The Introduction of a Tilted Volute Design for Operation with a Mixed Flow Turbine for Turbocharger Applications

S. P. Lee, M. L. Jupp
Turbocharger Research Institute
University of Huddersfield
Huddersfield, United Kingdom
Sam.lee@hud.ac.uk

A. K. Nickson
BorgWarner Turbo Systems
Bradford, United Kingdom

Abstract* - This paper introduces a tilted volute design for operation with a mixed flow turbine rotor. CFD results show an efficiency gain of up to 1.2% over the standard radial design at the highest tested turbine rotational speed. The efficiency gain was found to be the result of a reduction in separation from the blade suction surface. A reduction in the flow cone angle was also observed for the tilted housing, as a result an increase in negative incidence angles at the blade LE was observed. This work shows that optimization of the turbine housing specifically for mixed flow applications can yield significant performance benefits.

***Keywords* – Turbine Performance, Volute Design, CFD, Mixed Flow Turbine**

I. INTRODUCTION

Environmental issues have, over recent years, become increasingly important. Government legislation has aimed to reduce emissions in many industries. This is of particular importance in the automotive sector, as it is estimated that light duty vehicles contribute approximately 15%, and heavy duty vehicles approximately 6%, to the EU's total CO_2 emissions [1].

In an effort to reduce emissions, turbocharging has become a desirable option, not only in diesel powered engines, but increasingly in gasoline engines too. The main aim of turbocharging is to improve performance and efficiency over a wide range of operation by utilizing waste energy in the exhaust gas.

The purpose of the turbine is to extract energy from the high temperature exhaust gas to increase cylinder inlet pressure through the use of a compressor. In turbocharger applications, radial turbines are almost exclusively used. However, mixed flow turbines have received a significant amount of attention due a number of potential performance benefits over their radial counterpart.

It is widely documented in literature that the optimum incidence angle for radial turbines occurs between -10° and -40° [2]. In order to satisfy radial stacking of the blade fibres, the blade inlet angle must be set to zero degrees for a radial turbine. Alternatively, mixed flow turbines can achieve non-zero blade angles while maintaining radial stacking of the blade fibres. As a result, mixed flow turbines effectively operate like radial turbines with swept back blades. This is achieved through the introduction of a blade cone angle and the presence of a blade camber angle at the LE, this effect is reported in detail by [3] and [4].

As stated by [5], not only is the introduction of the blade cone angle enough, but a corresponding flow cone angle is also necessary for the mixed flow effect to take place. Without an axial flow component at the rotor inlet the mixed flow effect cannot be achieved. The resulting blade angle is given by –

$$tan\beta_B = tan\phi \cdot cos\lambda_{flow} \qquad (1)$$

Where β_B denotes the blade angle, ϕ the blade camber angle and λ_{flow} the flow cone angle. Therefore the stator design in a mixed flow turbine stage must produce the necessary inlet conditions; otherwise the advantages outlined above cannot be realized.

Due to a non-zero blade angle, mixed flow turbines have been found to achieve peak efficiency at lower U/c_s values than radial turbines [6]. This is of importance, as during the turbine inlet pulsations, the blade speed remains almost constant [7], but a significant variation in inlet pressure and velocity is experienced. The maximum energy in the exhaust pulse is available at higher pressure ratios that result in low U/c_s values [8]. High efficiency at these points in the pulse can potentially improve energy extraction.

Mixed flow turbines have also been shown to result in reduced flow separation at the shroud due to a reduction in flow turning through the rotor over radial wheel designs [9]. Furthermore, mixed flow turbines generally have lower inertia than radial turbines due to significant reduction in wheel mass and hence drastically improving transient response as shown by [8].

Only a small number of previous studies have looked at the specific requirements of the turbine stator for mixed flow applications. A mixed flow stage with a trapezoid volute was tested by [10], the nozzle was extended in an attempt to align the hub and shroud contours with the blade LE. The authors estimated the flow cone angle at the blade LE from the design layers through the meridional plane of the rotor, stating that these layers could be thought of as streamlines. Through this approach the flow cone angle was shown to vary significantly. To counteract this, the blade camber angle was varied to give a constant blade angle at the inlet. Investigation of the resulting flow cone angle from CFD simulation showed that the streamlines deviated from the design layers. This was particularly significant at the shroud edge and the authors stated that the flow was unable to follow the strong curvature at the shroud contour.

