International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-08, Special Issue, MAY 2022

Design and Analysis of Aircraft Engine Cooling Fan

Mr. R. Vinay*, Mr. R. Leela Satish Kumar*, Mr. J. Durga Vara Prasad*, Mr. T. Anish*, Mr. K. Gowri Shankar*, Mr. B Hari Krishna, Mr. M Sunil Raj****

***UG students, **Faculty, Department of Mechanical Engineering College, Pragati Engineering College (A)**

ABSTRACT - The current work is aimed to design an aircraft engine cooling fan. the family of turbo machines and they move air or gas conns belong continuously at desired velocity by action of a rotor. Flow investigation of the fan is planned to be carried out by using ANSYS-FLUENT software for different designed off design points of operation. The performance of the fan generated from the CFD analysis at the design point will be compared with that of the designed data assumed for calculation. This will also be compared with the best efficiency point of operation. For the analysis, an CAD drawing, and a 3-D model of the fan impeller and casing are developed for the designed fan. This is followed by the generation of Grid and aerodynamic analysis using the available CFD solver. The work is concluded by identifying possible zones of improvements in the design of impeller and casing and suggests suitable modifications.

Key words: Design, Aircraft, CFD, Ansys, Grid, Aerodynamic

I. INTRODUCTION TO TURBO MACHINES

Turbomachines used for the compression of gases are classified under radial, axial, or mixed flow types depending on the flow through the impeller. In a radial or centrifugal machine, the pressure increase due to the centrifugal action forms an important factor in its operation. The energy is transferred by dynamic means from the impeller to the fluid. The fluid because of centrifugal action is continuously thrown outwards making way for fresh fluid to be inducted in because of the reduced local pressure. Another characteristic feature of the centrifugal impeller is the angular momentum of the fluid flowing through the impeller is increased by virtue of the impeller outer diameter being significantly larger than the inlet diameter. In axial flow machines, a large mass of gas is set in motion by the rotating impeller and is made to move forward because of the aerodynamic action of the blades. A mixed flow machine encompasses the properties of both the above types. Depending on the pressure rise attained, these machines are named as fans and blower or compressors. There is however no distinct demarcation among the different types. Fans handle gases in large volumes without appreciable density variation. Pressure ratio attainable is of the order of 1.05. They are invariably single stage machines. Blowers cover pressure ratios from 1.05 to about 4. They are made either as single stage or two or three stages. No inter cooling is required. Compressors include pressure ratios from 3 to 12 or higher. They are invariably multistage with or without intercooling. For higher pressure ratios appreciable compression takes place followed by a reduction in volume. The calculations are done on the basis of mass flow in such cases. The selection of a type of impeller namely axial, radial, or mixed flow for a specified pressure rise, speed and flow rate follow from shape number considerations defined by

$$N_{shape} = n \sqrt{(v)} / w^{0.75}$$

The shape number is important to achieve an optimum efficiency. Radial machines have low shape numbers ranging from 0.033 to 0.12 and are known as slow running impellers. Axial flow types have shape numbers from 0.33 to 1.5. Mixed flow types have values in between those of radial and axial impellers. An idea of the shape of impeller can be obtained from the shape number. For example, slow running impellers have long and narrow vane channel passages and large shroud diameters. This increases the friction losses and lowers the efficiency high shape numbers are desirable. The energy which is converted into pressure in the impeller is indicated by the degree of reaction which is the ratio of specific pressure energy to the specific work of the machine. Blowers and compressors operate with degree of reaction greater than zero, and mostly than 0.5. The reason is that the static pressure can be generated more efficiently in the impeller than in the guide vanes as the centrifugal forces in the rotating channels of the impeller help in the suction of the boundary layer and dead zones. If the specified pressure rise cannot be obtained in one stage, two or more stages as required are built in series, the individual stages being joined by what are known as return guide passages or return channels. In such a multistage centrifugal compressor or blower, the chief problems encountered are regarding the design of efficient guide and return channel passages as well as carefully designed shroud and vane contours. Though compressors with more than eight or ten stages are in existence, the number of stages is generally restricted to two or three. The desired pressure rise is obtained by

 DOI : 10.35291/2454-9150.2022.0248

ISSN : 2454-9150 Vol-08, Special Issue, MAY 2022

employing high rotational speeds made possible by the steam and gas turbine drives and using high strength forged impellers with straight radial blades and devoid of front shroud in order to minimize the stresses in the hub and back shroud. In blowers and fans dealing with large volumes of gas but relatively low pressure rise, sheet metal construction is employed, with suitable hub design to take care of stresses and guide the flow. The sheets are suitably pressed to shape, and the joining is through riveting or welding. Blade loading, shroud or disc stresses and critical speed considerations impose serious restrictions on the dimensions of the machine to lower values. However, are the pressure rise increases with increasing peripheral speeds, minimum number of stages is preferred for a compact blower, thus necessitating the use of high peripheral speeds limited by the strength of the material.

CLASSIFICATION OF FANS:

Centrifugal flow fans:
a. Forward Curved
b. Radial Curved
c. Backward Curved
Axial flow fans:
a. Propeller type.
b. Tube-axial type
c. Contra rotating
d. Guide-vane type
e. Axial type
Mixed flow fans:
a. Axial Casing
Cross flow fans:
a. J-Casing
b. S-Casing
c. U-Casing

In any centrifugal machine, the most important requirement is that it should develop the required specific work with the desired static pressure rise. In other words, the specific pressure rise is directly dependent on the specific work developed by the machine.

