Journal of Current Research in Engineering and Science
Bi-Annual Online Journal (ISSN : 2581 - 611X)

Volume 5- Issue 2, Paper 31 August 2022

Analysis of Air Flow through cooling wheel using Computational Fluid Dynamics

Kunal Thakkar
Department of Mechanical Engineering
Vishwakarma Institute of Information technology
Pune, India
kunal.17u573@viit.ac.in

Navam Shah
Department of Mechanical Engineering
Vishwakarma Institute Of Information Technology
Pune, India
navam.17u633@viit.ac.in

Aniket Mujumdar
Department of Mechanical Engineering
Vishwakarma Institute of Information technology
Pune, Maharashtra
aniket.17u628@viit.ac.in

Shubham Gite
Department of Mechanical Engineering
Vishwakarma Institute of Information technology
Pune, Maharashtra
shubham.17u004@viit.ac.in

Dattatray Nalawade
Department of Mechanical Engineering
Vishwakarma Institute of Information technology
Pune, Maharashtra
dattatraya.nalawade@viit.ac.in

Sandeep Kore
Department of Mechanical Engineering
Vishwakarma Institute of Information technology
Pune, Maharashtra
sandeep.kore@viit.ac.in

***Abstract*— This paper presents the CFD analysis of a cooling wheel, designed to have blades inside so as to have a secondary function to create a forced conventional flow. This design is intended for used in reciprocating compressors as a compact and effective way of cooling the compressor. Experimental analysis using a prototype of the cooling wheel and a duct setup according to Air Movement and Control Association (AMCA) standards, an international body is considered as the base for CFD analysis. CFD analysis is done to get an estimate of parameters like total velocity, pressure and mass flow. The goal of this paper is to finalize the most optimum design and operating parameters for the cooling wheel for its effective utilization as a secondary cooling system as well as energy efficiency.**

Keywords — Cooling wheel, CFD, AMCA, forced conventional flow, contours

I. Introduction

The paper aims to compile and analyze various airflow parameters and patterns of the compressor cooling wheel using CFD analysis. The basic components of the reciprocating air compressor are the electric motor, pump, intercooler and receiver. The receivers can be vertical or horizontal and have different sizes and capacities. An electric motor powers the compressor pump. The motor then drives a pulley via a belt, which transmits power from motors to the compressor through a cooling wheel. A cooling wheel is a heavy wheel attached to a rotating shaft used to balance the forces from the crankshaft and to smooth out the delivery of power from a motor to the compressor. The inertia of the cooling wheel moderates fluctuations in the speed of the engine and stores the excess energy for intermittent use. To oppose speed fluctuations effectively, a cooling wheel is given high rotational inertia.

Compressor cooling wheels come in a variety of shapes and sizes. Often the vanes on the cooling wheel are oriented to move air over the pump to cool it. Thus it is necessary to analyze the performance of the cooling wheel. During the experimentation, we aim to analyze the flow rate generated and pressure difference due to rotation of the cooling wheel during the working of the compressor. The cooling wheel is powered by a three-phase induction motor through a belt drive. At the same time air flows through the duct passing through the straightener and various parameters like temperature, pressure, and velocity are sensed by sensors mounted inside the duct at specified locations which will be used as raw data as input for the CFD simulation. All design and selection procedures used are according to the AMCA standards. The results from the experimental analysis will be helpful to increase the efficiency of the compressor by analyzing the airflow parameters of the cooling wheel.

The experimental results are used to assign boundary conditions of the CFD solver. The computational domain is developed in the CAD model using experimental set up and mesh is generated in the Ansys Fluent. The various inputs gathered with the help of experimental data are then used as input for the boundary conditions for the simulation. The cooling wheel analysis is done to measure the accuracy of the model being used to simulate the flow and also to measure the flow at various points during the flow.

II. Literature Review

Nalawade, Divekar, Chandgude, Chandak& Bhagwat [1] presents an experimental study of an empirical approach in which a wind tunnel apparatus is used to improve the efficiency of power output by a small scale wind turbine. A custom-designed wind tunnel attachment was constructed to test wind turbines with three and five blades to record, analyze, and interpret both incoming and outgoing wind velocity readings. An investigation was conducted toevaluate the relationship between wind velocity outputs using a custom-designed wind tunnel attachment (WTA) with three & five blades wind turbines. Based on the statistical analyses the results showed that with increase in wind speed, generator output increases The experimental study proves that five blades wind turbine is more efficient than three blades wind turbine & it has been observed that wind turbine works more efficiently at the end of the duct as compared to the middle of the duct.

