International Journal of Heat and Mass Transfer 88 (2015) 165–180

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Influence of turbulence modelling on non-equilibrium condensing flows in nozzle and turbine cascade

Yogini Patel *, Giteshkumar Patel, Teemu Turunen-Saaresti

Fluid Dynamics Laboratory, LUT Energy, School of Technology, Lappeenranta University of Technology, PO Box 20, FIN-53851 Lappeenranta, Finland

ARTICLE INFO

Article history:
Received 17 December 2014
Received in revised form 23 April 2015
Accepted 23 April 2015
Available online 14 May 2015

Keywords:
CFD
Condensation
Steam
Turbulence modelling
Loss coefficient

ABSTRACT

The accurate analysis of a condensing flow plays an important role in the development of high-efficiency steam turbines. This paper presents an investigation of turbulence modelling influence on non-equilibrium condensing steam flows in a Laval nozzle and in a stationary cascade of turbine blades using a commercial computational fluid dynamics (CFD) code. The calculations were conducted by employing 2D compressible Reynolds-averaged Navier–Stokes (RANS) equations coupled with a two equation turbulence model. The condensation phenomena were modelled on the basis of the classical nucleation theory. The standard k–ε turbulence model was modified, and the modifications were implemented in the CFD code. The influence of inlet flow turbulence on condensing process was discussed. The impact of turbulence modelling on wet-steam flow was examined based on the experimental data available in the literature. The cascade loss coefficients were calculated numerically as well. The presented study of losses that occur due to the irreversible heat and mass transfer during the condensation process emphasised the importance of turbulence modelling for wet-steam flows in turbines. The paper demonstrates that the accurate computational prediction of condensing steam flow requires the turbulence to be modelled accurately.

1. Introduction

Two-phase wet-steam flows consisting of tiny liquid droplets have a fundamental importance in both scientific contexts and in industrial applications, such as the combustion of liquid droplets, meteorological processes, the formations of contrails from aircraft exhausts, chemical engineering applications, and condensation of steam in turbines. Particular attention has been given to wet-steam flow research in turbines to take into account the thermodynamical and mechanical losses, blade erosion, and the influence of the condensation process on blade aerodynamics. With a large role played by steam turbines in the area of power generation, any progress in understanding the condensation and loss mechanisms might lead to improved designs of steam turbines, and, as a result, yield handsome economic dividends. Therefore, detailed analysis of condensing steam flow, either by experiments or with numerical simulations, has great importance.

The condensing steam flow in nozzles and in turbines has been widely studied experimentally, theoretically, and numerically since the work of Stodola [1] in order to enhance knowledge about the complicated physics involved. Comprehensive experimental works have been organised for condensing flow in the nozzle by numerous researchers, that is, Barschdorff [2], Moore et al. [3], Bakhtar et al. [4], Moses and Stein [5], Skillings et al. [6], Bakhtar and Zidi [7,8], Gyarmathy [9]. The experimental work for condensing steam flows in turbine cascades has been performed as well. For example, Bakhtar et al. [10,11] and White et al. [12]conducted experiments of non-equilibrium condensing steam flow in turbine cascades, in which they provided a large set of measurement data for various parameters.

Many numerical studies over the past several decades have been directed toward modelling condensing steam flow utilising various approaches, where the vapour phase is always treated by the Eulerian method and the liquid phase is solved by the Lagrangian/Eulerian method. Much of the modelling work was initially performed on convergent–divergent (CD) nozzles with simplified one-dimensional flow, considering both the inviscid and turbulence conditions. Later on, studies were dedicated to two-dimensional flows in turbine cascades, with more sophisticated numerical models utilised to handle the additional dimension. For example, in the studies of Bakhtar and Tochai [13], Young [14,15], White and Young [16], Bakhtar et al. [17], White et al. [12], the most often used numerical approach was the

---

* Corresponding author. Tel.: +358 40 765 2489.
E-mail addresses: yogini.patel@lut.fi (Y. Patel), giteshkumar.patel@lut.fi (G. Patel), teemu.turunen-saaresti@lut.fi (T. Turunen-Saaresti).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2015.04.069
0017-9310/

Nomenclature

Latin alphabet

C_p	specific heat at constant pressure	$\mathrm{J\,kg^{-1}\,K^{-1}}$
C_v	specific heat at constant volume	$\mathrm{J\,kg^{-1}\,K^{-1}}$
h	specific enthalpy	$\mathrm{J\,kg^{-1}}$
H	total enthalpy	$\mathrm{J\,kg^{-1}}$
I	nucleation rate	$\mathrm{m^{-3}\,s^{-1}}$
k	turbulent kinetic energy	$\mathrm{m^2\,s^{-2}}$
K_t	thermal conductivity	$\mathrm{W\,m^{-1}\,K^{-1}}$
M	liquid mass	kg
M_m	molecular mass	$\mathrm{kg\,mol^{-1}}$
P	pressure	Pa
r	radius	m
$\bar{r}$	average radius	m
r_*	critical radius	m
R	gas constant	$\mathrm{J\,kg^{-1}\,K^{-1}}$
Re_λ	Reynolds number	
s	entropy	$\mathrm{J\,kg^{-1}\,K^{-1}}$
S_1	mass source term	$\mathrm{kg\,m^{-2}\,s^{-1}}$
S_2	momentum source term	$\mathrm{kg\,m^{-2}\,s^{-2}}$
S_3	energy source term	$\mathrm{W\,m^{-3}\,K^{-1}}$
t	time	s
T	temperature	K
u	velocity component	$\mathrm{m\,s^{-1}}$

