Periodicals of Engineering and Natural Sciences ISSN 2303-4521
Vol. 8, No. 1, March 2020, pp.413-422

Aerodynamic design and performance investigation of an axial turbocharger turbine for automotive application

Layth H. Jawad[1], Hussein Younus Razzaq[2], Hussein Mohammed Hasan[3]
[1,2,3] Department of Mechanical Techniques, Karbala Technical Institute,
[1,2,3]Al-Furat Al-Awsat Technical University

ABSTRACT

In recent years, developments of vehicle downsizing promote the developers to improve the performance of current turbocharging technology. Due to drawbacks transient response of conventional radial turbine used in turbocharging techniques, preliminary design of axial turbine was proposed, in order to achieve highest performance of turbocharger axial turbine. In this study, the optimal design methodology, based NACA profile blade of a single stage axial turbine for turbocharger system. Therefore, simulation analysis of steady state three-dimensional flow carried out by highly reliable for calculation, computational fluid dynamic (CFD), using ANSYS CFX, to be evaluated the stage overall aerodynamic performance of the axial turbine stage. Analysis results, gave a more details of flow behaviour such as, flow separation, vortexes and performance characteristics. Moreover, it is found the pressure load for less blades, it's too low on each blade, and a reasonable pressure blade load on a single blade was therefore seen to be too high for more blades, resulting in loss of Boundary layer of the blade, flow of tip leakage and Secondary flow. Hence, noticeably, it was observed that the aerodynamic performance of turbocharger axial turbine model was predicted numerically such as, total-total Polytropic efficiency (84.64) % and shaft output power (187) kW at (80k) rpm.

Keywords: Turbocharger; Axial Turbine; CFD

Corresponding Author:

Third Author Name, Layth H. Jawad
Department, Department of Mechanical Techniques
University, Al-Furat Al-Awsat Technical University
Address. 56001 Karbala, Iraq
E-mail: inkr.lyt@atu.edu.iq

1. Introduction

Nowadays, the turbocharger developments interest become the most broadly used in automotive industry, due to engines downsizing, reducing emissions, reducing fuel consumption, and highest performance. However, the transient response is one of drawbacks conventional turbocharging systems, promote to investigate a single stage axial turbine for turbocharger. In the last few years, a lot of studies performed to understand the unsteady state flow of centrifugal turbine. Moreover, improving efficiency of radial turbine it is still difficult under unsteady characteristics. The effect of a significant difference between unsteady and steady flow conditions, turbine efficiency decreases significantly. Therefore, the improving of the stable operating ranges of axial turbocharger turbine produce benefits in both steady and transient state efficiency of a single stage axial turbine for modern turbocharging systems. Therefore, the use of (CFD) computational fluid dynamic become useful tool for improvement, modification and resolving the complex of flow inside fluid domain of turbocharger turbine. One of the main problems, slow engine acceleration at low turbocharger exhaust flow, this is turbocharger lag. Thus, need to at low rate of exhaust flow, improving the turbocharger efficiency [1]. In order to reduce the turbocharger lag, (VGTs) variable geometry turbocharger for nozzles was used compared to a fixed geometry turbocharger, which produced highest speed of the rotor turbine [2]. Due to higher moment of inertia and weight of the turbine wheel, there is a simpler way to reduce the turbocharge lag by using electric motor to accelerate the shaft of turbine wheel [3]. At same flow conditions the axial turbine has lower moment of inertia compared with radial turbine, thus, in aviation and marine applications used the axial turbine, but still

yet in automotive applications the axial turbocharger turbine type need to use [4,5] . Moreover, the flow characteristics and pressure frequency pulsation, at different operating conditions were studied inside two nozzles radial turbocharger turbine used for automotive [6, 7]. The influence of guide vanes on distribution of flow angle and its oscillations were studied within a radial turbocharger turbine stage [8, 9]. The better understanding of fluid flow dynamic phenomena of radial compressor and radial turbine used for turbocharger were studied numerically by CFD analysis, results showed good agreement with experimental results [10, 11]. The developed turbine designs were tested to evaluate the performance and more understanding of unsteady flow characteristics [12, 18]. The quasi one-dimensional steady mixed flow volute of rotor turbine was studied [13]. Appropriate two nozzles radial turbine model developed in order to produce pressure drops across the rotor and stator, the prediction CFD results agree with experimental data of engine test [14, 15]. Unsteady performance prediction of radial turbine for turbocharger, quasi 3D flow CFD model was studied [16, 17].
In the literature reviewed above, more studied performed to improve and modify the design geometry of the radial turbocharger turbine, where CFD used to study the flow behaviour within radial turbine and nozzles. A model of the CFD in the current paper for a complete single stage axial turbine has been considered. The CFD model was tested with experimental data from other researchers. Using flow non-uniformity indices, a comparison of performance parameters (total efficiency) for different admission cases is discussed, these are described as possible performance parameters to be used to compare optimization of axial turbine designs. Thus, in this study was focused to develop the design methodology and investigate an axial turbocharger turbine single stage, in order to achieve highest aerodynamic performance based on computational fluid dynamic (CFD) analysis.