The specific work is developed in the impeller only through the energy transfer to the fluid through the vanes and is given by Euler's equation

$W = U_2C_2 – U_1C_1$

W= specific work developed by the stage (N-m/Kg)

U_1 = impeller speed at start of vane

U_2 = impeller tip peripheral speed

C_1 and C_2 are the components the absolute velocity in the tangential direction at points just before the inlet to the impeller vane and the exit from the impeller vane respectively.

The above Equation can be rewritten as:

$W = (U_2^2 – U_1^2 + C_1^2 - C_2^2 + W_0^2 - W_3^2)/2$

As the flow energy of the fluid comprises the pressure energy, the kinetic energy and that due to the geodetic head, the energy at any section of the passage (except where energy is being added) can be written as:

$E = P/\rho + C^2/Z + g.h$

BLADE ANGLES:

Inlet vane angle

As the temperature of the air at the inlet is less. The sonic velocity is also less. There is the danger of the velocity in this region reaching a sonic value. For incompressible flow, the relative inlet velocity is a minimum when β_1 =35°. In compressible flow, the relative inlet Mach number is a minimum when β_1 is in between 25° to 30°.

Exit vane angle

There are three considerations for β_{2b} namely forward curved blades if β_{2b}<90°, radial blades when β_{2b}=90° and backward curved blades if the angle β_{2b}>90°. In all the three cases β_{1b}, the fan speed, the inlet velocity c_m and size are kept the same. Therefore, the velocity triangles at 1 are the same for three cases. The velocity triangles at 2 are shown in the figures for each case. It can be seen c_{2u} increases with β_{2b} and likewise the specific work. As β_{2b} increases, the blades are more cambered finally resulting in the highly cambered impulse profile this means increase in the B_{2b} results in increase in C_{2u}, likewise the specific work. **The kinetic energy of the fluid at the impeller outlet becomes a smaller percentage of the total energy as blades become more backwardly curved. Therefore, a larger portion of the static pressure can** be **recovered in the impeller with backward** curved vanes.

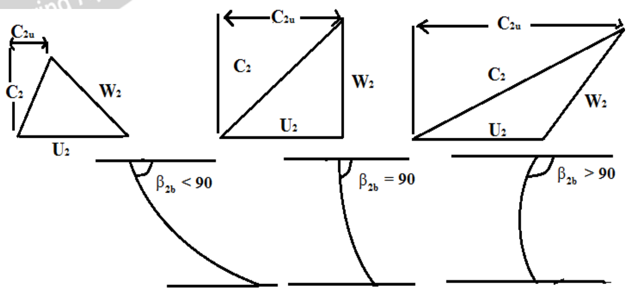


Fig 2.1 Effect of Exit Vane Angle on Outlet Velocity of Impeller

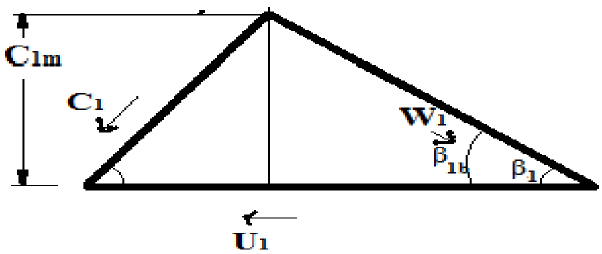


Fig 2.1. Effect of Exit Vane Angle on Outlet Velocity of Impeller

DOI : 10.35291/2454-9150.2022.0248

IMPELLER BLADE ANGLE AT THE SUCTION END (β1b)

β_{1b} used in impeller is within a limited range for all machines. It is the angle at inlet for pump/comp and at exit for turbines. For radial fans and blowers, values outside this range reducing up to 20° are found to be in use. In the case of turbines, a low β_{1b} would mean more flow deflection in the impeller blade row with corresponding increase in specific work. With decreasing β_{1b}, the blade tangential thickness t_{1u} at exit increases. From strength considerations, trailing edge thickness cannot be reduced to small values. Also, this causes formation of eddied behind the blade trailing edge and results in wider wakes and more losses values between 15° to 35° are used.

2.3 VELOCITY TRIANGLES:

The three velocities that make a velocity triangle are namely

I. Blade speed U
II. Absolute velocity C
III. Relative velocity W

Generally, the blade speed is taken as the base of the triangle, the direction of U1 and U2 follow the direction of In a radial machine U_2 greater than U_1.

Angle between C 'absolute velocity' and 'relative velocity' U is α and β is the angle between W and –U.

The flow velocities are resolved into two components with respect to U, the component along U is C_u {may be C_{1u} or C_{2u}} and perpendicular to U i.e., along meridional plane is Cm and similarly Wu and Wm are obtained. To get the volume flow rate at the particular section C_m can be multiplied by flow area at that section hence it is called the 'flow velocity'. If the pre whirl is 0 then $C_{1u} = 0$, hence it is desirable to design with consideration $C_{1m} = C_{2m}$ whenever possible which also helps to maintain the blade angle within considerable range.