A 3-dimensional nozzle vane ring was introduced by [11] in an attempt to supply optimum inflow characteristics to the mixed flow rotor. The steady state results showed an improvement in efficiency over a nozzleless design at higher vane angle settings. However, the unsteady performance showed that the nozzle ring resulted in substantial deviations in performance from the steady assumption. This was put down to the nozzle ring creating a second volume within the stator region that had a significant effect of the filling and empty characteristics.

While only a limited number of published studies on mixed flow stator design exist, these works indicate that optimization of the stator design specifically for mixed flow turbines can yield substantial performance improvements. It is therefore vital to consider the requirements of a mixed flow turbine during the volute design process.

This paper introduces a tilted volute design for operation with a mixed flow turbine. This design aims to introduce an improvement in flow cone angle across the LE and reduce losses associated with flow separation along the shroud contour. The new design flow characteristics are compared with that of a standard radial housing.

A. Mixed Flow Turbine Flow Angle Definition

As shown by [5] the axial flow component influences the inlet blade angle through the flow cone angle. In much the same way, the inlet flow angles of a mixed flow turbine are influenced by the axial flow component. For a radial turbine the flow enters the turbine purely in the radial plane. As such the absolute and relative flow angles can be easily calculated from the velocity components. In a mixed flow turbine, flow enters the rotor in a plane inclined to the flow cone angle, where a 90° angle is that of the purely radial operation.

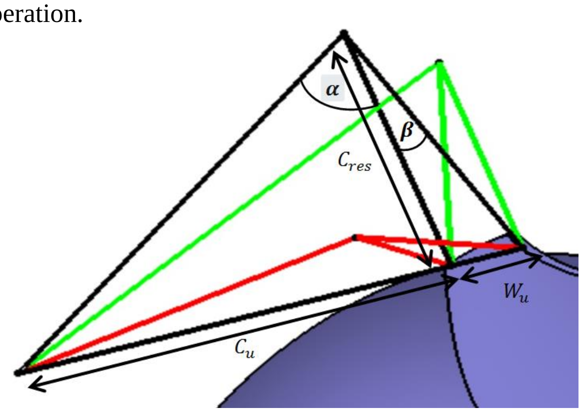


Figure 1 - Mixed flow turbine velocity triangles

Figure 1 shows the velocity components at the LE of a mixed flow turbine. The radial velocity components are shown in green, the axial components in red and the resultant components in black. The resultant velocity components are dependent on both the radial and axial velocities. The inlet flow angles are therefore dependent on both the axial and radial velocity components and care must be taken to calculate the angles correctly. The relative and absolute flow angles for a mixed flow turbine are therefore given by:

$$\beta = \tan^{-1} \frac{w_u}{C_{res}} \tag{2}$$

$$\alpha = \tan^{-1} \frac{C_u}{C_{res}} \tag{3}$$

II. INTORDUCTION OF A TILTED VOLUTE DESIGN

Volute design is driven by the ratio of the cross sectional area to the radius of the volute passage centre (A/r ratio). The A/r ratio controls the rotor inlet flow angle and usually varies linearly around the volute passage [12] and [13]. This relationship is based on one-dimensional, incompressible and free vortex flow. While numerous studies have shown the inaccuracies of these assumptions [13], [14], [15] and [16], this approach is still widely used in volute design.

To design a volute for use with a mixed flow rotor a ‘tilt’ angle was introduced. The aim of this geometric change is to improve the flow cone angle and reduce flow turning through the stage. It has was shown by [9] that the reduction in turning associated with mixed flow turbines can reduce shroud separation in the blade region. However, if the housing is positioned radially, the flow must still turn through 90° within the stage. As shown by [10] significant separation can still occur due to the shroud curvature. Tilting the housing to reduce flow curvature, particularly along the shroud contour, reduces losses and provides a larger flow cone angle to improve inlet flow conditions. The new design is defined in Figure 2.

