Propulsion and Power Research 2015;4(2):72–83

http://ppr.buaa.edu.cn/

Propulsion and Power Research

www.sciencedirect.com

Characteristics of transonic moist air flows around butterfly valves with spontaneous condensation

A.B.M. Toufique Hasan[a,*], S. Matsuo[b], T. Setoguchi[b]

[a]Department of Mechanical Engineering, Bangladesh University of Engineering and Technology (BUET), Dhaka-1000, Bangladesh
[b]Department of Advanced Technology Fusion, Saga University, Saga 8408502, Japan

Received 10 November 2013; accepted 29 April 2015
Available online 26 June 2015

KEYWORDS

Transonic flow;
Shock waves;
Spontaneous condensation;
Butterfly valves;
Reynolds-averaged Navier-Stokes (RANS)

Abstract Effects of spontaneous condensation of moist air on the shock wave dynamics around butterfly valves in transonic flows are investigated by experimental and numerical simulations. Two symmetric valve disk shapes namely- a flat rectangular plate and a mid-plane cross-section of a prototype butterfly valve have been studied in the present research. Results showed that in case with spontaneous condensation, the root mean square of pressure oscillation (induced by shock dynamics) is reduced significantly with those without condensation for both shapes of the valves. Moreover, local aerodynamic moments were reduced in case with condensation which is considered to be beneficial in torque requirement in case of on/off applications of valves as flow control devices. However, total pressure loss was increased with spontaneous condensation in both the valves. Furthermore, the disk shape of a prototype butterfly valve showed better aerodynamic performances compared to flat rectangular plate profile in respect of total pressure loss and vortex shedding frequency in the wake region.

1. Introduction

Investigation on the aerodynamic characteristics of butterfly valves has been performed for the last many years due to their technological and industrial importance. This type of valves are used intensively in aircraft outflow valves to regulate the cabin pressure in pressurized aircraft [1], in hydro-electric power schemes as safety valves [2] as well as in nuclear containment

*Corresponding author. Tel.: +(880) 1730714444.
E-mail address: toufiquehasan@me.buet.ac.bd (A.B.M. Toufique Hasan).
Peer review under responsibility of National Laboratory for Aeronautics and Astronautics, China.

http://dx.doi.org/10.1016/j.jppr.2015.05.003

Nomenclature

c	chord length of butterfly valve (unit: mm)
f	frequency of shock oscillation (unit: kHz)
g	condensate mass fraction (unit: -)
H	height of the channel (unit: mm)
I	nucleation rate (unit: $1/(m^3 \cdot s)$)
k	turbulent kinetic energy (unit: m^2/s^2)
Ma	Mach number (unit: -)
p	static pressure (unit: kPa)
R	undamped eddy viscosity (unit: m^2/s)
Re	Reynolds number (unit: -)
S	degree of supersaturation (unit: -)
RMS	root mean square (unit: -)
x	streamwise coordinate (unit: mm)
y	normal coordinate (unit: mm)
T	time period (unit: s) / temperature (unit: K)
t	time (unit: s) / thickness (unit: mm)

Greek symbols

β	integrated total pressure loss (unit: -)
θ	angle of attack (unit: °)

Subscripts

ave	average
b	back pressure
l	lower surface
u	upper surface
0	upstream condition
01	total/stagnation condition at inlet

purge valves as flow controllers [3,4] and so on. Butterfly valve disks behave in the same manner as airfoils that the angle of attack influences the flow field characteristics [5]. However, under a certain combination of valve disk opening angle and pressure difference across the valve, compressible flow effects can significantly alter the performance characteristics and flow field of a butterfly valve [6]. At these conditions, regions of transonic and supersonic flow can develop in the vicinity of the valve disk and downstream of it. Morris and Dutton [7] experimentally studies the flow field and torque of a compressible fluid in a butterfly valve. The 2-D experimental investigations were performed over a range of disk positions and pressure ratios. Recently, butterfly valve performance coefficients (such as lift, drag, resultant force and torque coefficients) in compressible flow field were predicted using computational fluid dynamics (CFD) [4]. However, in transonic or supersonic flow field, expansion of vapor/carrier gas mixture (moist air) or steam is often so rapid that the flow field gives rise to spontaneous (homogeneous and non-equilibrium) condensation process [8,9]. In this process, first the vapor molecules itself generate condensation nuclei spontaneously by molecular collision, and secondly, the condensation of the vapor molecules take place on these nuclei (droplet growth). This condensation process releases thermal energy to the surrounding gaseous medium and considerably modifies their thermo-fluid behaviors. This phenomena is evident by the presence of water droplet which was seen in the exhaust plume during the experimental testing performed by Leutwyler and Danban [4] and responsible for the difference between previous numerical and experimental results. Thus, to predict accurately the flow field around butterfly valve in compressible flows, working fluid must need to be considered as multiphase fluid.

In the present study, the effect of moist air with the occurrences of spontaneous homogeneous condensation on transonic internal flow around a symmetric disk butterfly valve is investigated. Two valve disk shapes have been studied in the present study: a flat rectangular plate profile and a mid-plane cross-section of a prototype butterfly valve. Results are presented for various aerodynamic aspects for the case of moist air. Further the results are compared with those of no condensation of dry air.

2. Numerical methods

2.1. Governing equations

The transonic flow through the symmetric disk butterfly valves is governed by the unsteady compressible Navier-Stokes equations written in two-dimensional coordinate system (x, y). To link the heat supply by condensation process, a rate equation of liquid-phase production [10] was coupled. As a turbulence model, a modified two-equation k-R model [11,12] was used in the present computation where k and R are the turbulent kinetic energy and the undamped eddy viscosity, respectively. These equations were discretized by the finite difference technique. A third order TVD (total variation diminishing) finite difference scheme with MUSCL [13] was employed to discretize the spatial derivative, and a second order central difference scheme for viscous terms, and a second order fractional step method was used for time integration. For simplicity of the computation, assumptions are as follows; no velocity slip and no temperature difference between condensate particles and gas medium, and the effect of condensate particles on pressure is negligible.

Figure 1 shows the computational domain of the flow field, details of test section and two different valve geometries (rectangular plate and mid-plane cross-section of a prototype butterfly valve). The prototype butterfly valve is similar to that used in previous study [3]. Chord length, c and the thickness of the valve, t are 48 mm and 7.2 mm (aspect ratio: $t/c = 0.15$), respectively. The height of the test section H is 60 mm ($H/c = 1.25$). Angle of attack of the valve is denoted by θ; where θ is the angle between x-axis (reference axis) and the line joining the leading and trailing edges of the valve (Figure 1(c)). Computational domain is