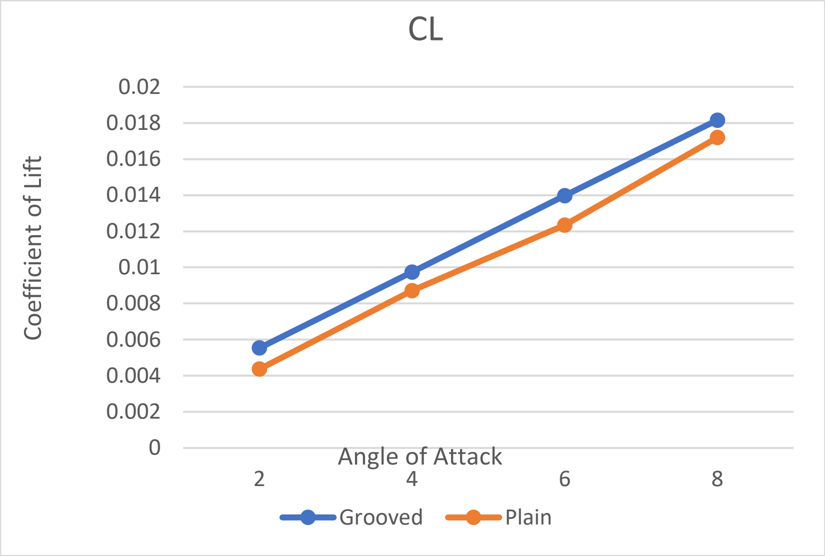
(This work is licensed under a Creative Commons Attribution 4.0 International License.)

It was found that the performance of Dimpled fan blade was better than the Non-dimpled fan.

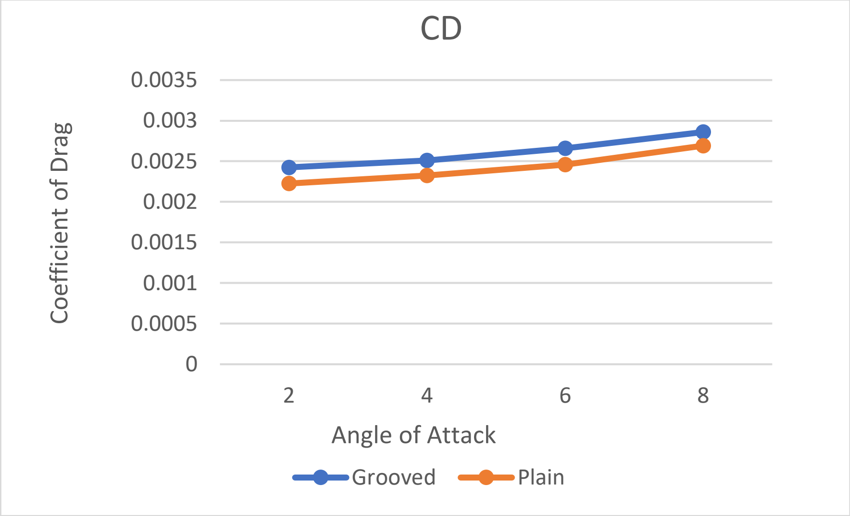
- The lift coefficient of dimpled fan blade was higher than non-dimpled one at all angle of attacks.
- The drag coefficient of dimpled fan blade was lower than non-dimpled one at all angle of attacks.

Comparison of Plain and Grooved fan blade:

The geometries of both, plain and grooved fan blades were almost similar, with the only major difference being the presence of grooved structure. This grooved structure resulted a less aerodynamic looking cross section, but the air flow across the grooved fan blade was better than that of the plain blade. It is also evident from the comparison of lift and drag coefficients of both fans at 50 rpm and different angles of attack:



Plot of lift coefficients for plain and grooved blades



Plot of drag coefficients for plain and grooved blades

It was observed that both, Coefficient of lift and drag are more in case of the grooved fan blade. Thus, the grooved fan blade would require more torque to overcome the drag force caused by the design, but it also has a better lift coefficient.

The rate of increment of the drag was more after 6 degrees. Therefore, at this stage, 6 degrees was chosen as the optimum angle of attack for both plain and the grooved fan for further studies.

4.4.3 Comparison of all fans at various speeds:

PLAIN FAN BLADE	Simple blade at 6 degrees angle of attack
GROOVED FAN BLADE	Fan blade with grooves at 6 degree angle of attack

Types of fan blades

The airfoil cross-section fan and the curved fan blades did not produce satisfying results. The drag produced by the curved