FAN APPLICATIONS

Power Plant Auxiliaries

Cooling of Motors, Generators and Engines

Air Circulation and Mine Ventilation

Steel Plants

DESIGN METHODOLOGY:

Blade generation

rotation of impeller and W and C's direction vary depending on that and such that W=C-U (In vectorial notation) is satisfied

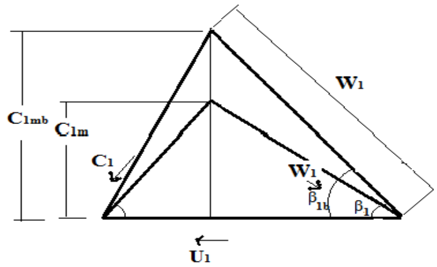


FIG 2.2: Velocity Triangle at Inlet of Impeller

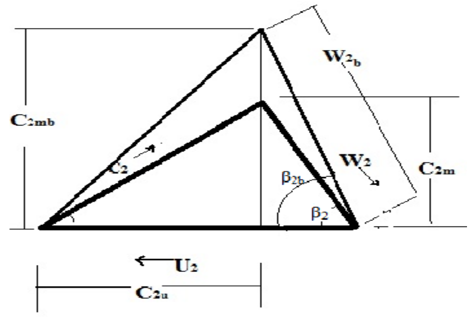


FIG 2.3: Velocity Triangle at Outlet of Impeller

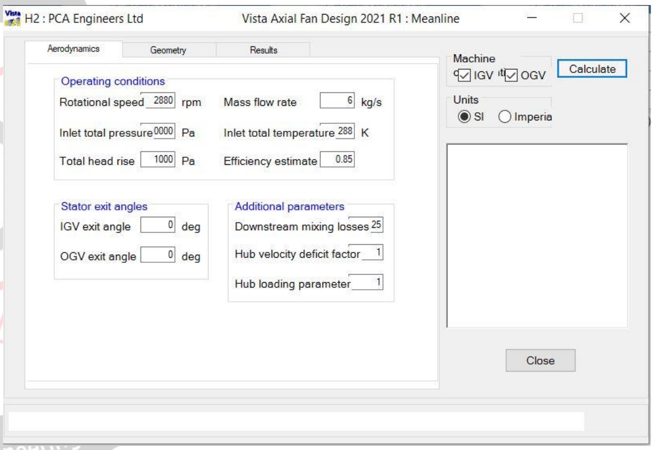


Fig 3.1: Blade Generation Input Parameters

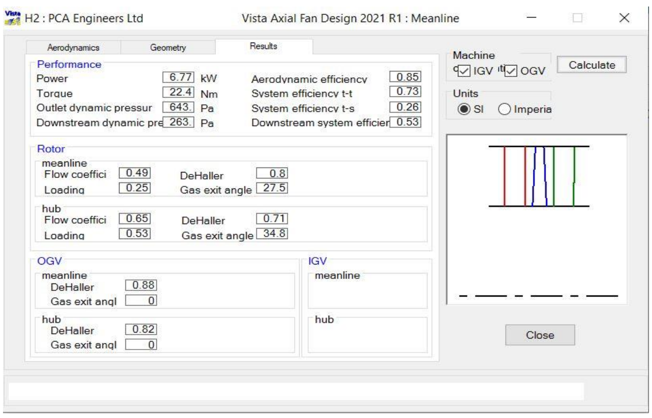


Fig 3.2: Calculating Parameters

DOI : 10.35291/2454-9150.2022.0248

International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-08, Special Issue, MAY 2022

