

# Design and Flow Visualization of Turbo Fan for Micro Gas Turbine

Shahirah Abduls<sup>1</sup>, Hamidon Salleh<sup>1,\*</sup>, Azwan Sapit<sup>1</sup>, Muneer Khalifa Alsayed<sup>2</sup>

<sup>1</sup> Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering  
Universiti Tun Hussein Onn Malaysia, MALAYSIA

<sup>2</sup> School of Applied Sciences and Engineering  
Libyan Academy for Postgraduate Studies, LIBYA

\*Corresponding Author

Received 15 August 2023;  
Accepted 13 Sept 2023;  
Available online 1 Nov. 2023

**Abstract:** The research presents the design and simulation of the flow of a turbofan in a micro gas turbine, with a specific focus on analysing the resulting thrust using Computational Fluid Dynamics (CFD) simulations in ANSYS. The objective of this research is to enhance the understanding of the flow characteristics and performance factors of turbofans at the micro scale, and to explore the impact of design parameters on the generated thrust. The study begins with a comprehensive literature review of micro gas turbines, highlighting the existing knowledge gap in understanding the flow behaviour of turbofans in micro gas turbines. The design process involves the fluid flow geometry, inlet conditions, and outlet conditions to achieve optimal performance. CFD simulations are conducted using ANSYS to analyse the flow behaviour, pressure distribution, and velocity profiles within the turbofan. The thrust generated by the turbofan is evaluated under various operating conditions, providing insights into the relationship between design parameters and thrust performance. The results highlight the importance of design, inlet conditions, and outlet conditions in optimizing thrust production. The lowest thrust produced is 152.341 N (design 1) and the highest thrust produced is 36711.71 N. Overall, this thesis provides valuable insights into the design and simulation of the flow of a turbofan in a micro gas turbine, specifically focusing on the resulting thrust using CFD simulations in ANSYS.

**Keywords:** CFD, Micro Gas Turbine, Thrust measurement

## 1. Introduction

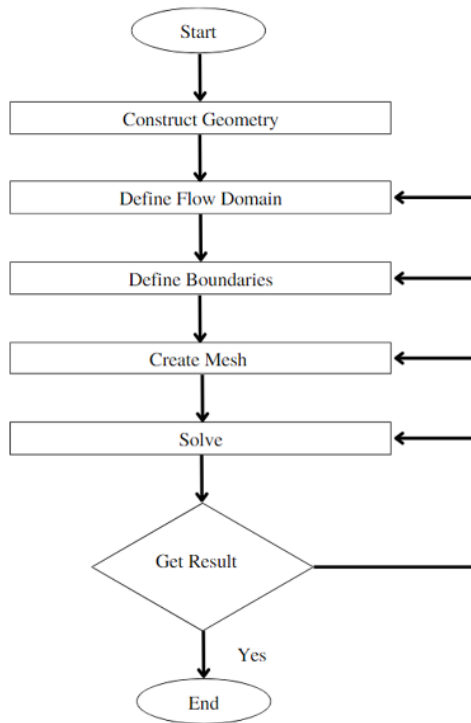
As the gas turbine industry advances, turbine concepts become increasingly efficient and powerful. However there exists a need to improve the performance of smaller engines, such as the micro gas turbines found in UAV engines. These power plants still have room for notable efficiency improvements, which in UAVs would lead to extended mission duration and range. Concepts exist to improve the power plant performance by converting existing micro-turbojet engines to turbofan engines [1-5]. Compact engine intakes must effectively disperse high-speed flows down to acceptable speeds for the remainder of the engine for these approaches to be practical. Other aerodynamic requirements for the engine, in addition to flow diffusion, are maximum total pressure recovery, flow homogeneity, and minimal boundary layer growth and separation.