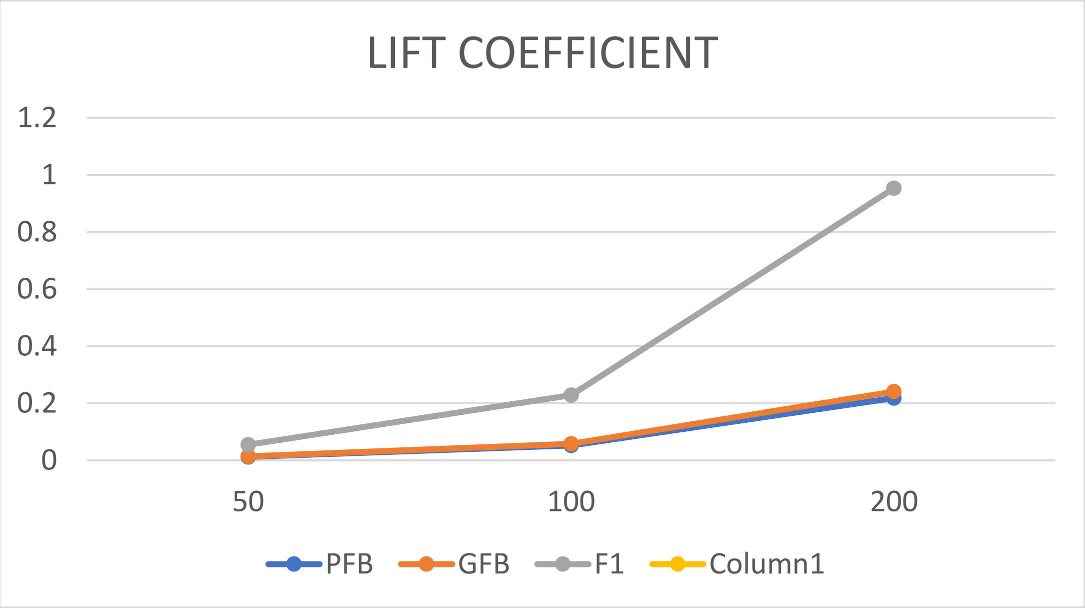
(This work is licensed under a Creative Commons Attribution 4.0 International License.)

fan blade was too high and the lift coefficient for the airfoil fan blade was too low to be included in the comparison graph. Therefore, they were excluded from the comparison.

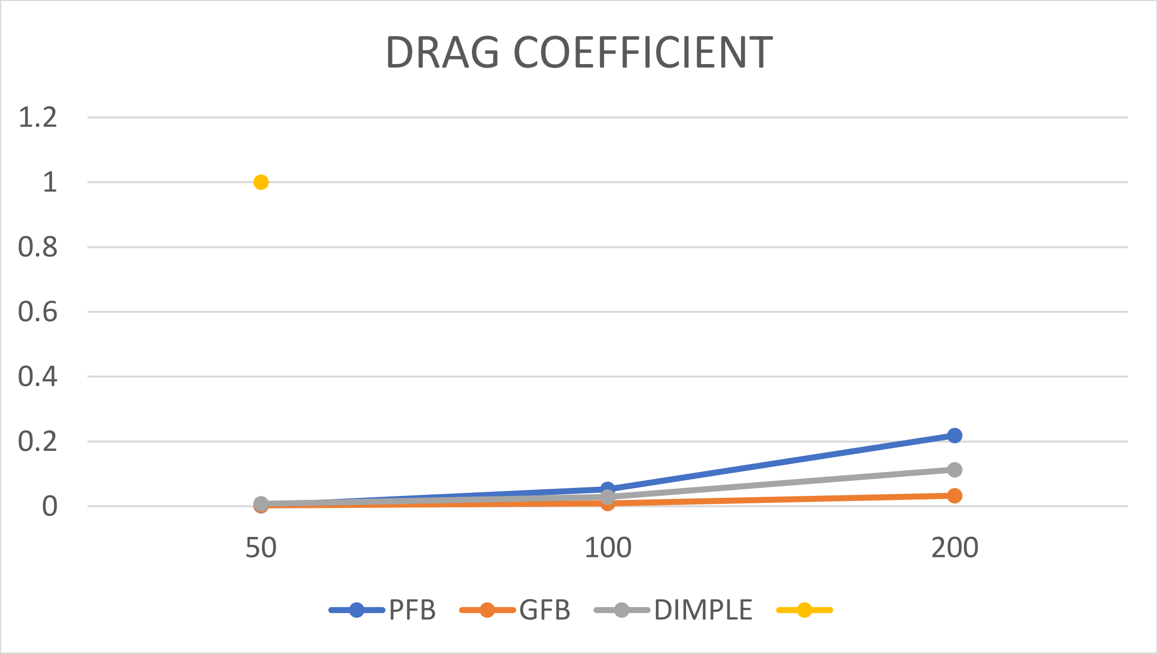
From the above table 13, the blades used for comparison were:

- Plain fan blade (At 6 degrees)
- Grooved fan blade (at 6 degrees)

Were compared in terms of lift and drag coefficients for 3 various speeds. And the results were as follows:



Plot of lift coefficient for 3 variousspeeds



Plot of drag coefficient for 3 various speeds

- The lift coefficient was almost similar for the plain and the grooved fan blade.
- The lift coefficient of F2 fan blade was the highest in all cases.
- The drag coefficient for plain fan blade was the highest and lowest for the grooved fan blade.
- The drag coefficient of the F2 fan blades did not increase as much as the others with the change in its speed.