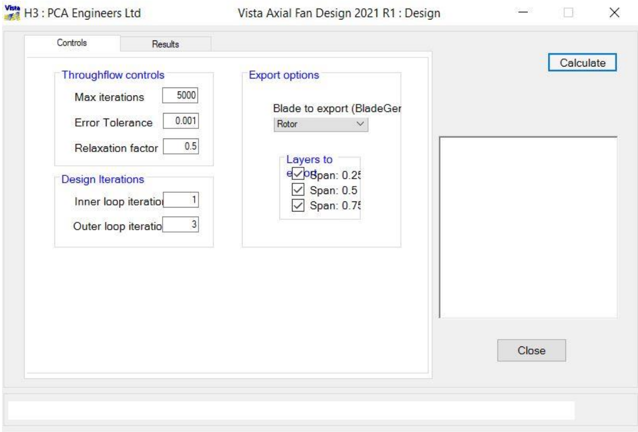


Fig 3.3: Performing The Analysis For Generated Results

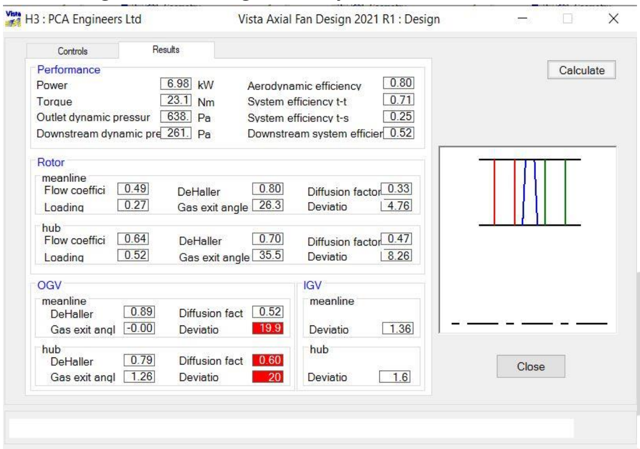


Fig 3.4: Design of Blade

Fig 3.5: Analysis of Blade Generation

3.2 Blade design in Solid works:

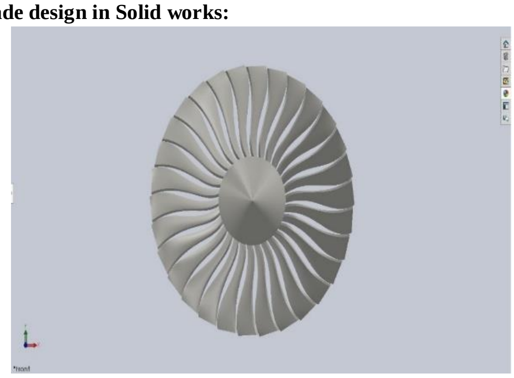


Fig 3.6: Front View of The Fan

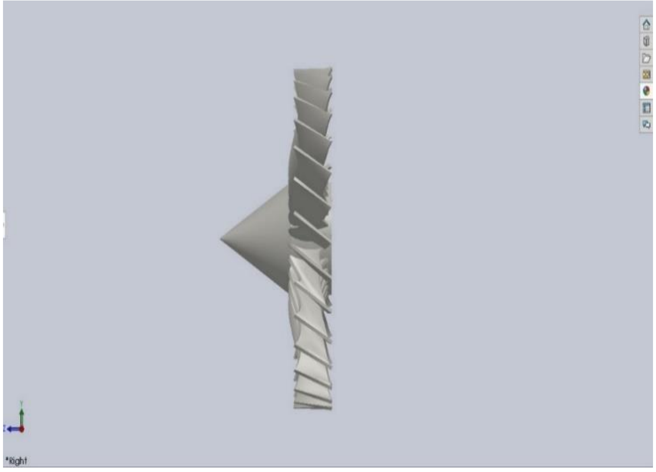


Fig 3.7: Side View of The Fan

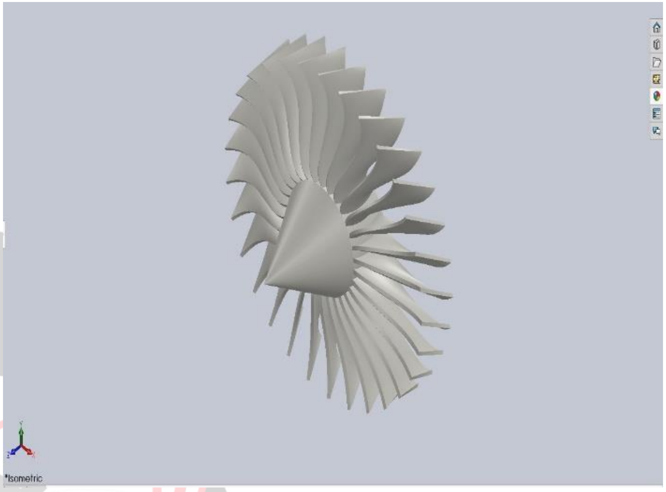


Fig 3.8: Isometric View of The Fan

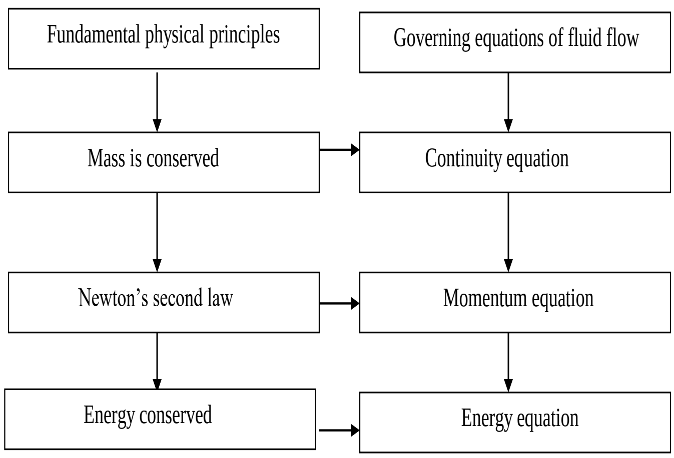
CFD THEORY:

CFD is playing a strong role as a design tool as well research tool. In CFD, the fundamental equations of fluid mechanics are based on the following universal laws of conservation:

1. Conservation of mass

2. Conservation of momentum

3. Conservation of energy.

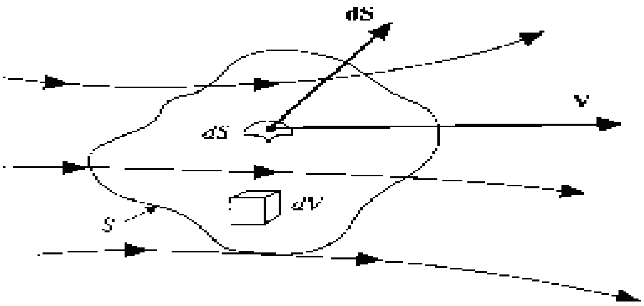


 DOI : 10.35291/2454-9150.2022.0248

ISSN : 2454-9150 Vol-08, Special Issue, MAY 2022

4.1.1 Continuity Equation:

Physical principle: Mass is conserved.



