Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 730–737

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 cascade of steam turbine blades: comparison with an experimental test case

Giacomelli Francesco[a*], Mazzelli Federico[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 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 Mixture-model approach. The model is compared with literature data for a 2D stationary cascade of a steam turbine blade.

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

The steam turbine still plays a key role in the production of electricity in power industry. One of the main problems concerning the efficiency of steam turbines are the losses stemming from the non-equilibrium condensation in the last stages, where irreversibility due to the non-ideal phase-change become significant. Since a significant share of the output power is produced in the last stages, the correct modeling of wet steam flow behavior is of great importance. Moreover, predicting the droplet formation and growth allows understanding, and possibly mitigating, some undesired phenomena such as the erosion of turbine blades due to droplet impacts.

* Corresponding author: francesco.giacomelli@unifi.it , +39 055 2758740

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.306

In the past years, the steam wetness losses were modeled considering a single-phase flow corrected with empirical correlations; to date, thanks to the improvement of CFD techniques for compressible multiphase flows, the accurate modeling of condensation has become accessible. The most advanced codes in this field are based on the Eulerian/Lagrangian approach, where the Eulerian phase calculations are coupled with explicit droplet integration (some examples can be found in [1] and [2]). However, the high computational cost of these methods make their use prohibitive in many circumstances. Consequently, many authors have developed Eulerian/Eulerian models [3] to enhance computational performances. These methods imply volume averaging of liquid phase equations and lead to a lower accuracy with respect to the direct tracking of droplet trajectories of the Eulerian/Lagrangian approach.

The validation of these models is generally performed by comparison with the several test-cases available on both supersonic condensing nozzles [4] [5] [6] and steam turbines cascades [7] [8] [9]. The results presented in this paper are obtained exploiting a Mixture model approach described in [10]. This kind of method is commonly embedded in commercial CFD solvers but the present approach has the double benefit of allowing great flexibility in the choice of the physical model and, at the same time, exploiting the capability of commercial software in terms of solver settings. The model has been implemented in the commercial CFD code ANSYS Fluent v.18.0 making extensive use of in-house developed routines. The results from steady simulations are compared with experimental data available from a condensing test case on a 2D stator blade of a steam turbine [7].

2. Numerical setup

The present model is validated against experimental data from a condensing, 2D, stationary steam turbine cascade [7]. The boundary conditions for the calculation are taken from the experimental tests conducted in a wet steam wind tunnel facility over a set of boundary conditions (the reader is referred to [7] for a more extensive description of the experimental facility). The geometry was obtained by a digitalization of the blade profile since the authors do not provide the exact shape.

The computational domain and mesh is depicted in Figure 1. The blade has a chord of 137.5 mm, a pitch of 87.6 mm and a stagger angle of 45.3°. The mesh is composed of approximately 130’000 quadrilateral elements. Simulations are performed using the commercial CFD package ANSYS FLUENT v18.0. Figure 1 also summarizes the type of boundary conditions used in the simulations.

The inlet total pressure is set to 40300 Pa with a total Temperature of 354 K while the outlet static pressure is 16300 Pa. The simulations are performed considering a two-phase compressible fully turbulent flow. The turbulence model adopted is a two equations k-ε realizable model [11] (in accordance to [12]) with enhanced wall treatment and the wall boundaries are assumed to be adiabatic. The solver is pressure-based with pressure-velocity coupling. A 3^{rd} order accurate QUICK scheme is used for the spatial discretization of all transport equations.

3. Results

The test case used for the validation of the model is a blade of a condensing steam turbine stationary cascade from reference [7]. The considered experimental case, named “L1” in ref. [7], was chosen because of the availability of more experimental results; moreover, the steam at the inlet is scarcely superheated, so that the effects of spontaneous condensation are clearly observable.

Figure 2 shows the pressure comparison of pressure measurements with different CFD schemes. The “WS-CFD” (red curve) refers to the results obtained with the standard Wet Steam Model available in ANSYS Fluent v.18.0 (for more details see [13]) and simulated on a coarser mesh of approximately 25000 elements. Differently from developed mixture model, the ANSYS Fluent standard scheme is built upon a density-based solver. The “CFD-Mixture” curve represents the results of the presented model on the coarser mesh. Finally “CFD-Mixture-Refined” is the pressure profile obtained with the mesh of 130’000 elements (see Figure 1).

Although the agreement between experiments and CFD is generally good, Figure 2 highlights the different sensitivity of pressure- and density-based solver to the mesh resolution. In particular, when focusing in the region of the “condensation-shock” (squared area on the suction side), the results for the pressure-based solver shows a comparable agreement with the density-based scheme only when using the refined mesh. This is probably due to the better capacity of the density-based solver to simulate flow conditions with steep gradients such as the presence of