Global Journal of Researches in Engineering: D
Aerospace Science
Volume 17 Issue 2 Version 1.0 Year 2017
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN:2249-4596 Print ISSN:0975-5861

Experimental and Simulation Performance for Fan Extraction System

By Dr. Klaudio Bari & Dr. Syed Hasan
University of Wolverhampton

Abstract- Computational Fluid Dynamics (CFD) is commonly implemented in industry to perform fluid-flow and heat-transfer analysis, however, rarely used in computational material engineering. The project aim is to select a biodegradable composite for impeller used in conventional domestic extraction fan. The analysis starts with a validation of the experimental values of thrust obtained from different materials used to manufacture impellers tested in a wind tunnel. An identical model was used in the CFD simulation using STAR-CCM+ software. The study compares thrust values of different impellers made from thermoplastic polymer such as Acrylonitrile Butadiene Styrene (ABS), Polylactic Acid (PLA) and composite materials such as Bronze-fill, fibre glass and wood fill composites. The results revealed that composite impellers like Bronze-fill and Glass fibre fill composites perform better in term of thrust than traditional ABS impeller. Also, biodegradable wood fill composite shows competitive alternative to replace the conventional AB Simpeller used in fan extraction units. A cost comparison shows that wood-fill composite impeller would be cheaper to manufacture using Polymer Injection Moulding (PIM). A crack growth can be monitor using Acoustic Emission (AE) during solidification of molten plastic.

GJRE-D Classification: FOR Code: 090199p

EXPERIMENTALANDSIMULATIONPERFORMANCEFORFANEXTRACTIONSYSTEM

Strictly as per the compliance and regulations of:

Experimental and Simulation Performance for Fan Extraction System

Dr. Klaudio Bari [α] & Dr. Syed Hasan [σ]

Abstract- Computational Fluid Dynamics (CFD) is commonly implemented in industry to perform fluid-flow and heat-transfer analysis, however, rarely used in computational material engineering. The project aim is to select a biodegradable composite for impeller used in conventional domestic extraction fan. The analysis starts with a validation of the experimental values of thrust obtained from different materials used to manufacture impellers tested in a wind tunnel. An identical model was used in the CFD simulation using STAR-CCM$^+$ software. The study compares thrust values of different impellers made from thermoplastic polymer such as Acrylonitrile Butadiene Styrene (ABS), Polylactic Acid (PLA) and composite materials such as Bronze-fill, fibre glass and wood fill composites. The results revealed that composite impellers like Bronze-fill and Glass fibre fill composites perform better in term of thrust than traditional ABS impeller. Also, biodegradable wood fill composite shows competitive alternative to replace the conventional AB Simpeller used in fan extraction units. A cost comparison shows that wood-fill composite impeller would be cheaper to manufacture using Polymer Injection Moulding (PIM). A crack growth can be monitor using Acoustic Emission (AE) during solidification of molten plastic.

I. Introduction

Traditionally, Computational Fluid Dynamics (CFD) is commonly used in industry to perform fluid-flow and heat-transfer analysis of turbo machinery equipment. The use of CFD plays an important role in fluid mechanics. Due to the progress of numerical methods and computer capability, the impeller design nowadays has been analysed by using numerical analysis and CFD software to predicttimpeller performance (Shah 2013). However, both manufacture and design cannot be done easily with out sufficient details about experimental outcomes. If there is discrepancy between result obtained from the CFD and experimental set up, the impeller cannot be manufactured in a real production line (Kaminsky, 2012).

The project used STAR-CCM+ as a simulation software tool of an identical model of domestic fan extraction placed in a wind tunnel, which was manufactured in the University of Derby workshop. Validation of the results in such case is necessary because the designs and implementers of publically used components could make errors in operation of the component. These errors will result in term of undetected faults in the impeller blades, numerous errors such as loss of system integrity, electric fire hazard in the motor, and worst of all loss of human lives. The mission of validating and verifying is therefore to find out and correct these errors that causes misinterpretation, as early as possible thus preventing exposure of the faulty components or systems to customers. In order to validate the experimental results, an identical design of3D parts was assembled to duplicate a real physical model of fan extraction system and test it in the wind tunnel. The challenge in this project that the simulation software does not possess a tool to select appropriate material of the impeller during simulation, thus a comparison between the simulation/experimental results will be limited to one scenario. Wood-fill/PLA composite impeller was proposed as an option for domestic fan extraction system.

