Technium Vol. 14, pp.115-119 (2023)
ISSN: 2668-778X
www.techniumscience.com

Radial Centrifugal Fan Redesign and New Technologies Using CFD Applications

Sanda Budea

Hydraulics, Hydraulic Machines and Environmental Engineering Department, University Politehnica of Bucharest
Bucharest, Romania
sanda.budea@upb.ro or ORCID 0000-0002-9503-7564

Ion-Bogdan Iordan

PhHidro, MEng student
University Politehnica of Bucharest
Bucharest, Romania
bogdan.iordan@stud.mec.upb.ro

***Abstract*—The purpose of this article is to redesign centrifugal radial fans used in HVAC installations, using the analysis of aerodynamic flows and velocities distributions in the fan. Numerical simulations based on the reference model from the Pumps and Fans laboratory were performed using the SolidWorks program, Flow Simulation module. Following the simulations, information regarding the fluid flow in the machine rotor and casing, the most suitable shape of the blades resulted, and redesign solutions through 3D printing technologies were proposed. Numerical simulations with Flow Simulation on three-dimensional models were carried out, for a better analysis of the volume of air passing through the fan, but also for the prediction of some negative effects: the appearance of vortices that lead to high noises/acoustic disturbances, the appearance of additional vibrations, but also sudden and uncontrolled variations in the pressure force. Following the numerical analysis, it was found that for the new blade profiles the velocities are uniform, which greatly reduces the noise and vibrations in the operation of the fan. The redesign also aims to reduce the energy consumption of these fans.**

Keywords—fan redesign, flow simulation, velocities spectrum, 3D printing.

I. Introduction

In the current energy context, energy saving in all fields of activity is very important, saving that can be achieved by redesigning and improving the execution technologies of aero and hydrodynamic machines. For energy savings and operation without noise and vibration, all fans in HVAC systems must be redesigned. The careful profiling of the blades, as well as new manufacturing technologies, must be considered in the redesign. The dimensioning of the rotor, the orientation of the blades, the number of the blades and their overlap were verified, based on the theoretical approach from the papers [1-4]. The effect of the guide vanes in the operation of the fans was considered in our analysis, in accordance with the theory from the works [1-2]. Aspects regarding the design optimization of the fans were also studied in works like [5-7]. The turbulence phenomena in the fluid flow were analyzed and compared with those described in [8]. All these analyzes were aimed at improving the aerodynamic and energetic performance of the analyzed fan, as can be seen in [9]. In the present study three cases were analyzed: i) the old model existing in the laboratory with simple curved blades, fixed by riveting; ii) the model with redesigned, short, profiled blades; iii) the model with redesigned short, profiled blades and guide blades between the short ones.

CFD technique for modeling and simulating the flow in fans was also used in previous works [10-12], in order to find the optimal geometry, blade profile, visualization of current lines and velocities distribution in the rotor. The interaction of the fluid with the rotor, but also with the spiral casing, the use of the results in the optimization of the blade profile and the interspersing of long blades with short blades is what the present study brings. The model of analyzed fan is centrifugal, with axial air inlet and radial outlet, the blades are curved backwards.

The operating characteristics of the analyzed fan are: flow rate Q=3000 m^3/h, pressure H=30 m, or pressure variation Δp=359.046 Pa, air velocities v = (4.5-10) m/s, see also the catalog with fan characteristics [13].

II. The 3D Model of the Fan

The dimensions of the studied rotor of the fan are suction diameter D_1 = 400 mm, discharge diameter D_2 = 285 mm. The rotor has 35 short simple curved blades (the shape of the initial blade is a circular arc) and 5 long guide blades, interspersed between the short ones. The blades are made of 2.5 mm sheet. The width of the aerodynamic channel is b_1=b_2=70 mm, the blade inlet angle β_1 =51°, and outlet β_2 =125°, see Table I. The fixing of the blades on the rotor discs is done by riveting.