1.1. Preliminary design of axial blade

To gain an understanding of the nature and important aspects of turbines, in-depth research on axial turbines will be conducted as part of the turbocharger investigation. Each section contains principles such as the design of the stator and rotor blade as well as methods to be followed in this analysis. Commonly used for the blading design of some types such as free vortex, constant nozzle angle and constant mass flow. There are free vortex blading stage characteristics such as constant annulus axial velocity, constant annulus stagnation enthalpy, and whirl velocity is inversely proportional to the radius. The nozzle angle is not twisted with a particular form of stator blade due to Constant nozzle angle blading that is one of the forms of blading. A constant nozzle angle type is the blading design used in this study, this type is chosen because of the simple stator contour geometry and will be influenced by the cost of the manufacturer with simple geometry contour.
The geometry of an axial blade with different blade profiles and multiple angles can be described. In addition, important tool such as velocity diagram to illustrate the inlet velocity of fluid within turbine blades. Figure 1 displays the diagram of velocity and the loads of the blades; conditions of work will work [17]. The stator and the rotor geometry were created using the ANSYS (BladeGen) software package. Blade profiles were cambered and drawn using tangential curves and straight lines as shown in Figure 2. The stator blade profile was designed to maximize the momentum will be transferred to the rotor blades. The blade profile selected was a general NACA (0006) for the stator as well as the rotor. The main geometry features and dimensions are given in Table 1.

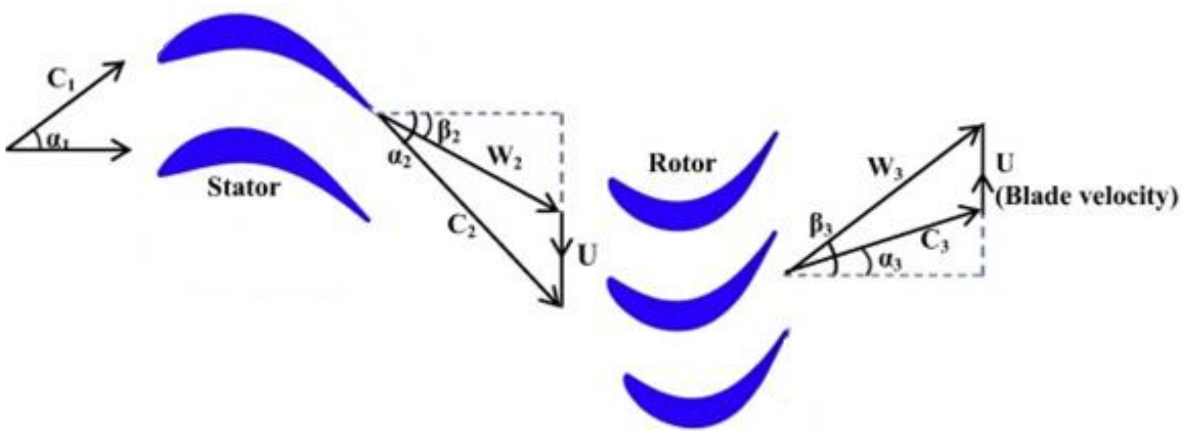


Figure 1. Velocity diagram of axial turbine

A three-dimensional geometric modelling of the fluid region of the planned single-stage axial turbine, involving inlet extension, single-stage cascade flow passage and outlet extension, is completed according to the profiles and flow passage of the key geometric parameters of turbine cascades, as shown in Figure 2.

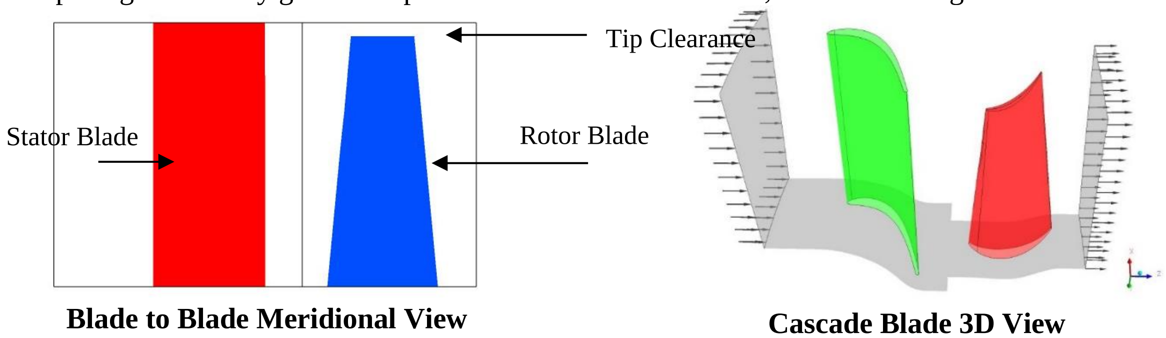


Figure 2. Fluid flow domain for stator and rotor blades view

Table 1. (Stator and rotor) axial features

Geometry of Axial Turbines	Rotor
Axial Width of the Blade	17 mm
Inner Turbine Radius at Inlet	63.5 mm
Outer Turbine Radius at Inlet	90 mm
Width of Blade at Leading Edge	9 mm
The Number of Blades	40
Clearance of the Shroud Tip (% Span)	95%
Nozzle Geometry	**stator**
Number of Blades	25
Nozzle's Inner Radius	63.5 mm
Nozzle's Outer Radius	90 mm
Inner Blade Width	11.5 mm

A single-stage turbine wheel with 40 main rotor blades and 25 nozzle blades, as shown in Figure 3. The design CFD computations were carried out on the stator and rotor geometries. All geometry surfaces, inlet, outlet, and periodic boundaries were defined through ANSYS (BladeGen) design.