CONCLUSION AND FUTURE SCOPE

Throughout the span of major project, various fan blades were analyzed in Ansys Fluent. The result was verified by performing validation beforehand using different airfoils for 2D and 3D.

The main basis of comparison between fan blades was coefficient of lift and drag. Thus, the study mostly focused on the performance of an individual fan blade rather than the

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

performance of the whole fan and its effect on the air circulation of the room.

➢ In 2D simulation of airflow across the dimpled fan blade cross-section was seen to be was observed to be slightly better than that of the non-dimpled fan blade cross-section. This opens a possibility to improve the performance by strategically introducing dimple(s) at certain places in the fan blade.

➢ In 3D simulation, it was concluded that the F2 fan blade has the best performance amongst all the variants. The reasons behind this conclusion were:

1. It had the highest lift coefficient at all speeds.
2. The increase in drag with increase in speed was considerably low.

➢ The performance of other unique fan blades (airfoil blade and curved blade) was not satisfactory during the study of lift-drag coefficients, thus they were not included in the comparison, but they are yet to be observed for their performance at a larger scale.

For the whole project, the bigger goal to achieve is to determine the performance of these fan blades in assembly with the hub. This can be achieved by performing full-scale simulation of a fan inside a room. This study requires a high computational power to be carried out and therefore was seen only as a future scope of the project.

REFERANCES

[1] Experimental and numerical investigations of indoor air movement distribution with an office ceiling fan by Wenhua Chen, Shichao Liu, Yunfei Gao, Hui Zhang, Edward Arens, Lei Zhao, Junjie Liu.
[2] Aerodynamic Characteristics of Different Airfoils under Varied Turbulence Intensities at Low Reynolds Numbers by Yang Zhang, Zhou Zhou, Kelei Wang and Xu Li.
[3] Studying the features of air flow induced by a room ceiling-fan Ramadan Bassiouny∗, Nader S. Korah Department of Mech. Power Eng. & Energy, Minia University, Minia 61111, Egypt
[4] Babich F, Cook M, Loveday D, Rawal R, Shukla Y, Transient three dimensional CFD modelling of ceiling fans, Building and Environment (2017), doi: 10.1016/ j.buildenv.2017.06.039
[5] Pushpesh Singh and Dr. Gajendra Vasantrao Patil, Design and CFD Analysis of Ceiling Fan for Regular Room Size. International Journal of Mechanical Engineering and Technology. 11(4), 2020, pp. 25-33
[6] Ehsan Adeeb, Adnan Maqsood and Ammar Mushtaq, Effect of Number of Blades on Performance of Ceiling Fans.
[7] P. D. M. P. Rajapakshe, M. H. H. G. Mapa, R. Thanushan and R. A. C. P. Ranasinghe, Investigating Aerodynamic Performance of a Ceiling Fan
[8] Human comfort and perceived air quality in warm and humid environments with ceiling fans by Yongchao Zhai, Yufeng Zhang, Hui Zhang, Wilmer Pasut, Edward Arens, Qinglin Meng
[9] https://en.wikipedia.org/wiki/Airfoil
[10] https://www.researchgate.net/profile/Mohammed-Salman-16/publication/331031704/figure/fig1/AS:809221851463685@1569944932859/Aerodynamic-parameters.ppm
[11] https://en.m.wikipedia.org/wiki/File:Airfoil_camber.jpg
[12] http://www.aviationchief.com/angle-of-attack.html
[13] http://avstop.com/ac/flighttrainghandbook/angleofattackandlift.html
[14] https://en.m.wikipedia.org/wiki/File:Airfoil_lift_and_drag.svg
[15] https://studiousguy.com/wp-content/uploads/2020/10/aeroplane.png
[16] http://airfoiltools.com/airfoil/naca4digit
[17] https://en.wikipedia.org/wiki/Ceiling_fan#MythBusters:_%22Killer_Ceiling_Fan%22
[18] https://blog.ceilingfan.com/5-factors-that-determine-ceiling-fan-airflow/
[19] https://grabcad.com/library/ceiling-fan-blade-1
[20] https://www.lightingdirect.com/minkaaire-artemis-led-artemis-58-3-blade-led-indoor-ceiling-fan-with-dc-motor-and-remote-control-included/p4031476
[21] https://www.dansfancity.com/hercules-pure-white-with-80-bahama-bent-walnut-blades-3
[22] Fan Size and Energy Efficiency by Richard Aynsley, School of Engineering Technology & Management Southern Polytechnic State University, Marietta, GA, USA
[23] Improvement of Human Thermal Comfort by Optimizing the Airflow Induced by a Ceiling Fan by Hsin-Hung Lin
[24] Efficiency improvement opportunities for ceiling fans by Nihar Shah & Nakul Sathaye & Amol Phadke & Virginie Letschert
[25] Measured Ceiling Fan Performance and Usage Patterns: Implications for Efficiency and Comfort Improvement by J.K. Sonne, and D.S. Parker, FSEC, Cocoa, FL

(This work is licensed under a Creative Commons Attribution 4.0 International License.)