Certificate of Attendance

VIRTUAL PRESENTATION

Name & Surname **Noureddine Menasri**

Paper Title **Retro design of impeller blade of the induced Draft fan (FN-280) using A Coordinate measuring machine (CMM).**

ICAME 2021

6TH INTERNATIONAL CONFERENCE ON ADVANCES IN MECHANICAL ENGINEERING

OCTOBER 20-22, 2021 ISTANBUL TURKEY

ICAME 2021 CONFERENCE CO-CHAIRMAN

Prof. Dr. Zehra YUMURTACI
Head
Mechanical Engineering Department

ICAME 2021 CONFERENCE CHAIRMAN

Prof. Dr. Ali Fuat GÜNERİ
Dean
Mechanical Engineering Faculty

RETRO DESIGN OF IMPELLER BLADE OF THE INDUCED DRAFT FAN (FN-280) USING A COORDINATE MEASURING MACHINE (CMM).

Amour Aissa, Menasri Noureddine, Bidi TArek, Bakhti Fatima Zohra, Menasri Youcef
Laboratory of Materials and Structural Mechanics LMMS, Faculty of Technology, University of M'sila
Corresponding author: Aissa.Amour@univ-msila.dz

ABSTRACT

Centrifugal fans are widely encountered in engineering practices, they are essential in the cement production process. The three dimensional, complex and turbulent flow in a centrifugal fan make the prediction of the performance of the centrifugal fan and the examination of the flow field very difficult. Computational techniques have made a great progress, because of time and cost that may be involved in experimental analysis of the flow, computational fluid dynamics (CFD) is intensively used in many industrial purposes. Nevertheless, the agreement between the numerical results and real data is a subject of many researches, because the numerical result is very sensitive to the numerical method being used, the boundary conditions applied, mesh generation and turbulence model selecting.

to consider the interaction between the main three parts of the fan namely the inlet, impeller, and scroll, it is necessary to carry out a three-dimensional Numerical simulation of the flow field for the whole centrifugal fan. The impeller is the important part of the centrifugal fan and the aerodynamic design of the impeller blades affects the flow passage, thus improving the aerodynamic design of the blades results in improvements of fan performance and reduction of flow separation. The selection of turbulence model depends on the flow separation that can occur, particularly in the blade passage, thus in order to get a good numerical investigation of the performance of the fan and predict the internal flow field in the impeller, a Coordinate measuring machine (CMM) machine is used as a reverse engineering tool to extract the original impeller blade design of the induced Draft fan (FN-280) by using Geomagic software for point cloud processing.

Keywords: induced Draft fan, impeller blade design, Three-dimensional measurement, point cloud processing.

INTRODUCTION

Over the last few years, centrifugal fan market has become hyper-competitive, with the expansion of its use in many industrial domains and the increasing priority being given to operation cost. High-powered draft fans used in the cement manufacturing process must be developed to deliver high aerodynamic efficiency today over a much wider operating range than before. Since the blades are the most important parts of the fan, their geometric properties and configuration determine the performance of the fan.

While experimental analysis continues to play an important role in fluid mechanics, numerical simulation is extensively used to predict the flow behavior and provide performance evaluation of a specific design. Experimental method is money and time consuming for this reason, The technique of numerical simulation of fluid flow CFD (Computational Fluid Dynamics), based on the resolution of the Navier-Stokes equations has more benefit, it is used to improve the overall performance of fans in the design phase and develop new products more quickly and with less uncertainty.

A large number of researches in this field mainly related to numerical simulation of flow in turbomachinery (centrifugal fan) are as follows:

A. Amjadimanesh, H. Ajam and A. Hossein Nezhad [1**]**, simulated numerically a 3D forward curved centrifugal fan. various turbulent models were compared and they examined the influence of blade shapes including (flat blade, circular blade and NACA4412 airfoil blade) on the fan performance.to determine the most efficient one. They introduced the k-epsilon and circular arc blade as the suitable model and most efficient blade shape respectively.