A reduced weight intake improves overall engine performance by boosting the specific thrust and specific fuel consumption of the engine [6-8]. Turbofan engines are air breathing jet engines with fan(s) located in front of compressors. In addition to the jet engines, turbofan engines have another turbine for rotating fans. This configuration is named as two-spool engine. Spool is used for defining the shafts that connect fan and its turbine, and compressor and its turbine [9,10]. The aim is to examine impact of various operating conditions on the flow characteristics and resulting the thrust. The objective is to propose improvements to enhance the performance of the turbofan system with respect to thrust production.

## 2. Design Process and Development

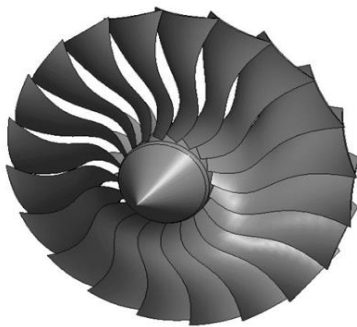
There are three main stages involved in CFD, which are pre-processing, solver, and post-processing [11-13].

The geometry of the Turbo fan can be constructed using SOLIDWORK software and then loaded into CFD during the pre-processing stages, which also entail the geometries and meshing analysis of the model [14,15]. Next, the geometry of the turbo fan is meshed, and the flow structure is verified. In post-processing stage, the solution solving using solver is employed to solve the governing equation and visualization of the data and result. The procedure and steps of the simulation process are shown in the form of flowchart in figure 1.

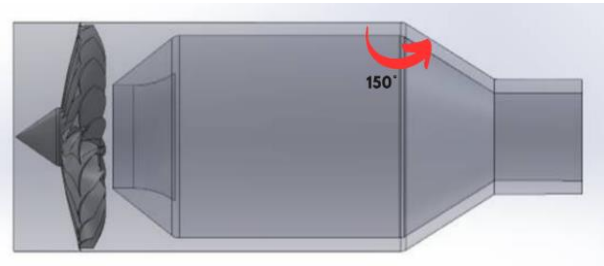


**Fig. 1 – Modelling Flowchart**

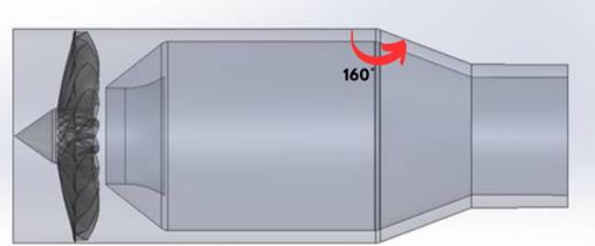
The 3D model of a turbofan and its associated fluid domain was created using SolidWorks software. The turbofan includes 20 fan blades and has a hub diameter of 20 mm (figure 2). Two designs were modelled with the outer diameter of the turbofan measures 150 mm and 160 mm respectively, as in figure 3. The fluid domain represents the primary flow, which is the airflow of the bypass ratio in a micro gas turbine.



**Fig. 2 – 20 turbine blade design**



**Design 1**



**Design 2**

**Fig. 3 – Fluid domain of the primary**

## 2.1 Computational setup

In ANSYS Fluent, the process of geometry setup and meshing involves several steps to prepare the computational domain for CFD simulations. Firstly, the geometry of the system or object is imported into the software or created using the built-in modelling tools. Next, boundary conditions are defined, specifying the behaviour of the fluid at various boundaries, such as inlet and outlet conditions and wall properties. Once the boundary conditions are set, the geometry is meshed, where a grid or mesh is generated over the computational domain.

Meshing can be done using different techniques such as structured, unstructured, or hybrid meshes. The mesh quality is assessed and adjusted if necessary to ensure accuracy and convergence of the simulation. Edge sizing has been applied in areas where higher mesh resolution is desired, such as near boundary layers and regions of flow separation. After mesh generation, the geometry setup is complete, and the computational domain is ready for simulation in ANSYS Fluent. The setup part of the ANSYS Fluent, the setting was set to base on our targeted simulation. The two-design examined with 3 variations of rotational speed which are 20000 rpm, 40000 rpm and 60000 rpm.

