International Journal of Heat and Fluid Flow 58 (2016) 19–29

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Unsteady influence of Self Recirculation Casing Treatment (SRCT) on high pressure ratio centrifugal compressor

Yang Mingyang[a,*], Martines-botas Ricardo[b], Deng Kangyao[a], Zhang Yangjun[c], Zheng Xinqian[c]

[a] Institute of Internal Combustion Engine, Shanghai Jiao Tong University, Shanghai 200240, China
[b] Mechanical Engineering Department, Imperial College London, London SW7 2AZ, UK
[c] State Key Laboratory of Automotive Safety and Energy, Tsinghua University, Beijing 100084, China

ARTICLE INFO

Article history:
Received 5 September 2015
Accepted 14 December 2015
Available online 13 January 2016

Keywords:
Turbocharger
Centrifugal compressor
Self-recirculation-casing-treatment
Unsteady

ABSTRACT

Self-Recirculation-Casing-Treatment (SRCT) is a widely employed method to enhance aerodynamic stability of a centrifugal compressor. This paper investigated unsteady effects of SRCT on the flow in a transonic centrifugal compressor via numerical method validated by experimental test. Firstly the static pressure distribution in the compressor without SRCT is measured for information of boundary conditions as well as validation. Then a 1-D unsteady model of a single passage is built and validated based on the experimental results. Next, the 1-D model of a passage with SRCT is built to investigate the unsteady influence of the SRCT on the flow in the passage. Finally 3-D unsteady CFD is employed to investigate the detailed influence of SRCT on the flow field in impeller passages. Results show that the topology of the passage with SRCT can remarkably damp the distortion propagating from downstream, hence depress the magnitude of the inlet flow distortion. Furthermore, the width of the rear slot in SRCT is the key factor for the damping effect. The 3-D simulation results further show that the fluctuations of the re-circulated flow rate via the front slot is depressed by the SRCT which is attributed to the damping effect of its configuration.

1. Introduction

The increasing demand of low carbon vehicle has forced the industry towards highly efficient and downsized engines. One of the key enablers for the engine downsizing is the turbocharger which reclaims the energy from exhaust gas to boost the intake air. Self Recirculation Casing Treatment (SRCT) is one of the most widely used methods to enlarge the performance map width of a high pressure ratio centrifugal compressor. Extensive researches have been carried out via the experimental method and CFD simulation to investigate the influence and the mechanism of SRCT for the stability enhancement (Hunziker et al., 2001, Hu et al., 2009, Sivagnanasundaram et al., 2010, Gao et al., 2010, Galindo et al., 2005). The recirculation flow in the device driven by the inverse pressure gradient is considered to be the main reason for the performance enhancement. It can compensate the inlet flow rate when the compressor is approaching to surge condition, thus postpones the surge limit.

It has been confirmed that the flow in a centrifugal compressor is not axial-symmetrical. The asymmetrical geometry of the volute in a compressor results in the distorted flow in the diffuser, especially at off-design flow rates (Ayder et al., 1993, Hagelstein et al., 1997, K. and Van den Braembussche, 1999). Intensive investigations have already been conducted to study the interaction between the volute and impeller, of which most are performed via steady or unsteady 3-D CFD method. Sorokes et al. (1998) and Sideris and Van den Braembussche (1987) have showed that the distortion resulted from the volute can reach the inlet of impeller which may degrade the stability of compressor. A simplified 3-D unsteady model was employed by Fatsis et al. (1997) to investigate the unsteady interaction between volute and impeller. A single passage was modeled and the static pressure at impeller outlet from experiment was imposed. At the meantime, a “phase lag” method was used to reduce the demand for computational resources. Results showed that the perturbation from impeller outlet had an obvious influence on the inlet incidence as well as other flow parameters. Gu and Engeda (2001) and Hunteler (2009) studied the interaction via steady 3-D simulation and concluded that the influence of volute made the performance of compressor quite different from that of the one without volute.