Chhagan Lal Kharol, Shankar Lal Suthar [2], investigated the effect of the shape of two different airfoil blade namely (NACA2412 and NACA2424) in the performance of centrifugal fan. They found that the blade profile plays a predominant role to reduce flow separation and recirculation that occurs at trailing edge of the fan. Thus, providing better performance and energy consumption.

Kyung Jung Lee et al [3], proposed an optimized plenum fan by designing a four-layer three-dimensional blade. It was concluded that the optimized model stabilized the flow and reduced the flow separation at the leading edge resulted in an improvement of both the static pressure and static efficiency.

Lin Wu1 et al [4] carried out and optimization of the blade design by employing the controlling velocity distribution method. Good enhancement of the fan performance, static pressure and stability.

Rui Rong, Ke Cui, Zijun Li, Zhengren Wu [5], analyzed numerically the aerodynamic properties of the blade (G4-73No.8D) of a centrifugal fan with slots cut along the blade pressure side to suction side. The study shows that creating slot in blades can improve the blade surface flow behavior by decreasing the blade surface resistance and boundary layer separation which in turn decrease energy consumption.

With the development of computer resources and computational techniques, great efforts have been made in the context of numerical simulation of complex flow in a centrifugal fan in order to improve the performance design of these machines. According to the findings of the above researches, the aerodynamic characteristics of the blade of a centrifugal fan has a great influence on the overall performance of the fan including: static pressure efficiency, pressure loss, flow separation and energy consumption.

In this paper, an impeller blade of the draft fan FN-280 is considered for reverse design. With the intent purpose of numerically simulating the flow field in the centrifugal fan. The CAD model of the blade was not available. Considering the fact that geometrical characteristics of the blade is crucial in order to validate the numerical results, therefore, reverse design using a Coordinate measuring machine (CMM) is adopted to develop the CAD model of the blade on one hand, on the other hand, remaining parts of regular contours are designed by SolidWorks.

1- REVERSE DESIGN METHODOLOGY

Reverse Design is the technology of converting digital data called 3D scan data collected by a 3D scanner of the geometrical information of a real object into a CAD model to be used in the manufacturing process.

Reverse design steps can be outlined as follow:

- **Scanning Phase:** This phase is involved with the scanning strategy–selecting the correct scanning technique, preparing the part to be scanned, and performing the actual scanning to capture information that describes all geometric features of the part such as steps, slots, pockets, and holes.
- **Point Processing Phase:** This phase involves importing the point cloud data, reducing the noise in the data collected, and reducing the number of points. A wide range of commercial software is available for point processing. The output of the point processing phase is a clean, merged, point cloud data set in the most convenient format.
- **Application Geometric Model Development Phase:** The generation of CAD models from point data is probably the most complex activity within RE because potent surface fitting algorithms are required to generate surfaces that accurately represent the three-dimensional information described within the point cloud data sets.[6]

In this work, coordinate measuring machine (CMM) is used for the collection of point cloud data which is shown in Figure [1]

Scanning the blade with CMM

Before starting the scanning, the workspace preparation must be taken into consideration. Blade alignment is the process of associating both the coordinate system of the CMM and the blade Figure [2]. Scan data is collected by mechanical method using touch-trigger probe. The CMM probe collects data by touching the surface along the complete profile of the part to provide continuous flux of data. Therefore, probe selection, position control, path followed by the probe and angle at which the probe approaches the surface is very important during the scanning phase.

Exporting 3D scan data to Geomagic design

All information collected during the scanning is converted into digital data (point cloud data) through (**PC-DMIS**) software then exported to any CAD modeling software (**Geomagic design X** in this case) to obtain a final CAD model.

