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.

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

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 3rd 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
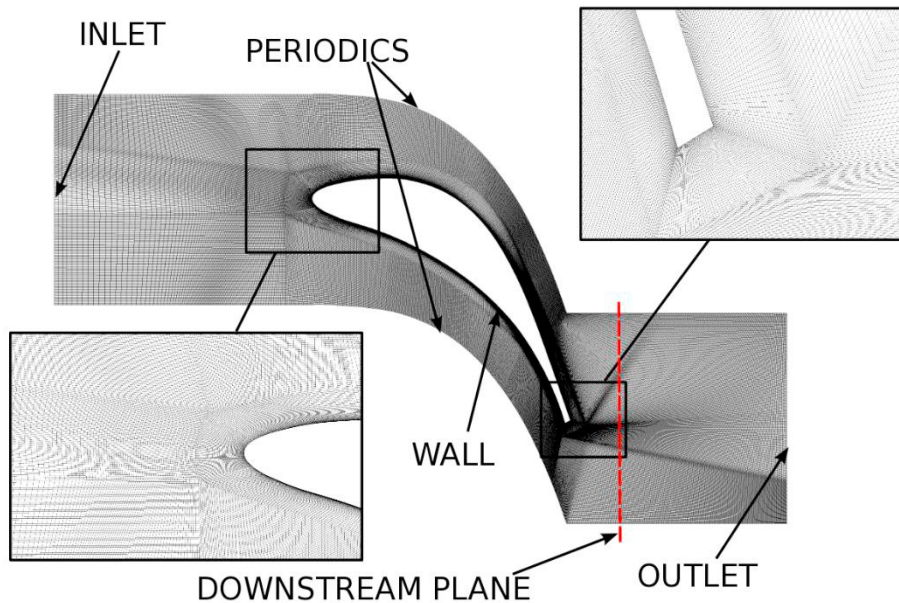
Figure 1: Computational grid with mesh details and boundary conditions

shocks or pressure rise due to spontaneous condensation. The differences between the numerical results and the experimental data upstream the condensation-shock position are probably due to the choice of the turbulence model which was chosen because it has been extensively validated and improves upon the standard k–ε model for a wide variety of flows. However, because the modification to the standard k– ε model is only valid for high Reynolds numbers, the model cannot be expected to provide any significant improvement in near-wall regions. The differences between the various CFD approaches downstream the condensation-shock are direct consequences of the different prediction of the pressure rise connected to the condensation-shock itself.

In what follows, only the results of the developed model on the refined mesh will be shown (which will be labelled simply as "CFD").

Figure 3 presents a comparison of the computed results with the experimental data obtained along a transverse section downstream of the turbine blade. The pressure measurements in reference [7] were made at one quarter of the axial chord length downstream of the blade trailing edge (the same position is used to obtain the CFD results and is highlighted in Figure 1 by means of the dotted red line). These results show the capability of the present model to predict the pressure rise across the downstream pressure shockwave (as can also be seen in the pressure contour of Figure 5). The sudden drop of liquid mass fraction plot in Figure 3 highlights the evaporation phenomenon of the droplets across the shock wave.

Figure 4 shows the density gradient contour plot (top) together with the nucleation rate map (bottom). The comparison of these two images allows to understand the different nature of the shocks caused by the condensation phenomenon and that stemming from adaptation of the supersonic stream to steep pressure gradients (the downstream pressure shockwave). Clearly, the condensation shock is connected with a strong rise in the nucleation rate, which is absent in the downstream shock.