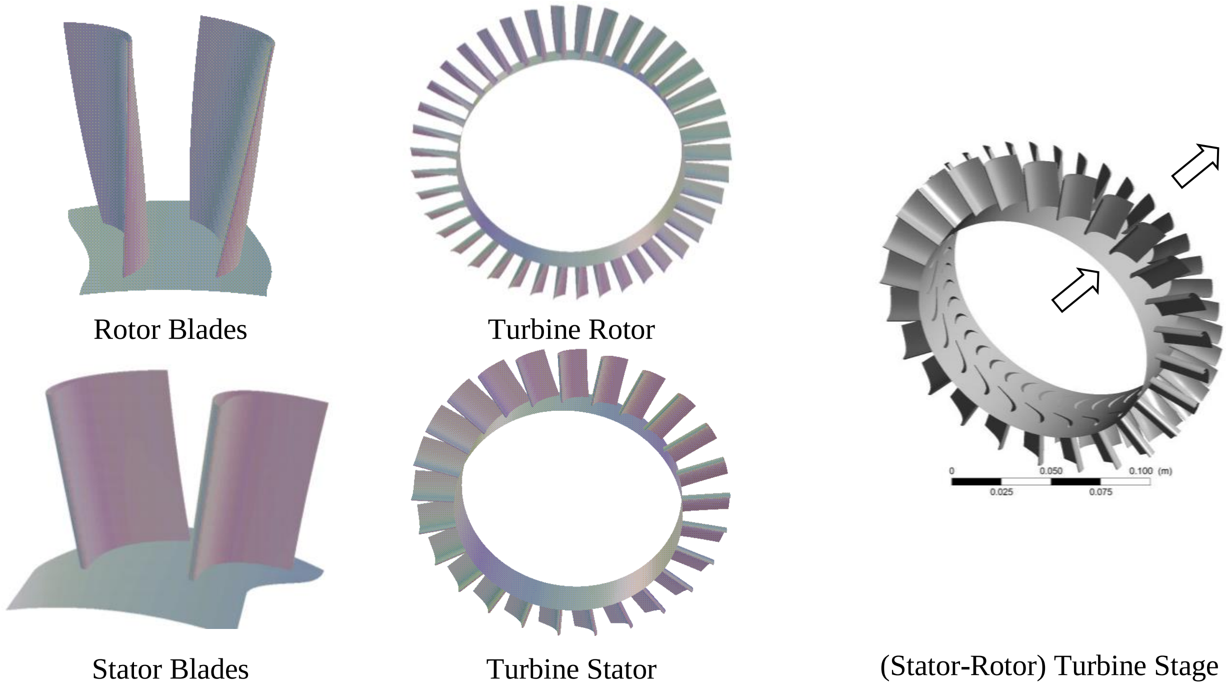
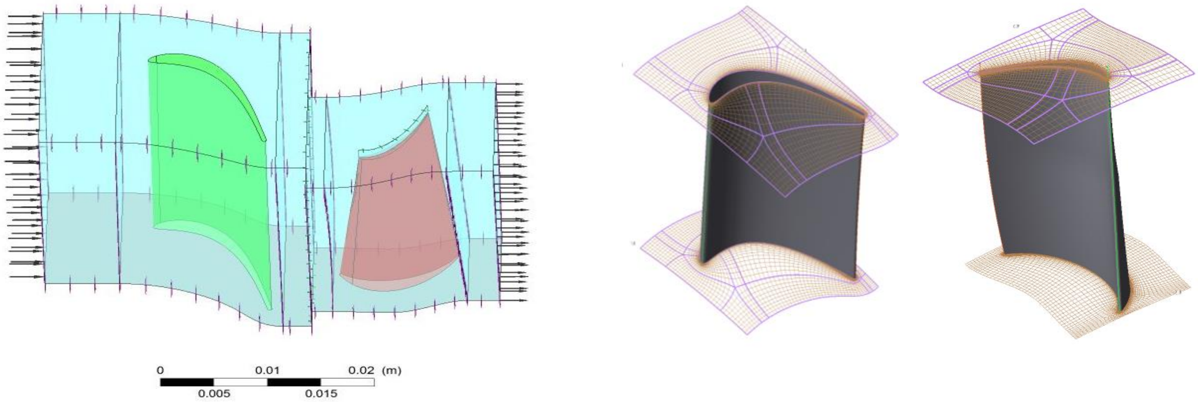


Figure 3. Three-dimensional view of (stator-rotor) stage axial turbine

2. Modelling setup of CFD methodology

Axial turbine modelling and simulation of following the performance target of the turbocharger is a very important factor in the development of new turbochargers. Each turbocharger's working conditions could have a major impact on engine performance and efficiency. Decreasing in power, efficiency and increasing fuel consumption due to the selection of the wrong turbocharger resulting in severe effects of turbo lag. Consequently, each turbocharger is designed based on specific engine features. The blade turbine's conceptual design is an essential first step in the development of a current turbine. Another preliminary advanced design used in combination with an advanced system of CFD will lead to a very powerful turbine. Uses the ANSYS CFX for preliminary design, which enables the development of three-dimensional geometry designs from initial ideas. ANSYS (Turbogrid), is a high-performance automated meshing tool, the first move was to import the single stator-rotor blade. To obtain accurate results with different parameters was carried out for several trials in order to test the meshing. The blade was in the middle of Extracted the fluid domain for one passage rotor blade. The hexahedral surface mesh on the stator and rotor surfaces was generated. The sufficient control tools for surface repairing analysis Enables the selection of components to be included in the meshing and excluded, the hexahedral as finite element mesh was automatically generated, and it consists of 8 nodes. The computational grid was developed using a dedicated Turbogrid program for the rotor blade passage consisting of exactly 456352 used in multi-block environment as structured hexahedral elements and for the nozzle passage consisting of 355875 hexahedral elements. The clearance area in the rotor tip, nearby the rotor blade, and at the walls of the hub and the shroud, sufficiently fine grid elements were developed. Grid size plays a major role in the solution's convergence and accuracy. In general, finer grids allow the solution to be independent of the grid size and generate more correct results, but always require lager computing resources and time. In this paper, a fine grid size of elements was used For the CFD simulations, as shown in Figure 4.



