Proceedings of the 1st Global Power and Propulsion Forum
GPPF 2017
Jan 16-18, 2017, Zurich, Switzerland
www.pps.global

GPPF-2017-54

UNSTEADY BEHAVIOURS OF A VOLUTE IN TURBOCHARGER TURBINE UNDER PULSATING CONDITIONS

Mingyang Yang
Shanghai Jiao Tong University
myy15@sjtu.edu.cn
Shanghai, China

Ricardo Martinez-Botas
Imperial College London,
r.botas@imperial.ac.uk
London, UK

Rajoo Srithar
UTM Centre for Low Carbon Transport in Cooperation with Imperial College London
srithar@fkm.utm.my
Johor, Malaysia

Seiichi Ibaraki
Nagasaki Research and Development Centre
seiichi_ibaraki@mhi.co.jp
Nagasaki, Japan

Takao Yokoyama
Nagasaki Research and Development Centre
takao_yokoyama@mhi.co.jp
City, State, Country

Kangyao Deng
Shanghai Jiao Tong University
kydeng@sjtu.edu.cn
Shanghai, China

ABSTRACT

Turbochargers are currently in their prime utilization period, which directly pushes for performance enhancement from conventional turbochargers and more often than not revisiting design methodology. It is well known that turbocharger turbine is subjected to pulsating flow, and how this feeds a steady-flow-design volute is a topic of interest for performance enhancement. This paper investigates the unsteady effect on the flow characteristic in a turbine volute under pulsating flow conditions by numerical method which is validated by experimental measurement. A single sinusoidal pulse is imposed at the turbine inlet for the instigation of unsteady behaviours. Firstly, pulse propagation along the volute passage, including pressure, temperature and mass flow rate, is studied by the validated numerical method. Unsteady effect by the pulsating flow on the flow angle upstream the rotor inlet is confirmed. The mechanism of this unsteady effect is understood by an analytical model and clearly demonstrates two factors for the flow angle distributions: the configuration of the volute A/R and the unsteady effect resulted from the pulse gradient. This paper demonstrates the unsteady behaviour of the turbine volute under pulsating conditions and the mechanism is unveiled, which can lead to improvement of the volute design methodology under pulsating flow conditions.

INTRODUCTION

Engine downsizing becomes a prevailing method to improve the fuel economy of Internal Combustion Engine. It is achieved via the reduction of displacement of cylinders while maintaining an equivalent power with the original engine by increasing the power density of the engine. One of the key enablers for the downsizing is turbocharging [1]. It pushes for demand of performance improvement from conventional turbochargers and more often than not revisiting its design methodology. A turbine of a turbocharger is an important component which recovers energy from the exhaust gas of an engine. This component is inevitably subjected to pulsating flow due to reciprocating internal combustion engine. How this fact feeds a steady-flow-design volute is a topic of interest for its performance enhancement.

Extensive researches have been carried out on turbine behaviour under pulsating conditions and results clearly demonstrated deviation of the performance from the equivalent steady performance [2-6]. The rotor extracts energy from the exhaust flow, thus stands in the centre of research topics. On the other hand, the volute, which feeds the flow into the rotor in a desired angle in order to optimize the performance, plays a key role of determining unsteady behaviours of the turbine. It is the volume of the volute that dominates the filling and emptying process of the turbine under pulsating conditions [7, 8]. Larger volume results in enlarged hysteresis loops of the turbine performance as the phase difference of those two processes are enhanced by the volume. The deviation of hysteresis loops from the steady performance is an indication of mass imbalance which is considered to be a reasonable parameter of unsteadiness evaluation [9]. Except for the volume, the length of the volute passage is another important factor influencing the unsteady behaviour. It was confirmed that an evident phase shift exists between the input energy and out power. Half length of the volute passage was suggested for the evaluation of the phase shift between the input isentropic energy at volute inlet and the output power from the rotor [10, 11]. This fact indicates the route of the pulse propagation in the volute before being absorbed by the rotor. However, the propagation of flow parameters or their unsteady behaviours which are related with the energy are unclear yet. Furthermore, researches of the topic mainly focuses on the gas dynamics effect as the filling and empting resulted from configurations of a volute, limited documented investigations have been carried out on the influence of pulsation on detailed flow in the volute. Instead, detailed flow analysis in turbine volute was mainly carried out at steady flow conditions [12-14].

Design procedures of turbine volute based on steady conditions have been developed over few decades [11, 12]. It is well known that the geometrical parameter A/R of a volute is the key parameter determining the flow direction at the volute exit thus has a significant impact on the turbine performance. The assumptions, such as that the flow follows the rule of angular momentum conservation (free vortex) and the uniformly distributed mass flow rate in annulus, result in linearly distributed A/R in circumferential direction, which is the widely accepted guideline for volute design [13-15]. Considering that those assumptions are all based on steady flow conditions, it is necessary to re-evaluate them under pulsating conditions and their influence on the design guideline if there is any.

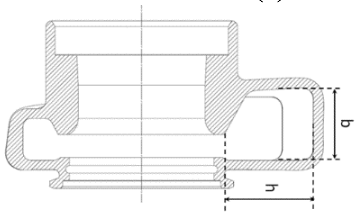
The current paper aims to study unsteady behaviours of a turbine volute under pulsating conditions through validated CFD method. Detailed flow parameters in the volute are analysed and an analytical model is built for understandings of the mechanism. The investigation demonstrates the unsteady influence of the pulse on flow parameters in the volute, which may shed light on the tailored design method of the volute targeting at turbine performance improvement under pulsating conditions.

NUMERICAL METHOD AND VALIDATION

Please follow the format above for the Title, header, paper number, author info, etc.