---

* Corresponding author. Tel.: +86 2134206850.
E-mail address: myy15@sjtu.edu.cn (Y. Mingyang).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2015.12.004

Nomenclatures

A	area mm^2
Am	amplitude
br	width of rear slot mm
bf	width of front slot mm
bb	width of the recirculation channel mm
C_r	radial component of velocity m/s
D	diameter mm
E	power through an area kW
F_{co}	coriolis force N
F_{cen}	centrifugal force N
F_{ps}	pressure force N
hb	height of the recirculation channel mm
I	power density at an unit area kW/m^2
L	length mm
LE	leading edge
N	design speed RPM
n	rotational speed RPM
P	pressure Pa
Sf	distance from LE to front slot mm
Sr	distance from LE to rear slot mm
TVD	total variables diminish
t	time s
U	blade velocity m/s
V	absolute velocity m/s
W	relative velocity m/s
Z	acoustic impedance Pa•s/m

Greek letters

α	sonic speed m/s
β	relative flow angle degree
θ	circumferential angle degree

Subscripts

ave	average
in	inlet
max	maximum
min	minimum
out	outlet

The interaction between volute and impeller notably influences the compressor performance thus the effect of volute should not be ignored in the investigation of SRCT influence by numerical method. Indeed Zheng et al. (2010) have numerically proved that SRCT can depress the distortion of inlet flow angle at main blade tip, which has a positive effect on the compressor stability. The asymmetrically distributed re-circulated flow rate at the impeller inlet is considered to be the reason for the distortion depression. The conclusion was drawn based on the steady simulation, where the interfaces between the volute, SRCT and the impeller are "frozen". However, in the real phenomenon, the impeller spins at high speeds when it is confronted by the flow distortion caused by the volute. Therefore, the propagation of the distortion in the impeller happens in the rotating coordinates instead of the fixed one as the assumption in the steady simulation. On the other hand, it is accepted that the unsteadiness effect shouldn't be ignored when the Strouhal number of the compressor is larger than 0.1, of which the characteristic length is the average length of the impeller passage (Fatsis et al., 1997). For a high pressure ratio centrifugal compressor, Strouhal number is most likely much larger than this criterion. As a result, the unsteady simulation method is necessary for the investigation on the interaction between the volute and the impeller in a high pressure ratio centrifugal compressor.

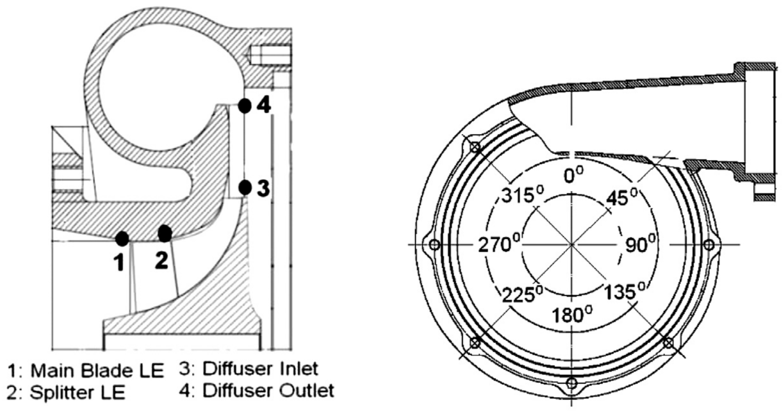
The 3-D unsteady simulation on the whole compressor stage with SRCT might be conducted for the investigation purpose, but no doubt that immense computational resources are required. An alternative route is to apply a method which can simplify the complexity of the computation based on the physics of the phenomenon without losing the correct pictures.

This paper focuses on the unsteady influence and its mechanism of the SRCT on the flow in a passage. 1-D unsteady model of SRCT was established based on the model of passage validated by experimental results. A key geometrical parameter of the impaction was confirmed by the model. Furthermore, a 3-D unsteady simulation on a single passage is carried out for detailed investigation on the evolution of the flow in the impeller with SRCT.

2. Experimental testing

The circumferential distribution of static pressure at different locations in the impeller and diffuser were measured by pressure transducers on the turbocharger test facilities. Fig. 1(a) shows a sectional view of the compressor. As shown in the figure, there are two measured locations in the impeller shroud (referring as 1 and 2), which are near main blade leading edge and splitter leading edge, respectively. Another two positions locate at the inlet and outlet of the diffuser respectively. To obtain the pressure distribution in annulus, eight pressure tapings are mounted evenly in circumference for each location, as shown in Fig. 1(b).

Figs. 2 and 3 show the static pressure distribution in annulus at diffuser inlet and outlet near surge for 4 rotational speeds as



(a) 4 measured locations (b) 8 measured positions

Fig. 1. Measurement points in the compressor.