Stator-Rotor Domain Mesh **Stator Domain Mesh** **Rotor Domain Mesh**

Figure 4. Computational Domain Mesh

2.1. Fluid flow modelling

The Several manufacturers have adopted the standard procedure for testing of axial turbocharger turbine flow conditions under steady, although it does not reproduce the actual operating conditions of the engine faithfully. The stator turbine rotor mesh has been imported into the CFX setup. The rotor domain choice was chosen to rotate, and air ideal gas was the working fluid. Depending on engine operating conditions, the boundary conditions for the pressure and temperature of the inlet and outlet were used. The heat transfer was set from the fluid model tab to total energy and shear stress transport for the turbulence. At the inlet of the stage the total pressure was 365 kPa and static temperature was 850 K. With a static pressure was 130 kPa, at the outlet of rotor was established. The included inlet walls were set as smooth and no-slip, due to the mesh movement, the rotor shroud frame type was rotating. Therefore, the residual target was fixed to 0.0001 from the solver control before the simulation was run. Through the use of a specific method for designing the radial rotor, the axial turbine was designed using a different approach. Results were obtained from a single stag axial turbine, so the correct axial design conditions could be determined. As mentioned earlier, in the automation industry, axial turbines are not widely used; thus, a different design method has been used. To obtain the efficiency of rotor-stator configuration, Parametric simulations have been carried out on a 3D turbulent computational fluid dynamic (CFD). The domain of fluid consists of one area that covers the rotating parts (hub and blades) and the stationary parts (nozzle, inlet, outlet and shroud). The periodic boundaries were set to the right and left of fluid domain, the model k-ω-SST was used as a Turbulence model. Technically, it's a combination of k-away from the walls and k-away nearby the boundary layer wall. In addition, any surface is defined as adiabatic, the axial turbine design boundary conditions are described in table 2.

Table 2: CFD Boundary Conditions

Inlet Total Temperature (K)	850
Inlet Total Pressure (bar)	3.65
Speed of Rotation (rpm)	80000
Static Pressure at Outlet (bar)	1.30
Residual Target	0.0001
Interface of Mixing Plane	(Stage) General Connection
Direction of Flow	(1,0,0) Cylindrical

As mentioned earlier, the preliminary design angles were used for the first blade sketches of the blades with regard to the axial turbocharger turbines blade design; hence, some optimization was required to achieve excellent efficiency and turbine performance. Because of the use of BladeGen, The axial turbine design optimization was different through ANSYS. Optimization capabilities were minimal, and the improvements had to be made one at a time, whereas, the axial optimization of the blade designed in Geometry directly enabled more than one parameter at a time to be optimized.

3. Results and discussion

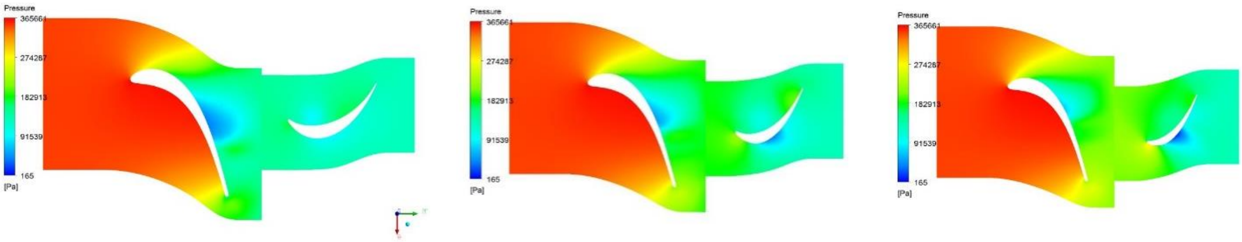
The experimental test data used in calculations for the axial turbine design as boundary conditions. The mass flow rate at a specified turbine speed is one of adopted variables, basic turbine equations could then be resolved. This could make it easier to conclude at each turbine stage, the calculations of the temperature and pressure. The solver part of the CFD simulation software requires an iterative process in order to achieve a solution. In general, the residual magnitude will decrease as the solution converges. Overall performance CFD results prediction for the designed an axial turbine at specific speed 80k rpm, as shown in table 3.

Table 3. Overall performance results for an axial turbine preliminary model

Rate of Mass Flow (kg/s)	0.324084
Power Shaft (W)	187240
Coefficient of Inlet Flow	0.3462
Ratio of Total Pressure	1.3765
Ratio of Total Temperature	1.1231
Coefficient of Nozzle Loss	0.8718
Nozzle Efficiency %	91.1704
T-T Isen. Efficiency%	85.1327
T-T Poly. Efficiency%	84.6431

Therefore, it can be seen a little difference among the passage of rotor, because the rotor outlet conditions are not uniform, the computational time was lower for one passage axial turbine and the accuracy of solution CFD modelling results was not significantly affected.