A MHI radial turbine in a turbocharger is employed for the investigation. The rotor, the volute and its cross section are shown in figure 1. There are 11 blades in the rotor. Main geometrical parameters are list in Table 1.

(a) radial turbine rotor (b) volute



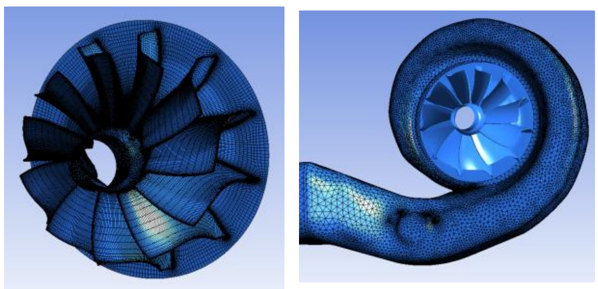
(c) cross section of the volute

Figure 1. the turbine rotor and volute

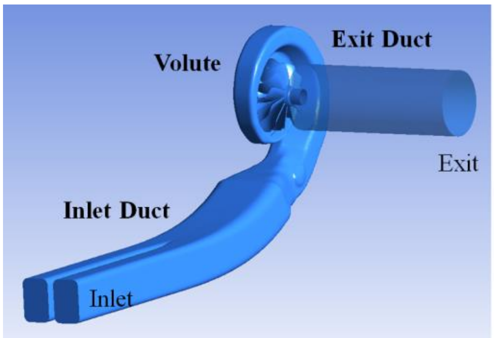
Table 1 Main geometries of the rotor

Rotor geometries	
Inlet diameter of rotor	74.0 mm
Blade height of the rotor inlet	12.0 mm
Hub diameter at rotor exit	21.0 mm
Tip diameter at rotor exit	64.9 mm
Blade number	11

Computational fluid dynamic (CFD) method is employed for detailed flow investigation on the volute under steady and pulsating conditions. Domains from the measurement plane to the exit pipe, including inlet ducts, the volute, the rotor and the exit pipe, are all modelled for the investigation. Configurations such as blade fillets and sculptures are not modelled for the convenience of meshing. The volute and the ducts are meshed by unstructured meshes for convenience and high mesh quality, while the rotor is meshed by structured ones via the TurboGrid module. There are 4.5 million nodes in all domains in total, where there are about 800k nodes in the volute and about 3 million nodes in the rotor which can well guarantee the grid independence. The computational meshes are shown in figure 2.



(a) Mesh of rotor (b) Mesh of volute



(c) Computational domains of the turbine
Figure 2. Meshes of the turbine for CFD Analysis

The instantaneous total pressure and temperature from the measurement are given as the inlet boundary conditions; the instantaneous static pressure is imposed as outlet boundary condition. Adiabatic, non-slip wall condition is applied for solid wall surfaces. 'Frozen rotor' method is applied for the treatment of rotor/stator interface between the volute and the rotor, which can reduce the amount of computational time immensely compared to fully transient interface (sliding mesh) treatment. The flow phenomenon, such as the unsteady disturbance caused by the tongue, in the rotor is difficult to be accurately predicted because the relative location of the rotor to the volute is assumed to be 'frozen' in this method. However, unsteady behaviours of the volute can be well captured because it is much less influenced by the unsteady interaction. As the flow in the volute is the interest of the current investigation where the rotor can be recognized as a downstream boundary to some extents, the method can well meet requirements of the reliability of the simulation. Furthermore, this method has already been proved to be reasonably good for the turbine performance and flow prediction in the turbine [2].

RANS equations were solved by commercial CFD code ANSYS-CFX. Stress Transport (SST) model was used to model the turbulence flow, which has been confirmed as a reliable model for the prediction of turbine performance as well as the flow phenomenon [2, 16]. Dual time stepping method is applied for the unsteady simulation. The physical time step of the iteration is set to be1.5X10-4s, which counts for 112 steps for a pulse period or 45 degrees of rotation for the rotor. It is considered to be rough for the resolution of the flow phenomenon in the rotating passages but adequate enough to resolve the flow evolution in the stationery volute. 20 inner iterations are conducted in each time step to guarantee a reasonable convergence (global residual as 10-4). Well converged steady results are employed as initial conditions of the unsteady calculation. The simulation is considered to be converged when the predicted pulse of flow parameters is well repeated for at least 3 pulse periods.

For the validation of the CFD method, the test was carried out on turbine facilities in Imperial College London. The pressure ratio (PR) and the mass flow rate parameter (MFP) for steady condition are defined as equation (1) and (2):

$$PR = \frac{P_{t-i}}{P_{s-e}} \tag{1}$$

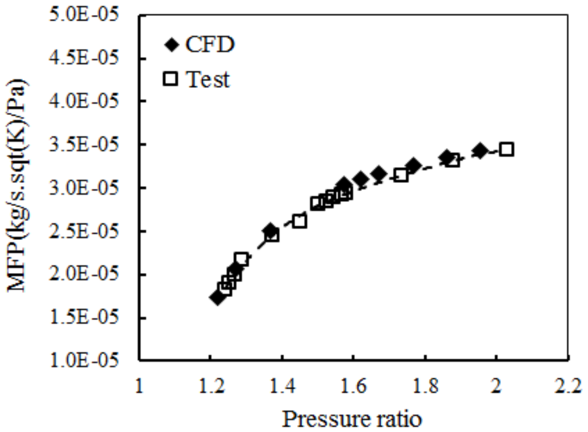
$$MFP = \frac{m\sqrt{T_{t-i}}}{P_{t-i}} \tag{2}$$

The efficiency (η) and the velocity ratio (VR) are defined as equations (3) and (4):