Figure 1. Coordinate measuring machine

Figure 2. Impeller Blade alignment

2- Impeller Blade Reverse Design workflow

3D scan data consists of point cloud and a mesh. Point cloud is a set of three-dimensional vertices. Each vertex corresponds to one position on the surface of the blade, whereas mesh is defined by vertices, edges and faces. The process of converting point cloud into mesh is performed by the scanning software (PC-DEMIS) of the coordinate measuring machine (CMM).

scan data is raw information, mesh optimization is needed before proceeding to use 3D scan data provided by scanning machine. Features available in Geomagic design x for processing scan data were used in order to optimize mesh globally. Figure [3] shows scan data and figure [4] shows mesh regions created in order to use the surface fitting technology.

The reverse design strategy depends on the scan data type, the shape complexity of the model and the design intent. Because the section of the blade of the fan FN-280 is not uniform and the surface of the leading and trailing edge is deformed causing the scan data includes deformities also, the method adopted in the reverse modeling process is Hybrid modeling which mixes with parametric modeling and surface fitting methods.

The surface fitting technology provides a powerful way to create freeform surface from a specific freeform region on a mesh [7]. The Mesh fit tool is used to generate the surface of both the suction and pressure side of the blade.

Figure [6] shows the surfaces created by setting the allowable deviation to (0.0008mm), number of control points to (70) for better control of iso-lines and minimizing the smoothness in order to envelope the entire mesh region. figure [6] illustrates the deviation from the mesh which shows that is within a good range.

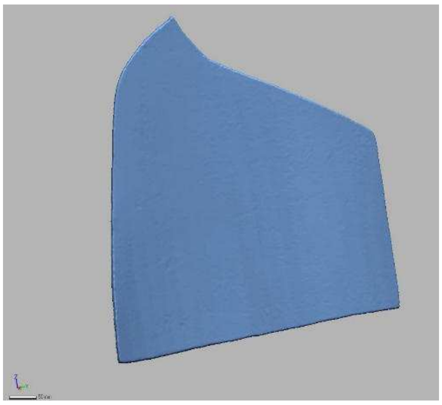

Figure 3. Cloud data from CMM

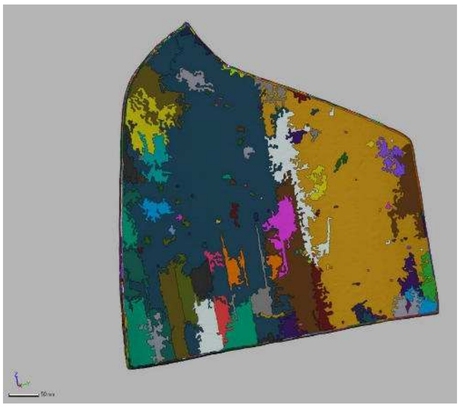

Figure 4. Mesh regions

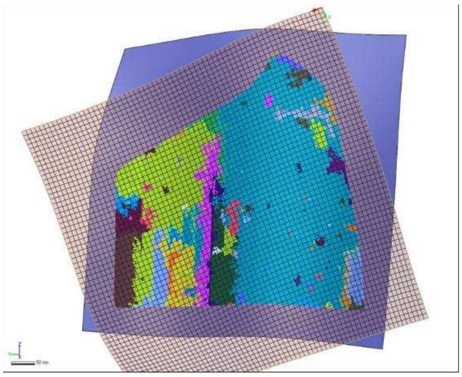

Figure 5. iso-lines created in the surface

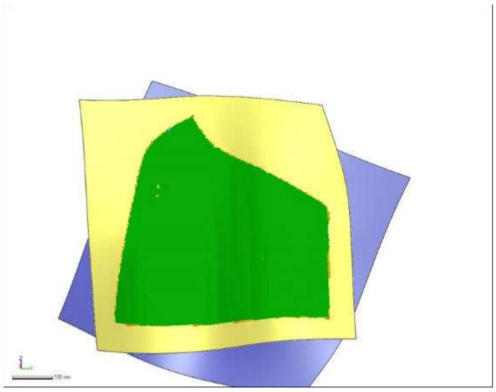

Figure 6. surface fitting of pressure and suction side

Three-dimensional mesh sketch curve with nodes connected to only concave mesh regions was created. Offset curve of (0.03mm) width is created to generate the Surface portion of the shroud and the leading-edge surface.