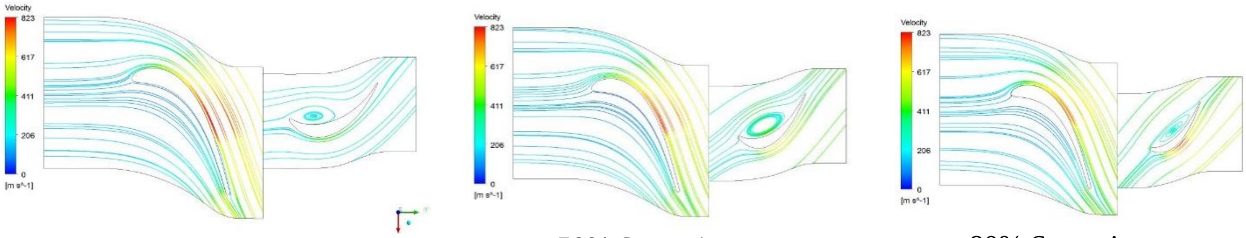
The pressure distribution at the 20%, 50%, and 80% spanwise blade height section as shown in Figures 5. The pressure indicates a downward trend from the inlet nozzle to the rotor outlet, the pressure distribution at each cascade's outlet is uniform. The pressure on the pressure surface in the rotor cascade of each stage is higher than that on the suction surface, thus producing efficient differential pressure force, causing the rotating blade to rotate. Because the fluid on the suction side has a higher velocity in the nozzle, the pressure here is lowered and they rise again after the fluid is combined with that on the outlet pressure side. From the entrance to the exit, the pressure on the pressure side slowly decreases and has a uniform gradient. The working fluid in the rotor expands through the direction of flow, and the pressure decreases slowly, reaching a pressure at the rotor outlet. While, there is relatively uniform pressure distribution of the cascades along the direction of the blade height. Therefore, the pressure gradient created by the fluid is concentrated to the tip position along the direction of the blade height.



20% Spanwise 50% Spanwise 80% Spanwise

Figure 5. Blade to blade pressure distribution contour

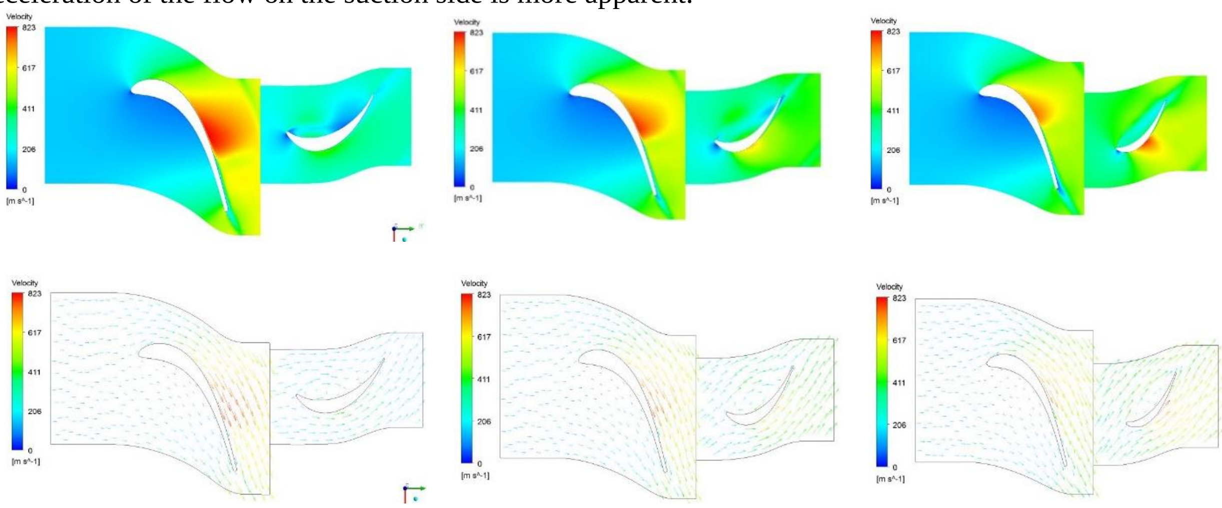
The velocity streamline height part and the relative extension of the leading edges and trailing edges of the cascades of each stage shown in Figure 6 at different spans. Due to the large curvature of the nozzle suction surface, the fluid on the suction side has a higher flow rate. The mixed fluid enters the rotor inlet after being mixed with the fluid on the pressure surface side close to the nozzle trailing edge. The curvature suddenly change causes a local effect of acceleration on the suction side when the fluid passes through the rotor's leading edge. The leading edges of the nozzle and rotor, where the fluid has a lower flow rate, have a small range of stagnation zones. From the streamline distribution, it is easy to see that each cascade's outlet velocity is substantially greater than the inlet velocity. Due to the expansion of the working fluid in the cascade, enthalpy decreases and portion of it is converted into kinetic fluid energy. In addition, the entire cascade flow channel does not have a significant flow separation, and the flow region is relatively uniform.



20% Spanwise 50% Spanwise 80% Spanwise

Figure 6. Blade to blade velocity streamline distribution contour

Additionally, the incoming wake and velocity vectors, can also be viewed from the upstream blade boundary layer as low-energy fluid shedding. In the blade row the transport of wakes also influences the evolution of these vortices. Under the typical boundary conditions, Figure 7 shows the evolution of the vortex system in the blade to blade axial turbine cascade. It indicates that the wake primarily comprises small-scale vortex structures prior to entering the rotor. Through acceleration, the broad eddies in the stream slowly dominate the wake area after entering the main stream. These eddies accumulate on the suction side because of the backflow, then the acceleration of the flow on the suction side is more apparent.