a) Aim

The goal of this study is to analyse the thrust of different types impellers in a wind tunnel and examine the performance and in particular wood-fill composite impeller. The reuse and recyclability of impellers were considered in the selection of the best bio degradable material, thus have gaining a positive environment impact due to recycling and reuse.

b) Star CCM+ Simulation

The use of STAR-CCM+ is becoming more and more important, as it is one of the most powerful CFD software. Ideally when choosing a CFD software, a compromise had to be made between accessibility and powerful functionality. However, STAR-CCM+ integrates a set of tools that perform design, mesh and analyse simulations without having to compromise the accuracy of the boundary conditions. It was chosen in this study because of its ability to solve complex boundary conditions involving several laws of physics with minimum user effort. The software usage in this study is based solely on its rapid expansion through the ranks of CFD and other mechanical engineering fields.

(Kaewnai, 2009) has an objective of using the CFD technique in analysing and predicting the performance of a radial flow-type impeller of centrifugal pump. Although currently there is limited information available of the physics models used axial flow type

Authors α σ: *Carbon Fibre Research Laboratory, School of Engineering, University of Wolverhampton, Telford Innovation Centre, Telford, United Kingdom TF2 9NT. e-mail: k.bari@wlv.ac.uk*

Global Journal of Researches in Engineering (D) Volume XVII Issue II Version I Year 2017

impeller, there is comprehensive research detail for simulation centrifugal impellers in water flow in pumps.

This information can be used at some point in term of comparison criteria and source of error which may encountered during both experimental and simulation process. In the present study, the main concern is the noise and vibration impact on the multi pressure sensors deployed in our wind tunnel. Most research on impellers using STAR-CCM+ was performed ontide, wave and wind energy generation turbines using open air media (Kumawat, H. (2014).However, the validation of low speed air stream (< 10 m/s) a wind tunnel has never been investigated due to lack of interest.

II. Methodology

This research investigates the thrust of an axial impeller rotated at low rotation speed (RPM=10,000). Most of other sresearches were performed mainly on predicting flow pattern of air in axial impellers at above 350 m/s (Kumawat, 2014)

Impeller: The impeller parts have been manufactured using an Ultimaker2 3D printer via a process of fused filament fabrication (FFF), also known as fuse deposition modelling (FDM) as shown in Figure 1. This process deposits a polymer filament into a heated extrusion nozzle. The nozzle moves to a build plate and constructs the 2D cross section of the 3D model; the cross section moves down and allows the nozzle to print another 2D cross-section of the model on the initial 2D cross-section, using heat as an adhesive. The layers are stapled till the 3D model is completed.

Figure 1: 3D Printer Ultimaker allocated to manufacture five different impellers

Before printing the CAD model of the impellers, it had to be prepared using software called CURA; it is associated with Ultmaker2 (the 3D printer model). The Ultmaker2 does not support files from other software, hence a change of file extension to STL was done in order to open the file in CURA. CURA was used in changing the printing conditions such as; size and shape of components, and Layer height (the movement of the control plate between each layer). Note that the higher the layer height the lower the printing quality, just as pixels in 2D picture printing. There are two other parameters that determine the finishing of the components being printed; fill density and wall thickness. Fill density, is the internal cavity, a fill density of 100% would create a solid part while a fill density of 0% would create a component with hollow cavities. Then the impellers were aligned to find the accurate centroid of the rotation in order to prevent vibration at all rotation speeds as shown in Figure 2.