Figure 5 adds to this interpretation by showing the pressure (Top) and supercooling level (Bottom) contour. In particular, both types of shockwave imply a steep pressure rise but the sign of supercooling variation is reversed in the two cases. Whereas the condensation shows a decrease in the supercooling upstream of the shock (due to the high expansion of the flow), the supercooling levels downstream of the dynamic shock reaches its highest value, proving that there is a partial evaporation of droplets in this region. This is in accordance with the liquid mass fraction plot of Figure 3 that shows a drop of the liquid mass fraction after the shock.
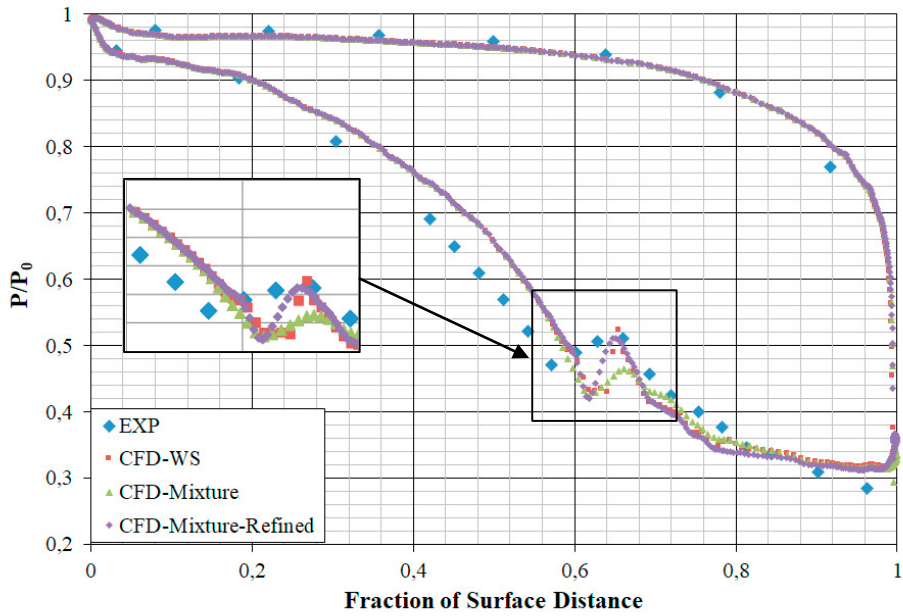
Figure 3: Numerical and experimental values of pressure on the surface of turbine blade
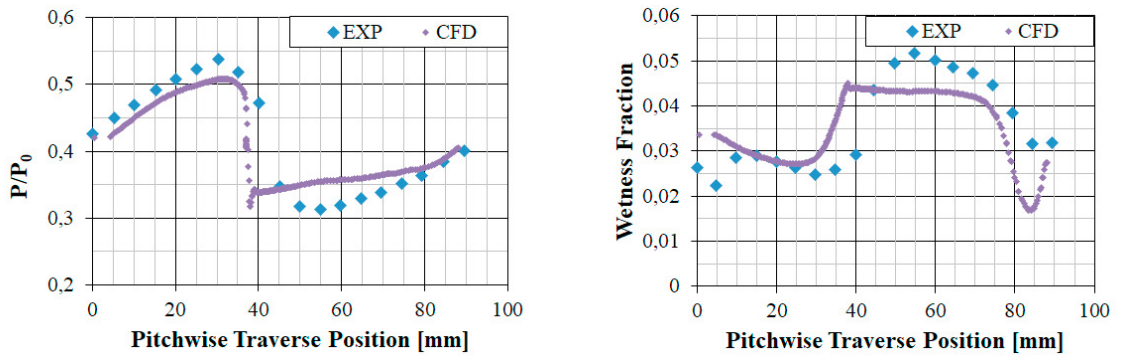

Figure 2: Numerical and experimental pitchwise data comparison. Pressure (LEFT) and Wetness Fraction (RIGHT)