**Table 1 – Meshing Nodes and elements**

| Design   | Statistics | Values    |
|----------|------------|-----------|
| Design 1 | Nodes      | 408 382   |
|          | Elements   | 2 439 735 |
| Design 2 | Nodes      | 856 923   |
|          | Elements   | 3 982 582 |

Table 2 outlines the necessary setup steps before conducting the simulation. In this research, the SST  $k-\omega$  turbulence model was selected due to its suitability for simulating viscous sub-layer flow and providing more accurate predictions of flow separation compared to most Reynolds-Averaged Navier-Stokes (RANS) models. This model is particularly advantageous in resolving boundary layer behaviours, especially under unfavourable pressure gradients. Each design is tested at three different rotational speeds: 20000, 40000, and 60000 rpm. The boundary conditions for the inlet consist of a mass flow rate of 0.0765 kg/s and a pressure of 0 Pa. The pressure value is set equal to atmospheric pressure. Before initiating the calculation, the number of iterations is defined as 2000 to ensure accurate results. It is essential for the residual graph to converge to the lowest possible value, indicating a stable and reliable solution.

**Table 2 – Computational Setup**

| Parameters             | Values                  |
|------------------------|-------------------------|
| Analysis Type          | Steady State            |
| Rotational Speed       | 20 000 – 60 000         |
| Inlet (mass flow rate) | 0.0765 kg/s             |
| Outlet                 | Pressure                |
| Turbulence Model       | SST $k-\omega$          |
| Initializations        | Standard Initialization |
| Number of Iterations   | 2000                    |

## 2.2 Calculation for the thrust

Thrust is the force generated by the engine that propels an aircraft or vehicle forward. It is directly related to the mass flow rate and velocity of the exhaust gases. The mass flow rate represents the amount of air passing through the engine per unit of time. In the context of a turbofan, it corresponds to the airflow entering the engine's inlet. Increasing the mass flow rate means more air is entering the engine, resulting in a higher thrust potential. The velocity of the exhaust gases, specifically the exhaust jet velocity, is another critical factor. The exhaust jet's speed is determined by the engine's design and the combustion process. The faster the exhaust gases are expelled, the greater the thrust generated. The mass flow rate is constant (1.225 kg/s) so, the thrust force can be calculated by multiplying the exit mass flow by the exit velocity. This simplification allows for a more straightforward analysis of the thrust generated by the engine through the Eqns. (1),

$$T = \rho AV^2 \quad (1)$$

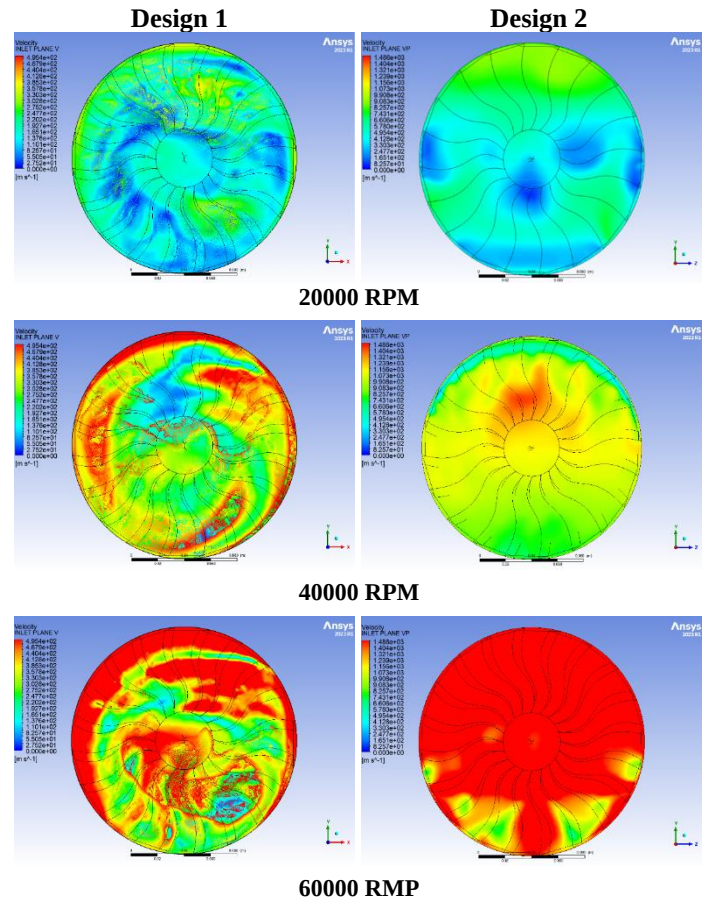
where,  $T$  = Thrust (N),  $\rho$  = density (kg/m<sup>3</sup>),  $A$  = area (m) and  $V$  = velocity (m/s).