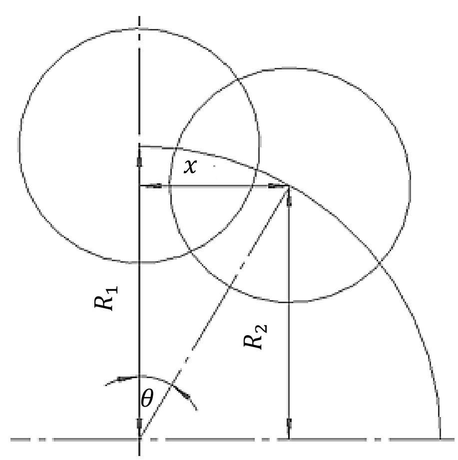


Figure 2 - Tilted volute definition

The tilted volute is positioned at an angle of θ to the radial volute. The axial position (x) of the tilted housing in relation to the radial housing at the base section is defined as:

$$x = sin\theta R_1 \tag{4}$$

$$R_2 = cos\theta R_1 \tag{5}$$

With progression around the volute azimuth angle the axial position (x) changes. This can be expressed as -

$$x^* = sin\theta r_1 - (r_1 - r^*) \tag{6}$$

Where x^* is the axial distance at a given azimuth angle, r_1 is the volute cross sectional radius at the base and r^* is the volute cross sectional radius at a given azimuth angle. This method produces a spiraled volute centroid.

Tilting the volute reduces the radius from R_1 to R_2 as illustrated in Figure 2. Clearly, increasing the volute tilt angle causes a larger change in volute radius. The implication of this is that the volute A/r increases. From a design perspective this can mean one of two things:

That the volute area can be reduced correspondingly to maintain a constant A/r, therefore reducing frictional and dissipative losses.

The volute area can remain constant resulting in a larger A/r, this can improve inlet flow angles without increasing frictional and dissipative losses associated with a larger volute area.

The tilted volute design has therefore introduced an extra degree of freedom. In the current study the volute area remains constant and the A/r experiences a small increase as a result of the tilt. The tilt angle (θ), of the volute throughout this study was kept equal to the blade cone angle.

