Energy Conversion and Management 98 (2015) 449–462

Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

Influence of pulsating flow frequencies towards the flow angle distributions of an automotive turbocharger mixed-flow turbine

M.H. Padzillah [a,b,*], S. Rajoo [a], M. Yang [b], R.F. Martinez-Botas [b]

[a] UTM Centre for Low Carbon Transport in Collaboration with Imperial College London, Universiti Teknologi Malayisa, 81300 Skudai, Malaysia
[b] Turbocharger Research Group, Imperial College London, SW7 2AZ London, United Kingdom

ARTICLE INFO

Article history:
Received 10 November 2014
Accepted 6 March 2015
Available online 18 April 2015

Keywords:
Pulsating flow
Turbine
Computational Fluid Dynamics
Flow angle

ABSTRACT

This paper aims to provide a detailed understanding of the flow interaction within the complex geometry of turbine stage coupled with reflection and superposition of the imposed unsteady pressure waves. The defining performance characteristic of a turbomachines is the inlet flow angle at its leading edge, which in most cases limit its optimum operational range. Pulsating flow field, typical to turbocharger turbine due to opening and closing of the exhaust valves, further deteriorates the flow angle distribution around its circumference, hence performance. To investigate this phenomena and how it differs from the steady state equivalent, a fully validated 'cold flow' Computational Fluid Dynamics (CFD) analysis was carried out. For this purpose, 4 unsteady CFD simulations of 20 Hz, 40 Hz, 60 Hz and 80 Hz together with the steady state simulations have been conducted at 30,000 rpm turbine speed. Discussions in this paper aim to provide a better insight on the flow angle behaviour in pulsating flow field. Evidences are shown where flow angle swings over a wide range (300% more than the steady state condition) in pulsating flow field as it propagates through the guide vanes. It was also found that the flow angle fluctuations during pressure drop period is significantly lower as compared to that during pressure increment period by (19°). This work also leads to a recommendation of correction factor in applying quasi-steady assumption of the rotor wheel. Without this correction, the prediction results could lead to underestimation of mass flow parameter up to 8% at high pressure ratio.

1. Introduction

It is now a common practice to turbocharge engines to achieve higher power-to-weight ratio by increasing the density of the inlet air. This allows more fuel to be injected and burned, and more power delivered thus leading to engine downsizing. However, according to Watson and Janota [1], turbo machines such as turbines can only performs efficiently at a limited operation range which depends heavily on the incidence flow angle at the rotor inlet. Japikse and Baines [2] indicated that the optimum incidence angle that results in the smallest lost due to flow separation on the blade surfaces is between -20° and -30°. It is well known that the axial turbine is capable of operating more efficiently than the radial turbine but suffer massive efficiency penalty as its size is reduced. For that reason, in the automotive turbocharger, radial turbine is commonly used as compared to axial turbine due to the under hood space limitation. However, the application of radial turbine has introduced additional geometrical constrains for the turbine designer in a way that the inlet blade angle is not adjustable due to the radial fibre requirement[1]. Spence et al. [3,4] has performed numerical investigations for a non-radial fibre design for radial turbine. However its application of non-radial fibre turbine is currently limited in the market. As the result of this limitations, it is difficult to match the incidence angle at the rotor inlet to its optimum value. In the automotive turbocharger application where pulsating flow is imposed to the turbine inlet, it is desirable to have optimal operating efficiency at high energy/high pressure instances in a pulse. This has led to the application of mixed flow turbine instead of a pure radial turbine.

A mixed flow turbine is capable of working more efficiently at higher pressure ratio compared to radial turbine due to its ability to be adapted to the incoming flow angle by modifying its inlet blade angle. A comprehensive review of mixed flow turbine is

* Corresponding author at: UTM Centre for Low Carbon Transport in Collaboration with Imperial College London, Universiti Teknologi Malayisa, 81300 Skudai, Malaysia.

E-mail addresses: hasbullah@mail.fkm.utm.my (M.H. Padzillah), srithar@fkm.utm.my (S. Rajoo), m.yang09@imperial.ac.uk (M. Yang), r.botas@imperial.ac.uk (R.F. Martinez-Botas).