## 3. Results and Discussion

In this section, the simulation results obtained from ANSYS were discussed, encompassing the distribution of velocity, pressure, and induced velocity for the purpose of calculating the total thrust. The analysis for the pressure increase, velocity and thrust obtained from the simulation.

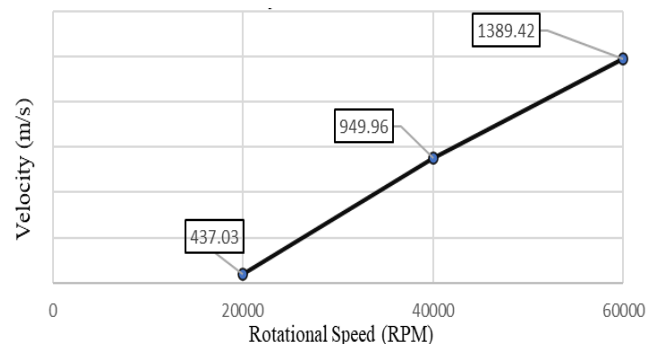
## 3.1 Velocity distribution

The velocity plays a crucial role in accurately simulating the flow dynamics and ensuring the correct direction of airflow in various engineering applications. As in MGTs, the air enters through the front and gets squeezed or compressed as it moves through the engine. This compression increases the pressure and temperature of the air. The velocity at the inlet for both design shows in figure 4. From the figure, it clear shows that the velocity has a significant effect on the design.



**Fig. 4 – Comparison of velocity inlet for both design**

The velocity distribution for both designs were plot in the figure 5 and 6 below to show the significant of the design on the velocity for the micro gas turbine.



**Fig. 5 – Velocity distribution for Design 1**

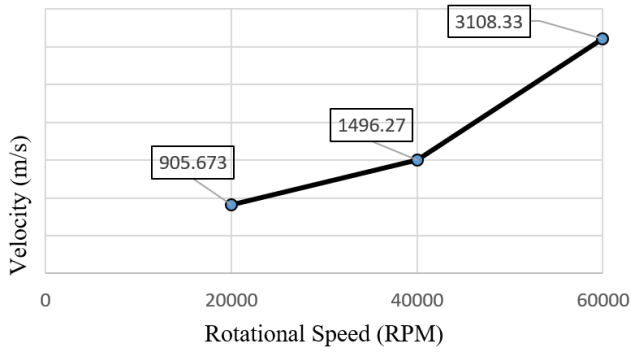


Fig. 6 – Velocity distribution for Design 2

Based on the figures above, that Design 2 shows higher velocity values in comparison to Design 1. This serves as evidence supporting the idea that adjusting the size of the nozzle, thereby influencing the angle of air flow, that change the angle of by-pass ratio air flow leads to an increase in velocity. Conversely, Design 1, which features a bigger nozzle size, demonstrates lower velocity values. These findings strongly suggest that the variations in nozzle size directly impact the flow characteristics of the air, resulting in corresponding changes in velocity.

Other results that interesting to discuss on an efficiency for each design was the velocity at the outlet of the turbine. The velocity distribution at the outlet for the design were shows in figure 8.

