Application of a single-fluid model for the steam condensing flow prediction

K Smolka¹, S Dykas¹, M Majkut¹ and M Strozik¹

¹ Silesian University of Technology, Institute of Power Engineering and Turbomachinery, Konarskiego 18, 44-100 Gliwice, Poland

E-mail: Krystian.Smolka@polsl.pl

Abstract One of the results of many years of research conducted in the Institute of Power Engineering and Turbomachinery of the Silesian University of Technology are computational algorithms for modelling steam flows with a non-equilibrium condensation process. In parallel with theoretical and numerical research, works were also started on experimental testing of the steam condensing flow. This paper presents a comparison of calculations of a flow field modelled by means of a single-fluid model using both an in-house CFD code and the commercial Ansys CFX v16.2 software package. The calculation results are compared to in-house experimental testing.

1. Introduction
It is a long-known fact that a considerable part of losses arising in the steam flow through the turbine low-pressure stages is due to the two-phase character of the process. Despite the impressive progress made in the development of the computational fluid dynamics (CFD) methods, modelling the two-phase flow in the turbine blade-to-blade cascade remains a complex task. First attempts to perform two-dimensional numerical modelling of steam flows with homogeneous condensation were made by Bakhtar and Tochai [1]. In the last few years a number of new publications concerning the flow of wet steam have appeared, which proves how topical the problem is worldwide. The progress made in theoretical studies of steam flows is reflected in a series of papers concerning the extension of the computational algorithm by the viscous gas model [2] and by the polydispersed model which can be applied in the Euler-Lagrange description of the liquid phase flow [3]. An interesting survey of selected works concerning the research on wet steam was presented by Bakhtar [4]. He pointed to the confirmed substantial share of thermodynamic losses resulting from water vapour condensation in the total loss value. He also indicated the impact of droplets and their size on generation of losses due to primary and secondary condensation. A survey of issues related to the formation of a water film on the surfaces of the steam turbine blades was made by Crane [5]. Apart from offering an explanation of the phenomenon physical aspects, he turned his attention to the impact of the droplet size on the film formation. Because the film on the blade moves due to the steam-related resisting force and the centrifugal force (on the rotor), the liquid phase on the trailing edge returns to the main flow in the form of large-sized droplets. These droplets are essential for the blade erosion process.

Many works on modelling the steam flow have been devoted lately to the problem of including the droplet size distribution in the analysis, for example the studies of Gerber [6, 7] or White [8]. Describing the droplet distribution, they all used the transport equation for the dispersed phase size according to the concept presented by Hill [9]. In the calculations, all these approaches draw on the
assumption of no slip between the phases. A proposal was put forward to extend this method for aerosol flows, taking the slip between the phases into account [10]. The possibility of analysing two-phase systems with different velocities of the phases and different temperatures is discussed in the work of Zywotiajin et al. [11], where conditions for the Godunov method application are defined and a one-dimensional analysis of a sudden outflow of water with high parameters is presented.

The commercial CFD tools which are now in common use, such as the Ansys CFX package, create an opportunity to perform numerical calculations of the steam condensing flow using two-phase flow models with a different degree of complexity.

One of the results of many years of research conducted in the Institute of Power Engineering and Turbomachinery of the Silesian University of Technology are computational algorithms for modelling steam flows with a non-equilibrium condensation process [12, 13]. The considered computational models either assume that droplets are small enough to omit the slip between the phases (single-fluid model) or take the slip between the gaseous and the liquid phase into account by solving all conservation equations for the two phases separately (two-fluid model) [14]. In parallel with theoretical and numerical studies, works were also started in the Institute on experimental research on the steam condensing flow [15].

The numerical investigations into the two-fluid model are still in the early stage of development and its numerical algorithm is much more complex compared to the single-fluid model. Due to that, it is difficult to carry out calculations for the two-fluid model for complex geometries and even the application of the Ansys CFX commercial code is not effective enough. Generally, the single-fluid model has been adopted for engineering applications of the steam flow with homogeneous condensation. Such calculations may be performed using in-house CFD codes or the Ansys CFX program.

This paper presents a comparison of calculations of a flow field modelled by means of a single-fluid model using both an in-house code and the commercial Ansys CFX v16.2 software package. Both codes use the same flow model, i.e. the model based on the Reynolds-averaged Navier-Stokes equations (the RANS model) supplemented with steam properties as specified in the IAPWS'97 standard, the same viscous turbulence model, the same condensation model based on the classical models of nucleation and droplet growth as described in [16] and [17], respectively, and the same numerical grids. Additionally, the calculation results are compared with those obtained from in-house experimental testing performed for the de Laval nozzle and the blade channel.

2. Single-fluid, no-slip model
Steam expansion may involve the occurrence of homogeneous condensation. One of the ways to describe such a flow process is to use a no-slip model, where the two-phase system is treated as a homogeneous mixture. The no-slip two-phase flow model describing the flow of steam with phase transitions is composed of:

- the mass, momentum and energy conservation equations for the mixture,
- the turbulence model equations,
- the transport equations for the liquid phase arising due to homogeneous condensation,
- the relations modelling the condensation process,
- the real gas equation of state.

The presented model of a no-slip flow of steam as a two-phase medium assumes that the droplet-steam relative velocity is omitted, i.e. a droplet moves at the same velocity as steam. Another assumption is that, compared to the volume of the gaseous phase, the liquid phase volume is small and may therefore be ignored. The steam pressure is equal to the liquid phase pressure, \( p = p_v = p_l \). This makes it possible to determine the mixture specific volume from the following relation:

\[
\nu = \nu_v (1 - y) + \nu_l y |_{\nu_l \rightarrow 0} \approx \nu_v (1 - y)
\] (1)

where \( y \) denotes the non-equilibrium moisture degree (liquid phase mass content).
Moreover, in the single-fluid model the interaction between the liquid phase and the walls limiting the flow is not taken into account, which makes modelling the water film on the walls impossible.