Net mass flow out of control volume through surface S = Time rate of decrease of mass inside control volume

Partial differential equation form of the continuity equation in differentiable conservative form can be expressed as

$$\left[\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}\right] dx\, dy\, dz = -\frac{\partial \rho}{\partial t}(dx\, dy\, dz)\frac{\partial \rho}{\partial t}$$

$$+\left[\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}\right] = 0$$

x, y, z → Cartesian Coordinates

u, v, w → velocity vectors in x, y, z directions.

L.H.S → Net mass flow out of the control Volume

R.H.S → Time Rate of Decrease of mass inside the control volume

The basic continuity equation of fluid flow is as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0$$

The first term in this equation represents the rate of increase of density in the control volume and the second term represents the rate of mass flux passing out of the control surface, which surrounds the control volume. This equation is based on Eulerian approach. In this approach, a fixed control volume is defined and the changes in the fluid are recorded as the fluid passes through the control volume. In the alternative Lagrangian approach, an observer moving with the fluid element records the changes in the properties of the fluid element. Eulerian approach is more commonly used in fluid mechanics. For a Cartesian coordinate system, where u, v, w represents the x, y, z components of the velocity vector, the continuity equation becomes

Vortex structures in a four-stroke engine just after injection of fuel and intake valve opening.

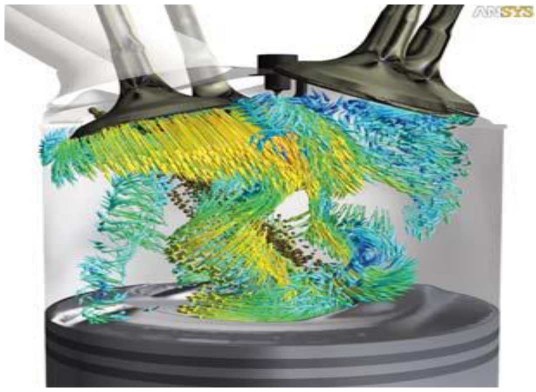


Fig 5.6.1

2. Nucleate boiling downstream of spacers in a fuel rod bundle assembly.

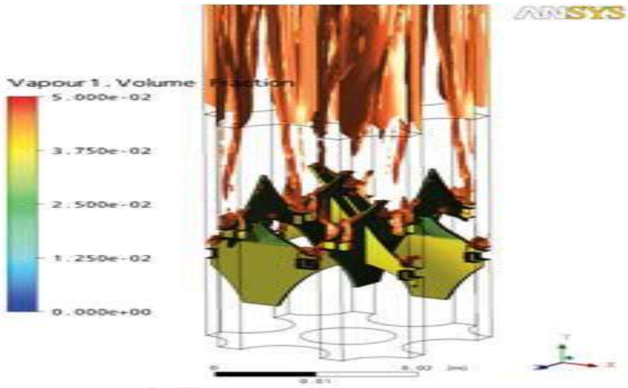


Fig 5.6.2

3. Prediction of heat transfer distribution in a shell and tube heat exchanger

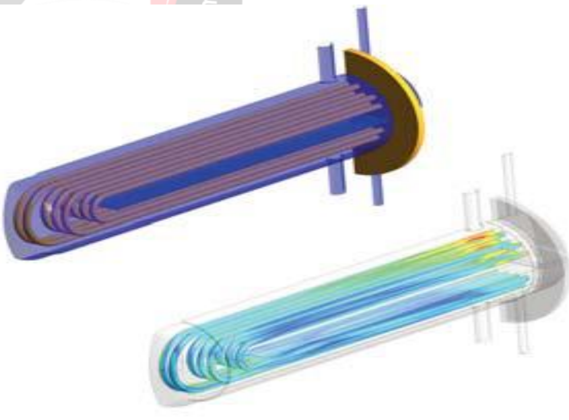


Fig 5.6.3

4. Prediction of wetness dispersion under non-equilibrium conditions for quantification of thermo-dynamic performance in a low- pressure steam turbine.

 DOI : 10.35291/2454-9150.2022.0248

International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-08, Special Issue, MAY 2022