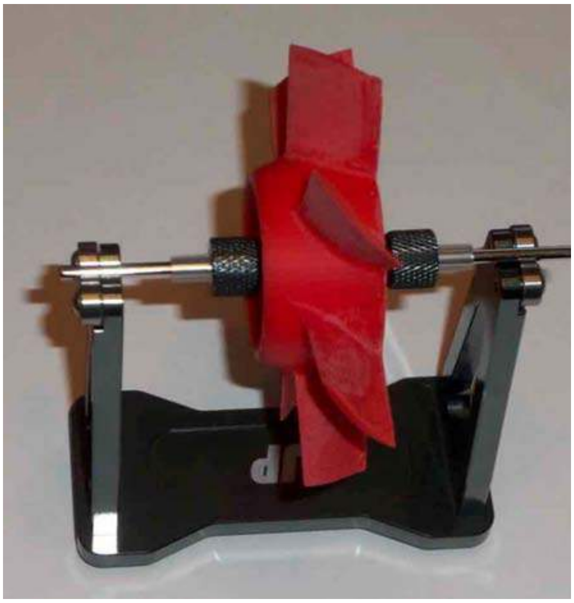


Figure 2: ABS impeller produced by 3 D printing device fixed by alignment shaft

After successfully printing the impeller in ABS and PLA, three other composites were used. Wood-fill (70% PLA, 30% recycled Wood-fibre), Bronze-fill (a composite of PLA and fine bronze particles) and Glass fibre (70% PLA, 30% E-Glass fibre). The same layer height and infill parameters were used for Bronze-fill (0.1mm layer height, 25% infill), however Wood-fill required a layer height of 0.25mm due to its lower printing temperature. All five impellers and the backward stator are shown in Figure 3. The material properties of the impellers are listed in Table 1.

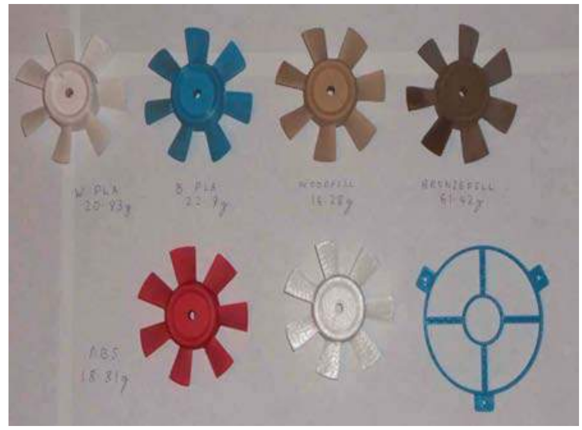


Figure 3: Image shows five Impellers and stator

Table 1: The material properties of different impellers tested in the wind tunnel

Material	Fan Size	Tolerance	Density g/cm^3	Melting Point(°C)	Tg(°C)	Impact Strength KJ/m^2
ABS	90 mm	±0.10mm	1.03	-	105	36
PLA	90 mm	±0.10mm	1.24	150-160	60-65	7.5
Wood-fill/PLA	90 mm	±0.05mm	1.29	*220*	60-65	17.4
Glass fibre fill/PLA	90 mm	±0.05mm	3.1	*n/a*	*n/a*	21.1
Bronze-fill/PLA	90 mm	±0.05mm	3.9	*n/a*	*n/a*	22.5

a) Wind Tunnel

The wind tunnel was manufactured in our workshop using laser cut machine and assembled using expert technicians to fit all components inside. It is a cylindrical geometry tunnel of synthetic smooth PVC plastic in order to reduce intrinsic drag from air streams and supported by ply wood frames to reduce vibration. It was mainly designed to minimise the turbulence created by the rotating impeller. The outlet air streams measured by 0.5 mm^{2}highly sensitive pressure sensor that measures the velocity and mass flow rate. The impeller, fan case and the other components used in the experiment shown in Figure 4.

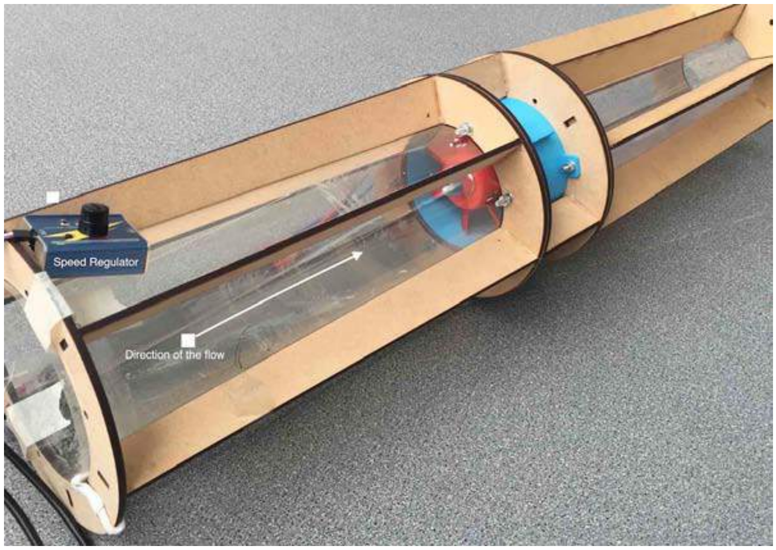


Figure 4: Image shows a Wind tunnel includes impeller and fan case

Speed Regulator:

