



Numerical Calculation of Aerodynamic Shape of an Axial Turbine Blade

Norizzuddin Mat Daud¹, Abdulhafid M A Elfaghi^{1,*}, Alhadi A Abosbaia²

¹ Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

² Faculty of Engineering,
Universiti of Zawia, LIBYA

*Corresponding Author

Received 2 January 2023;
Accepted 14 February 2023;
Available online 1 March 2023

Abstract: Aerodynamic characteristics, structural integrity, vibration elements, combustion, cooling, acoustics, and environmental impact are all active research and development topics in gas turbine design. On the other hand, turbine aerodynamics is a challenge due to high inlet gas temperature, secondary flow, separation, and transonic flow, as well as their interactions. Numerical methods and rising computational power have aided turbomachinery design in recent decades. With their increasing ability to predict flow features, CFD tools have become integral to the design process. Initially, the project was about calculating or measuring the efficiency of the blade by using the numerical method or CFD simulation. Based on the step of the CFD modelling, which includes the geometry, solver, and post processes. One of the most crucial parts was the designing phase which determined the obvious result to export to the geometry that needed to be in the perfect model to be mesh. At the end of this study, the efficiency calculation was determined based on the result achieved and the following steps, such as the pressure drop. The high-pressure drop may cause a cavity in the finished model, and better get the lower pressure drop to show a good blade profile performance.

Keywords: CFD Aerodynamic, CFD Modelling, Numerical methods, Small Scale Turbine

1. Introduction

Aerodynamic characteristics, structural integrity, vibrational elements, combustion, cooling, acoustics, and environmental impact are all active research and development topics in gas turbine design [1,2]. Compressor and turbine aerodynamics in gas turbines are two types of flow functions. Because of the stability, compressor aerodynamic design is difficult, particularly when maximum efficiency is within the surge margin [3-5].

Turbine aerodynamics, on the other hand, is a challenge due to high inlet gas temperature, secondary flow, separation, and transonic flow, as well as their interactions. Numerical methods and rising computational power have aided turbo machinery design in recent decades [6-8]. With their increasing ability to predict flow features, CFD tools have become an integral part of the design process. It is both economically and scientifically justified to use them in architecture [9,10]. However, experimental validation is still needed.

These methods have greatly reduced the size of the design framework and the number of experimental cases needed to validate a given design. With the rise in fuel prices and tighter environmental standards, the push for increased performance, improved reliability, reduced noise, and lower costs of gas turbines, especially in the aerospace industry, is stronger than ever [11]. Calculation approaches involving CFD flow simulations are a realistic technique for achieving these objectives.

Turbine blade construction has progressed from two-dimensional to quasi-three-dimensional to three-dimensional. Aerodynamic calculation necessitates a functional parameterize that allows for the regulation of flow characteristics. Its functional development necessitates prior experience and is a critical component of performance enhancement. After that, an analysis of the literature is conducted with an emphasis on parameterize and calculation methods.

2. Method and Numerical Modelling

The design of an axial turbine was constructed by using SolidWork software. Specifications and properties of materials are set in Solidwork. The model of axial turbine simulated are shown in the following figure 1.

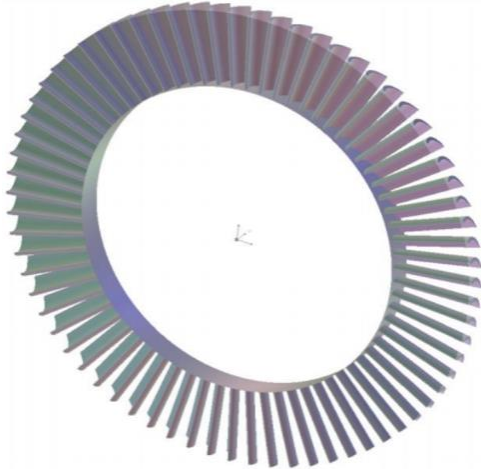


Fig. 1 – Designed of the axial turbine.

2.1 Axial turbine

The introduction of the gas-turbine engine, which is completely reliant on sufficient justification efficiencies for cost-effective operation, and the need for novel materials that can endure severe strains at temperatures even higher than those seen in gas turbines cause engineers are now thoroughly scrutinizing the design of turbines both from an aerodynamic and a mechanically aspect [12-14]. Axial flow turbines have blades on one end of their rotors, which are mounted on a vertical beam attached to the seabed. The turbine begins to spin and generate electrical energy as the current flow spins the blades.

When the sweep area of the blades confronts the direction of current flow, the energy extraction from an axial flow turbine can be maximized consequently, in some designs of axial flow turbines, the rotor can yaw according to the direction of current flow [15]. In many cases, however, rotors that do not yaw with the current flow are nonetheless used to simplify the design.

