Experimental Thermal and Fluid Science 78 (2016) 137–146

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

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

Experimental and numerical flow field investigation through two types of radial flow compressor volutes

Mohammad Mojaddam [a,*], Ali Hajilouy-Benisi [b]

[a] Department of Mechanical and Energy Engineering, Shahid Beheshti University, Tehran, Iran
[b] School of Mechanical Engineering, Sharif University of Technology, Tehran, Iran

ARTICLE INFO

Article history:
Received 29 February 2016
Received in revised form 21 May 2016
Accepted 25 May 2016
Available online 26 May 2016

Keywords:
Radial flow compressor
Volute
Flow field investigation
Five-hole probe

ABSTRACT

Being able to optimal design and fabrication of compressors requires understanding of the flow structure through the compressor components. Volutes are commonly the last main component in radial flow compressors. The stream passes through inlet, impeller and diffuser and finally is collected and discharged to the downstream pipeline by a volute. Volute shape has a direct and non-negligible effect on the compressor performance and its stable operating range. This component causes distorted pressure field in the upstream flow passages which could lead to the aero-mechanical forces acting on the impeller. As the flow inside the volute is fully three dimensional and turbulent, better understanding of flow mechanism has priority before modifying the volute design procedure.

In this research the experimental and numerical flow investigation through a radial flow compressor volute are performed to recognize the flow structure. The whole compressor components including the inlet, impeller, vane-less diffuser and volute are modeled and the flow structure inside the volute is captured. The three dimensional flow field model of the compressor was obtained numerically solving Navier–Stokes equations with shear stress transport turbulence model. Flow field investigation through volute cross section is performed utilizing a five-hole probe.

As the original volute is overhang type volute, the new external type volute is designed and fabricated based on the optimum design procedure. The results which are used for numerical model verification show that stage pressure ratio and total to total isentropic efficiency are increased by 2.5% and 1.9%, respectively at 70,000 rpm rotational speed using the new fabricated volute. The flow field inside the volute is also captured and compared to the experimental results obtained from the original volute setup and also for verifying the numerical model.

1. Introduction

Radial flow compressors are broadly used in the turbochargers for transportation vehicles. Normally turbocharger radial flow compressors have three major parts, the impeller, the vane-less diffuser and the volute. Exploring the flow structure and loss mechanism inside each compressor component helps to improve the design procedures to increase the machine performance.

As the flow field inside the volute is fully three dimensional and complicated, measurement technology for capturing flow field patterns are limited and expensive regarding the strong swirl of fluid entering the volute and the turbulent flow inside it [1]. Implementing the measurement devices and CFD techniques allows prediction of the flow structure inside this component.

Ayder et al. used a five hole probe to investigate the flow inside an overhang elliptical cross section volute to give a better insight into the three-dimensional flow structure at different mass flow rate conditions [2,3]. They developed a three-dimensional volute calculation method for this purpose to predict the velocity and pressure distribution inside and at the upstream parts of the volute [3].

Hagelstein et al. investigated an outward volute experimentally and numerically to gain further knowledge about the flow structure in the volute. A Cobra probe is used to recognize the flow pattern at each cross section [4].

Gu et al. performed a numerical simulation on a single-stage centrifugal compressor using CFX–TASC flow and reported the flow mechanism which results the higher energy loss at off-design mass flow rates [5]. Kim et al. implemented CFD simulation to investigate the effect of volute inlet by comparing velocity distribution,

* Corresponding author.
E-mail address: m_mojaddam@sbu.ac.ir (M. Mojaddam).

http://dx.doi.org/10.1016/j.expthermflusci.2016.05.016
0894-1777/

Nomenclature

C_Y	yaw coefficient (–)
C_p	pitch coefficient (–)
C_v	dynamic pressure coefficient (–)
C_{PT}	total pressure coefficient
m	Pfleiderer constant (–)
MP	mass parameter $= \dot{m}\sqrt{T_{0in}}/P_{0in}$ (kg s^{-1} K$^{-0.5}$ bar^{-1})
P	pressure (bar)
r_c	cross section radius (m)
r_d	diffuser discharge radius (m)
V	velocity vector (m s^{-1})
Z	volute cross section traverse position (m)
Z^*	non-dimensional traverse position (–)
β	pitch angle
γ	yaw angle
σ^2	the variance (the average of the squared differences from the mean)

Subscript

ave	arithmetic mean value
$1, 2$	first and second conditions
$P1 - P5$	the five hole probe pressures
r, θ, z	cylindrical coordinate
Y	related to yaw angle
P	related to pitch angle

static pressure recovery and total pressure loss coefficients through different planes of the volute cross sections [6].