[1] This requirement is necessary in order to limit excessive bending stress on the rotor blade due to high centrifugal load during its operations

http://dx.doi.org/10.1016/j.enconman.2015.03.028
0196-8904/

Nomenclature

Greek Symbols
ρ density
γ specific heat ratio
Λ Lambda parameter
∇ vector differential operator
ν kinematic viscosity
Π Pi parameter

Roman Symbols
L_0 domain length
$\dot{m}$ mass flow rate
N rotational speed
P pressure
St Strouhal number
T temperature
t time
t_0 domain residence time
$\langle U \rangle$ mean flow velocity
U_0 domain velocity
U flow velocity
x location along streamwise direction

Subscripts
01 total condition at inlet

made available by Rajoo and Martinez-Botas [5]. It offers turbine designers the ability to alter the inlet blade angle to suit their applications, while still maintaining the radial fibre requirement. The improvements that have been discovered therefore highlight the importance of flow angle towards the turbocharger turbine performance. However, in-depth analysis for the flow angle distribution within the turbine stage under pulsating flow conditions is still lacking. Such analysis could lead to further advancement of turbine design in order to take advantage of the incoming pulsating flow. Flow angle distribution could be studied through experiments, but one of the limiting factor is the small size of the turbine assembly which made the experimental measurement of flow angle rather difficult. Karamanis [6] utilized the LDV approach to investigate the flow angle in a vaneless mixed flow rotor inlet and recorded aggressive changes in the flow angle throughout the pulses (-60° to 90°). This finding has prompted Palfreyman [7–9] to conduct a full-stage CFD simulations to further the understanding in [6]. This enables in-depth flow visualizations during pulsating flow conditions but unfortunately only 40 Hz flow frequency was simulated due to limited computational resources.

Rajoo [10] in 2007 has opted to modified a vaneless volute designated Holset H3B to accommodate a nozzle vane row upstream the rotor inlet. Rajoo [10] also proposed a new vane concept that has a lean geometry in order to match the cone angle of the mixed flow turbine. The application of nozzles ultimately changed the flow field entering the turbine. Although Rajoo [10] conducted extensive testing, there is currently no analysis involving the flow angle due to experimental limitation as described earlier. Chiong [11,12] extended the work of Rajoo [10] and performed 1D numerical calculations to predict the turbine performance and 'filling and emptying' phenomena using ONDAS, originally developed by Costall [13]. Chiong [11] was able to predict the mass flow hysteresis loop with sufficient accuracy thus ensuring that the model and experimental data are correct. The work of Rajoo [10] has also opened a platform for a full 3D numerical investigation on a single entry vaned mixed flow turbine. Padzillah et al. [14] modeled this configuration and shown several interesting analyses, but limited to only a single flow frequency of 20 Hz. This prevented any further analysis with regards to the relationship of flow angle and frequency. Padzillah et al. [14,15] later extended their work to include different turbine speed and flow frequency. In this work, relationship of the speed and the turbine performance under pulsating flow was clearly shown. However there were no direct relationship of the flow frequency and turbine performance attributed to insufficient data set (only 2 frequencies were modeled). This issue is addressed in the current work where intermediate frequencies are added in order to visualize its effect on the evolution of flow angle, and as such, the turbine performance.

This paper provides visual explanation of the flow angle behaviour in a single entry vaned mixed flow turbine during pulsating flow operations. This is achieved by simulating 4 different frequencies (20 Hz, 40 Hz, 60 Hz, and 80 Hz) at a constant turbine speed of 30,000 rpm. These frequencies corresponds to 800 rpm, 1600 rpm, 2400 rpm and 3200 rpm of a 4-stroke, 3-cylinder engine connected to a single exhaust manifold. Simulations of different flow frequencies enable general characterization of the flow angle behaviour at least for this particular type of turbine.

2. Research background

This section details the experimental and numerical works contributed to this paper. Even though work in this paper is dominantly numerical, the experimental data from [10] is invaluable to validate the simulation. Thus it is felt that a brief introduction on the experimental facility is necessary. The full explanation with regards to the experimental works can be found in [10].

The numerical and experimental works are setup for a representative cold-flow condition. This is done to compliment the turbocharger test-rig facility available at Imperial College London. Gaussmann in 1972 developed the 'similarity approach' that is used to scale down the actual turbine temperature to the testing temperature and maintaining the isentropic power available to the turbine. This relationship is shown in Eqs. (1) and (2).

$$\left(\frac{\dot{m}\sqrt{T_{01}}}{P_{01}}\right)_{test\_rig} = \left(\frac{\dot{m}\sqrt{T_{01}}}{P_{01}}\right)_{actual} \tag{1}$$

$$\left(\frac{N}{\sqrt{T_{01}}}\right)_{test\_rig} = \left(\frac{N}{\sqrt{T_{01}}}\right)_{actual} \tag{2}$$

2.1. Experimental setup

Schematic of the cold-flow turbocharger test facility and its instrumentations are shown in Fig. 1. Compressed air for the test rig is supplied by three screw-type compressors that can deliver up to 1 kg/s at maximum absolute pressure of 5 bars. Two successive heaters are used to heat the gas up to 345 K, mainly to prevent condensation during turbine expansion. Subsequently, the flow is channelled into two limbs with individual diameter of 81.4 mm, thus enabling experiments for single entry and twin or double entry turbines [19,20]. For single entry turbine in the current