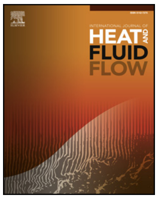
International Journal of Heat and Fluid Flow 71 (2018) 170–178

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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



Investigation of using multi-shockwave system instead of single normal shock for improving radial inflow turbine reliability

Zhao Ben[a,b], Harold Sun[a], Shi Xin[a,*], Qi Mingxu[a], Guo Sinan[a]

[a] *School of Mechanical Engineering, Beijing Institute of Technology, Beijing 100081, China*
[b] *Michigan State University, East Lansing 48824, USA*

ARTICLE INFO

Keywords:
Variable nozzle turbine (VNT)
Shock wave
High cycle fatigue (HCF)
Efficiency

ABSTRACT

A variable nozzle turbine (VNT) has become an important technology for diesel engine applications. During the development and design process, one of the most important tasks is to minimize the risk of high cycle fatigue (HCF) failure. When a VNT operates with high inlet pressure and small nozzle opening, a shock wave may appear near nozzle vane trailing edge. The shock wave introduces strong excitation force on the downstream turbine blades and has a potential to damage the turbine wheel. To improve turbine reliability, a method of generating multi-shockwave system by using grooved surface nozzle was investigated. It was found that, in the multi-shockwave system, the intensity of the shock wave is weakened in comparison to original single normal shock wave, and its impact on the turbine wheel is therefore lowered. However, with the use of the grooved surface, penalty on the turbine efficiency is numerically found, but very small at the research point.

1. Introduction

Turbocharger is one of the most important parts of an advanced internal combustion engine. One of its advantages is to downsize an internal combustion engine without reducing engine output power (Feneley et al., 2017; Tang et al., 2015). Besides, working with a turbocharger, an engine usually has better fuel economy. In order to recover more energy from exhaust gas in a wide operation range, a variable nozzle turbine (VNT) has been developed and became a popular technology for diesel engine applications. Compared to a fixed geometry turbine, a VNT improves transient response at low engine loads (Pesiridis, 2014; Rakopoulos and Giakoumis, 2009; Saidur et al., 2012), but has strong aerodynamic excitations downstream of the nozzle vanes that may cause high cycle fatigue (HCF) failure of a turbine wheel. Therefore, the HCF issue is one of the most important challenges for the development of a VNT (Kulkarni and LaRue, 2008).

In the turbocharger industry, the HCF may be caused by unsteady aerodynamic excitations with frequencies close to natural frequencies, or their harmonics of turbine blade or disk. Usually, HCF damages may be rapidly accumulated at a rate typically higher than 8000 Hz (Heuer et al., 2008). As one of the most important aerodynamic exciting sources, shock waves may cause the serious HCF damage of turbine wheel. Chen (2006) pointed out that when a VNT operates in engine braking mode, the shock wave generated near the nozzle exit may damage downstream turbine wheel.

Besides the shock wave, the nozzle end-wall clearance leakage flow (LF) is closely associated with the HCF issue of a turbine wheel. At a small nozzle vane open condition, the combination of high loading of nozzle vane and its large setting angle inevitably causes the strong nozzle end-wall clearance leakage flow. With high radial momentum and vortices structure, the leakage flow enhances the unsteadiness of the flow not only at the inducer inlet (Zhao et al., 2015) but also inside the turbine wheel (Hu et al., 2013), leading to a strong aerodynamic exciting force on the turbine wheel, especially around the inducer blade tip and root. Liu et al. (2014) reported that, with the increase of nozzle end-wall clearance size, the unsteadiness of the aerodynamic loading obviously increases at both the inducer blade tip and root.

Previously, using grooves on nozzle vane surface to mitigate the shock wave excitation near the nozzle vane trailing edge was first proposed by Sun et al. (2016). And then the grooved surface's effectiveness for mitigation of the shock wave intensity and for improving the turbine wheel reliability was numerically investigated and found to be promising (Zhao et al., 2016). A more in-depth experimental work was done by Lei et al. (2017, 2017) by analyzing the shock wave variations with the grooved surface in a linear turbine nozzle. Based on the previous work, this paper analyzed two styles of grooved surface design on the nozzle vane surface, and compared them against the original nozzle vane to know if there is an optimal groove design that can generate multi-shockwave system waves instead of original single shock wave to mitigate shock wave intensity and to decrease the risk of

---

[*] Corresponding author.
E-mail address: shixin@bit.edu.cn (X. Shi).

https://doi.org/10.1016/j.ijheatfluidflow.2018.03.018
Received 12 June 2017; Received in revised form 27 March 2018; Accepted 28 March 2018

Nomenclature

LE	leading edge
TE	trailing edge
T	time period of one wheel blade passing over a nozzle vane
SL	strong leakage
WL	weak leakage
LF	leakage flow
Amp	amplitude
AP	acceleration process

turbine wheel suffering from HCF damage with very small turbine efficiency penalty.