20% Spanwise 50% Spanwise 80% Spanwise

Figure 7. Blade to blade velocity vectors distribution contour

The Mach numbers contour in the cascade row blades at different span Displayed in Figure 8. If the number of Mach is 0.7 at outlet, the flow is subsonic. As outlet Mach number increases to 0.9 as backpressure decreases (remain unchanged the inlet angle flow and inlet total pressure), blade loading slowly increases, and a small supersonic region can be seen in the axial chord range. This feature of the flow is very typical of most current generation axial turbines. When backpressure decreases further, the number of Mach outlets would gradually increase and reach 1, while the number of Mach inlets may gradually increase to the upper limit and no longer change. When backpressure continues to decline, the distribution of pressure on a large part of suction pressure surface and pressure surface would no longer change, then a shock wave would travel from near the edge of the trail to the suction surface of the adjacent blade and then reflect the suction surface.

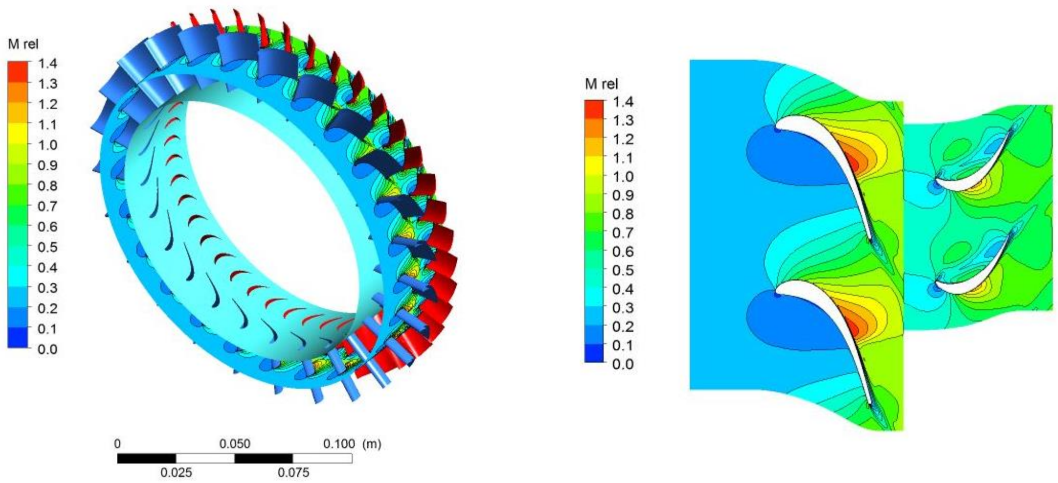
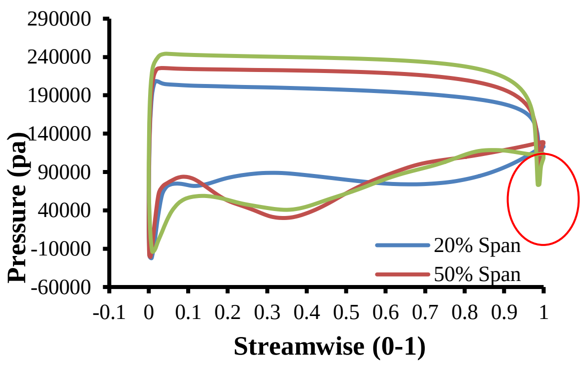
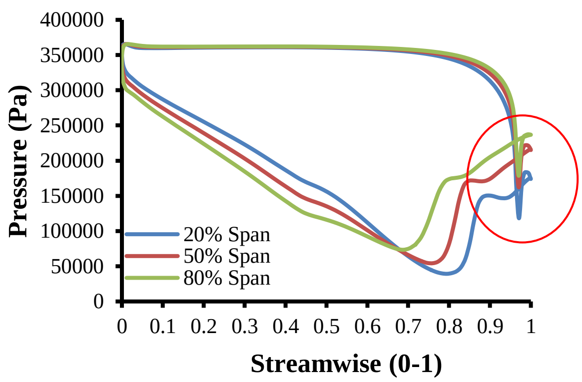


Figure 8. Mach number distribution contour

Blade loading at different spans through the stator-rotor channel, as shown in Figure 9 that indicates the turbine stage loading. In fact, the load on a single blade also worries designers. If the load on a single blade is too high, then a turbine stage needs more blades, thus increasing the turbine's weight. In addition, if the load is too high on a single blade, increased boundary layer of blade loss, tip leakage flow, secondary flow, and other turbine passage flows may occur. Moreover, the degree of blade loading should be correctly chosen for a given design problem in order to balance aerodynamic efficiency versus turbine weight, Based on a reasonable standard for blade load determination. The magnitude of the loading of the pressure blade, defined as the ratio of the actual circumferential force on the blade to the ideal circumferential force, obtained on the condition that both the static surface on the pressure surface and the suction surface are uniform and equivalent to the total inlet pressure and the static outlet pressure. The red circle was seen as the area where the flow is normally forced to reverse direction because the pressure at the surface of the suction is greater than that on the surface of the pressure. The reversal of the pressure can be avoided in this design and prevents turbine losses.