Greek alphabet

β	liquid phase mass fraction	
γ	specific heat ratio	
Γ	mass generation rate	$\mathrm{kg\,m^{-3}\,s^{-1}}$
Γ_E	thermal diffusion coefficient	$\mathrm{W\,m^{-1}\,K^{-1}}$
ε	turbulence dissipation rate	$\mathrm{m^2\,s^{-3}}$
η	number of liquid droplets per unit volume	$\mathrm{m^{-3}}$
μ	dynamic viscosity	Pa s
μ_t	turbulent viscosity	$\mathrm{kg\,m^{-1}\,s^{-1}}$
ρ	density	$\mathrm{kg\,m^{-3}}$
σ	liquid surface tension	$\mathrm{N\,m^{-1}}$
τ	viscous stress tensor	Pa
τ_p	droplet response time	s
χ	turbulence intensity	

Subscripts

d	droplet
m, l, v	mixture, liquid phase, vapour phase
i, j	cartesian tensor notation
x	cartesian coordinate
$0, 1, 2$	total, inlet, outlet condition of cascade

inviscid time-marching scheme of Denton [18] for turbomachinery flows.

An Eulerian–Lagrangian approach for two-phase steam flow was adopted by Gerber [19] to conduct numerical simulations of low-pressure (LP) CD nozzle and turbine cascade. He utilised the k–ε turbulence model by assuming oneway coupling between the gas phase turbulence and dispersed droplets. Later on, Gerber and Kermani [20] presented an Eulerian–Eulerian multi-phase method for non-equilibrium condensation in nozzles. Moreover, Senoo and White [21,22] utilised a coupled numerical approach, considering the two-phase flow as a mixture, to simulate inviscid wet-steam flow in LP steam turbine stator cascade and in Laval nozzle. Wróblewski et al. [23], Dykas and Wróblewski [24,25] have developed in-house CFD codes for modelling non-equilibrium wet-steam flow. They validated their code using the single-fluid and two-fluid approaches coupled with both one-equation and two-equation turbulence models as well. Moreover, numerical work concerning wet-steam flows through multistage stator rotor cascade channels in a low-pressure steam turbine were performed by Yamamoto et al. [26–28], Starzmann et al. [29], Miyake et al. [30], who solved flow turbulence using the SST k–ω turbulence model.

Turbine flows include a variety of complex flow phenomena, including laminar-to-turbulent transition, flow separation, secondary flow mixing, rotor–stator interaction, and heat transfer. A common thread among all of these phenomena is turbulence. The turbulence plays an important role in the processes of mass, momentum, and heat transfer in boundary layers on the surface walls, especially on the possible deposition of condensed liquid droplets. Turbulence may have some direct/indirect influence on shock wave generation under the conditions of subcooled steam flow [31]. Additionally, the accurate prediction of absolute losses requires the turbulence to be modelled accurately [32,33].

However, published work on the influence of turbulence on the condensing steam flow is rather sparse. White [34] presented a numerical method based on a simple stream function technique for the prediction of condensing steam flow in a CD nozzle to analyse the influence of the viscous effect on condensation within compressible boundary layers. Moreover, Simpson and White [35] conducted a numerical study performing viscous calculations for a steady flow condition with CD nozzle using the standard k–ε turbulence model, and they concluded that the growth of the boundary layer has a significant impact on the predicted pressure distributions and droplet sizes. Additionally, Avetissian et al. [36] investigated the influence of the turbulence level and inlet wetness on the process of spontaneous condensation in Laval nozzles, utilising the moment method and the Delta-approximation method to determine the droplet size spectrum. They concluded that the effect of both high-level turbulence and inlet wetness causes the shock of spontaneous condensation to disappear. Later on, the effect of turbulence was investigated by Avetissian et al. [31]. Their study emphasised the steady and unsteady spontaneously condensing transonic turbulent flows in 2D flat nozzles and round shape nozzle dealing with and without initial moisture at the nozzle inlet. Additionally, the influence of turbulence parameters and real gas models in condensing steam flow in a CD nozzle has been studied by Patel et al. [37]. The performance of various turbulence models for the wet-steam flow has been studied by Patel et al. [38]. In the work of Patel et al. [38], the SST k–ω model was modified to predict steam condensing flow and losses in turbine cascade. It was concluded that the prediction of steam condensing flow in turbine cascade is influenced by turbulence.

The aim of this work is to investigate the influence of turbulence modelling on the process of spontaneously condensing flow in a nozzle and turbine blade cascade using the Eulerian–Eulerian approach. The significance of turbulence modelling on the loss mechanism is discussed, as well. The achieved numerical results are analysed with the available experimental data.

2. Governing equations

2.1. Conservation equations

All results presented in this paper were obtained by means of ANSYS Fluent 14.0 CFD code. The mixture of vapour and liquid phases was governed by Reynolds-averaged Navier–Stokes