International Journal of Heat and Fluid Flow 74 (2018) 118–129

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Compressed dynamic mode decomposition for the analysis of centrifugal compressor volute

Yang Xiaojian, Zhu Xiaocheng*, Hu Chenxing, Du Zhaohui

Department of Mechanical Engineering, Shanghai Jiao Tong University, 800 Dongchuan RD. Minhang District, Shanghai 200240, China

ARTICLE INFO

Keywords:
Numerical simulation
Unsteady flow
Centrifugal compressor volute
Dynamic mode decomposition
Compressed dynamic mode decomposition

ABSTRACT

The compressed dynamic mode decomposition (compressed DMD) is used to analysis the unsteady characteristics of a centrifugal compressor. Firstly, to extract the unsteady flow structures of the volute under the mild surge condition, dynamic mode decomposition (DMD) method is applied to the flow field snapshots. The results indicate that the characteristic frequencies relating to the blade passing frequencies (BPFs) and corresponding modes are captured. Besides, the perturbation oscillating with the characteristic frequency (233.5 Hz), which is generated by the non-synchronic unsteady flow, is also obtained accurately. The non-synchronic unsteady flow is vividly shown by the reconstruction of the mild surge mode. Ten, the unsteady flow of the volute is also analyzed by the compressed DMD method, which is performed on the much compressed datasets and thus saves the calculation time. The dominant characteristic frequencies and corresponding modes can also be extracted accurately by the compressed DMD method when adopting an appropriate compression ratio. The compression ratio has an important influence on the results of the compressed DMD method. Considering the dominant frequencies and the energy ratio spectrum, a conservative recommendation of the compression ratio is 1% in this case. The energy ratio scaling method performs better than the amplitude scaling method.

1. Introduction

For the centrifugal compressor, analyzing the features of unsteady flow is one of the fundamental research problems since it has an important effect on the performance of the machine (Day, 2016; Jansen, 1964). The unsteady flow of turbomachinery can be divided into synchronic unsteady flow, which can be attributed to the rotor-stator interaction, and non-synchronic unsteady flow, which is caused by rotating stall or surge and generally occurs under the small mass flow operating condition. With the reducing of mass flow rate, the amplitude of the oscillation increases and even lead to a catastrophe for the whole compressor system, including the impeller, diffuser and volute. Therefore, it is necessary to deeper understand the features of the unsteady flow, especially the non-synchronic unsteady flow.

As one of the main components, the volute has a significant effect on working performance of the compressor system. In previous research, the influence of volute design and geometry parameters on the working performance of compressor has been investigated (Hassan, 2007; Kim et al., 2010). The structure of volute is circumferentially asymmetric, which further increases the complexity of the flow field of the volute (Ayder et al., 1993). K. Hillewaert et al. investigated the interaction between the impeller and volute at off-design condition, and the research showed the relation between this waviness and the unsteady flow (Hillewaert and den Braembussche, 1999).

Numerical simulation is a very useful tool to analysis fluid phenomenon and can provide comprehensive insight into the fluid. Many researches have been carried out by steady or unsteady numerical simulation, and the results indicate this method is reliable by comparing with experiment results (Fatsis et al., 2006; Zemp et al., 2010). J. Galindo et al. studied the effect of the tip clearance of a centrifugal compressor on the flow field by numerical simulation (Galindo et al., 2015). The noise generation mechanism of a turbocharger compressor is analyzed under the surge and stall operating conditions (Broatch et al., 2016). C. S. Tan et al. investigated the features of spike-type stall of an axial compressor with unsteady numerical simulation, and the results show the formation and growth characteristics of this aerodynamic instability flow (Tan et al., 2010). Taking numerical simulation, W. Huang et al. focused on the flow characteristics of the interactions between centrifugal impeller and vaned diffuser under stall working condition (Huang et al., 2007). Q. Guo et al. investigated the effect of volute on the unsteady flow and analyzes the characteristics of the internal flow of volute under different working conditions (Guo et al., 2007).

Dynamic mode decomposition (DMD) is an effective method to analyze the dynamic system. DMD was firstly proposed by

* Corresponding author.
E-mail addresses: xiaojianyang@sjtu.edu.cn (X. Yang), zhxc@sjtu.edu.cn (X. Zhu), ryanhu@sjtu.edu.cn (C. Hu), zhdu@sjtu.edu.cn (Z. Du).

https://doi.org/10.1016/j.ijheatfluidflow.2018.09.013
Received 4 June 2018; Received in revised form 17 September 2018; Accepted 21 September 2018
0142-727X/ Published by Elsevier Inc.

Schmid (2010) and then has been widely used in many fields. The relationship between DMD and other methods, such as Koopman operator theory and linear inverse modeling (LIM), has been further discussed (Rowley et al., 2009; Tu et al., 2013). In the linear stability theory, the small-amplitude perturbation which determines the stability or instability of the flow could be obtained (Theofilis, 2011). Yet the DMD is more useful to analyze the unsteady features of the complex flow. When applied in the fluid dynamics, DMD method can decompose sequential series of velocity field, which usually is called as “snapshot sequence”, into spatial modes which oscillate with a single temporal frequency (Tu et al., 2013). Therefore, the characteristic frequency of the flow and the corresponding spatial structures can be identified, which is helpful to understand the underlying flow structure we interest. Kou discussed the application of DMD on many complex flow phenomena (Kou and Zhang, 2018).