Figure [7].and [8] shows the deviation for mesh using the accuracy analyzer and Figure [9] and [10] shows the surfaces generated respectively.

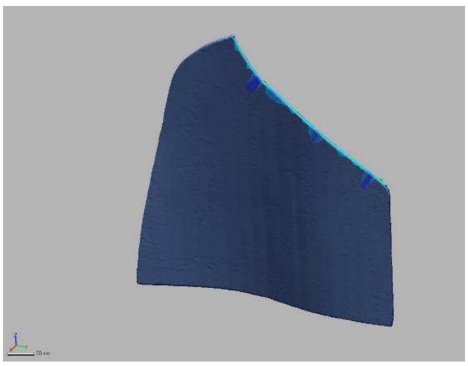

Figure 7. shroud side curve deviation from mesh

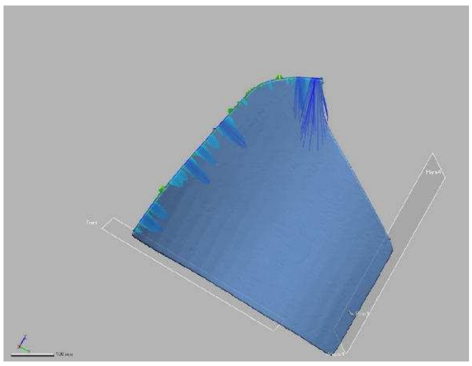

Figure **8**. leading edge curve deviation from mesh

Figure 9. surface-cut of shroud part

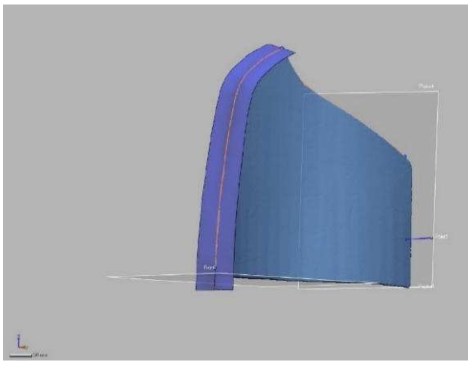

Figure 10. surface-cut of leading edge

The process of creating the trailing edge surface is as follow:

- Extract a tangent plane (plane 1) to the surface of the pressure side of the blade near the trailing edge mesh region
- Create a plane (plane 4) that is orthogonal to both (plane 1) and the front plane.
- (Plane 4) must contains the first point of the 3D mesh sketch on the trailing edge mesh region contour.

Figure [11] and [12] Shows the surface cut plane (plane 4)

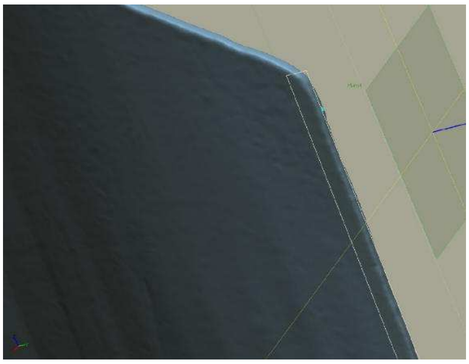

Figure 11. 3D sketch on surface cut plane

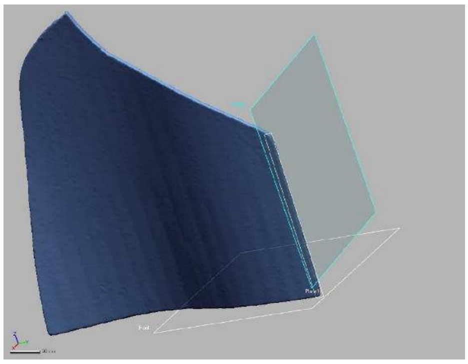

Figure 12. surface cut plane (plane 4)

The impeller CAD model is finally obtained by trimming all surfaces and planes created. Accuracy analyzer is used to control the deviation of the CAD model from the model.