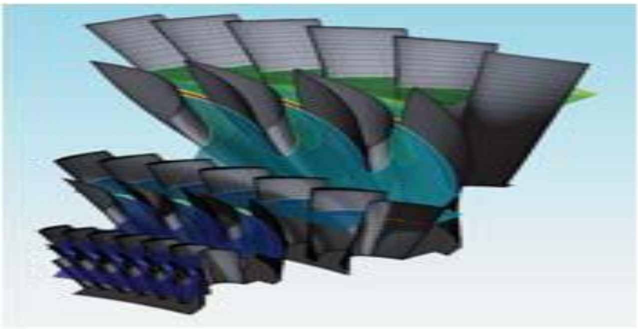


Fig 5.6.4 Ansys Fluent results

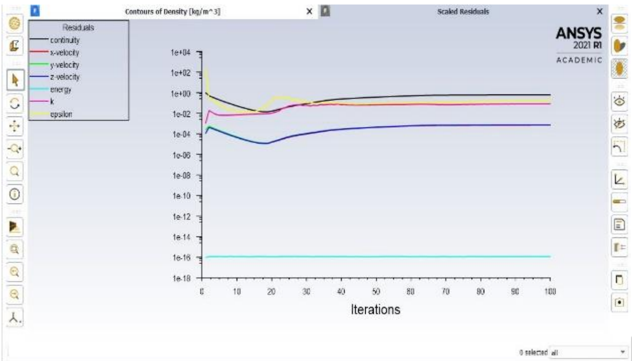


Fig 7.1: Scale Residuals

The continuity equation applies to all fluids, compressible and incompressible flow, Newtonian, and non-Newtonian fluids it expresses the law of conservation mass at each point in a fluid and must be satisfied at every point in a flow field

Solution converges as the graphs getting closers.

More no of iteration will give the closer and accurate.

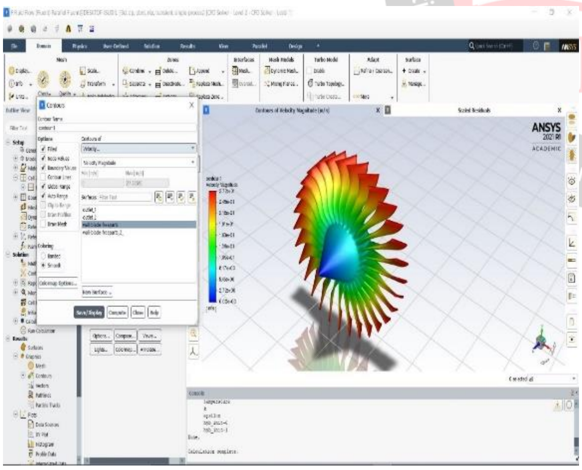


Fig 7.2: Velocity Over Blade

The maximum velocity is acting over the blade for designed condition. The blue color indication shows the very low velocity and the maximum velocity shown by red color region. The maximum velocity shown at the end of the blades due to impeller rotating and air flows outside of the blade.

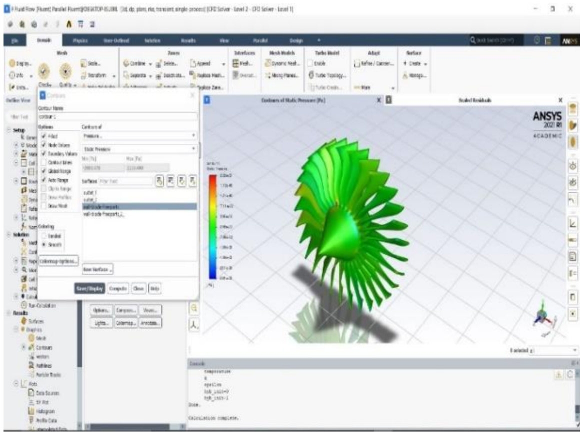


Fig 7.3: Pressure Acting on Blade

The maximum pressure is acting over the blade for designed condition. The blue color indication shows the very low pressure and the maximum pressure shown by red color region. Maximum pressure shown at the rotor and blade connection. Due to vortex flow at the joints more pressure is generating.

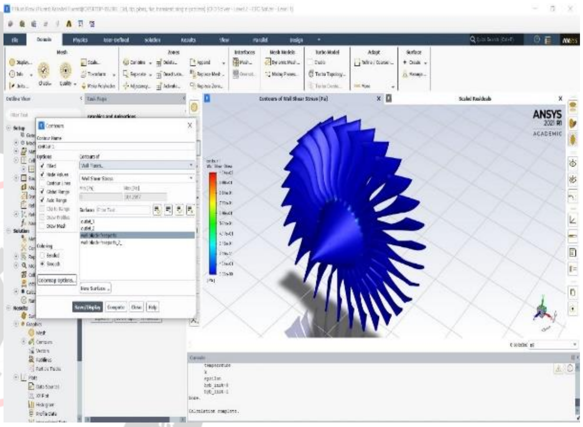


Fig 7.4: Shear stress over blade

The maximum shear stress acts at the blade and rotor joints. The shear stress acting very low.

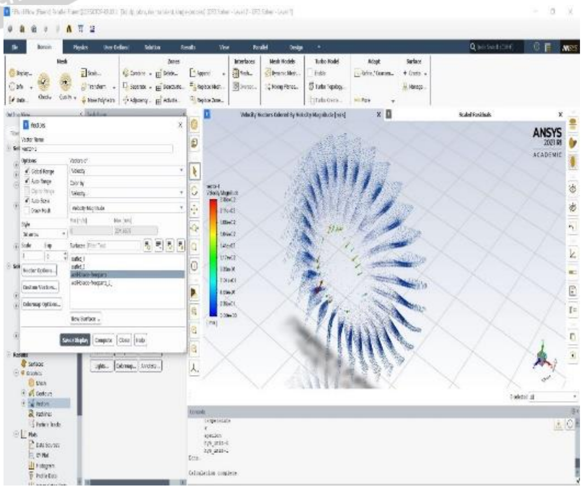


Fig 7.5: Velocity vector

A velocity vector represents the rate of change of the position of an object. The magnitude of a velocity vector gives the speed of an object while the vector direction gives

 DOI : 10.35291/2454-9150.2022.0248

International Journal for Research in Engineering Application & Management (IJREAM)
ISSN : 2454-9150 Vol-08, Special Issue, MAY 2022

its direction. Velocity vectors can be added or subtracted according to the principles of vector addition. velocity vectors magnitude.