The device shown in Figure 4 is an electric servo regulator adjusting the impeller speed during the experiment. The fan was power up by 12 DC battery able to run the impeller up to 40 hours, the regulator would switch off the fan in case of motor overheat in the wind tunnel using servo consistency master. The motor rotating the fan is connected to a car battery, to provide stable DC power for long period of time and constant rotation of 10000RPM. that Controlled by a digital Tachomter shown in Figure 5

Figure 5: Image shows the digital Tachometer

Guided Vanes:

This is a non-rotating fan stationed in the tunnel to laminate the level of turbulence in exited at the outlet. This stator acts like a frame fastened to the walls of the container, to make sure all air streams are aligned to horizontal axis in order to prevent turbulence at the outlet end. Figure 6 shows the horizontal view of outlet end of the wind tunnel.

Figure 6: Image shows the Outlet view of the wind tunnel

III. Results

a) Experimentation Case

A mass rate and velocity meter is fixed on the outlet of the tunnel to measure the outlet mass rate and air velocity. By replacing the impeller for each batch process, the thrust can be measured for different impellers using the equation1.

$$Thrust = Velocity\ x\ mass\ flow\ rate \quad (1)$$

The velocity meter is connected to PC. The probe has very high sensitivity and an accuracy of ± 0.3%. With that kind of sensitivity and accuracy, the application of the device on the tunnel would require a level of precision; any sudden vibrations could affect the accuracy of the readings and cause a variation in results as shown in Figure 7.

Figure 7: Image shows Manometer is connected without fixing the ends, which cause vibrations

The averaged outlet velocity, mass flow and thrust measured using the manometer are listed in Table 2.

Table 2: The list of outlet velocity, mass flow and thrust of different impellers tested in the wind tunnel.

Material	Outlet velocity m/s	Outlet mass flow rate Kg/s	Thrust (N)
ABS	12.50	0.041	0.51
PLA	10.30	0.032	0.33
Wood-fill/PLA	11.91	0.039	0.465
Glass fibre fill/PLA	12.94	0.044	0.57
Bronze-fill/PLA	13.96	0.049	0.68

b) Simulation Case

The first step is use of a CAD based software (Solid Works) to draw the structure and topology of complete wind tunnel includes the internal parts. As the impellers were already designed in a CAD based software, a simple transfer of the drawing would merely suffice to Solid Works. The second step is to import the identical model geometry from solid work file (STL format) to star CCM+. The wind tunnel, fan case, motor, cables and impeller were identically represented and were applied to the same boundary conditions used in the experiment case. After that, appropriate physics models (RANS with K-epsilon correction) were applied to specify the behaviour of the turbulence produced as shown in Figure 8.

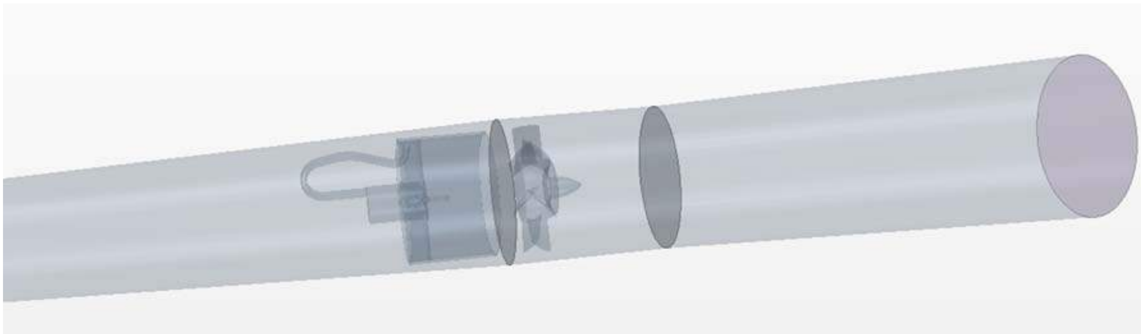


Figure 8: The model configuration shows the exact real case, motor, guide vain, wires, fan and wind tunnel case.

c) Simulation Results

The model meshed using different size of Polyhedral size for the fluid region (1-10 mm) and fixed Prism layer size of 1 mm for surface structure region.