### 3.2 Pressure distribution

In a micro gas turbine, it is expected that the pressure at the inlet is relatively low and remains low at the outlet. This pressure change is primarily influenced by the corresponding velocity change along the flow path. The comparison for pressure distribution at the inlet were shown in figure 9 below.

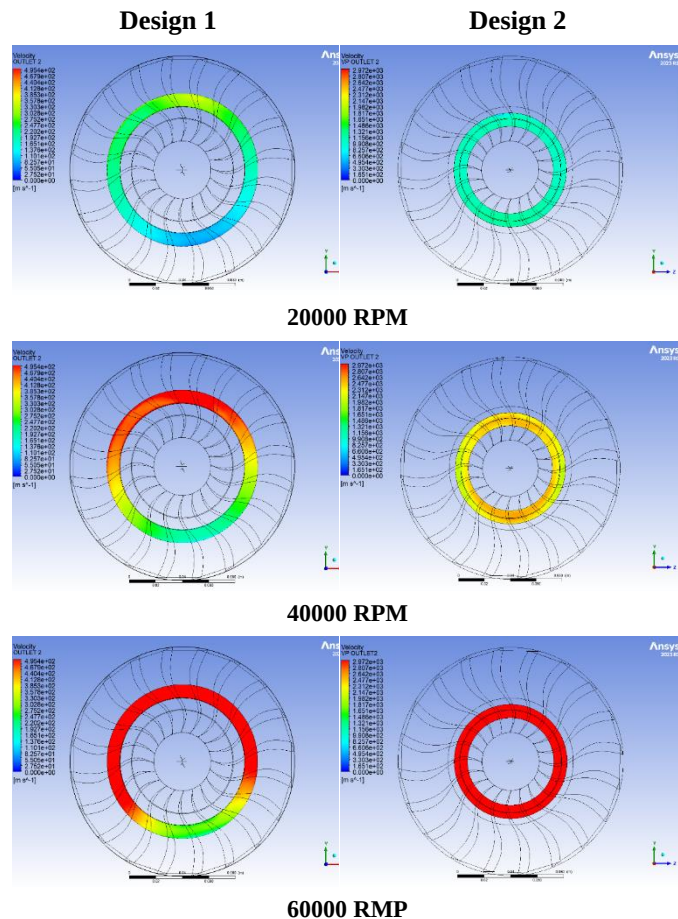


Fig. 8 – Comparison of velocity outlet for both design

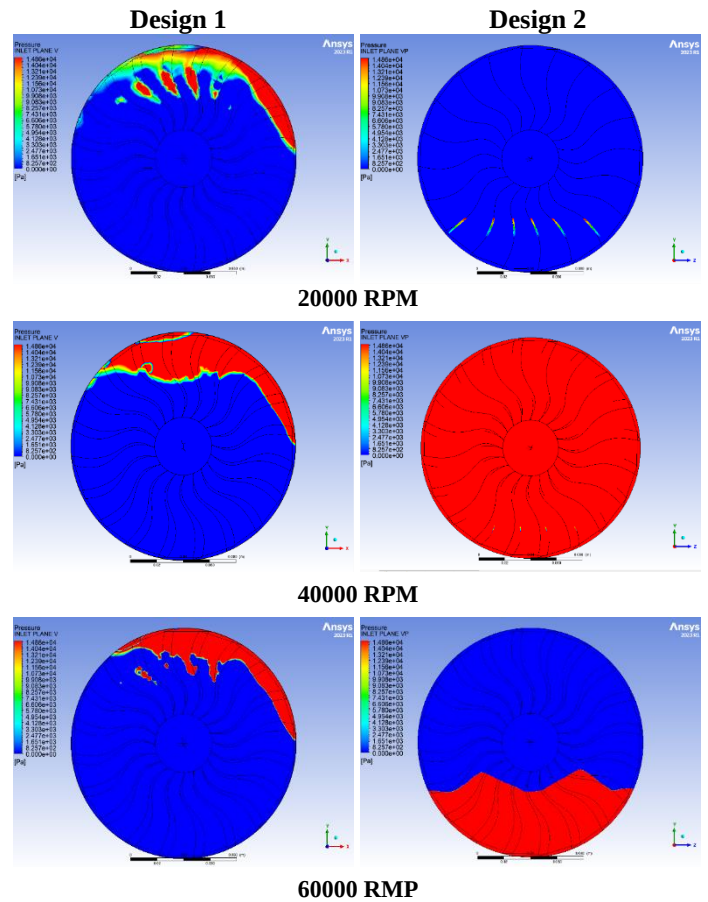


Fig. 9 – Comparison of pressure inlet for both design

The decrease in pressure occurs along with the increase in velocity. At the outlet, the high-velocity flow has gone through the expansion process and, as a result, the pressure remains low. The converging relationship between the high velocity and low pressure can be attributed to the principles of fluid dynamics and the conservation laws governing the energy transfer within the micro gas turbine. Based on the table above, the result shows that the pressure remains constantly low or not having much change from the inlet to the outlet of the MGTs.