Dakeev [2] studied a new approach in which a wind tunnel apparatus is used to identify the efficiency of power output by a wind turbine with a 400W rating. Moreover, the study addresses significant issues concerning the turbulence formed by a natural wind which can be eliminated or reduced with the use of the proposed wind tunnel. By performing a number of case studies, wind power characteristics were studied which includes power output versus wind velocity. The case studies include normal operation of the experimental wind turbine at variable wind velocity values with and without proposed wind tunnel. A certain level of turbulence is formed and the wind turbine power output is measured and recorded.

Ulan Dakeev, Connie Lam, James Pung [3] further develops another approach using a wind tunnel apparatus to improve the efficiency of power output by a small-scale wind turbine. The statistical t-test and one-way ANOVA analyses resulted in a 60% increase in wind power output with the use of the custom-constructed design.

Soren Madsen, Kim Bertelsen&BóasEiríksson [4] presents an experimental study of lightning verification tests for wind turbines. Based on extensive use of the test methods, the paper addresses both high voltage testing: strike attachment tests and swept stroke tests and the high current tests in terms of the conducted current test, the arc entry test and the charge injection test.

Suryawanshi &Bhaskar[5]presents an experimental investigation on the Dual Mass Flywheel (DMF) which is primarily used for damping of oscillations in automotive powertrains and to prevent gearbox rattling. They explained a detailed initial model of the DMF dynamics is presented. This mainly includes the two arc springs and two masses in the DMF and their behaviour. An experimental the DMF model is compared to conventional flywheel. Finally the observation of the engine torque using the DMF is discussed. For this purpose the DMF is manufactured and done experiment or testing to see the results. And then results are compared with the conventional flywheel.

Ulan Dakeev, Connie Lam, James Pung [3] further develops another approach using a wind tunnel apparatus to improve the efficiency of power output by a small-scale wind turbine. The statistical t-test and one-way ANOVA analyses resulted in a 60% increase in wind power output with the use of the custom-constructed design.

Soren Madsen, Kim Bertelsen&BóasEiríksson [4] presents an experimental study of lightning verification tests for wind turbines. Based on extensive use of the test methods, the paper addresses both high voltage testing: strike attachment tests andsweptstroketestsandthehighcurrent tests in terms of the conducted current test, the arc entry test and the charge injection test.

Suryawanshi &Bhaskar[5] presents an experimental investigation on the Dual Mass Flywheel (DMF) which is primarily used for damping of oscillations in automotive powertrains and to prevent gearbox rattling. They explained a detailed initial model of the DMF dynamics is presented. This mainly includes the two arc springs and two masses in the DMF and their behaviour. An experimental the DMF model is compared to conventional

Journal of Current Research in Engineering and Science
Bi-Annual Online Journal (ISSN : 2581 - 611X)

Volume 5- Issue 2, Paper 31 August 2022

flywheel. Finally the observation of the engine torque using the DMF is discussed. For this purpose the DMF is manufactured and done experiment or testing to see the results. And then results are compared with the conventional flywheel.

III. Procedure

A. Setup Introduction

The setup used for the testing and measurements of the cooling wheel airflow parameters is designed with the help of the AMCA standards. This Setup is a normal long cylindrical duct of diameter 250 mm and length of 2500mm and also a specific Converging section which is designed with the diameter of each cooling wheel taken into consideration. This exact setup is made in the CAD model and used for simulation as a wall for the flow.

B. Geometry Editing

The Geometry can be easily imported from different CAD modelling software into Solidworks, but it needs some editing to be used for the simulation. The open space between the vanes of the cooling wheel is not considered as the flow domain by the software so an enclosure needs to be made to cover the cooling wheel but it also should not intersect with the model itself or there will be issues with the meshing. To do this we use the extrude command available in the Solidworks. In Solidworks, while using extrude, we deselect the merge results option in extrude command.

C. Meshing

The model needs to be meshed to select how the calculations are made for the flow. The mesh is made using tetrahedral shape and using an element size of 8mm. An element is the fundamental part of the model on which the computer will run the calculation using either the initial input parameters or the previous cell calculations.

D. Simulation Setup

For the Simulation setup, we need to first select the material used in the flow. The fluid used is air and the solid used for the wall and cooling wheel is mild steel. This all is selected in the Add-ins option under “Flow Simulation” under “New wizard study” in Solidworks. Then according to the model, we chose an “External type” simulation. After that, the Computational Domain, which is by default 1m^3 cube, is reduced to fit in the cad geometry and reduce computational time. After that we define the rotating domain and give the rpm. Expectations, which in Solidworks are goals, are defined and simulation is then solved. We then check the pressure readings at four points, which is 500 mm away from the end of the outlet in inward direction.