Rotor blade loading



Stator blade loading

Figure 9. Stator-rotor pressure blade loading

As shown in Figure 10, when the tip clearance is 5%, the span-wise distribution of the angle of flow can be seen at the blade tip, corresponding to the leakage tip. The average flow angle distribution value in stator and rotor

(TE) along the distance from hub to shroud. Moreover, if the flow structure is considered, the study of the flows will become much more complicated. For example, the axial direction of the outlet flow angle varies from the hub to the shroud of the three-dimensional flow parameters, in addition, tip leakage flows, and rotor wakes and secondary vortexes flow are the main sources of aerodynamic losses and have a major impact on exit flow conditions. Therefore, the flow angle among the flow condition parameters is one of the most important factors. Therefore, the shroud's boundary layer is more stable, instead of the hub's boundary layer, due to the flow angle impact. The rotor outlet flow angle was observed to show significant turning over and under near the (TE) region due to vortices. The major vortex pair can also result in a large fluctuation of the flow angle in the outlet of the rotor with the amplitude of fluctuation being as large. Incidence loss in downstream blade rows is clearly fluctuating as a result of these unstable fluctuations.

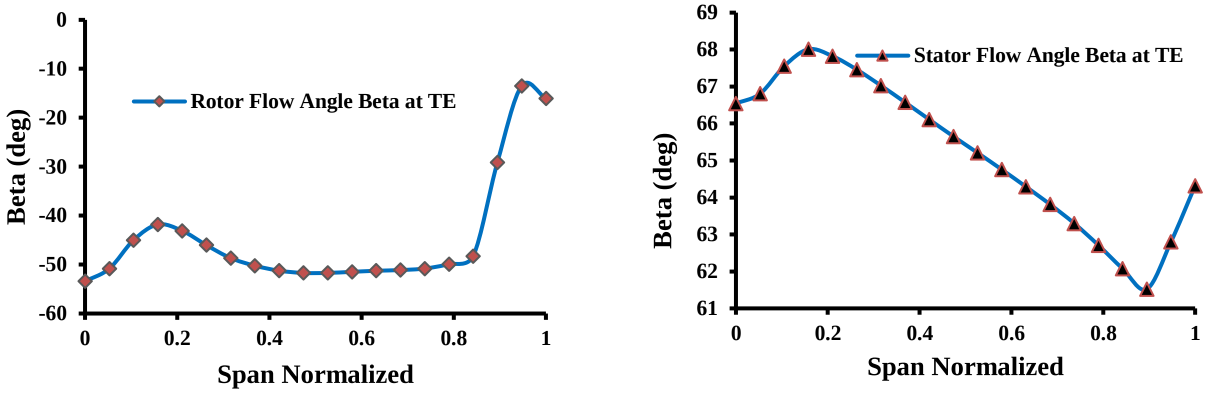


Figure 10. Stator-rotor flow angle distribution at blade trailing edge (TE)

In addition, Figure 11 shows the difference among the inlet stator and the exit rotor Mach number across the stream position; it is clear that the Mach number is higher between the stator exit of and the rotor inlet; therefore, taking into account the effect of the space ratio in the design is very important. It is obviously seen that the number of Mach is less than one and there is no flow chocking, as mentioned earlier that the ratio of the space area is very significant to change.

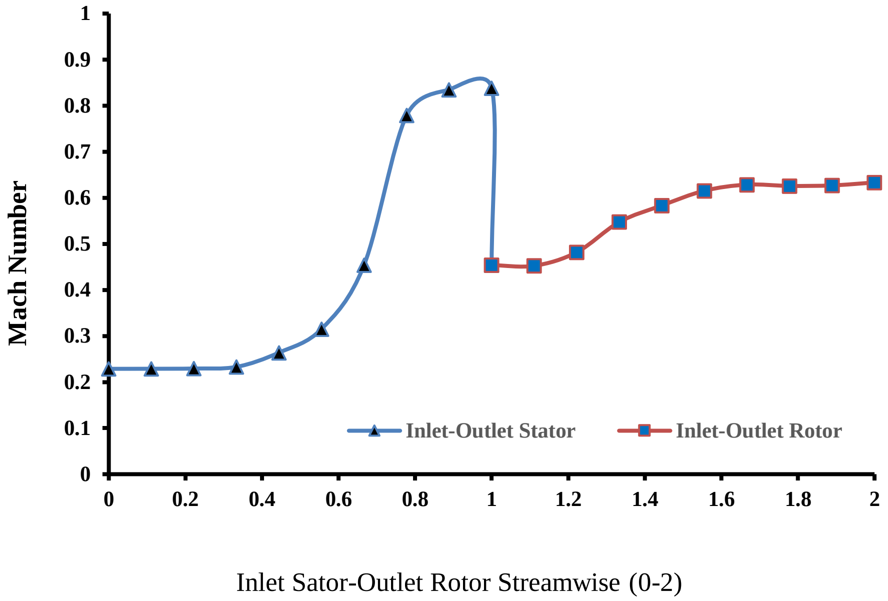


Figure 11. Mach number with streamwise location

Computational fluid dynamics (CFD) Modelling provides a much better understanding of the flow inside the single-stage axial turbine, let’s solve a lot of problems quickly and easily. The stator-rotor flow passages were taken into account only in the numerical analysis. An axial turbocharger turbine's preliminary design provides better performance or a wide range of operations.