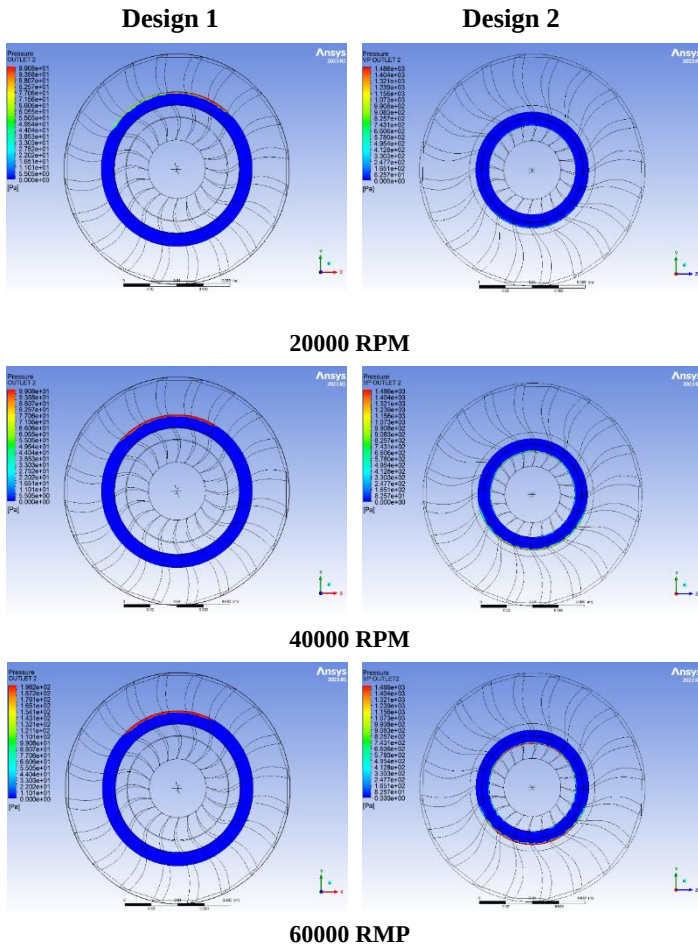


Fig. 10 – Comparison of pressure outlet for both design

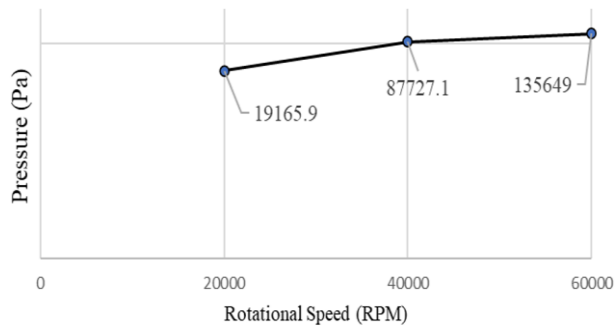


Fig. 11 – Pressure distribution for Design 1

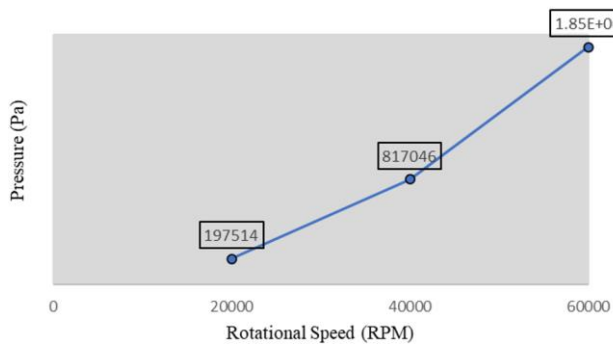


Fig. 12 – Velocity distribution for Design 2

The figure demonstrates that the pressure value for Design 2 is higher compared to Design 1. This observation provides evidence supporting the notion that the change in angle has an impact on the bypass ratio airflow. In a turbofan engine, the bypass ratio refers to the ratio of the mass of air that bypasses the engine core (the fan's bypass flow) to the mass of air that goes through the core (the primary flow). The bypass ratio affects the overall engine performance and efficiency.

### 3.3 Thrust calculation

In this section, the calculations were conducted using data obtained from the ANSYS Fluent software. The trust calculation results show in Table 3.

Table 3 – Computational Setup

| Design   | Speed (RPM) | V-induced (m/s) | Thrust (N) |
|----------|-------------|-----------------|------------|
| Design 1 | 20 000      | 199.9871        | 152.341    |
|          | 40 000      | 393.1234        | 609.269    |
|          | 60 000      | 587.736         | 1370.817   |
| Design 2 | 20 000      | 1176.181        | 3991.555   |
|          | 40 000      | 2371.474        | 16226.71   |
|          | 60 000      | 3567.019        | 36711.71   |