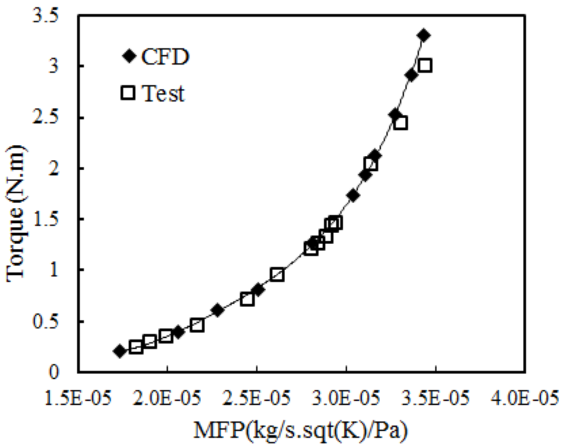
$$\eta = \frac{\tau\omega}{m\sqrt{C_p T_{t-i}(1 - PR^{\frac{\gamma}{\gamma-1}})}} \tag{3}$$

$$VR = \frac{U}{\sqrt{2C_p T_{t-i}(1 - PR^{\frac{\gamma}{\gamma-1}})}} \tag{4}$$

Figure 3 compares the steady performance of the turbine at the rotational speed as 50 kRPM between the experimental measurement and CFD prediction. The predicted swallowing capacity is in high accordance with the experiment over all operation conditions. The maximum discrepancy between two results is less than 1.5%. The predicted output torque matches quite well with the experiment at low load and middle load conditions, but is over predicted moderately at high load conditions. Simplification of configurations in the model, such as fillets at the hub and tip gap, are considered to contribute to discrepancies at high loads. The frozen-rotor treatment and the turbulence model at high load conditions when there is stronger rotor-stator interaction and flow separation in rotor passages may also contribute to the discrepancies. Nerveless, both the swallowing capacity and the output torque are considered to be reasonably well predicted at steady conditions by the numerical method.



(a) Swallowing capacity



(b) Output torque

Figure 3. Turbine performance at 50 kRPM under steady conditions

Figure 4 compares the unsteady locus of swallowing capacity by prediction and the experiment under the pulsating condition at 60 Hz, high load, 50kRPM. Steady performance is also shown in the figure for the comparison. Both the prediction and the experimental results encapsulate the steady curve.

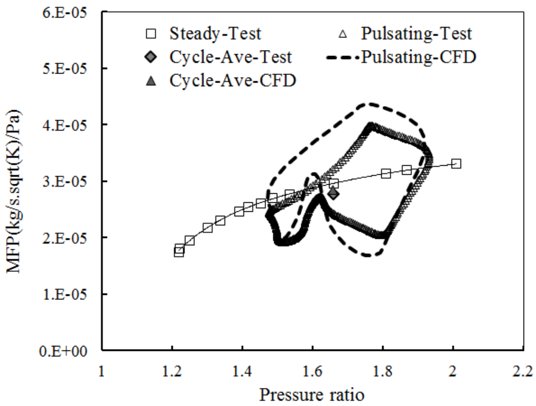


Figure 4. swallowing capacity at 60 Hz, middle load, 50 kRPM

It can be observed that the maximum mass flow rate parameter is moderately over-predicted by about 6%, while values at the tail of the pulse are in satisfied accordance with the experiment. The discrepancy of the magnitude between two results is considered to be caused by several reasons, among which the frozen-rotor method applied for the treatment of the rotor/stator interface is considered to the main one. The unsteady disturbance on the flow in the rotor caused by the volute tongue can't be predicted accurately as the relative location of the rotor is 'frozen', thus the throttling effect of the rotor can't be captured with high quality, especially for high frequency pulsating conditions. Except for the pulse magnitude, the shape of the loop, which results from the wave dynamics in the turbine, is quite similar between two results. Specially, a folder near the tail of the loop which is apparently caused by the wave reflection can be noticed at similar location for both the CFD and the experiment. Furthermore, cycle averaged values which are evaluated via values averaging over the pulse period are also compared. It can be observed that the averaged values are nearly overlapped with each other, indicating a very good matching of two results. It is also noticed that they are both slightly below the steady curve due to the unsteadiness.

According to the comparison between experiment and CFD, unsteady wave dynamics in the turbine is well captured by the numerical method. Discrepancies are observed for the pulse magnitude of mass flow rate, but it is considered to be in a reasonably good range. Furthermore, the flow phenomenon in the volute instead of the rotor is the main interest of the paper, which is considered to be in capability of the numerical method. Therefore, the CFD method is credible for the flow investigation under pulsation conditions.

Pulse propagation in the volute

The volute has the longest length and the largest volume among all components in a turbine, thus is the key player for turbine unsteadiness under pulsating conditions. In order to study conveniently the propagation of a pulse in the volute, a single pulse of pressure and temperature with sinusoidal shape is imposed at the inlet of the turbine. By this simple pulse, the interaction between the backward pulses from reflection and followed forward pulses can be avoided, which will simplify the analysis dramatically. Unsteady behaviours of the turbine are predicted by the same CFD method as discussed above except the pulsating inlet boundary condition.

The shape of the inlet single pulse is shown in figure 5. The occupation of the pulse period is 1/180s which is equivalent to 60 Hz pulsating condition in the experiment (the occupation coefficient is 3, discussed in reference [8]). In order to be relevant to the real pulse, pulse magnitudes of both the pressure and the temperature are set to be the same with experimental results at 60 Hz frequency, high load condition. A period of time (1/180s) with the steady boundary conditions is experienced before the pulse begins to appear for full development of the flow in the turbine, therefore the appearance of the pulse can be clearly tracked in the turbine. Those steady conditions are determined by averaged values from the unsteady experimental measurement. The description of the inlet pulse is given by equations shown in the figure 5. Constant static pressure is imposed as the outlet boundary condition for the convenience.