Dai et al. studied the interaction between an impeller with one large and one small overhang volute by computational fluid dynamics simulation. The effect of volute tongue on the flow structure in the volutes was also investigated in the detail [7].

Zheng et al. investigated the influence of the volute on the flow in a high-pressure ratio radial flow compressor at the off-design conditions using CFD and computed the compressor performance characteristics with and without an overhang volute [1].

Experimental investigation by Bartlet et al. addressed the compressor behavior for reduced dimensions of volutes with rectangular cross sections. Five-hole probe is used to measure the velocity and pressure fields to verify the numerical CFD code at some operating points [8].

Mojaddam et al. investigated the effect of volute cross section shape for the external volutes using the numerical investigation which was verified through experimental data [9]. They also integrated a procedure for external volute design to define the volute area at exit, circumferential cross section variation, exit cone design and tongue area considerations by studying the effects of these parameters on four characteristics, including volute total pressure ratio, static pressure recovery coefficient, total pressure loss coefficient and the net radial force on the impeller [10].

In current study, the flow field measurements are performed for an intermediate volute of a turbocharger compressor using a five-hole probe through the volute cross sections in the different mass flow rates. The volute is utilized at the turbocharger compressor test rig at Turbocharger Laboratory of Sharif University of Technology where the compressor characteristic curves are obtained and flow field measurements are performed for different operating conditions.

The flow field study conduct on a new volute which was designed fabricated and installed on the same impeller and vaneless diffuser section. The numerical model is prepared and the results are verified using experimental data which shows the good agreement. The results are used to explain a mechanism to increase the velocity non-uniformities in the intermediate type volute which could be considered as a source of further loss.

2. Compressor geometry

A compressor of a turbocharger is investigated as the reference geometry which is primary installed and tested. The geometry of compressor components is obtained using the multi slice CT scan. 3D point cloud of scanning is used for geometry construction. The blade profiles and variable thickness must be captured precisely, hence, the mesh generated by the point cloud was processed at hub, mid and shroud spans and their respective profiles are extracted by digitizing [11].

The compressor impeller has six full blades and six splitter blades which start at 30% of impeller axial length. The impeller discharge diameter and height are 82 mm and 5.5 mm, respectively with 60° outlet angle related to radial direction. Inlet tip and hub diameters are 54 mm and 22 mm, respectively and the inlet angle is 30° related to a plane normal to the machine axis. The vane-less diffuser contains two parallel walls with radius ratio of 1.61.

Commonly the volutes are classified considering the locations of cross section area center (r_c) related to diffuser outlet radius (r_d). In external type volutes the whole area passage is located outside the diffuser exit radius, however internal volute cross sections are inside the diffuser exit radius (Fig. 1). The latter type is used to reduce the compressor size which introduces the new source of losses by adding a bend in the flow path. Intermediate or semi-external volutes are placed between two mentioned limits, where the cross section mean radius is nearly equal to the diffuser discharge radius [12].

The tested compressor volute, named here as the original volute, is an intermediate type. For investigating the effect of volute externality on flow field and performance, a new volute is designed, fabricated and tested, named here as a new volute.

3. Flow modeling

The flow field through compressor components is modeled and analyzed using a three dimensional viscous flow solver. The discretization of the equations is done via segregated implicit method in which energy equation is solved independent of momentum and mass equations. SIMPLEC algorithm is used for decoupling the pressure and the velocity and the Reynolds stress terms in the

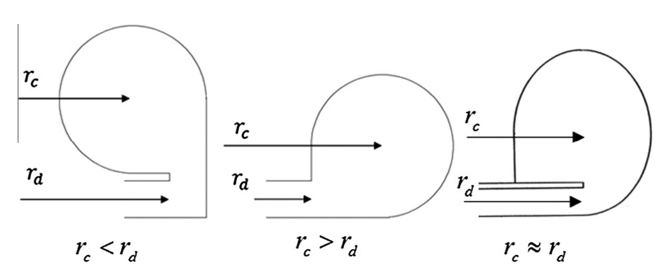

Fig. 1. Internal, external and intermediate volute. [12]