As indicated by the graph, the results demonstrate that Design 2 yields a higher thrust value compared to Design 1 when subjected to three different rotational speeds. This data supports the notion that the design modifications implemented in Design 2 have effectively enhanced thrust performance. It is essential to note that the relationship between design modifications and resulting thrust values may vary depending on factors such as the specific engine, rotational speeds, and operational conditions. Further analysis and testing would be required to validate and fully comprehend the observed differences in thrust between the designs.

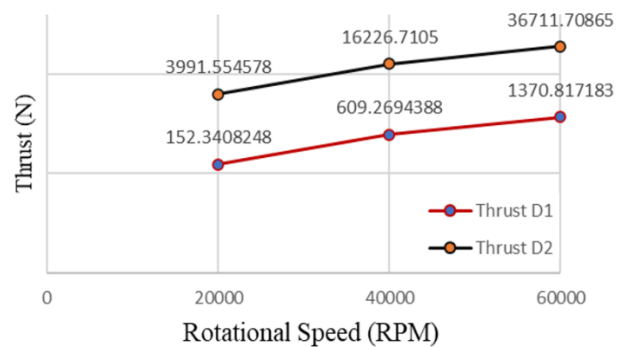


Fig. 13 – Thrust comparison

### 4. Conclusion

After obtaining the results, pressure and velocity distribution graphs were plotted to analyze the changes in velocity and pressure along the primary flow, which represents the bypass ratio airflow. Comparing the results from both designs, it was observed that in Design 2, both pressure and velocity values were higher compared to Design 1. Additionally, the thrust produced in Design 2 was found to be significantly better, measuring at

36,711.71 N, which surpassed the thrust generated in Design 1. These findings suggest that Design 2 outperformed Design 1 in terms of both pressure and velocity distribution as well as overall thrust production. This outcome confirms the effectiveness of the proposed enhancements and suggests that Design 2 has the potential for superior performance compared to Design 1. In conclusion, the analysis of pressure and velocity distribution, along with the comparison of thrust values, demonstrates the superior performance of Design 2 over Design 1.