III. COMPUTATIONAL SET-UP

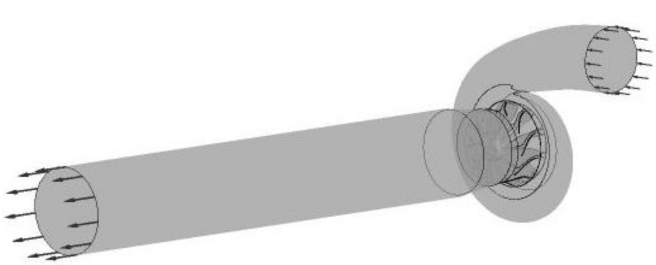


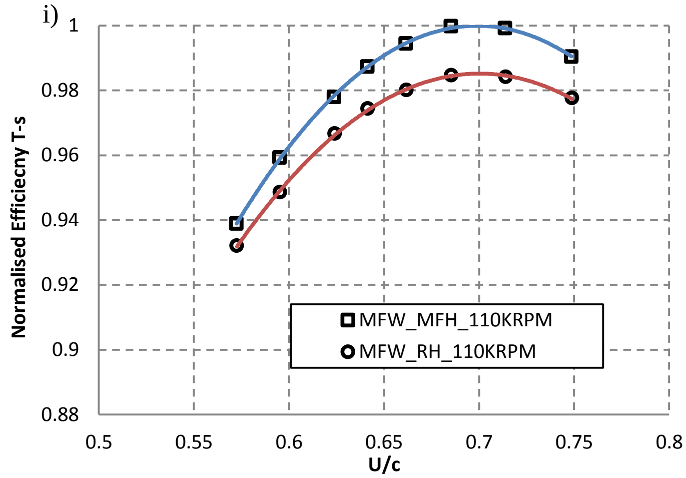
Figure 3 - Computational domain set-up

The computational set-up is shown in Figure 3, this consists of three domains, the volute, the rotor and the outlet. A full domain specific mesh study was completed similar to that recommended by [17], this was extended to include a full boundary mesh independence study. The Y+ throughout the model was kept below 3. The volute and outlet regions were meshed using an unstructured tetrahedral mesh developed in ICEM CFD. The rotor region consists of a structured hexahedral mesh completed in ANSYS TurboGrid. The computational set-up was carefully validated against experimental gas stand testing. All CFD simulations were completed in CFX 14.5 using the Shear Stress Transport (SST) turbulence model, with the frozen rotor approach to account for the turbine rotation which was shown to be valid by [18]. The convergence criterion used was when the RMS residuals fell below $1e^{-4}$ and the efficiency values fluctuated by less than 0.05%.

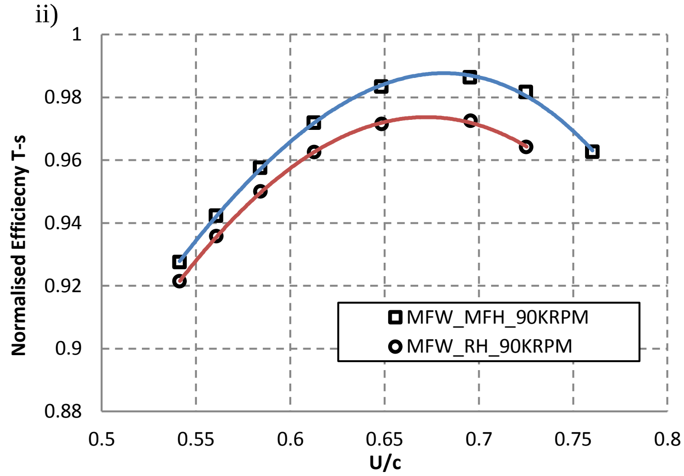
IV. TILTED VOLUTE RESULTS

The tilted housing design was tested at three turbine speeds, 70KRPM, 90KRPM and 110KRPM. Two turbine housings were tested; a standard radial housing and the new tilted housing design. The efficiency results are presented in Figure 4 i), ii) and iii).

i)



ii)



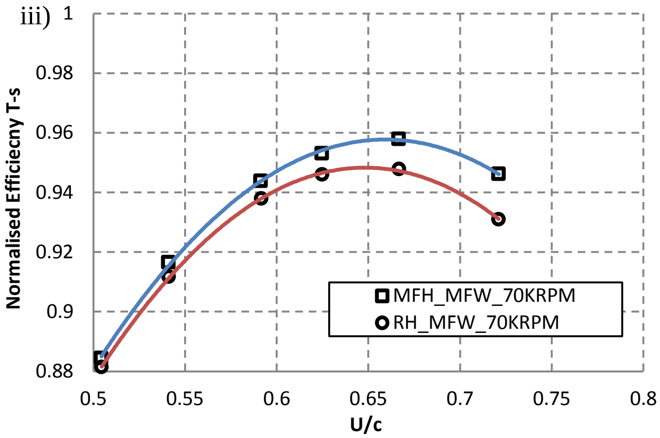


Figure 4 - Normalised Efficiency for Radial and Tilted housings. i) 110KRPM ,ii) 90KRPM, iii) 70KRPM

The tilted housing results in higher peak efficiencies than the radial housing stage at all tested turbine speeds with a maximum efficiency improvement of 1.2% at 110KRPM.

The breakdown of volute and rotor efficiencies for the 110KRPM turbine speed are shown in Figure 5 and Figure 6 respectively. With decreasing U/c_s the volute efficiency increases; [19] and [20] found a similar result with volute losses decreasing almost linearly with increasing mass flow rate. The tilted volute efficiency was found to be less than that of the radial volute at any given U/c_s.

The improvement in stage efficiency for the tilted housing was found to occur in the rotor region. A maximum efficiency improvement of approximately 1.45% is achieved in the rotor domain of the tilted housing. This efficiency gain was found to be primarily a result of reduced separation on the blade suction surface in the tilted housing design. Figure 7 shows contour plots of static entropy in the blade passage for both designs at the 0.7 and 0.8 span positions. Flow separation on the blade suction surface is delayed to larger spans for the tilted housing design. A reduction in LE separation was also observed for the tilted housing design particularly at lower span positions due to a variation in blade camber over the LE.