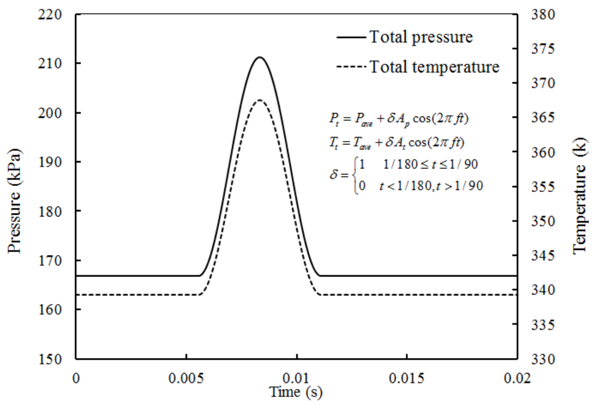


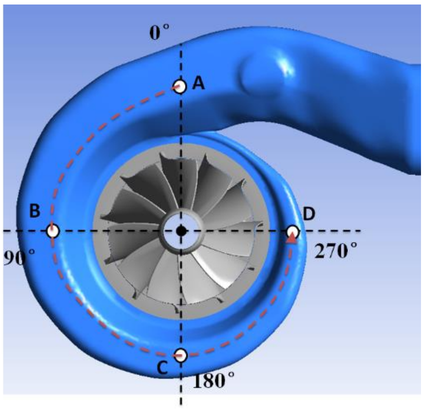
Figure 5. single pulse imposed at inlet of the turbine

$$P_t = P_{ave} + \delta A_p \cos(2\pi ft) \quad (5)$$

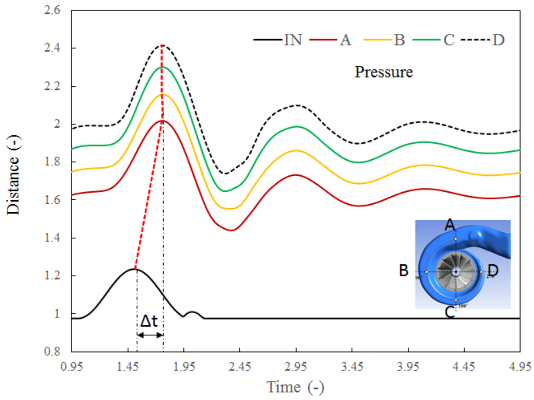
$$T_t = T_{ave} + \delta A_t \cos(2\pi ft) \quad (6)$$

$$\delta = \begin{cases} 1 & 1/180 \le t \le 1/90 \\ 0 & t < 1/180, t > 1/90 \end{cases} \quad (7)$$

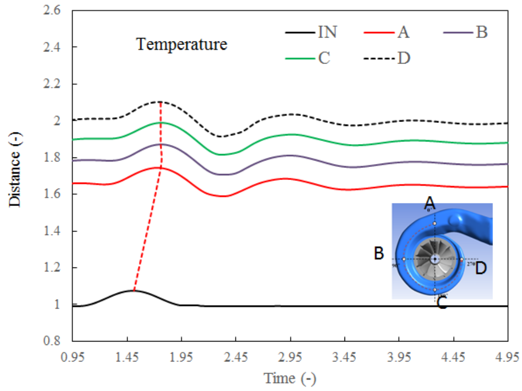
Figure 6 shows variations of different flow parameters versus the time at 5 different locations, of which four are uniformly distributed (referring as A, B, C, D) along the volute passage (subfigure (a)). Distances from those four locations on the volute to the inlet plane are all dimensionless by the value from the inlet to the location D (the maximum distance). All pulses of flow parameters are dimensionless by the corresponding averaged values over the pulse period. In order to demonstrate the propagation of pulses in the turbine, the variations of flow parameters at each location are plot in the figure. Instead of plotting with actual values, the pulses at five locations are arranged according to the dimensionless distance.



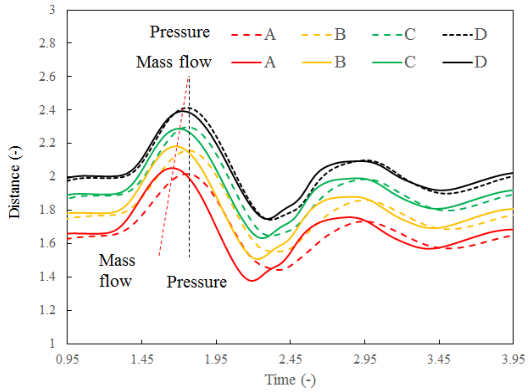
(a) Locations in the volute for flow analysis



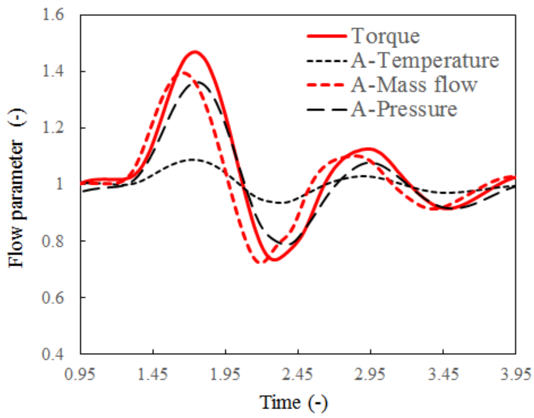
(b) pressure



(c) temperature



(d) Mass flow rate



(e) Output power

Figure 6. instant flow parameters in the volute

Subfigure (b) traces variations of static pressure in turbine. The propagation of the pressure pulse is demonstrated by the phase shift of the pulse between the inlet and other four locations. Furthermore, the propagation speed can be evaluated from the figure where the time and the distance of the propagation can be read. Specifically, the speed of the pressure propagation is about 385m/s on average in the duct, which is quite similar as the sum of the sonic speed (about 356m/s) and the bulk velocity (about 40 m/s). It is a direct proof that the pressure pulse travels at the local sonic speed. Besides, it can be observed that two fluctuations with smaller magnitudes appear following the main fluctuation. Apparently they are resulted from reflections of the main pulse by the interface of the volute/rotor.