The velocity value-location from the outlet wind tunnel pipe of from different mesh size are shown in Figure 9a, b, c, d, e.

It was found that mesh size between 1-3 mm was appropriate and it would produce repeatable results. The similar outlet velocity profile is evident when comparing the outlet velocity profile in Figure 10 and Figure 11.

The quality of the model mesh was over 99.9%. This was achieved using automated mesh which uses surface and geometry mesh control.

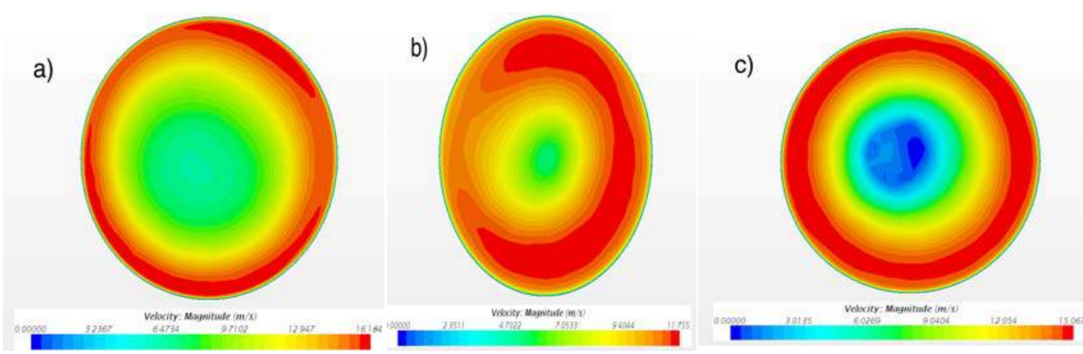
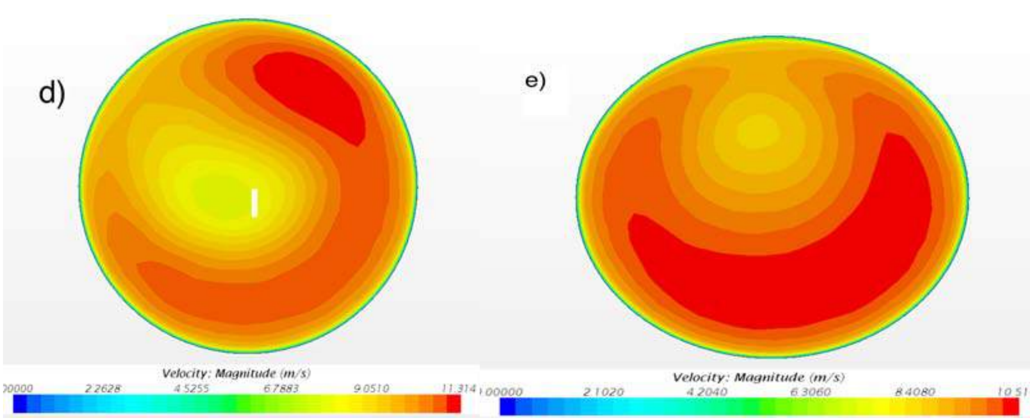