4. Conclusions

Some preliminary CFD results obtained with a Mixture-model for simulating the non-equilibrium condensation of steam flows are presented and compared with experimental data on a stationary cascade of steam turbine blades available from literature. The model is based on in-house developed routines and implemented in the commercial software ANSYS Fluent v.18.0, this approach allows great flexibility in the choice of the numerical settings and future studies will address the analysis of different phase change and phase interaction models. The capability of the model to predict the main features of spontaneous condensation of steam was considered. Comparison of numerical results with experiments have shown a satisfactory agreement both in terms of pressure trends and of liquid mass fraction data. Future developments will focus on further validation of the presented approach and on its extension to the Eulerian-Eulerian multiphase model.
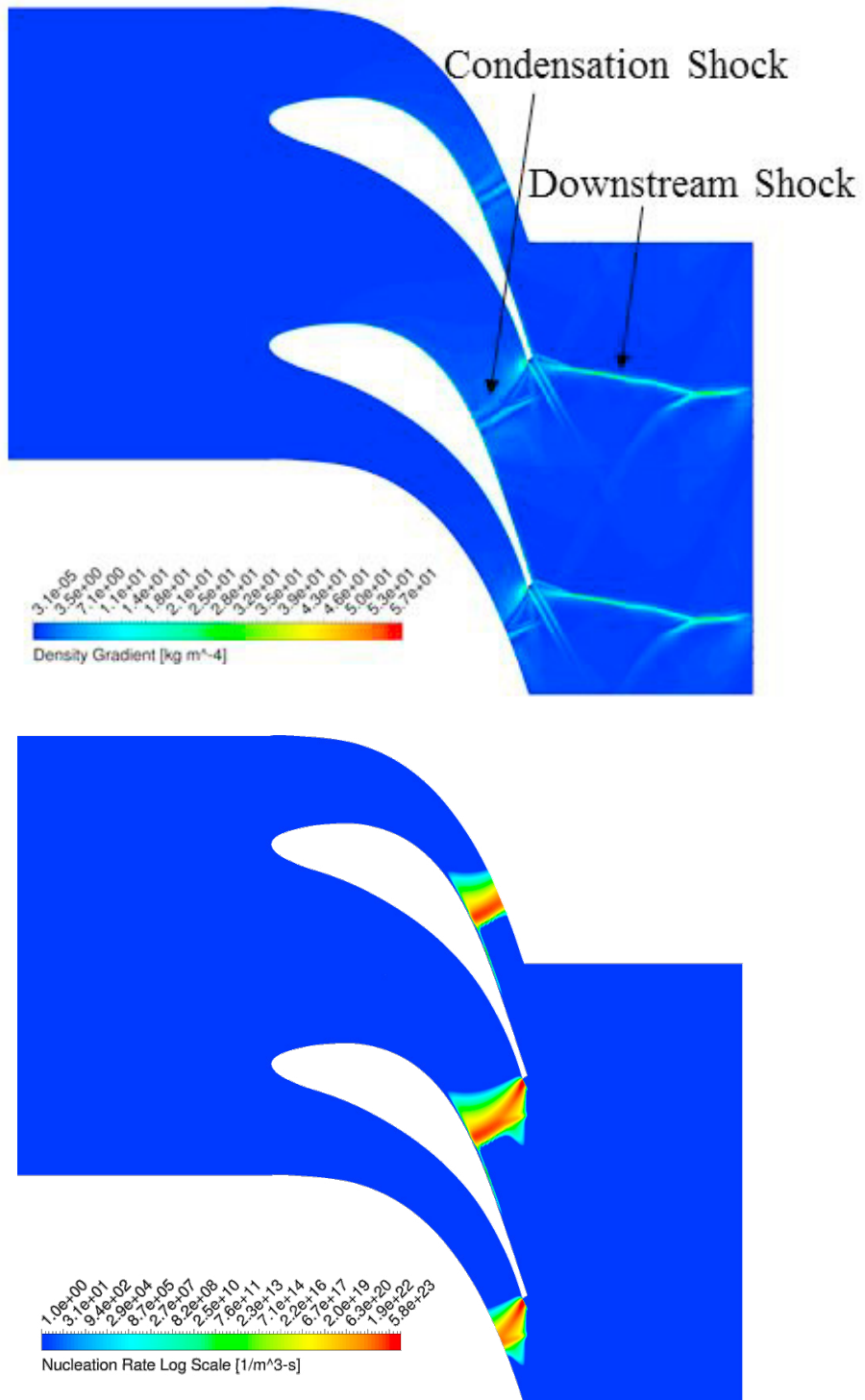
Figure 4: Density gradient contour (TOP) and nucleation rate contour (BOTTOM)
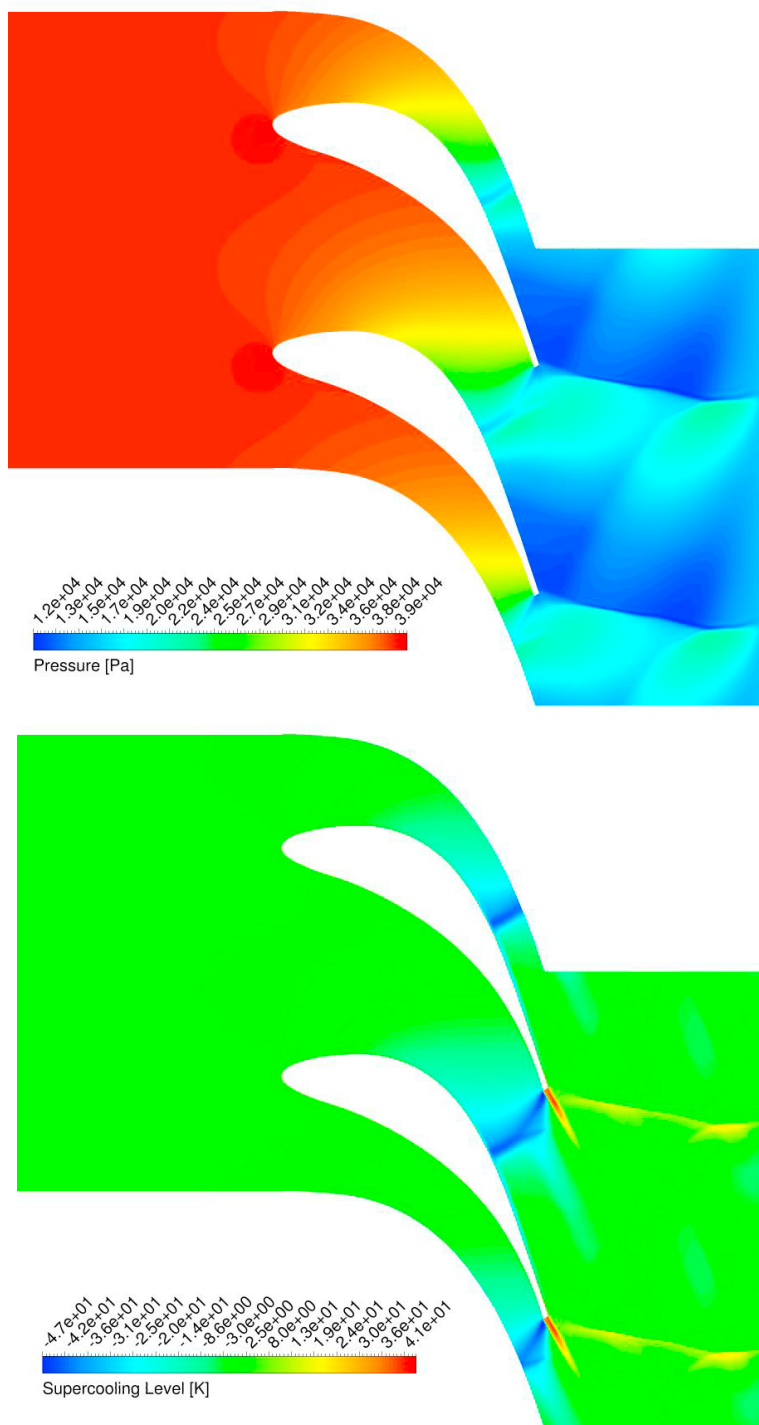
Figure 5: Pressure contour (TOP) and supercooling level contour (BOT)