Regarding the discharge casing / volute, it is made of 5 mm sheets, with radius from 225 mm to 395 mm and rectangular discharge flange 320 x 182 mm, like in Fig. 1.

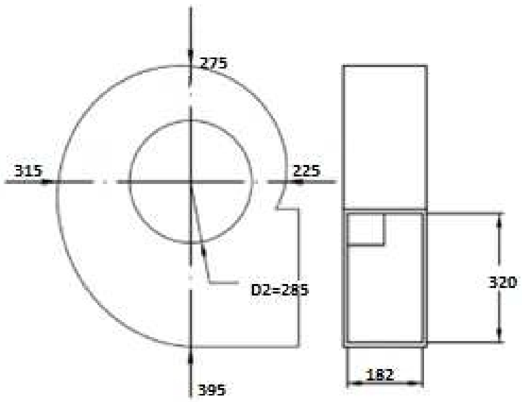


Fig.1 Spiral casing dimensions

TABLE I. Rotor geometry

D_1 (mm) inlet diameter	D_2 (mm) outlet diameter	$b_1=b_2$ (mm) channel width	β_1	β_2	s (mm)
400	285	70	51°	125°	2.5

In the following, we present the creation of the fan model with SolidWorks and the numerical analysis with Flow Simulation [14] of the three cases i) the old model existing in the laboratory with simple curved blades, fixed by riveting; ii) the model with redesigned, short profiled blades; iii) the model with redesigned short profiled blades and guide blades between the short ones. In fig. 2 is the model made in Solid Works of the reference fan rotor. The discretization of the air flow domain in a spatial network, the mesh, is made based on the 3D model. This numerical procedure assumes an external analysis, of the flow of the fluid both inside and outside the 3D model, and it is necessary to introduce some initial conditions. Defining a computing domain with initial and boundary conditions: the gas pressure inside the domain equal to the atmospheric pressure; the temperature of the gas inside the domain is equal to 20°C; the working fluid is air; the wall temperature is equal to the gas temperature, and its roughness is zero; the fluid velocities in each of the 3 directions, x ,y, z were set to zero, defined by 3D vectors.
See also Table II.

TABLE II. INITIAL AND BOUNDARY CONDITIONS

fluid	pressure	Temperature for air and walls	Initial velocities x,y,z
air	$p_{atmospheric}$	20 °C	0

Fig. 2. 3D reference centrifugal fan rotor

In Fig. 3 is represented the computational domain consisting in the rotor and volute casing of the fan.

With Flow Simulation module, for the previously created geometry, we obtained the velocities distributions and the flowing streamlines through the fan, as we can see in the following figures.

In Fig. 4 it can be seen that the velocities distribution in the area of the rotor blade is not uniform, there are areas with high air velocity, around the value of 30 m/s (in red) and similar areas with a lower air velocity, around 20 m/s. These uneven distributions can cause loss of load and the appearance of additional, unpredictable vibrations that can lead to damage to the fan.

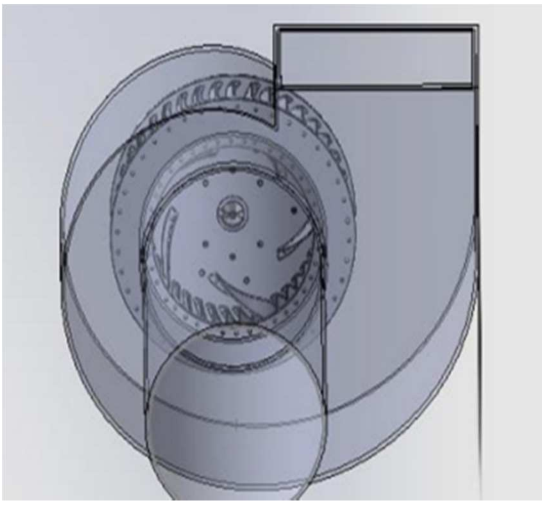


Fig. 3. Computational domain for the initial model i)