IV. Using the Template

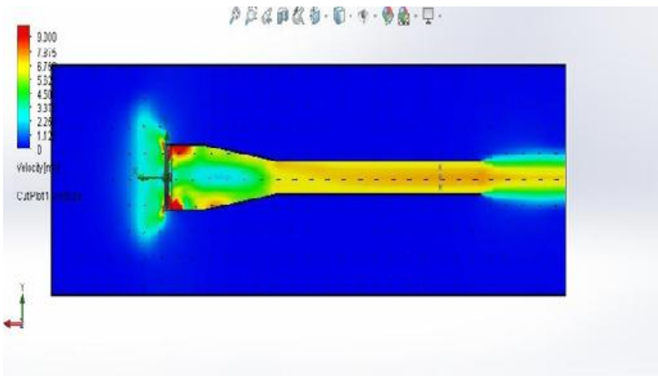

Fig. 1. Velocity contours for the computational domain

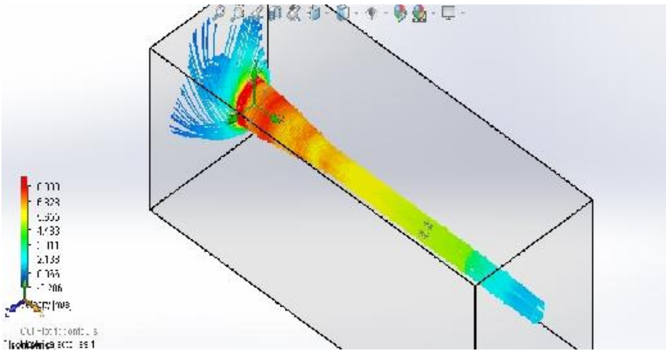

Fig. 2. Streamlines of velocity vectors

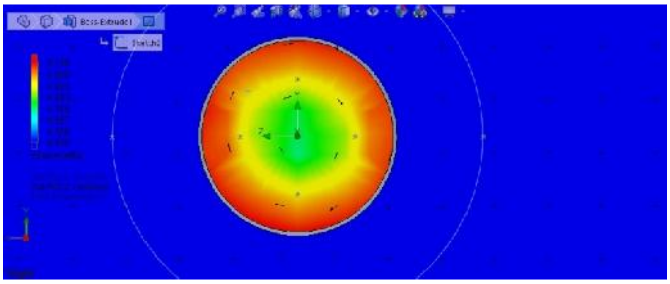

Fig. 3. Pressure Contours across the computational domain

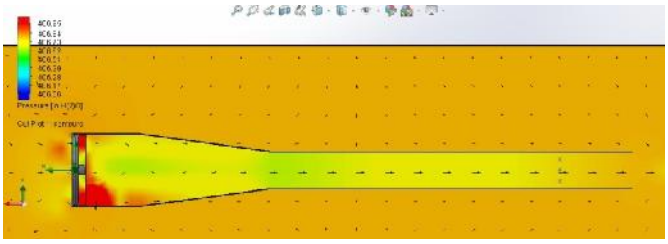

Fig. 4. Pressure Contours across the computational domain

Journal of Current Research in Engineering and Science
Bi-Annual Online Journal (ISSN : 2581 - 611X)