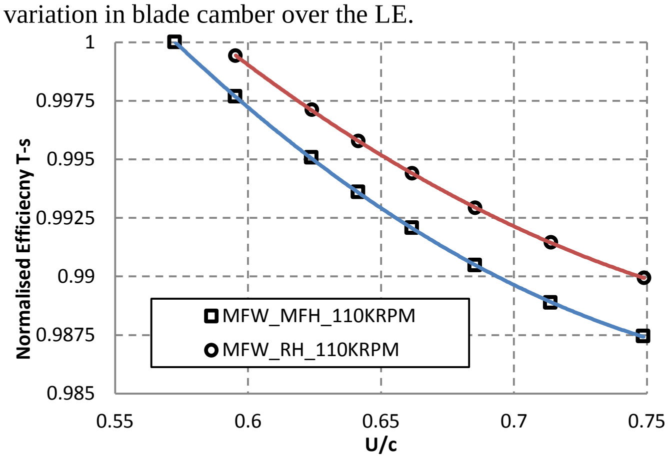


Figure 5 - Normalised volute efficiencies for the tilted and radial volute designs at 110KRPM.

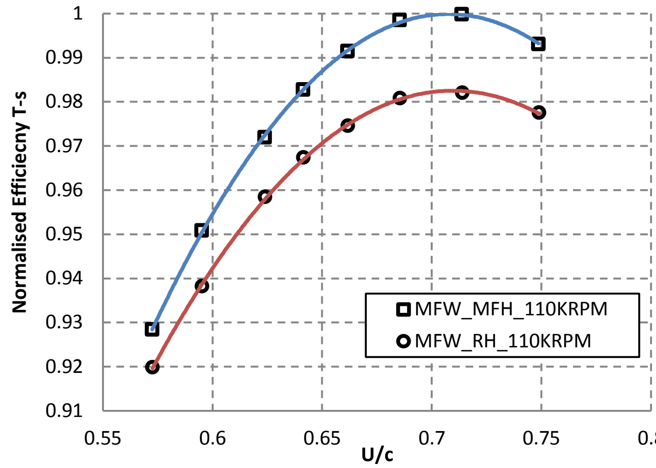


Figure 6 - Normalised rotor efficiencies for the tilted and radial volute designs at 110KRPM.

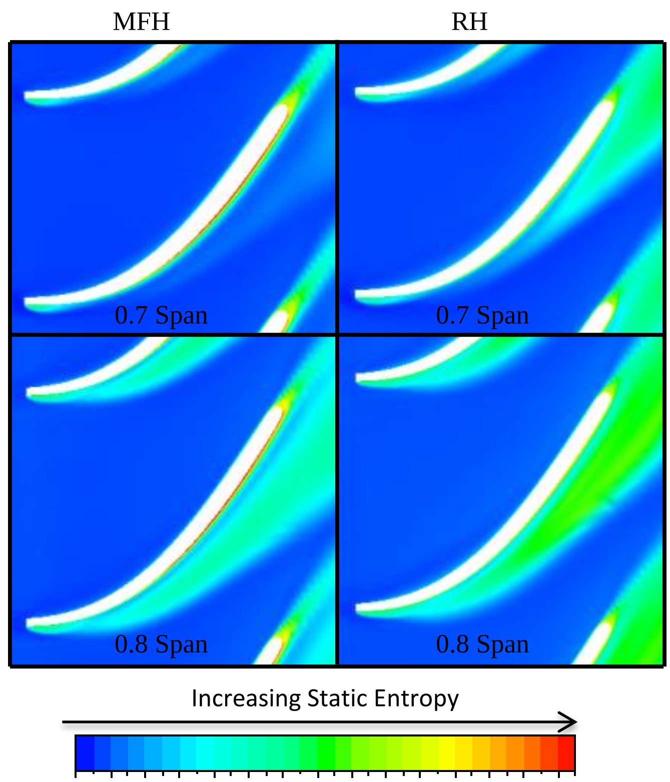


Figure 7 – Static entropy contours through the blade passage for both the tilted and radial designs at the peak efficiency point, 110KRPM.