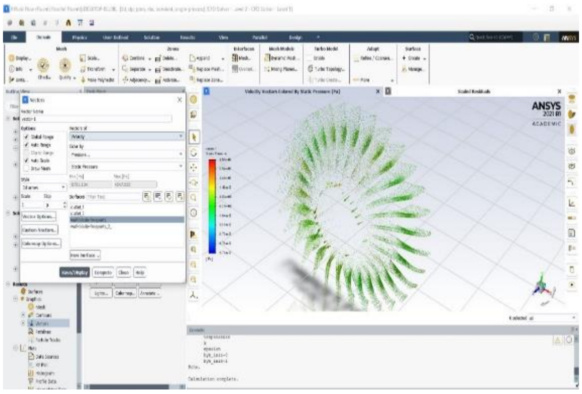


Fig 7.6: Pressure vector

Although pressure itself is a scalar, we can define a pressure force to be equal to the pressure (force/area) times the surface area in a direction perpendicular to the surface. The pressure force is a vector quantity. Pressure forces have some unique qualities as compared to gravitational or mechanical forces.

Velocity vectors at 50% span: -

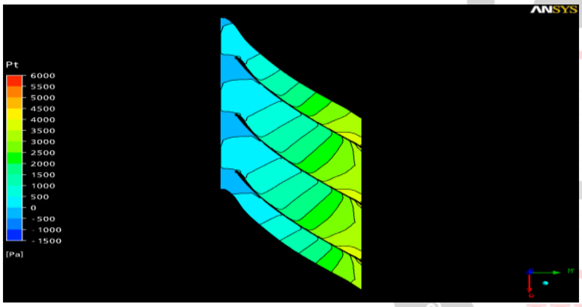


Fig 7.7 Total pressure for 980 RPM at 70% flow

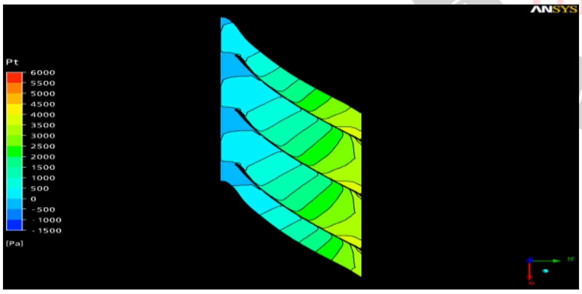


Fig 7.8 Total pressure at 980 RPM at 100% flow

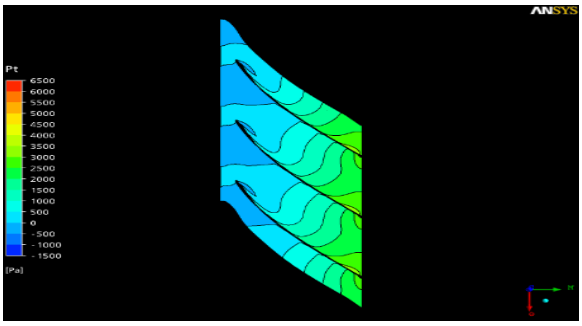


Fig 7.9 Total Pressure For 980 RPM At 130% Flow

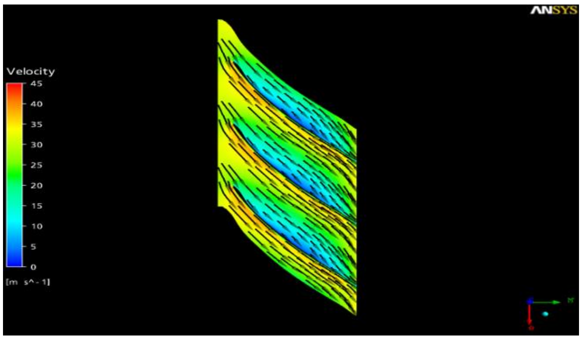


Fig 7.10 Velocity at 980 RPM at 70% flow

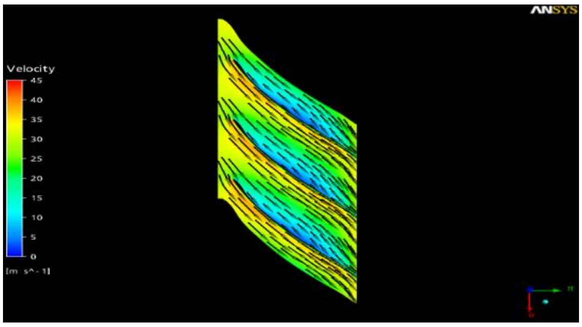


Fig 7.11 Velocity At 980 RPM At 100% Flow

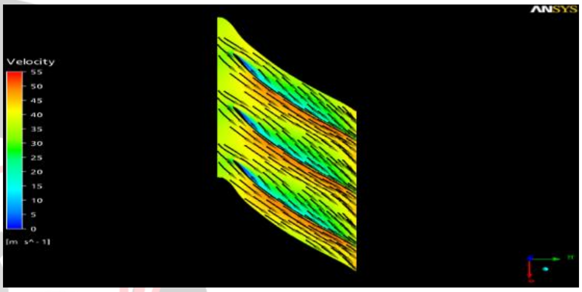


Fig 7.12 Velocity At 980 RPM At 130% Flow

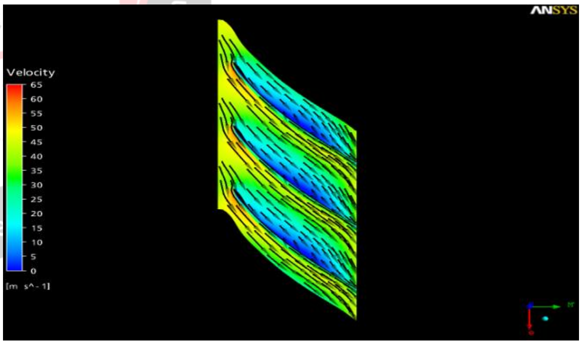


Fig 7.13 Velocity At 1450 RPM At 100% Flow

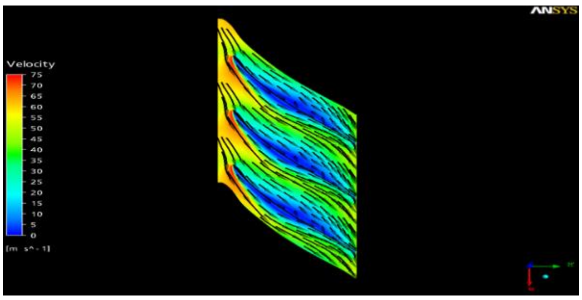


Fig 7.14 Velocity At 2000 RPM At 100% Flow

II. CONCLUSION

1. Aircraft propeller was designed for the given flow and maximum rotation. The fan impeller was designed by

 DOI : 10.35291/2454-9150.2022.0248

using ANSYS Turbo Grid, and it was analysed using FLUENT package.

2. The fan performance was evaluated and studied for different flow conditions converging design and off-design points of operation.
3. The performance is seen to be following the normal trend for a low specific speed fan and the flow and head curve shifts upwards with increasing speed.
4. The impeller efficiency seen to be maximum at the design point and decreasing at off-design conditions. The efficiency is found to be above 90%, this is because the windage losses, frictional losses have not been accounted.
5. The different contour and vector plots as well as the blade loading curve are included for typical cases of design and off-design conditions.
6. The pressure rise is seen to increase uniformly along the impeller passage.
7. This work may be extended by varying the number of impeller blades and by including the volute casing to get the total fan performance.