## References

- [1] Onal, O., and Osman, T., "Calculation and Comparison of a Turbofan Engine Performance Parameters with Various Definitions." *World Academy of Science, Engineering and Technology International Journal of Aerospace and Mechanical Engineering*, 4 Aug. 2016,
- [2] Jingqi, L., and Yulong L., "Micro Gas Turbine: Developments, Applications, and Key Technologies on Components." *Propulsion and Power Research*. 12(1), (2023): 1–43,
- [3] Fisher, U., Sugarmen, C., Ring, A., & Sinai, J., "Gas Turbine Solarization Modifications for Solar/Fuel Hybrid Operation." *Journal of Solar Energy Engineering-transactions*. 26, (2004): 872-878.
- [4] Braun, Hans-Joachim. "The Chrysler Automotive Gas Turbine Engine, 1950-80." *Social Studies of Science*, 22(2), (1992): 339–351.
- [5] Al-attab, K.A., and Z.A. Zainal. "Externally Fired Gas Turbine Technology: A Review." *Applied Energy*. 138, (2015): 474–487
- [6] Pasupuleti, H.K., and Chandra S.K., "Design Analysis and Development of a Micro Gas Turbine Components." *International Journal for Modern Trends in Science and Technology*. 8(09), (2022): 33-41.
- [7] Pakle, S., & Jiang, K. "Design of double curvature radial turbine blades for a micro gas turbine." *Applied Mathematical Modelling*, 67, (2019): 529–548.
- [8] Song, P., Wei, Z., Zhen, H., Liu, M., & Ren, J., "Effects of pre-whirl and blade profile on the hydraulic and cavitation performance of a centrifugal pump." *International Journal of Multiphase Flow*, 157, (2022): 104261.
- [9] Coppinger, M., & Swain, E. "Performance prediction of an industrial centrifugal compressor inlet guide vane system." *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 214(2), (2002): 153–164.
- [10] Deshmukh, Ashutosh. B., Lotake, Aditya. A., Paricharak, Hrushikesh. N., Jadhav, Subhash. V., Kashid, Digambar. T., & Wangikar, Sandeep. S. "Comparative CFD Analysis of Mini Impeller Using Different Materials." *Materials Today: Proceedings*, 24, (2020): 2115–2122.
- [11] De Santis, A., Ingham, D. B., Ma, L., & Pourkashanian, M. (2016). CFD analysis of exhaust gas recirculation in a micro gas turbine combustor for CO<sub>2</sub> capture. *Fuel*, 173, (2016): 146–154.
- [12] Aghaei tog, R., Mesgharpoor Tousi, A., & Soltani, M. "Design and CFD analysis of centrifugal compressor for a micro gas turbine." *Aircraft Engineering and Aerospace Technology*, 79(2), (2007): 137–143.
- [13] Mendonça, F., Baris, O., & Capon, G. "Simulation of Radial Compressor Aeroacoustics Using CFD." *ASME Turbo Expo 2012: Turbine Technical Conference and Exposition*.
- [14] Le Sausse, P., Fabrie, P., Arnou, D., & Clunet, F. "CFD comparison with centrifugal compressor measurements on a wide operating range." *EPJ Web of Conferences*, 45, (2013): 01059.
- [15] Arntz, A., Olivier Atinault, & Alain Merlen. (2015). Exergy-Based Formulation for Aircraft Aeropropulsive Performance Assessment: Theoretical Development. *AIAA Journal*, 53(6), (2015): 1627–1639.

## Acknowledgement

This research was supported by Universiti Tun Hussien Onn Malaysia (UTHM) through Tier 1 (Vot H921) grant. The authors also would like to express their sincere gratitude to the Faculty of Mechanical and Manufacturing Engineering supporting staff for the continuous support throughout the research process including access to facilities, provision of materials, and analysis of the findings in the successful completion of this research.