By scrutiny of the subfigure, an interesting phenomenon can be observed: pulses are nearly in phase for the four locations (A, B, C and D) although there are considerable distances among them, which is evidently different from the propagation in the inlet duct where an obvious phase shift can be observed. This phenomenon is caused by the much larger bulk velocity (about 170m/s) in the volute passage than in the duct because of the significant acceleration. As a result, the propagation velocity which is contributed by the bulk velocity is much faster than in the inlet duct. In fact, the phase shift in the passage produced by the propagation is less than 1/3 of the one in the inlet duct. Furthermore, this phase shift can hardly be captured by the CFD because it is at the similar order of the time step of the numerical method. According to the phenomenon, it can also be concluded that the phase shift of the pressure pulse in the volute passage can be ignored and the pressure in the passage varies almost simultaneously.

Subfigure 5 (c) shows variations of the temperature at five locations. It is shown that there is no phase difference between the pulse of the pressure and the temperature. Same with the propagation velocity of the pressure pulse, the temperature pulse also travels at the speed as the sum of the sonic speed and the bulk velocity instead of the bulk velocity itself. Therefore, the temperature pulsates and responses in time with the pressure. This phenomenon implies that the variation of the temperature is not caused by the convection of flow particulars, but the result of the variation of the pressure thus governed by the equation of gas properties.

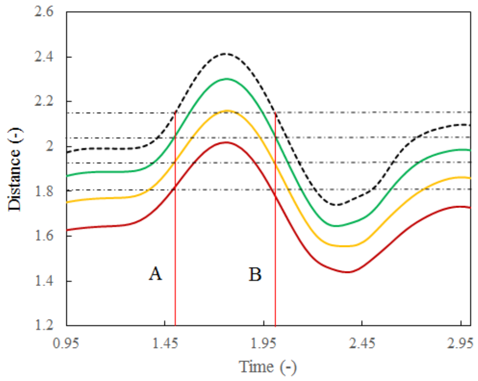
Subfigure 5 (d) shows variations the mass flow rate at cross-sections at four locations in the volute. The variation of the pressure is overlapped in the figure for the comparison. Different phenomenon can be observed for the mass flow rate: the phase shift among four locations are much more evident than for the pressure. The slope of the line which connects the peak of the pulses at four locations is evidently smaller for the mass flow rate than for the pressure. It is implied that the velocity of the propagation of the mass flow rate is smaller than that of the pressure. By evaluation of the distance and the time, it is found that the propagation velocity of the mass flow rate is at a speed (about 280m/s) between the local sonic speed with the bulk velocity. The mass flow rate is determined by both the density and the velocity of the flow, thus the propagation of the mass flow rate is contributed by both the propagation of the density (the temperature and the pressure) and the flow velocity. Therefore, the propagation speed of the mass flow rate is lower than that of the density (local sonic speed) and the flow particulars (bulk velocity).

Finally, subfigure 5(d) overlaps the mass flow rate, pressure, temperature at the location ‘A’ in the volute and the output torque by the rotor. A moderate phase shift among four parameters can be observed at the location. As discussed previously, the mass flow rate is out of phase with the pressure or temperature due to the different propagation speeds. The input energy into the rotor is determined by the mass flow rate, the pressure and the temperature, thus the phase of the energy is between the mass flow rate and the pressure or the temperature. As the rotor has been confirmed as a proximately quasi-steady component, the fed-in energy is almost in-phase with the output power for the turbine rotor, thus results the phenomenon demonstrated in the figure where the phase of the output torque is between the that of the mass flow rate and the pressure or the temperature.

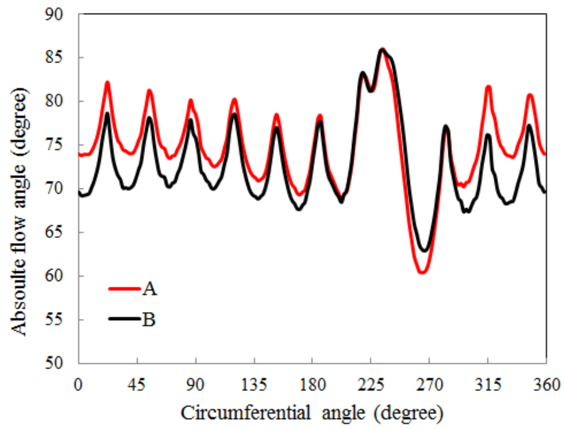
Unsteady effect on flow distribution at inlet of the rotor

The inlet flow condition of the rotor is determined by the flow field coming out of the volute. Therefore, unsteady behaviours of the volute under pulsating conditions influences flow distributions imposed at the rotor inlet, thus has direct influence on the performance of the rotor.

Figure 7 shows flow angle distributions in circumferential direction at two instant times, when their instant pressure is the same at the inlet of the rotor but locate at different sides of the pulse peak, as shown in the subfigure (a). The pressure is experiencing the increase at time A while the reduction at time B. The important phenomenon can be observed that flow distributions are different between those two instant times: the flow angle at time A is evidently higher at most of circumferential locations except for the sections near the volute tongue. It is worth mentioning that the instant pressure and temperature at those corresponding times are the same. The flow angle distributions are expected to be the same at steady conditions. Therefore, the only reason for those discrepancies is the unsteady effects of the flow under the pulsating conditions.