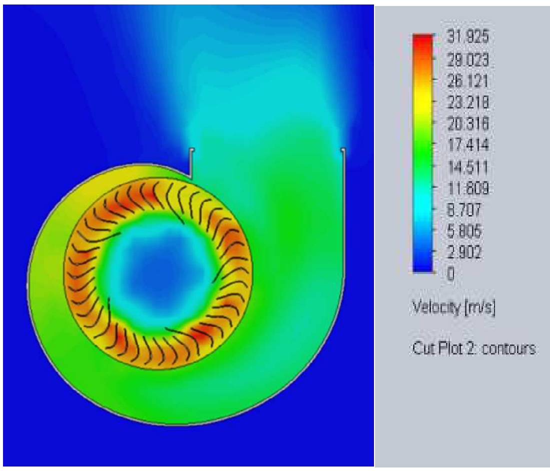


Fig. 4. Velocities spectrum i)

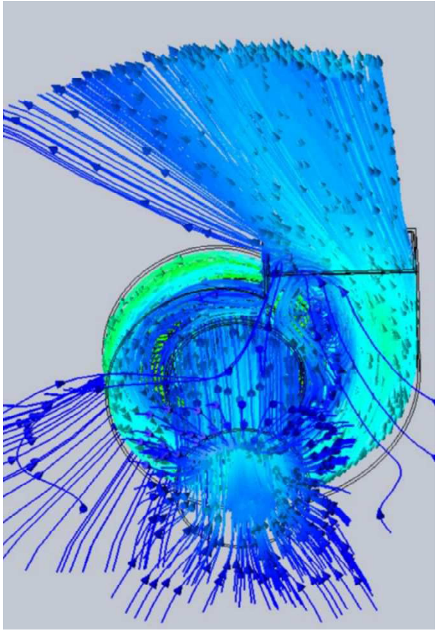


Fig. 5. Flowing streamlines for the initial rotor i)

The flowing streamlines from the suction to the discharge of the fan were represented in Fig. 5. The vortices formed at the sudden change in flow direction are highlighted.

III. Rotor Redesigned

A. Case ii) – Profiled Blades

The rotor has been redesigned in the same dimensions, but with 40 blades with curved aerodynamic profiles, without guide blades. Curved airfoils, similar to the profile of the reference fan blades, were used. We used the airfoil cp-180-050-gn, according to [15], with its chord length of 57.5 mm and maximum thickness of 2.5 mm located at 3.2% of the chord. The airfoil is curved, in the shape of a circular arc, its radius is 0.784 of the length of the chord as illustrated in the image below, like in Fig. 6.

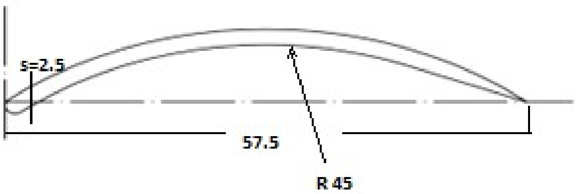
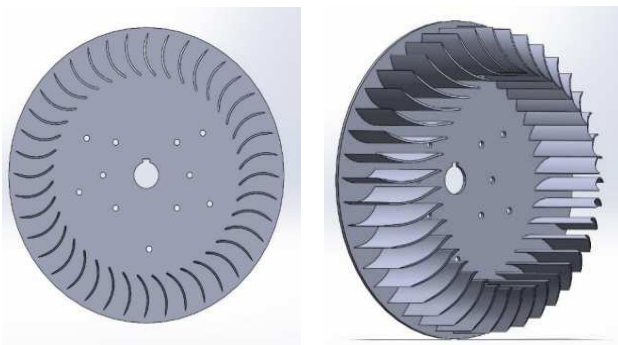