Volume 5- Issue 2, Paper 31 August 2022

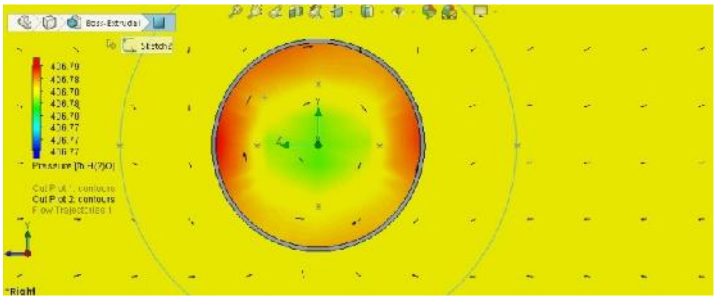

Fig. 5. Pressure Contours Front view

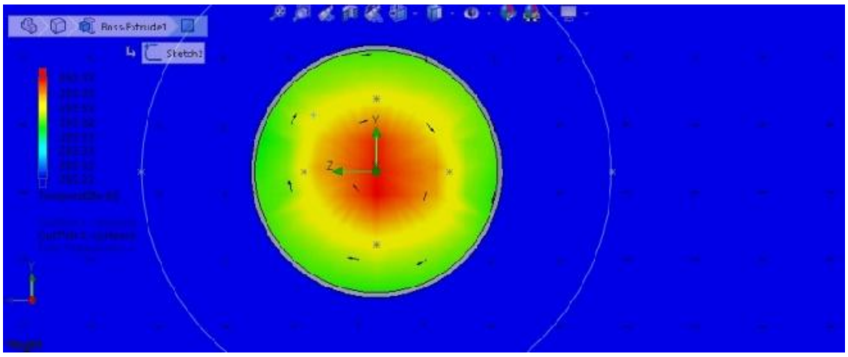

Fig. 6. Temperature Contours Front View

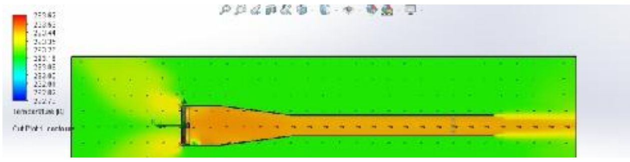

Fig. 7. Temperature Contours for the computational domain

V. Collected Data Analysis

For the Pressure Contours, the pressure seems to be maximum at the start of the cooling wheel with a value of 0.35 inch of H_2O which then decreases as we move further into the duct and close to the end where we have a pressure of 0.19 inch of H_2O which is similar to the raw data that shows the value to be 0.15 inch of H_2O. The rate of pressure decrease in the long straight seems to be less compared to the part in the converging part. The pressure measurement from the CFD analysis also is very close to the actual reading when taking into consideration the accuracy of the device used for the measurement purposes.

For the Velocity Contours, the velocity is less at the wall and higher in the middle section as seen in the YZ Plane. The velocity also decreases as we move further into the converging part of the duct. The initial velocity at

Journal of Current Research in Engineering and Science
Bi-Annual Online Journal (ISSN : 2581 - 611X)

Volume 5- Issue 2, Paper 31 August 2022

the cooling wheel has a value of close to 8 m/s, while the velocity at the end of the converging part is around 6.05 m/s which continues towards the end with a value of 2.35 m/s which is similar to the given boundary condition as expected. The velocity from the raw experimental data is calculated using the dynamic pressure reading and also cross verified using an anemometer.

The Temperature Contours show a lot of fluctuations throughout the cross-section, but as the scale used is very small there is a negligible amount of change in temperature in reality. The value used for temperature for the simulation is the default value which coincides with the temperature at the time when raw data was collected, having a value of 293 K.

VI. Results and Discussions

A. *Analytical Results*

The Results that have been collected from the simulation show that the airflow parameters measured change with a different cooling wheel at different RPMs, but the flow pattern seems to remain the same. This could be due to the low accuracy of the model used or lack of user-defined transport equations. The pattern might also be similar since the flow is not in a turbulent state but actually in the transient state which isconcluded from Reynolds's Number for the flow near the end of the duct.

B. *Use of the Data Collected*

The numerical and experimental data files can be used by wind tunnel designers, engineers and researchers when validating the prediction of their closed-loop wind tunnel models (theoretical, computational methods, etc.) with an empty test section. The numerical and experimental data can be used to test different turbulence models, boundary conditions, mesh design, discretization scheme, steady-state and transient simulations, etc. The value of the data is its use in improving the comparability between the results of other researcher's models by providing a common benchmark. The data can be used for training CFD users and contribute to an overall improvement of the prediction accuracy of CFD modelling of wind tunnels. The data can be used to explore different optimization of the full closed-loop wind tunnel design and also its separate components.

Conclusions

This method of simulating the flow to compare against real-world data seems to be accurate to a certain degree. This method can be useful to know the different parameters at different locations throughout the setup. The accuracy of the model is satisfactory for comparison purposes, but the model is not accurate enough to be used for the verification purpose of experimental findings. The data collected from the simulation is useful to get a range between which the expected parameter should be, rather than give the absolute true value of the parameter. For the scope of the project, the accuracy is more than enough and satisfies the conditions that were required from it.

References

[1] D.B. Nalawade , R.S.Divekar , N. R. Chandgude , P.D.Chandak , P.B.Bhagwat“An experimental study of wind power generation with wind tunnel attachment” International Journal of Research in Advent Technology, ISSN: 2321-9637(Online)

[2] Ulan Dakeev, “Management of Wind Power Generation with the attachment of wind Tunnel” IBSU Scientific Journal, 5(2011): 71-82.

[3] Ulan Dakeev, Connie Lam, James Pung “Analysis of wind-power generation with windguide attachment” Proceedings of the 2014 IAJCISAM International Conference ISBN 978-1-60643- 379-9

[4] Soren Madsen, Kim Bertelsen&BóasEiríksson, “An experimental study of lightning verification tests for wind turbines' GlobalLightning Protection Services A/S HI Park 445, 7400 Herning, Denmark.

[5] Suryawanshi, N.N., Bhaskar, D.P. and Kopargaon, S.R.E.S., 2015. An Experimental Study of Dual Mass Flywheel on Conventional Flywheel on Two stroke petrol engine. International Journal of Engineering Research and General Science (Part2).