Figure [13] shows all the surfaces and planes used in the CAD model design process and Figure [14] shows the final CAD model of the impeller Blade.

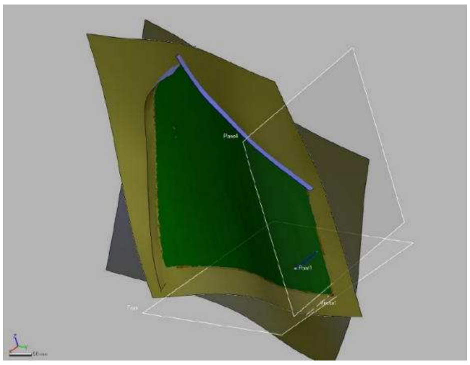

Figure13. planes and surfaces

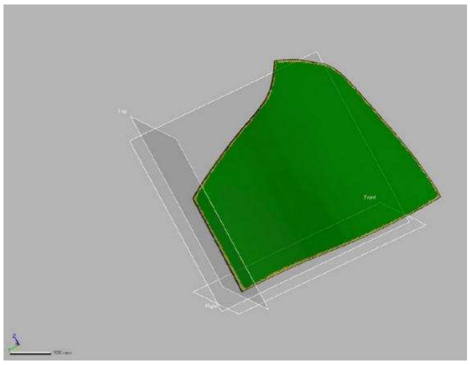

Figure 14. Final CAD model

CONCLUSION

All main parts of the centrifugal fan FN-280 including the inlet duct and the casing were designed by soldiworks. the impeller blade is selected for reveres design because the CAD model is not available and the complexity of its geometrical characteristics.

Scanning of the impeller blade is done using Coordinate Measuring Machine (CMM),3D scan data obtained is exported from the software bundle machine (PC-DMIS) to Geomagic design X to obtain CAD model of the impeller blade.

This study shows that revere design using coordinate measuring machine (CMM) helps to get all details about complex geometrical information of any object with high level of accuracy resulting in minimizing the design process time, provided the scanning technique and the modeling strategy are selected properly.

REFERENCES

[1] A. Amjadimanesh, H. Ajam and A. Hossein Nezhad. Effect of blade type on a 3D FC centrifugal fan. Journal of the Serbian Society for Computational Mechanics / Vol. 8 / No. 1, 2014 / pp. 23-35

[2] Chhagan Lal Kharol, Shankar Lal Suthar. Performance Analysis of Centrifugal Fan with Airfoil Blade using CFD. international journal of research culture society Volume - 3, Issue - 5;2019.

[3] Kyung Jung Lee, Wung Park , Ki Suk Bang , Yeong Min Kim and Young Chu Ahn.Optimal Design of a Plenum Fan with Three-Dimensional Blades. Appl. Sci. 2020, 10, 3460.doi:10.3390/app10103460.

[4] Lin Wu1, Hua-Shu Dou, Yikun Wei, Yongning Chen, Wenbin Cao, Cunlie Ying. International Journal of Fluid Machinery and Systems Vol. 9, No. 1, January-March 2016.http://dx.doi.org /10.5293/IJFMS .2016.9.1.095

[5] Rui Rong, Ke Cui, Zijun Li, Zhengren. Numerical study of centrifugal fan with slots in blade surface. Procedia Engineering 126 (2015) 588 – 591.doi:10.1016/j.proeng.2015.11.309.

[6] Gayatri D. Barai1, Sunil S. Shete2, Laukik P. Raut. design and development of a component by reverse engineering. International Journal of Research in Engineering and Technology Volume: 04 Issue:05;May-2015.

[7] Geomagic fesign X user guide

[8] PC-DMIS- training manual n-I (2012 MEI-M’sila).

[9] PC-DMIS- training manual n-II (2016 MEI-M’sila).