Fig. 6. Blade shape profiled – cp-180-050-gn



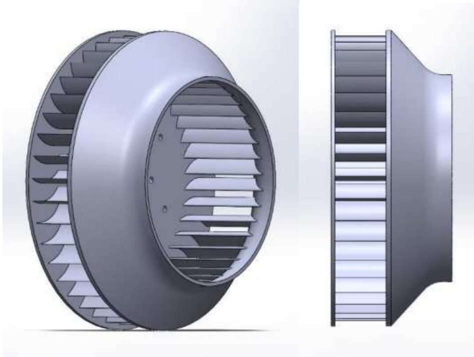


Fig. 7. 3D images with the fan rotor in case ii) profiled blades

With the new centrifugal fan rotor, modeled in 3D, like in Fig. 7, the numerical simulation of the air flow inside the centrifugal fan followed the same steps, resulting in the velocity distribution and flowing streamlines in Fig. 8 and 9.

It can be observed in the rotor blades, an accentuated uniformity of the velocities, compared to case i), the average value of the velocities in that area being approximately 29-30 m/s (shaded in green). Likewise, in the fan discharge area, the velocities distribution is uniform, which indicates high performance both in the rotor and in the spiral casing.

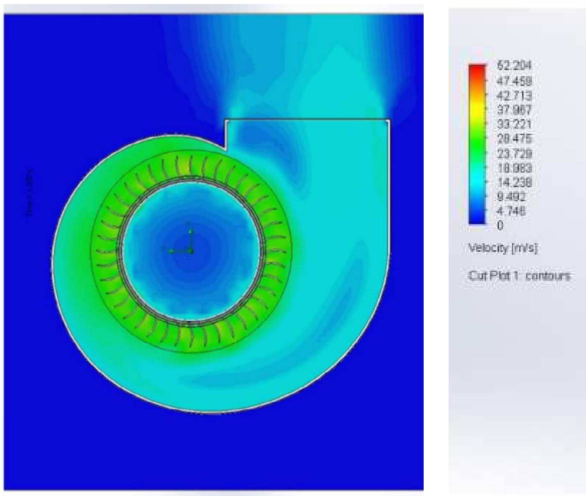


Fig. 8. Velocities spectrum in case ii)

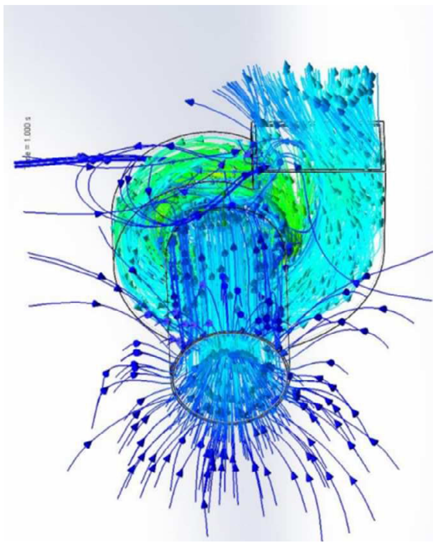


Fig. 9. Flowing streamlines in the fan

Technium Vol. 14, pp.115-119 (2023)
ISSN: 2668-778X
www.techniumscience.com
2023 11th International Conference on Thermal Equipment, Renewable Energy and Rural Development (TE-RE-RD)

B. Case iii) – Profiled Blades and Guidance Blades

The rotor is made with 35 blades in the form of short curved aerodynamic profiles, as in the previous model and with 5 guide blades, interspersed between the short ones. The addition of the guide blades has an effect in dissipating vortices that appear in the suction of the rotor. Part of the air flow that enters the fan axially is led to the blades, but a good part of it hits the wall of the rotor disk. Thus, the use of guide blades for efficient air routing is advantageous. The guide blade profile is illustrated in Fig. 10 and the rotor with 35 short blades plus 5 redesigned guide blades, in Fig. 11.