2.2 CFD turbine modelling

All the major components of gas turbines can be investigated using CFD. The aerodynamic element of the compressor is the most essential since it seeks great efficiency in converting available mechanical work into total air pressure. CFD is a sophisticated tool that may be used to investigate the complicated processes that occur in a gas turbine. Also, the temperature of the turbine blade was higher than the blade material's melting point in the gas turbine engine. The requirement for CFD modeling is to set up governing equations; the three basic governing equations are mass, momentum, and energy conservation equations.

Numerical approaches can also be utilized for various design parameter evaluation and flow analysis tasks. However, these techniques are usually not able to provide complete information about a design space. Design Calculation and inverse design are two commonly used strategies for achieving blade performance. The goal is to find the optimal geometry and design for the given system.

3. Results and Discussion

The result obtained from this study came from the CFD or precisely Ansys software which will be given eventually as an answer from the analysis of data. There will be the data of Turbogrid which is used for meshing over fluid domain to make a simpler way to mesh. Moreover, there will also select material to capture drag fraction at the end of the blade. Also from result, the torque and shaft power were obtained directly from the data analysis. All of this include the rotor and stator involvement in aerodynamic efficiency. There is also the data of pressure drop as the result from single stage turbine calculation.

Designing phase was a crucial part in producing the expected result or defining the best product to improve the efficiency. Blade Generation was involved in creating the design of the turbine blade since it was on CFD software. Solid Modelling also can be used to design the specific part and need to export into the software. Figure 1 shows the velocity profile occurred through the turbine blade.

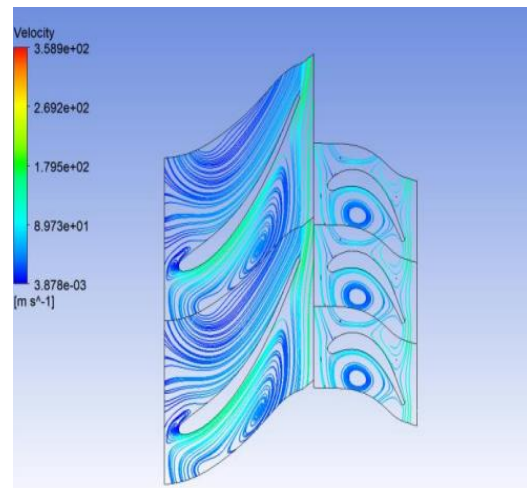


Fig. 2 – Velocity profile across the turbine blade.

Table 1 and 2 below shows the result from liquid turbine performance report which contains a lot of data such as user input and mass average. The torque and power output can be taken directly from this result. The power produced by this turbine is $2782.09 \text{ kgm}^2\text{s}^{-3}$. Another design which is $1076.52 \text{ kgm}^2\text{s}^{-3}$. For this turbine, the turbine will run at low pressure as for mass flow inlet and static pressure outlet.

Table 1 – Performance results for design 1

Torque	531.34 kgm ² s ⁻³
Power	278209 kgm ² s ⁻³
Unit discharge	0.197837 s ⁻¹
Efficiency from power	-283.624
Total head	1.34713 m

Table 2 – Performance results for design 2

Torque	2.05601 kgm ² s ⁻³
Power	1076.52 kgm ² s ⁻³
Unit discharge	0.2 s ⁻¹
Efficiency from power	-1.09667
Total head	-0.0353 m

3.1 Pressure drops over single stage turbine.

The pressure decreases across the turbine because of the turbine obtaining energy from the flow. The flow is kept linked to the turbine blades thanks to this pressure gradient. Therefore, the pressure drop across a single turbine stage may be significantly greater than the pressure rises across an adjacent compressor stage. The efficiency of the turbine is determined by a sizable pressure drop, which contributes to the creation of the most power. The turbine's aerodynamic design and proper operation are determined by the pressure drop inside the turbine and power generation.

Table 3 – Pressure drops for single stage turbine.

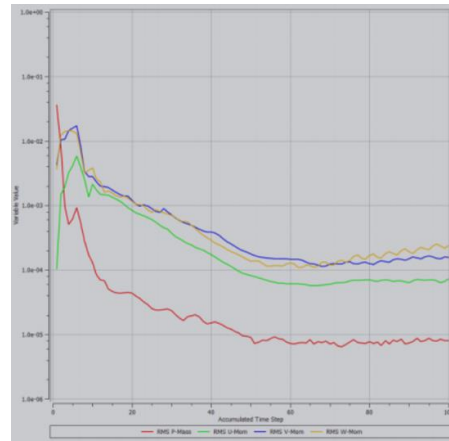
	Design 1 (Pa)	Design 2 (Pa)
Average pressure at inlet	-2609.66	3271.61
Average pressure at outlet	53.9754	1991.59
Pressure drops	2663.63	1280.02

Figure 3 shows the residual calculates the local imbalance of a conserved variable in each control volume in a CFD analysis. The residual directly quantifies the inaccuracy in the system of equations' solution, making it one of the most fundamental indicators of how well an iterative solution has converged. The residual calculates the local imbalance of a conserved variable in each control volume in a CFD analysis. The residual in an iterative numerical solution will never be precisely zero.