Figure 9: 10 mm mesh 8 mm mesh 6 mm mesh



3 mm mesh 1 mm mesh

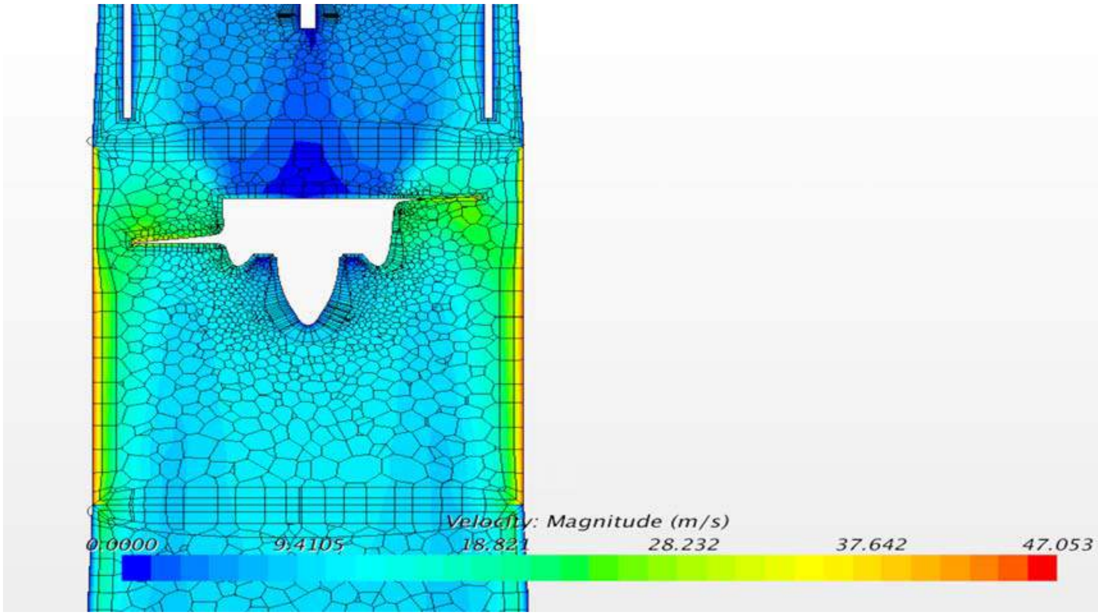


Figure 10

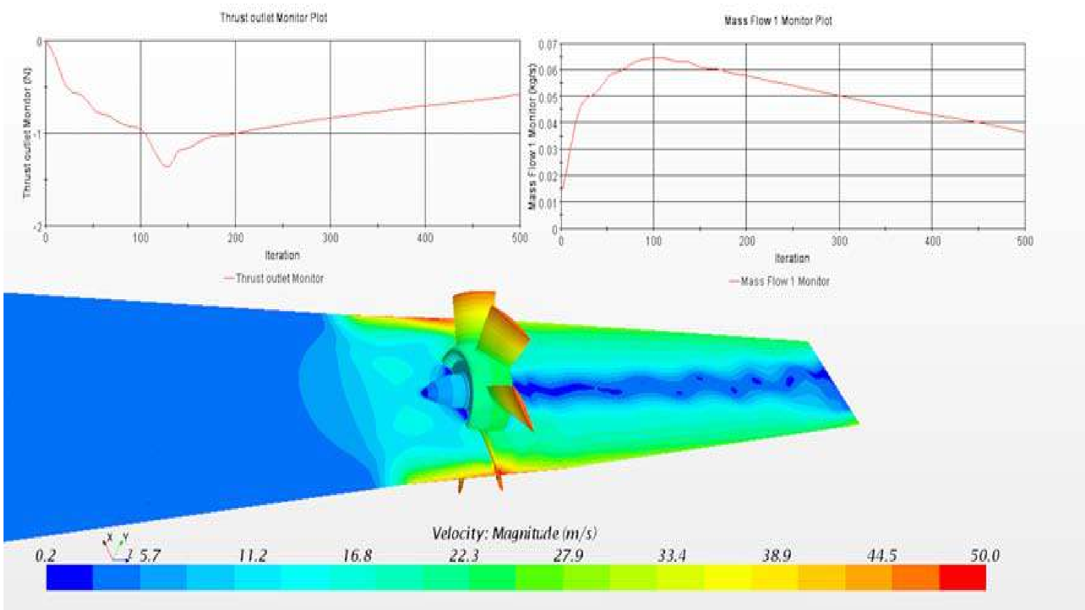


Figure 11: Image shows the final results from the simulation

IV. Experimental Results

Similarly, the outlet mass rate and velocity of different impellers, from the wind tunnel were measured using the bespoke manometer. The averaged of 10 readings were obtained and the final results are summarized in Figure 12.