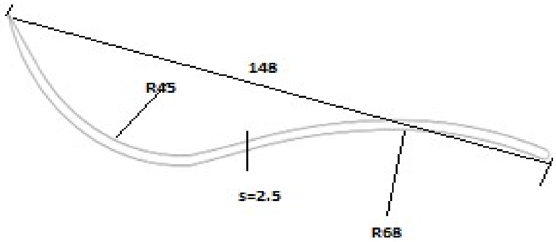


Fig. 10. Guide blade profile

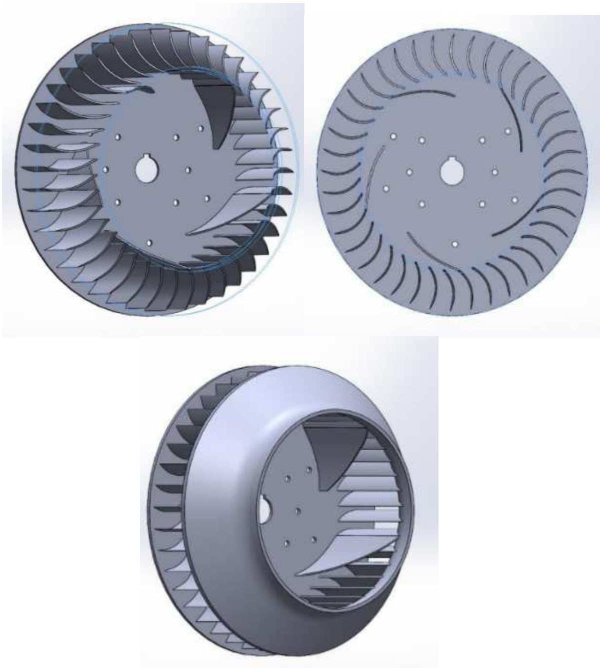


Fig. 11. Redesigned rotor with short aerodynamic curved blades and guide blades

The results of numerical simulations based on the rotor with 35 short blades, curved airfoil shape cp-180-050-gn and 5 guide blades, using the general initial conditions, as for all previous cases, are presented below – Fig.12 and Fig.13.

Analyzing the information obtained from the numerical simulation in this case, a better uniformity of velocities can be observed both in the profiled blades and around the guide blades. Airflow is directed to the blade, supported by the azure-green color gradient from the tip of the guide blades to the short ones. Similar results with reference to the flow in fan rotors are also presented in works [10-13]. The profile chosen by the authors of this article, cp-180-050-gn, is a special one, with good aerodynamics.

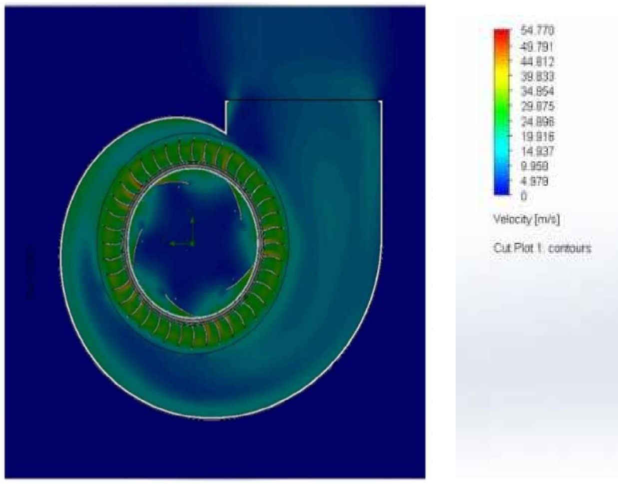


Fig. 12. Velocities spectrum for case iii)