Figure 8 shows the flow cone at the LE of the blade for both the radial and tilted housing configurations. The flow cone angle at the LE was found to be insensitive the operating pressure ratio. The tilted housing design produced smaller flow cone angles at all positions across the LE than that of the radial housing. The flow cone angle distribution of the radial housing follows a similar contour as that observed by [10] with a distinct increase in flow cone angle near to the shroud. It should be noted that the blade cone angle in the current study is substantially smaller than that tested by [10]. The tilted housing design gave a more linear variation in flow cone angle at the LE with the flow cone angle decreasing continuously up to the shroud.

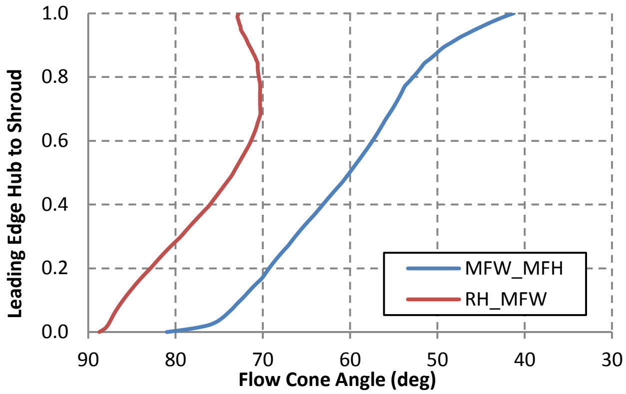


Figure 8 - Flow cone angle distribution at LE for the tilted and radial volute designs.

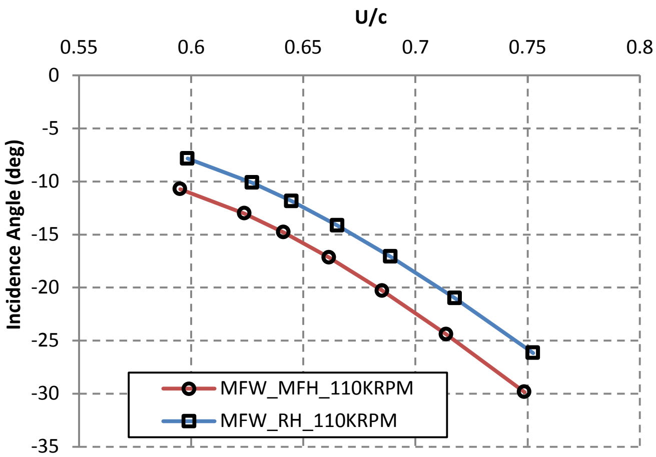


Figure 9 - Incidence angles at LE at for the tilted and radial volute designs 110KRPM.

The resulting incidence angles at the LE are presented in Figure 9. The incidence was calculated from the LE average cone and camber angles. It was observed that the tilted housing results in more negative incidence angles at all tested U/c_s due to an increase in the axial flow component. However, a shift in peak efficiency U/c_s was not observed. This result indicates that while the LE incidence was found to improve, other losses must dictate the optimum U/c_s.

V. Conclusion

This paper has introduced a titled volute design for operation with a mixed flow turbine wheel. It was found that the tilted housing design resulted in efficiency gains at all tested turbine speeds with a maximum efficiency increase of 1.2% at 110KRPM. This efficiency improvement was primarily a result of the reduced flow separation from the blade suction surface towards the shroud. The flow cone angle generated by the tilted housing was found to be significantly reduced when compared to the radial housing, leading to a reduction in incidence angle across the turbine map.

Further work will be undertaken to develop the tilted housing design further and to investigate the effect of different tilt angles and shroud contours. Pulsed gas stand testing will also be completed in order to validate the results presented in this study.

Acknowledgment

The authors would like to thank BorgWarner Turbo Systems for their support throughout this project.