(a)



(b)

Figure 7 (a) two instant times with the same pressure at the rotor inlet; (b) absolute flow angle distributions in annulus at two instant times

Figure 8 further shows variations of the absolute flow angle at the exit of the volute (2mm upstream the rotor inlet) and the local pressure at the position which locates at the circumferential angle as 90 degrees. It is clearly demonstrated in this figure that the flow angle varies evidently with the time, indicating that the pulsating incoming flow results in an unsteady effect on the flow angle distribution at the exit of the volute. The magnitude of the variation is about 2 degrees from the minimum value (72.2°) to the maximum one as (74.3°). Because the relative flow angle at the turbine rotor inlet is very sensitive to the absolute angle, the variation of the latter one as 2 degrees will be amplified dramatically for the relative flow angle at rotor inlet. As a result, this fluctuation will impose an evident influence on the flow capacity and even the efficiency of the turbine. Moreover, the variation of the flow angle is obviously out of phase with the pressure at the same location, which is profoundly different from behaviours of the temperature or even the mass flow rate as discussed previously. The mechanism of this unsteady behaviour of the absolute flow angle distribution at the volute exit is going to be discussed in following sections.

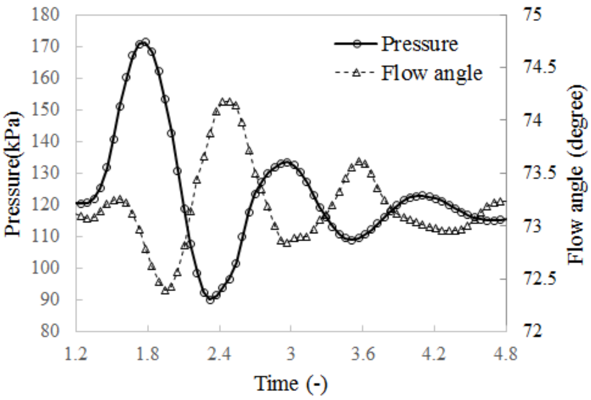


Figure 8 absolute flow angle distributions under pulsating conditions

The ratio of the section area (A) divided by its area centre (R_c) is the most important geometrical parameter for a turbocharger volute. This ratio determines the direction of flow fed into the rotor thus the swallowing capacity and the efficiency of the turbine. Usually, the flow in a volute is assumed to be distributed uniformly in annulus and governed by the conservation of angular momentum (free vortex). The mass flow rates coming-in and leaving a sector of the volute are balanced at any time (mass flow rate equilibrium). Therefore, linear distributed A/R_c in circumferential direction can be deduced from those two assumptions. This forms the fundament of the volute theory and methodology of design. However, according to previous discussions, it has been confirmed that the mass flow rate equilibrium is not valid anymore because of the variation of mass flow rate under pulsating conditions, especially for highly unsteady flow conditions. Unsteady effects are imposed on flow distributions in the volute by pulsating conditions. As a result, the conventional correlation between A/R_c and the flow angle at exit of the volute which is based on the steady situation doesn't valid for pulsating conditions. New correlation has to be developed based on the unsteady conditions.

A sector of the volute with a small circumferential angle as $d\theta$ is shown in figure 9.

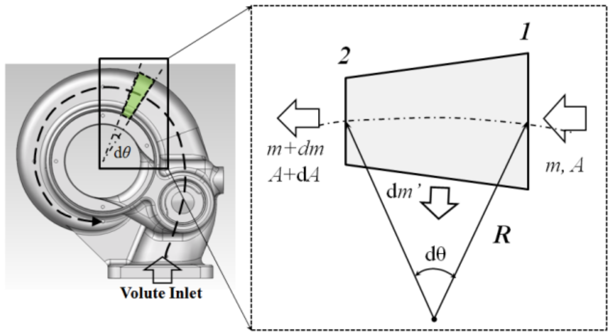


Figure 9 a sketch of volute sector

Mass conservation equation in the section is:

$$\frac{dm}{dt} = \rho(C_\theta + dC_\theta)\cdot(A+dA) + bC_r R_c \rho d\theta - \rho C_\theta A \quad (9)$$

Where, $\frac{dm}{dt}$ is the gradient of mass variation in the sector, C_θ and C_r are the tangential and radial components of the velocity, respectively, A and R_c are the area of the section and the radius of the section centre, respectively. b is the width of the section in axial direction.

For the control volume, the variation of the mass flow rate can be further calculated as:

$$\frac{dm}{dt} = R_c \cdot A \cdot d\theta \cdot \frac{d\rho}{dt} \quad (10)$$

According to the angular momentum conservation:

$$C_\theta \cdot R_c = K \quad (11)$$

Where K is a constant value for the flow in the volute.

Combine equations (9) ~ (11) and status equation of perfect gas, the following equation can be deduced:

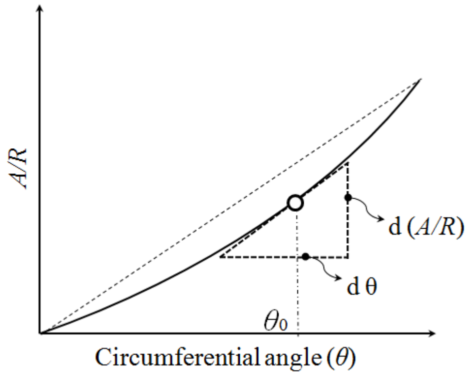
$$tg\alpha = -\frac{K}{b}\cdot\frac{d\left(\frac{A}{R_c}\right)}{d\theta} + \frac{A\cdot R_c}{b}\cdot\frac{d\left(\frac{P}{RT}\right)}{dt} \quad (12)$$

Where, α is the absolute flow angle at the exit of the volute.