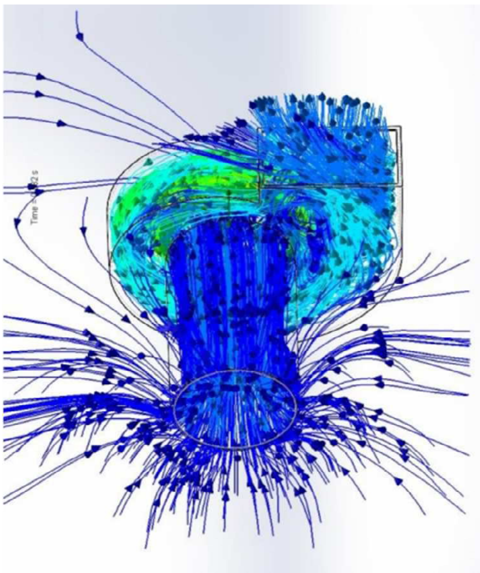


Fig. 13. Flowing streamlines in case iii)

The intercalation of long profiled blades between short ones in the design and simulation of the flow in the fan is an element of novelty. The advantage of this solution is to reduce the vortices from the suction, the air being better guided. In this study, unlike the previous ones, the flow is analyzed in the rotor and the casing, the simulations showing the interaction of the fluid with the set of aerodynamic landmarks. Similar application for turbomachines design and flowing simulation can be realized also with application [16].

IV. Technological Aspects

It is often preferred to make the rotor from pieces of thin sheet metal, to simplify the manufacturing process, after which they are assembled by riveting, as in the reference version. However, for high-pressure fans, the rotor can also be made by casting, aluminum being generally chosen as the construction material, both due to its high resistance characteristics and its low specific weight.

Technologies 3D printing [17], like FDM-Thermoplastic Extrusion Modeling was used to make the redesigned rotor, PET-G was used to make the rotor, a thermoplastic polyester that offers excellent mechanical and chemical resistance, durability, and good casting structure – Fig. 13.

Filament thickness: 0.8 mm, Layer height: 0.32 mm, Printing time: 15:45 h.

For a better mechanical resistance, the rotor is made by aluminum casting, with an aluminum density of 2700 kg/m^3, resulting a light fan rotor mass of 4.96 kg.

Through the new technology, 3D printing, fixing the blades with rivets or by welding is avoided, so that areas of aerodynamic turbulence are eliminated. This leads to the reduction of noises and vibrations, but also to the improvement of the fan's performance / efficiency.

These performances will be demonstrated through experimental tests of the fan with the rotor iii) and made by 3d printing technology. The experimental results will be the subject of another article.

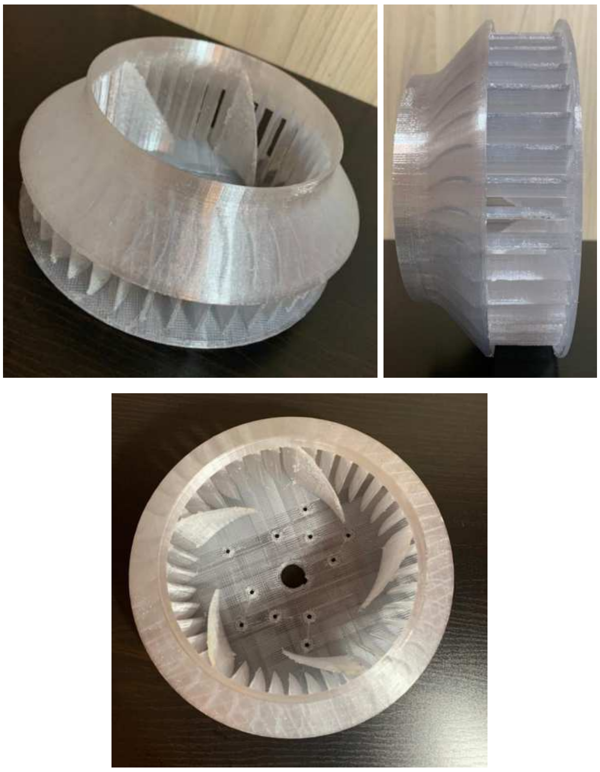


Fig. 13. 3D printed model of the rotor - PET-G

V. Conclusions

In this paper, the authors propose a model of a centrifugal axial fan with a redesigned rotor with short, profiled blades and long guide blades, based on the results of the CFD analysis with Flow Simulation in SolidWorks. Three cases were studied:

i) the old model with simple curved blades, fixed by riveting;
ii) the model with redesigned, short profiled blades;
iii) the model with redesigned short profiled blades and guide blades between the short ones.