Nomenclature

β_B	Blade Angle
ϕ	Blade Camber Angle
λ_{flow}	Flow Cone angle
U/c_s	Velocity Ratio
β	Inlet Relative Flow Angle
α	Absolute Flow Angle
C_u	Absolute Tangential Velocity
W_u	Relative Tangential Velocity
C_{res}	Resultant Flow Cone Velocity
θ	Volute Tilt Angle
SST	Shear Stress Transport
RMS	Root Mean Squared
CFD	Computational Fluid Dynamics
MFW	Mixed Flow Wheel
MFH	Mixed Flow Housing (Tilted Housing)
RH	Radial Housing
LE	Leading Edge
T-s	Total to static

References

[1] European-Commission. *Road Transport: Reducing CO2 emisions from vehicles*. 2015.

[2] Japikse, D. and N.C. Baines, *Introduction to Turbomachinery*. 1994, Oxford: Oxford University Press.

[3] Rajoo, S. and R. Martinez-Botas, *Mixed Flow Turbine Research: A Review*. Journal of Turbomachinery, 2008. 130(4): p. 044001.

[4] Lüddecke, B., D. Filsinger, and J. Ehrhard, *On Mixed Flow Turbines for Automotive Turbocharger Applications.* International Journal of Rotating Machinery, 2012. 2012.

[5] Leonard, T., et al., *A Numerical Study of Automotive Turbocharger Mixed Flow Turbine Inlet Geometry for Off Design Performance.* International Conference on Pumps and Fans with Compressors and Wind Turbines, 2013. 6th Proceedings.

[6] Watson, N. and M.S. Janota, *Turbocharging the internal combustion engine*. 1982, London: Macmillan.

[7] Szymko, S., R. Martinez-Botas, and K. Pullen. *Experimental evaluation of turbocharger turbine performance under pulsating flow conditions*. in *ASME Turbo Expo 2005: Power for Land, Sea, and Air*. 2005. American Society of Mechanical Engineers.

[8] Roclawski, H., M. Bohle, and M. Gugau, *Multidisciplinary Design Optimization of a Mixed Flow Turbine Wheel.* ASME Turbo Expo, 2012.

[9] Palfreyman, D. and R. Martinez-Botas, *Numerical study of the internal flow field characteristics in mixed flow turbines.* ASME, 2002.

[10] Leonard, T., et al., *A Numerical Study of Inlet Geometry for a Low Inertia Mixed Flow Turbocharger Turbine.* ASME Turbo Expo, 2014.

[11] Rajoo, S. and R. Martinez-Botas, *Variable Geomtry Mixed Flow Turbine for Turbochargers: An Experimental Study.* International Journal of Fluid Machinery and Systems, 2008.

[12] Chapple, P.M., P.F. Flynn, and J.M. Mulloy, *Aerodynamic Design of Fixed and Variable Geomtry Nozzleless Turbine Casings* Journal of Engineering for Power, 1980.

[13] Gu, F., A. Engeda, and E. Benisek, *A comparative study of incompressible and compressible design approaches of radial inflow turbine volutes.* Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 2001. 215(4): p. 475-486.

[14] Hussain, M. and F.S. Bhinder, *Experimental study of the performance of a nozzle0less volute casing for a turbocharger turbine.* SAE International, 1984.

[15] Scrimshaw, K.W., T., *Size Effects in Small Radial Turbines.* ASME, 1984.

[16] Benisek, E.S., W., *Laser Velocimeter Mesurements at the Rotor Inlet of a Turbocharger Radial Inflow Turbine* ASME, 1987. 55.

[17] Galindo, J., et al., *Set-up Analysis and Optimisation of CFD Simulations for Radial Turbines.* Engineering Application of Computational Fluid Mechanics, 2014.

[18] Aymanns, R.S., J; Uhlmann, T; Luckmann, D, *A Revision of Quasi Steady State Modeling of Turbocharger Turbines in the Simulation of Pulse Charged Engines.* Aufladetechnische Konferenz Dresden, 2011.

[19] Abidat, M., et al. *Design and flow analysis of radial and mixed flow turbine volutes*. in *European Conference on Computational Fluid Dynamics*. 2006.

[20] Benson, R.S.C., W. G.; Das, S. K., *An Investigation of the Losses in the Rotor of a Radial Flow Gas Turbine at Zero Incidence Under Conditions of Steady Flow*. Proceedings of the Institution of Mechanical Engineers, 1968.