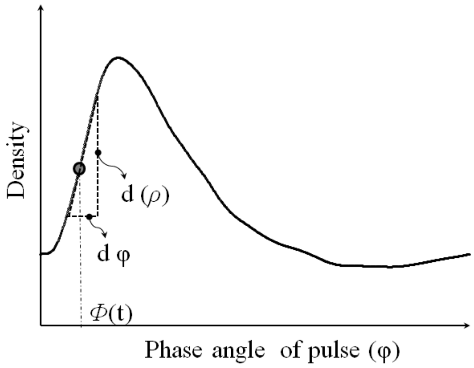
This equation describes the relation of the volute configuration and the pulsating condition with the exit flow angle. The first item on the right side of the equation is the impact of the geometrical parameter A/R_c on the flow at certain annulus location of the volute exit. This geometrical item determines the flow distribution at both steady and pulsating conditions. It should be mentioned that A/R_c at the throat is the key parameter for the volute which determines the flow angle at the exit for conventional volute theory. However, it can be seen from the equation that it is the local gradient of A/R_c distribution ($\frac{d(\frac{A}{R_c})}{d\theta}$) that contributes to flow angle distribution at the volute exit. The conclusion that A/R_c at the throat determines the flow angle at the exit is valid only when the parameter distribution is linear in annulus.

Importantly, the second item on the right side is the impact of the pulsating flow on the flow angle at the volute exit. It can be seen that the gradient of the pressure divided by the temperature (or $\frac{d\rho}{dt}$) has a direct contribution to the absolute flow angle in the volute. This equation clearly demonstrates unsteady effect of pulsating flow on the flow distribution, which is not be experienced under steady conditions. Moreover, it is concluded that the unsteady effect becomes more profound as the gradient increases which can be achieved by increase of pulse frequency or the magnitude. This mechanism is considered to be one of the reasons for the enhancement of the rotor unsteadiness by the frequency or the magnitude which has been confirmed in literatures. Furthermore, by the scrutiny of the unsteady item, it is understood that the gradient is actually the result from the density thus the mass imbalance in the sector. Therefore, the unsteady effect of the pulsating flow on the flow distribution in the volute is a result from the fact of mass imbalance at an instant time.

In conclusion, the influence of two items which are the volute configuration and the pulsating conditions on flow distributions is demonstrated in the subfigures of figure 10, respectively.



(a) effect of geometrical parameter



(b) unsteady effect by pulse

Figure 10 Impact on the flow angle by the volute under pulsating conditions

The analytical model is further applied for the understanding of flow behaviours in the volute as discussed in figure 7 and 8. In figure 7, the gradient of the density is positive for the instant time ‘A’ but negative for ‘B’. According the model (equation 12), the contribution by the pulsating flow to the flow angle is positive for the time ‘A’ while negative for the time ‘B’. As a result, the absolute flow angle should be evidently higher for the former, which is demonstrated in the figure. Moreover, the unsteady effect on the flow angle is proportional to the gradient of the pulsating flow parameter P/T (thus the density). Therefore, the unsteady flow angle should be in phase with the variation of the gradient of the parameter instead of the parameter itself. Figure 11 shows the flow angle variation and the gradient of the P/T. It can be seen that the two flow parameters are almost in phase which is a direct comparison with the phenomenon in figure 8, thus is a satisfied validation of the analytical model.

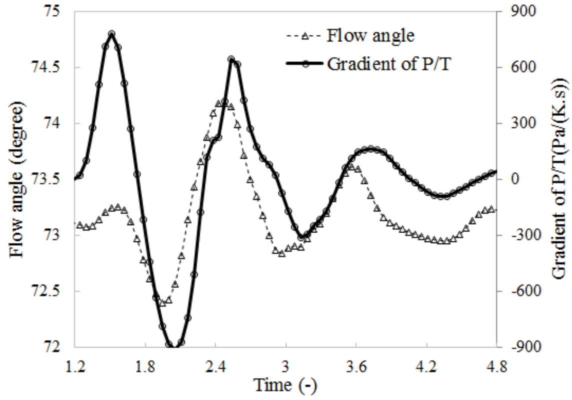


Figure 11 flow angle and the pressure gradient under the pulsating condition

CONCLUSIONS

A turbocharger turbine is confronted by pulsating flow conditions due to reciprocating internal combustion engine. In order to improve turbine performance in the real operational environment, it is important to understand the flow phenomenon happening in the turbine under such highly unsteady conditions. This paper investigates unsteady behaviours of the turbine volute under pulsating conditions via validated numerical and analytical methods. Three conclusions can be drawn from the investigation:

1. Both pulses of pressure and the temperature propagate at the velocity as the sum of sonic speed and the bulk velocity. The phase shift of the pulse along the inlet duct is evident but can hardly be observed along the volute passage mainly because of the larger bulk velocity. Mass flow rate propagates at the bulk velocity, which produces a moderate phase shift with the pulse of pressure or temperature. As a result, the output power by the rotor is notably out of phase with the both the mass flow rate and the pressure (or temperature) at the inlet of the rotor;
2. The pulsating flow imposes an evident unsteady effect on the absolute flow angle at the exit of the volute. The pulse magnitude of the flow angle is about 2 degrees and

significantly out of phase with the pressure at the same location for the investigated pulsating condition;

3. The analytical model is established for the absolute flow angle at the exit of the volute at pulsating conditions. The flow angle is determined by two factors under pulsating conditions: the geometrical parameter $\frac{d(\frac{A}{R_c})}{d\theta}$ and the gradient of the pressure divided by the temperature $\frac{d(\frac{P}{T})}{dt}$. The latter one is the result from the mass imbalance in the volute and the root for the unsteady behaviour of the flow angle distribution in the volute.