More uniform velocity distributions resulted from cases ii) and iii). The flowing spectrum through the fan showed that the vortices from the rotor suction decreased with guidance blades. These have good effects on reducing the level of noise and vibrations from the fan.

The proposed 3D printing manufacturing technologies and the analyzed materials, both the replacement of the riveting of the blades with casting of light materials, simultaneously with the aerodynamic profiling, represent a new trend, to be followed in the manufacturing of fans, with wide applications in the HVAC industry.

In future research, the redesigned rotors will be experimentally tested and the effect of the redesign will be followed according to the operating conditions.

References

[1] Frank P. Bleier, Fan Handbook Selection, Application, And Design. New York: McGraw-Hill, 1998.

[2] V. V. Tcacenco, Mechanical ventilation installations. Vol II, Air channels and mechanical equipment, București: Hydraulics and Hydraulic machines Chair, 1996.

[3] S M. Yahya, Turbines Compressor and Fans, 4th Edition. New Delihi : Rashtriya Printers, 2011, pp.618-623.

[4] Yang Yue, s.o., Hydrodynamic Analysis of Gas Flow in Centrifugal Ventilator. Front Sci., 2018. http://frontsci.com/journal/article?doi=10.32629/rwc.v1i1.3

[5] K. J. Lee, s.o., "Optimal Design of a Plenum Fan with Three-Dimensional Blades", Applied Sciences. 10, 3460, 2020.

[6] S. Ni, W. Cao, J. Xu, Y. Wang, W. Zhang, "Effects of an inclined blade on the performance of a sirocco fan", Applied Sciences, 9, 3154, 2019.

[7] K.Patel, Prajesh M Patel, "Performance analysis and optimization of centrifugal fan", Int Journal of Emerging Trends in Engineering and Development 2(3) , 2013. Available online http://www.rspublication.com/ijeted/ijeted_index.htm

[8] David C. Wilcox, Turbulence Modeling for CFD 3th Edition, San Diego : Birmingham Press, 2006.

[9] N. N. Bayomi, and Ahmed A. Osman, "The Effect of Inlet Configurations on Centrifugal Fan Performance", Energy Conversion and Management 47, 2006, pp.3307–3318.

[10] T. Siwek, J. Górski, F.Stanisław, "Numerical and Experimental Study of Centrifugal Fan Flow Structures and Their Relationship with Machine Efficiency", Pol. J. Environ. Stud. Vol. 23, No. 6, 2014, pp.2359-2364.

[11] Hemant Kumawat, "Modeling and Simulation of Axial Fan Using CFD", World Academy of Science, Engineering and Technology, International Journal of Aerospace and Mechanical Engineering, Vol:8, No:11, 2014.

[12] Ali Sahili, Bashar Zogheib, Ronald M. Barron, "3-D Modeling of Axial Fans", Applied Mathematics, 2013, 4, 632-651 http://dx.doi.org/10.4236/am.2013.44088 (http://www.scirp.org/journal/am)

[13] https://www.hindustrialfan.com/multi-blade-blower/ (accessed in feb 2023).

[14] How to model an interlinked star geometry in SOLIDWORKS? | LearnSOLIDWORKS.com (accessed in feb 2023).

[15] http://airfoiltools.com/airfoil/details?airfoil=cp-180-050-gn. (accessed in feb 2023).

[16] https://www.cfdsupport.com/centrifugal-fan-design-and-simulation.html (accessed in feb 2023)

[17] O.P.Gupta, "3D Printing - Manufacturing Technology of Present and Future,. Conference: Seminar on Engineer's Day, Bareilly, India. 2015. https://www.researchgate.net/publication/281782380