2. Investigation case and numerical method

2.1. VNT illustrations

A commercial VNT was used as research model by scanning a production turbine. The original turbine has 9 nozzle vanes, 13 wheel blades, and 1 volute. In this paper, the numerical model used for CFD simulations only included nozzle vanes and turbine wheel in order to reduce the requirement of CPU and memory.

2.2. Computational mesh

The multi-block structure mesh of the nozzles and the turbine wheel was generated by a mesh-generator, IGG/AutoGrid. The O4H topology structure, an O mesh surrounded by four H type mesh blocks, was employed, because of its advantages in capturing the geometry of vane/blade and improving mesh quality. Mesh sensitivity analyses were conducted by performing steady simulations to get a suitable computational mesh that is not only independent of mesh density but under affordable computation resource limit as well.

Following the steady simulation that was used to initially validate computational mesh, an unsteady simulation was performed to predict shock wave in nozzles and to obtain transient aerodynamic loading on the turbine blades. It is well known that the fluid property parameters (e.g. pressure, density, Mach number, and others) experiences larger change across a shock wave than in most other flow domains. Therefore, the computational mesh around the shock wave was refined again after the shock wave's position had been located. The research model and the computational mesh with locally enlarged zone are illustrated in Fig. 1, and the evaluations of refining the mesh in the local region are shown in Fig. 2 (the static pressure was normalized by dividing the nozzle inlet pressure). The tiny deviations of 0.86% in Mach number and 0.65% in static pressure prove the low sensitivity of the space discretization in the region around the shock wave to the shock wave prediction, which means that the mesh with local refined distribution should be good enough for this research. The total number of grid cells is more than 4.5 million in the computational domain and the average y plus value is about 2.68 (its maximum is about 5.18). To speed up the convergence, a full multi-grid technique was applied.

In the computational mesh, the nozzle vane spindle was neglected to simplify the generation of mesh. With the domain scaling method, only 1/3 whole revolution was used for the unsteady CFD simulations by changing the turbine blade count from 13 to 12. Though steady CFD results showed small deviation in the turbine performance, the changed wheel blade count has little influence on the conclusion stated in this paper, for the discussion is mainly about the shock wave from the nozzle exit and the excitation force resulted from the nozzles.

2.3. Mathematical model

The commercial code, EURANUS integrated in Fine/Turbo interface, was used in the current numerical simulation. It solves the time dependent Reynolds averaged Navier–Stokes equations using the central difference and a 4-stage explicit Runge-Kutta integration schemes. The one equation turbulence model, the Spalart–Allmaras turbulence model, was used because of its advantages in time-efficiency and accuracy. For the unsteady simulation, a dual time stepping technique was used, and the convergence criterion and the maximum inner iterations for each time step were set to 1e − 4 and 50, respectively. A time step sensitivity analysis was performed with three sets of physical time step, 2.08e − 6 s, 1.04e − 6 s and 6.93e − 7 s, for the impinging of a shock wave on the leading edge of a wheel blade is transient. After comparing the numerical results corresponding to the different physical time, as shown in Table 1 (the peak value was normalized by dividing the value when the physical time is 2.08e − 6 s), the minimal time scale was selected for the unsteady simulations, because the change of the peak value of the alternating loading (its location will be introduced in Section 3.5) exerted on a turbine blade decreased to 0.07% of the peak loading when the physical time was decreased from 1.04e − 6 s to 6.93e − 7 s. In addition, other evaluations of the calculation convergence can be found in literature (Qi and Sun, 2018).

2.4. Validation of CFD method

Fig. 3 shows the comparison of turbine performance at a fixed rotational speed. The test data was obtained from the literature

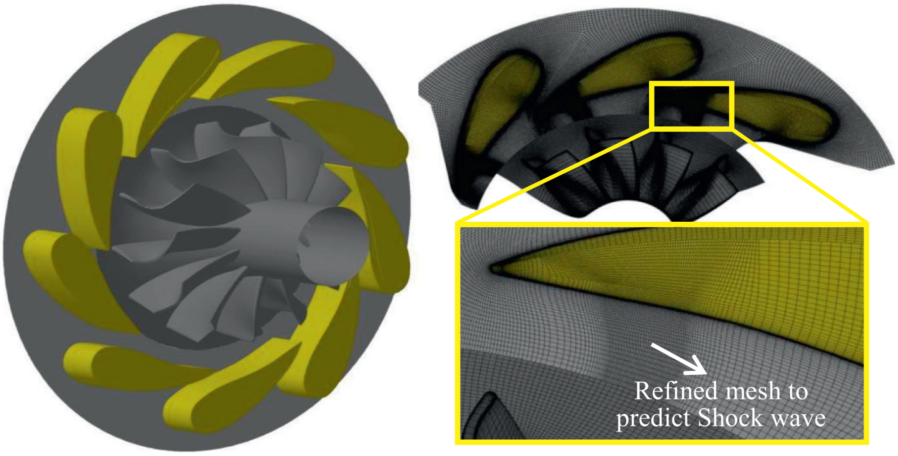


Fig. 1. Variable nozzle turbine and computational mesh.