It is well known that a relatively small variation of the absolute flow angle upstream the turbine rotor can produce a significant variation of the relative flow angle fed to the rotor. Therefore, the unsteady effect on the flow angle at the volute exit further enhances fluctuations of incidence angle at turbine rotor inlet. As a result, turbine performance is profoundly influenced by pulsating conditions which can't be experienced at steady conditions. It is implied from the study that the conventional guideline for volute design which is based on the assumption of mass balance might be mended for the application for pulsating conditions.

NOMENCLATURE

A	Area	mm^2
A_p	Magnitude of pressure pulse	pa
At	Magnitude of temperature pulse	pa
b	Blade height at inlet	mm
C	Flow velocity	m/s
CFD	Computational Fluid Dynamic	-
C_p	Specific heat	J/(kg.° C)
f	Frequency	Hz
MFP	Mass flow rate parameter	kg/s.√K/Pa
m	Mass flow rate	kg/s
P	Pressure	Pa
PR	Pressure ratio	-
Rc	Radius of the area centre	mm
T	Temperature	K
t	Time	s
U	Blade speed	m/s
VR	Velocity ratio	-

Greeks

α	absolute flow angle	degree
η	efficiency	-
θ	circumferential angle	degree
τ	torque	N.m
ρ	density	Kg/m^3
ω	angular velocity	Rad/s

Subscripts

ave	average
e	exit
i	inlet
r	radial component
s	static
t	total
θ	tangential component

ACKNOWLEDGE

This work is supported by National Natural Science Foundation of China [grant number: 51606121] and Science and Technology Commission of Shanghai Municipality [grant number: 16PJ1404400].

REFERENCES

[1] Martinez-Botas, R., Pesiridis, A., and Yang, M. Overview of Boosting Options for Future Downsized Engines, Science China Technological Sciences, v 54, n2, p318-331, 2011.

[2] Yang, M., Martinez-Botas, R., Rajoo, S., Yokoyama, T., and Ibaraki, S. An Investigation of Volute Cross-Sectional Shape on Turbocharger Turbine under Pulsating Conditions in Internal Combustion Engine. Energy Conservation and Management, 105, p167-177, 2015.

[3] Baines, N. C., Harjilouy-Benisi, A., and Yeo, J. H., "The Pulse Flow Performance and Modelling of Radial Inflow Turbines", Proc. IMechE, 180, Part 3J, 1994, Paper 23.

[4] Chiong M., Rajoo S., Romagnoli A., Costall A., and Martinez-Botas R. Integration of Meanline and One-dimensional Methods for Prediction of Pulsating Performance of a Turbocharger Turbine. Energy Conservation and Management, Vol.81, p270-281, 2014.

[5] Cao T., and Xu L. A Low Order Model for Predicting Turbocharger Turbine Unsteady Performance. ASME paper, GT2014-26913, 2014.

[6] Capobianco, M., and Gambaraotta, A. Unsteady flow performance of turbocharger radial turbines, Journal of Engineering for Gas Turbines and Power, Vol. 114, 1992, pp553-560.

[7] Szymko S., McGlashan N R., Martinez-Botas R., and et el. The Development of a Dynamometer for Torque Measurement of Automotive Turbocharger Turbines. Journal of Automobile Engineering, Vol. 221, 2007.

[8] Costall, A. Szymko, S., Martinez-Botas, R., and Filsinger, D. Assessment of Unsteady Behavior in Turbocharger Turbines. ASME paper, GT2006-90348, p1023-1038, 2006.

[9] Copeland, C D., Newton, P., and Martinez-Botas, R. F. A Comparison of Timescales with a Pulsed Flow Turbocharger Turbine. 10th Int. Conf. on Turbocharging and Turbochargers, Instn. of Mech. Engrs, London, 2012.
[10] Costall, A. and Martinez-Botas, R. Fundamental Characterization of Turbocharger Turbine Unsteady Flow Behavior. ASME paper, GT2007-28317, 2007.
[11] Arcoumanis, C., Hakeem, I., Khezzar, L. and Martinez-Botas, R. F., Performance of a Mixed Flow Turbocharger Turbine Under Pulsating Flow Conditions, Transc ASME 95-GT-210,1995.
[12] Chen, H. A Disscussion on Volute Design Method for Radial Inflow Turbines. ASME paper, GT2009-59110, 2009.
[13] Whitefild, A., MacGregor, S. A., and Mohd Noor, A. B. Design and Performance of Vaneless Volutes for Radial Inflow Turbines Part 2: Experimental Investigation of the Meanline Performance – Assement of Empirical Design Parameters. Journal of Power and Energy, Vol. 208, 1994.
[14] MacGregor, S. A., Whitefild. A., and Mohd Noor, A. B. Design and Performance of Vaneless Volutes for Radial Inflow Turbines Part 3: Experimental Investigation of the Internal Flow Structure. Journal of Power and Energy, Vol. 208, 1994.
[15] Whitefild. A., and Mohd Noor, A. B. Design and Performance of Vaneless Volutes for Radial Inflow Turbines Part 1: Non-dimensional Conceptual Design Considerations. Journal of Power and Energy, Vol. 208, 1994.
[16] Padzillah, M. H., Rajoo, S., Yang M., and Martinez-Botas R. F. Influence of Pulsating Flow Frequencies towards the Flow Angle Distributions of an Automotive Turbocharger Mixed-flow Turbine. Energy Conservation and Management, 98, p449-462, 2015.