5. Conclusions

In this research, an axial turbine was proposed, evaluated, designed and computationally tested for turbocharger application. The aim of the analysis was to replace the radial turbine of a current turbine with an axial blade turbine. The main objective was to decrease the axial turbine inertia and preserve an otherwise reasonable level of performance would lead to reducing the turbo-lag phenomenon. In addition, the turbine's performance can be increased by recovering the downstream system's static energy. The turbine design established from a single primary stage to validate the best of an axial blade turbine and was followed by a detailed design and dynamic analysis of 3D computational fluid through a single passage stage. The results of the CFD showed that the preliminary new axial turbine is operating at a reasonable level. In a wide range of operations, the axial turbine has high efficiency, thereby, it increases the turbine's overall efficiency. The results of the simulation analysis show that the preliminary design of an axial turbine is based on the axial blade configuration of the NACA profile, Could still ensure the same power and efficiency output as the original, in both the original radial turbine and the original axial turbine should maintaining the higher efficiencies for the best response system . Conclusively, the aerodynamic results showed that, firstly, simulation of computational fluid dynamics (CFD) technology can predict and improve axial turbine performance, and results created through the axial turbine stage for a better understanding of fluid flow. Secondly, the nature of the flow in the inlet stator-outlet rotor in the passage axial direction was uniform and the back-flow and vortex near the pressure surface slowly diminished.

References

[1] Zhao, H. Advanced Direct Injection Combustion Engine Technologies and Development, 1st ed.; Woodhead Publishing: Cambridge, UK, 2009; ISBN 9781845693893.

[2] Feneley, A.J.; Pesiridis, A.; Andwari, A.M. Variable Geometry Turbocharger Technologies for Exhaust Energy Recovery and Boosting-A Review. Renew. Sustain. Energy Rev. 2017, 71, 959-971.

[3] Terdich, N.; Martinez-Botas, R.; Pesiridis, A.; Romagnoli, A. Mild Hybridization via Electrification of the AirSystem: Electrically Assisted and Variable Geometry Turbocharging Impact on an Off-Road Diesel Engine.J. Eng. Gas Turbines Power 2013, 136, 031703.

[4] Ferrara, A.; Chen, H.; Pesiridis, A. Conceptual design of an axial turbocharger turbine. In Proceedings of the ASME Turbo Expo 2017, GT2017-64825, Charlotte, NC, USA, 26–30 June 2017.

[5] Pesiridis, A.; Saccomanno, A.; Tuccillo, R.; Capobianco, A. Conceptual Design of a Variable Geometry, Axial Flow Turbocharger Turbine. In Proceedings of the 13th International Conference on Engines & Vehicles, Napoli, Italy, 10–14 September 2017.

[6] J. Galindo, *et al.*, "Effect of the numerical scheme resolution on quasi-2D simulation of an automotive radial turbine under highly pulsating flow," *Journal of Computational and Applied Mathematics*, vol. 291, pp. 112-126, 2016.

[7] S. Zhu, *et al.*, "Modeling and extrapolating mass flow characteristics of a radial turbocharger turbine," *Energy*, vol. 87, pp. 628-637, 2015.

[8] M. Padzillah, *et al.*, "Experimental and numerical investigation on flow angle characteristics of an automotive mixed flow turbocharger turbine," *Jurnal Teknologi*, vol. 77, 2015.

[9] R. Capata and G. Hernandez, "Preliminary design and simulation of a turbo expander for small rated power organic Rankine cycle (ORC)," *Energies*, vol. 7, pp. 7067-7093, 2014.

[10] L. H. Jawad, *et al.*, "Numerical study on the effect of interaction vaned diffuser with impeller on the performance of a modified centrifugal compressor," *Journal of Mechanics*, vol. 30, pp. 113-121, 2014.

[11] L. H. Jawad, "Numerical Prediction of a Radial Turbine Performance designed for Automotive engines Turbocharger," *university of babylon Journal, vol.26, pp.132-146*, 2018.

[12] Newton, P.; Martinez-Botas, R.; Seiler, M.A. Three-Dimensional Computational Study of Pulsating Flow inside a Double Entry Turbine. J. Turbomach. 2015, 137, 031001.

[13] Chen, H.; Hakeem, I.; Martinez-Botas, R.F. Modelling of a turbocharger turbine under pulsating inlet conditions. Proc. Inst. Mech. Eng. Part A J. Power Energy 1996, 210, 397–408.

[14] Serrano, J.R.; Arnau, F.J.; Dolz, V.; Tiseira, A.; Cervelló, C. A model of turbocharger radial turbines appropriate to be used in zero- and one-dimensional gas dynamics codes for internal combustion engines modelling. Energy Convers. Manag. 2008, 49, 3729–3745. [CrossRef]

[15] Galindo, J.; Fajardo, P.; Navarro, R.; García-Cuevas, L.M. Characterization of a radial turbocharger turbine in pulsating flow by means of CFD and its application to engine modeling. Appl. Energy 2013, 103, 116–127.
[16] Cao, T.; Xu, L. A Low-Order Model for Predicting Turbocharger Turbine Unsteady Performance. J. Eng. Gas Turbines Power 2016, 138, 072607.
[17] Zungia, Y.S.P. Design of an Axial Turbine and Thermodynamic Analysis and Testing of a ko3 Turbocharger; Massachusetts Institute of Technology: Cambridge, MA, USA, 2011.
[18] Mohammed Hamza Abdulsada," Operational gas turbine swirl combustors design map for pure methane and different outlet configurations", Periodicals of Engineering and Natural Sciences, Vol. 7, No. 4, December 2019, pp.1886-1891.