REFERENCES

[1] Ravi Anant Kishore, Thibaud Coudron, Shashank Priya “Small-scale wind energy portable turbine (SWEPT)” United States, 2013.

[2] Kim B, Kim J, Kikuyama K, Rooij V, Lee Y 3D numerical prediction of horizontal axis wind turbine power characteristics of the scale’s delft university T40/500 model, the fifth JSME-KSME fluids engineering conference, Japan.

[3] Mandas N, Cambuli F, Carcangiu CE, Numerical prediction of horizontal axis wind turbine flow, European wind energy conference, Athens, Grecce.

[4] Mr. Monir Chandrala, Prof. Abhishek Choubey and Prof. Bharat Gupta, 2012, Aerodynamic Analysis Of Horizontal Axis Wind Turbine Blade, International Journal of Engineering Research and Application, Vol. 2, Issue 6, Nov.-Dec. 2012, pp. 1244-1248.

[5] Ansys CFD tutorial.

[6] Aditya Rachman, La Aka Muhtar, Silvianus Levi, and Natanael Salea, 2013, An Experiment on Horizontal and Vertical Wind Turbines with Incorporation of Rounded Shroud Device Using Wind Simulation in a Vehicle, International Journal of Innovation and Applied Studies ISSN 2028-9324 Vol. 3 No. 2 June 2013, pp. 365-374.

[7] Srinivas G, Mahesha G T, Chethan K N and Arjun N, 2014, Analysis of Wind Turbine Blade, International Journal of Current Engineering and Technology E-ISSN 2277-4106, P-ISSN 2347-5161.

[8] Dr. Abdullateef A. Jadallaha, Dr. Dhari Y. Mahmoodand Zaid A. Abdulqader, 2014, Optimal Performance of Horizontal Axis Wind Turbine for Low Wind Speed Regime, IISN: 2321-3124.

[9] Darshil U. Shah, Peter J. Schubel and Mike J. Clifford,2013, Can flax replace E-glass in structural composites? A small wind turbineblade case study, Composites: Part B 52 (2013) 172-181.

[10] Cláudio Tavares Silva and Maurício Vicente Donadon, 2013, Unsteady Blade Element-Momentum Method Including Returning Wake Effects, J. Aerosp. Techno lManag. Sao Jose dos Campos, Vol.5, N 1. Pp. 27-42, JanMar. 2013.

[11] Y. Guo, J. Keller, T. Moan and Y. Xing, 2013, Model Fidelity Study of Dynamic Transient Loads in a Wind Turbine Gearbox, Conference paper NREL/CP-5000-58414 april 2013.

 DOI : 10.35291/2454-9150.2022.0248