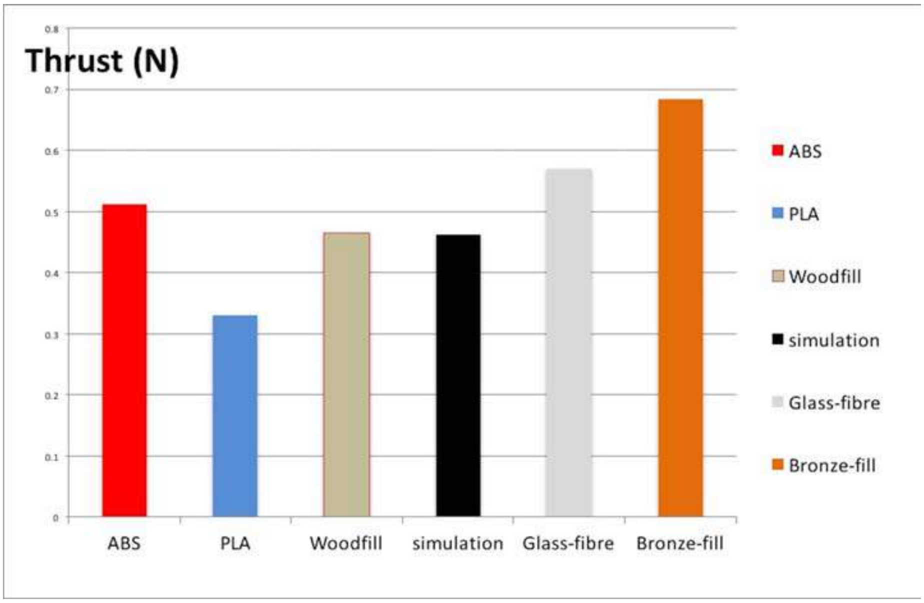


Figure 12: Chart shows a comparison between the experimental and simulation thrust results

V. Discussion

The discussion of the results can be summarised as below:

- Composite impellers like Bronze-fill/PLA and glass Fibre fill/PLA composites showed better performance in term of thrust compared to traditional ABS impeller used in domestic extraction fan system.
- Wood fill/PLA composite as a biodegradable material to shows similar thrust values to simulation results. This makes it attractive to be used in as base line in simulation using Star CCM+.
- A modification was proposed to CD-Adapco to add-on a Rigidity Factor (RF) in the rotation section in the software. This will enable the user to choose an appropriate material factor in the simulation using rotation reference frame.

Plastic Impellers are manufactured using one of our Pre-Tooled Plastic Impeller molds. This method saves molds development time and money by using our existing tooling inventory. Also, the time from placement of order to delivery of actual parts is significantly reduced. The cost unit of wood fill/PLA composite impellers will be reduced to half using the advance air plastic injection molding.

Advanced Air produces plastic injection molded impellers for a wide variety of OEM, commercial and industrial applications. The early results show that the design and manufacture high quality impellers have meet our exact specifications. The use innovative manufacturing facilities to produce plastic impellers and impeller products that give us 100% recyclability and zero impact on the environment, maximum durability and high performance.

In order to detect and monitor crack growth during solidification of the impeller, Acoustic Emission (AE) could be a micro-scale detection tool to monitor crack and porosity of the final product. AE technique also uses relatively simple but effective means of monitoring the time progress development of different failure mechanisms during air injection molding. The data of the emission can be analysed and differentiated to provide instantly alarm for shrinkage rate during solidification of the impeller in the mold (Spasova, 2008).

VI. Conclusion

Biodegradable impellers are promising material to replace the current ABS used in domestic fan extraction system. This will have a significant positive environmental impact by recycling the used impellers. An early result shows that wood-fill/PLA impeller was viable to be manufactured using traditional extrusion process to reduce cost of production. Plastic Mold Injection (PIM) was selected as a cost effective manufacture process in the mass production of wood-fill/ PLA composite impeller.

References Références Referencias

1. Shah R.S., Jainb S.V., PatelbRN ,Lakherab V.J (2013) "CFD for centrifugal pumps: a review of the state-of-the-art" Conference Procedia Engineering 51 (715-720)
2. Kaminsky C., Filush A., Kasprzak P. and Mokhtar W (2012) "CFD study of wind turbine Aerodynamics" Proceedings of ASEE North Central Section Conference, American Society for Engineering Education.
3. Kaewnai, S., Chamaoot, M. and Wongwises, S. (2009) "Predicting performance of radial flow type

impeller of centrifugal pump using CFD" J Mech Sci Technol, 23(6), pp.1, 620-1627.
4. Kumawat, H. (2014) "Modeling and Simulation of Axial Fan Using CFD" World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:8, No:11.
5. Amano R., Lee E., Xu C., Xie J. (2005) "Investigation of the Unsteady Flow Generated by an Axial Fan Experimental Testing and Simulations" International Journal of Rotating Machinery 3, 256–263, 2.
6. Spasova L.M, Ojovan M.I. (2008) "Characterisation of Al corrosion and its impact on the mechanical performance of composite cement wasteformsby the acoustic emission technique" Journal of Nuclear Materials 375 347–358.

This page is intentionally left blank