However, the answer is more numerically precise the lower the residual value is. A solution's stability can also be inferred from the residual plot. The presence of abrupt spikes indicates that the solution ran into difficulties. Such simulations do not produce trustworthy results.

References

- [1] Li, Y.G., "Performance-analysis-based gas turbine diagnostics: a review." *Proc Instn Mech Engrs*, 216 Part A: J Power and Energy IMechE 2002.
- [2] Douvi, E.C., and Margaritis, D.P., "Aerodynamic characteristics of s809 vs. naca 0012 airfoil for wind turbine applications," *5th International Conference from Scientific Computing to Computational Engineering*, Athens, 4-7 July 2012
- [3] Singh, P., and Shukla, O.P., "Heat transfer analysis of gas turbine rotor blade cooling through staggered holes using CFD," *Int. J. Eng. Res.*, 4(7) (2015): 538-545.
- [4] Lapworth, L., and Shahpar, S., "Design of gas turbine engines using CFD," *European Congress on Computational Methods in Applied Sciences and Engineering*, 24-28 July 2004
- [5] Misirlis, D., Vlahostergios, Z., Salpingidou, C., Flouros, M., Donus, F., and Yakinthos, K., "CFD analysis of the heat transfer and fluid flow around a low-pressure turbine blade," *Chem. Eng. Trans.*, 81 (2020): 583-588.
- [6] Zhang, Y., Han, Z.H., Shi, L.X., and Song, W.P., "Multi-round surrogate-based optimization for benchmark aerodynamic design problems," *54th AIAA Aerosp. Sci. Meet.*, (2016): 1-17

**Fig. 2 – Velocity profile across the turbine blade.**

4. Conclusion

The computational study of this simulation consumes a very low cost and time based on the software process. It could solve very complex mesh or model just in time compare the manual calculation which might brought to any fatal. Blade angle became one of the most important factors in producing the most efficient turbine blade. As for stator and rotor, the result was happened quiet fast which indicated the power, torque, velocity and else. The after effect itself could show the profile of any good or bad blade's profile.

Comparison between models could not be strongly agreeable because it had been created only based on parameters. Real life situation is needed to produce highly efficient blade to meet the most demanding source of aerodynamic turbine blade. Also, a very deep study needs to be pursued as the Ansys software itself contain a lot of applications that could help in every way and every model could produce. Need to improve on designing because it became the source of the mesh to complete meshing.

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, and Flow Analysis, Simulation and Turbulence Research Group (FASTREG) for its support

- [7] Agromayor, R., and Nord, L.O., "Preliminary design and optimization of axial turbines accounting for diffuser performance," *Int. J. Turbomachinery, Propuls. Power*, 4(32) (2019), <https://doi.org/10.3390/ijtpp403003>
- [8] Manjunath L.N., Karanth, K.V., and Madhwesh N., "A critical review on the application of computational fluid dynamics in centrifugal turbomachines." *J. Phys.: Conf. Ser.* 1473 012012
- [9] Pinto, R.N., Afzal, A., D'Souza, L.V. *et al.* "Computational fluid dynamics in turbomachinery: a review of state of the art." *Arch Computat Methods Eng*, 24 (2017): 467–479. <https://doi.org/10.1007/s11831-016-9175-2>
- [10] Morgese, G., Torresi, M., Fortunato, B., and Camporee, S.M. "Optimized aerodynamic design of axial turbines for waste energy recovery," *Energy Procedia*, 82 (2015): 194-200.
- [11] Ainley, D. G. and Garry Mathieson. "A method of performance estimation for axial-flow turbines." (1951).
- [12] Cao, H., "Aerodynamics analysis of small horizontal axis wind turbine blades by using 2D and 3D CFD modelling," *Master Thesis*, 2011.
- [13] Elqussas, N., Elzahaby, A.M., Khalil, M.K., and Elshabka, A.M., "Axial flow turbine aerodynamic shape design optimization." *International Research Journal of Engineering and Technology (IRJET)*, 7(10) (2020): 1092–1101.
- [14] Moustapha, S., Kacker, S. and Tremblay, B., "An improved incidence losses prediction method for turbine airfoils." *Journal of Turbomachinery*, 112(2) (1990): 267-276.
- [15] Massardo, A., and Satta, A., "Axial flow compressor design optimization: Part I-Pitchline analysis and multivariable objective function influence." *Journal of Turbomachinery*, 112(3) (1990): 399-40