In computational fluid dynamics (CFD), the larger number of meshes usually represents the more detailed flow information obtained by the numerical simulation. Therefore, with the permission of computing resource, a larger number of meshes for the research object is preferred in some cases. However, computer resource consuming for DMD method is dominated by the singular value decomposition (SVD). The computational cost of SVD depends on the total mesh number of flow field. Therefore, it is likely that the number of mesh is so large that DMD method consumes pretty huge computing resource and time, or even could not be performed with the limitation of computing ability.

Compressive sensing is an effective tool for the problems that have a sparse distribution in frequency space (Candes et al., 2006; Donoho, 2006). Compressed dynamic mode decomposition and compressive sampling dynamic mode decomposition are the methods which is developed form the standard DMD method and compressive sensing (Brunton et al., 2013). In addition, when the full measurement data is accessible, compressed DMD can be a better choice than the standard DMD. Since the main feature of compressed DMD is that the datasets of snapshot has been compressed firstly before performing the DMD algorithm, it can be more efficiently performed than standard DMD method and thus save the computation resource and computing time. S. L. Brunton applied compressed DMD method on two problems relating to fluid dynamics and oceanographic science, and the results indicate that the DMD eigenvalues and coherent modes can be accurately reconstructed by a few spatial measurements (Brunton et al., 2013). The compressed DMD method was applied to uses the background of videos, and the optimal modes are selected to characterize the background (Erichson et al., 2016).

The rest of this paper is organized as followed. In Section 2, we present the configuration of the centrifugal compressor GT70 and the details of the numerical simulation. Besides, the results of this numerical simulation are given as well. In Section 3, a brief introduction of the DMD and compressed DMD method is given. In Section 4, the results of DMD and compressed DMD applied to the flow field of the volute are presented and the influence of compression ratio on the results of compressed DMD method is analyzed. Finally, the conclusions are presented in Section 5.

2. Configuration and numerical simulation

2.1. Introduction of the centrifugal compressor

Firstly, a brief introduction about the research object is presented. It is a single-staged centrifugal compressor GT70, which includes three main components: impeller, vaneless diffuser and volute. To regulate the mass flow rate of the compressor, an exit pipe is added at exit of the volute in numerical simulation. The impeller consists of 12 blades, including 6 main blades and 6 splitter blades. Geometry parameters of this centrifugal compressor are listed in Table 1. Under design condition, the rotational speed of this compressor is 44,198 rpm and the mass flow rate is 1.8 kg/s.

Table 1
Geometry parameters of this centrifugal compressor.

Parameter	Value
Impeller inlet diameter (tip)/mm	133.15
Impeller inlet diameter (root)/mm	44.60
Impeller outlet diameter/mm	182.88
Diffuser outlet diameter/mm	304.80
Impeller axial width(tip)/mm	58.00
Impeller axial width(root)/mm	61.40
Diffuser axial width/mm	12.13

2.2. Mesh and numerical simulation

The commercial software Turbogrid and ICEM are used to generate the mesh of this compressor system. Structured H-type mesh is used for impeller passage, vaneless diffuser and exit pipe. In addition, unstructured tetrahedral mesh is employed for the volute, and volute tongue region is refined partly. Fig. 1 presents the grids of these main components. The total number of computational mesh for the whole compressor system is 2.9 million.

The three-dimensional Reynolds-averaged Navier–Stokes equations (RANS), based on the SST turbulence model, were adopted in this work, and they are solved by the commercial software CFX. The inlet boundary condition is total pressure (300 K) and total temperature (101,325 Pa); and the outlet boundary condition is static pressure. The adiabatic with non-slip boundary condition is applied to all the solid walls. In the steady simulation, the interface between impeller outlet and diffuser inlet is adopted by frozen rotor interface, whereas it is adopted by transient rotor-stator interface in the unsteady simulation. A physical exit pipe is employed at downstream of the diffuser outlet, which is used to control mass flow rate of the compressor. The steady simulation is implemented firstly, and then the results of steady simulation are used as the initial condition for unsteady simulation. In the unsteady simulation, the physical time step is considered as 1.131e-5s. More detailed description about this numerical approach has been documented in previous work by Zhu et al. (2015).

2.3. Results and analysis

Fig. 2 presents the performance results obtained by the numerical simulation. The results obtained by the numerical simulation agree pretty well with the experiment data. In unsteady simulation, the mass flow rate gradually decreases from 2.2 kg/s. As shown in Fig. 2, the performance curve of total pressure ratio peaks when the mass flow rate is 1.49 kg/s. Then, with the reducing the mass flow rate, the total pressure ratio decrease as well, which means the occurrence of surge (Davis and Yao, 1971).

In this case, the rotational speed of the shaft is 44,198 rpm, and thus the shaft rotating frequency is 736.6 Hz. Considering that both the number of main blade and splitter blade are 6, we can obtain that the main blade passing frequency (BPF) is 4419.8 Hz and the total blade passing frequency is 8839.6 Hz. Frequency analysis is applied to the pressure signal collected by the monitor point, which is disposed in vaneless diffuser. Fig. 3 shows the FFT results of static pressure when mass flow rate is 1.4 kg/s and 1.8 kg/s (design condition). For the 1.8 kg/s working condition, the results indicate that there are two dominant frequencies (4420.9 Hz and 8841.7 Hz) in the FFT spectrum, which are the corresponding values of the main BPF and total BPF. However, besides the two blade passing frequencies (BPFs), another dominating frequency 233.5 Hz can also be found when the mass flow rate is 1.4 kg/s, and this specific frequency is approximately 0.3 times of the shaft rotating frequency (736.6 Hz). Therefore, this operating condition is regarded as mild surge condition (Hoying et al., 1998). The following research in this paper is based on this mild surge condition (mass flow rate = 1.4 kg/s).