References

[1] A. Gerber, «Two-phase Eulerian/Lagrangian model for nucleating steam flow,» *ASME J. Fluids Eng.,* pp. 465-475, 2002.

[2] M. J. Kermani e A. G. Gerber, «A general formula for the evaluation of thermodynamic and aerodynamic losses in nucleating steam flow,» *International Journal of Heat and Mass Transfer,* pp. 3265-3278, 2003.

[3] A. Gerber e M. Kermani, «A pressure based Eulerian-Eulerian multiphase model for non-equilibrium condensation in transonic steam flow,» *International Journal of Heat and Mass Transfer,* pp. 2217-2231, 2004.

[4] C. A. Moses e G. D. Stein, «On the growth of steam droplets formed in a Laval nozzle using both static pressure and light scattering measurements,» *Journal of Fluids Engineering,* pp. 311-322, 1978.

[5] M. J. Moore, P. T. Walters, P. I. Crane e B. J. Davidson, «Predicting the fog-drop size in wet-steam turbines,» 1973.

[6] J. Starzmann, F. R. Huges, A. J. White, J. Halama, V. Hric, M. Kolovratnik, H. Lee, L. Sova, M. Statny, S. Schuster, M. Grubel, M. Schatz, D. M. Vogt, P. Y., G. Patel, T. Turunen-Saaresti, V. Gribin, V. Tishchenko, I. Garilov e e. al., «Results of the International Wet Steam Modelling Project,» Prague, 2016.

[7] A. J. White, J. B. Young e P. T. Walters, «Experimental validation of condensing flow theory for a stationary cascade of steam turbine blades,» *Philos. Trans. R. Soc. Lond.,* pp. 59-88, 1996.

[8] F. Bakhtar, M. Ebrahimi e R. A. Webb, «On the performance of a cascade of turbine rotor tip section blading in nucleating steam, part1: surface pressure distribution,» in *Proc. Inst. Mec. Enngrs. Part C*, 1995.

[9] F. Bakhtar, M. Ebrahimi e B. O. Bamkole, «On the performance of a cascade of turbine rotor tip section blading in nucleating steam, part 2: wake traverses,» *J. Mech. Eng. Sci.,* pp. 169-177, 1995.

[10] F. Mazzelli, F. Giacomelli e A. Milazzo, «CFD modelling of the condensation inside a supersonic nozzle: comparison an experimental test case,» *Energy procedia (under review),* 2017.

[11] T. H. Shih, W. W. Liou, A. Shabbir, Z. Yang e L. Zhu, «A new k-ε eddy viscosity model for high reynolds number turbulent flows,» *Computer & Fluids,* vol. 24, n. 3, pp. 227-238, 1995.

[12] L. Zori e F. Kelecy, «Wet Steam Flow Modeling in a General CFD Flow Solver,» in *35th AIAA Fluid Dynamics Conference and Exhibit*, Toronto, Ontario, Canada, 2005.

[13] ANSYS Inc., ANSYS Fluent Theory Guide, Canonsburg, PA: release 18.0, 2016.