Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 34–41

Energy Procedia

www.elsevier.com/locate/procedia

72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6-8 September 2017, Lecce, Italy

CFD modelling of the condensation inside a Supersonic Nozzle: implementing customized wet-steam model in commercial codes

Mazzelli Federico[a]*, Giacomelli Francesco[a], Milazzo Adriano[a]

[a] *University of Florence, Department of Industrial Engineering of Florence (DIEF), via di S. Marta 3, 50139, Firenze, Italy.*

Abstract

Non-equilibrium condensation of steam occurs in many jet and turbomachinery devices, such as supersonic nozzles, ejectors and across the last stages of steam turbines. Wet steam models are available in many commercial CFD codes and can represent the metastable behaviour of the flow with reasonable accuracy. Unfortunately, the use of built-in models does not allow freedom in the choice of model parameters and settings. In the present paper, a numerical model for the simulation of wet steam flow has been developed and implemented within a commercial CFD code (ANSYS Fluent) via user defined functions. The scheme is based on a single-fluid approach and solves the transport equation for a homogeneous mixture flow coupled with conservation equations for the number of droplets and liquid mass fraction. The model is compared against a well-known steam nozzle test-case.

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association

Keywords: CFD; steam; non-equilibrium; condensation; steam turbine.

1. Introduction

Non-equilibrium condensation of steam occurs in many jet and turbomachinery devices, such as supersonic nozzles and across low pressure stages of steam turbines. Normal operation of these devices involves flow expansions which leads to states that are well within the saturation dome. In the ideal case of a reversible transformation, the attending condensation process would follow a path of equilibrium states, and no losses occur. In real conditions, the very limited residence time and high cooling rates lead to a substantial departure from the equilibrium process. As the steam rapidly

* Corresponding author. Tel.: +39 055 275840.
E-mail address: federico.mazzelli@unifi.it

1876-6102
Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.053

expands inside a nozzle or blade vane, thermodynamic equilibrium is not maintained and, at a certain degree of expansion, the vapor state collapses and condensation takes place abruptly as a shock-like disturbance [1]. This is generally called the "condensation shock". This sudden change of state of aggregation leads to an instantaneous and localized heat release that increase the pressure and temperature and reduce the Mach number [1]. More than this, the condensation shock implies large gradients between the phases that cause irreversibilities. Moreover, downstream the condensation shock, the flow contains a considerable number of tiny liquid droplets (of the order of $10^{19}/dm^3$, [2]) that can interact in non-trivial ways with shock waves and turbulent structures. A reliable CFD scheme should be able to account for all these effects.

In the past decades, several methods have been devised to simulate wet steam flows, with different levels of complexities and accuracy. The simplest and perhaps most used is the so-called "single-fluid" approach. This is basically a fully Eulerian method that solves the continuity equation for both phases separately, whereas the momentum and energy equations are computed for the average properties of the mixture. In addition, a further transport equation is needed to describe the conservation of the number of droplets in the unit volume. This method is commonly employed by commercial codes (e.g. ANSYS Fluent or CFX) and has been used by several research teams [3] [4] [5].

Although commercial codes dispense from developing complex in-house solvers, the use of wet-steam built-in models generally do not allow much freedom in the change of the physical parameters and settings. This work represents an attempt to overcome this limitation through the development of a customized model within a widely used CFD commercial code (ANSYS Fluent, [6]). This approach has the double benefit to allow great flexibility in the choice of the physical model setting (especially for phase change and phase interaction models) and, at the same time, to exploit the capability of commercial software in terms of selection of algorithms and solver settings. The developed scheme is based on a single-fluid approach (mixture model) and is tested and compared against a well-known steam nozzle test-case and a 2D stationary blade cascade.

Nomenclature

h	latent heat [J kg^{-1}]
J	nucleation rate [s^{-1} m^{-3}]
k	Boltzmann constant [J K^{-1}]
m	mass [kg]
n	number of droplets per unit mass of mixture [kg^{-1}]
p	pressure [Pa]
R	specific gas constant [J kg^{-1} K^{-1}]
r	radius [m]
T	temperature [K]
u	velocity [m s^{-1}]
greek letters	
α	volume fraction [-]
β	mass fraction [-]
Γ	liquid mass generation rate [kg m^{-3} s^{-1}]
γ	specific heat ratio [-]
ρ	density [kg m^{-3}]
σ	surface tension [J m^{-2}]
φ_{ss}	supersaturation ratio [-]
subscripts	
d	droplet
m	mixture, molecule
v	vapour