The flow of steam as a compressible and viscous medium is described using the Reynolds-averaged Navier-Stokes (RANS) equations. In the no-slip model, conservation equations are formulated for parameters of a fictitious fluid – a steam-water mixture, for which the following relations between water and steam quantities are valid:

\[
\alpha = \frac{V_s}{V_w},
\]

\[
\rho_m = (1 - \alpha) \rho_s + \alpha \rho_v,
\]

\[
h_m = (1 - y) h_s + y h_v,
\]

\[
y = \alpha \frac{\rho_v}{\rho_m},
\]

where \(\alpha\) is the volume content. The mixture density \(\rho_m\) is a function of steam density \(\rho_s\), water density \(\rho_v\) and \(\alpha\). The mixture enthalpy \(h_m\) is defined in a similar manner. The mass content (the moisture degree) \(y\) depends on the volume content and on the water-to-mixture density ratio. In the case under analysis, the mass content value is about 10^3 times higher than the volume content.

A detailed description of the single-fluid model applied in the in-house code can be found in [13] and [14]. In the case of the Ansys CFX code, the model is described in the Ansys CFX-solver Theory Guide.

3. Flow analysis

3.1. Experiment

The experimental analysis of the flow field in the blade channel presented in this paper was conducted using the steam facility installed in the Institute of Power Engineering and Turbomachinery of the Silesian University of Technology (cf. Fig. 2).

![Figure 1. Steam experimental facility with fittings](image)

The wide range of changes in the channel inlet parameters (total inlet absolute pressure may vary in the range of 70–150 kPa(a), total inlet temperature – in the range of 70–150°C, steam maximum mass flow rate is about 3 kg/s) makes it possible to carry out tests of transonic steam condensing flows in
different types of nozzles and/or flat blade cascades. The experimental facility and the measurement system used during the experiments are described in detail in [15] and [18].

3.2. De Laval nozzle

The nozzle geometry is described using a fourth degree polynomial $y = \sum_{i=0}^{4} a_i x^i$, where for the nozzle inlet part and for the interval of $-0.05m < x < -0.01m$, the polynomial coefficients are $a_0=1.9E-4$, $a_1=1.4E-3$, $a_2=3.67E0$, $a_3=0$, $a_4=0$, respectively. For the nozzle critical area, i.e. for $-0.01m < x < 0.02m$, the values of the polynomial coefficients are $a_0=0$, $a_1=-1.365E-2$, $a_2=3.665E0$, $a_3=-4.688E1$, $a_4=-7.268E2$, whereas for the divergent part, for $x > 0.02m$, the values are: $a_0=-3.88E-4$, $a_1=5.45E-2$, $a_2=0$, $a_3=0$, $a_4=0$. The distance between the nozzle bottom and top walls in the nozzle critical cross-section is $y^* = 0.04m$.

The experiment results of inlet and outlet parameters measured for two selected measuring series in the nozzle are presented in Table 1. The parameters were directly implemented in the numerical TraCoFlow and Ansys CFX codes.

| Case                        | $p_0$, Pa(a) | $t_0$, °C | $\Delta t$, °C | $p_1$, Pa(a) | $Ma_{1,s}$ |
|-----------------------------|--------------|-----------|-----------------|--------------|------------|
| NS1 (nozzle, superheated steam) | 107000±250  | 113±0.25  | 11.5            | 21000±250    | 1.78       |
| NS2 (nozzle, superheated steam) |             |           |                 | 81700±250    | 0.72       |

Fig. 2 presents an example comparison of pressure distributions along the nozzle centre obtained from numerical calculations performed using the in-house (TraCoFlow) code and the commercial (Ansys CFX v16.2) program. It should be noted that due to difficulties that arise in numerical analyses using the Ansys CFX code, many correction factors modifying the nucleation rate are proposed in reference literature. One of them is the nucleation bulk tension factor $\beta$, which may be applied to limit the nucleation rate value. The numerical testing results indicate that if factor $\beta$ is assumed within the limits of 1.0–1.2, better convergence between the Ansys CFX code results and the experimental data can be achieved. Because no correction factors are used in the in-house (TraCoFlow) code, the value of factor $\beta$ assumed for calculations performed by means of the Ansys CFX program is 1.

Fig. 3 presents Schlieren images obtained from the experiment and from numerical testing. It can be noticed that, like in the static pressure distribution, both the intensity and the location of the condensation wave right downstream the nozzle throat are modelled well by the applied codes, except that the in-house TraCoFlow results are closer to those obtained experimentally. Better convergence of the TraCoFlow numerical data can also be seen for the NS2 measuring series at the place where the shock wave is generated. Additionally, a violent rise occurs on this wave in the gaseous phase temperature, which involves evaporation of a part of the droplets and a reduction in the moisture degree (Fig. 4, right).
Figure 2. Static pressure distribution along the nozzle – Case NS1 (left) and Case NS2 (right)

Figure 3. Schlieren distribution – Case NS1 (top), Case NS2 (bottom) – Experiment (left), TraCoFlow (centre) and Ansys CFX (right)

Figure 4. Moisture degree and droplet radius distribution along the nozzle – Case NS1 (left) and Case NS2 (right)
3.3. Blade channel

The test section intended for analyses of the flow field in blade-to-blade channels was designed as a flat blade cascade. The geometry of the blades corresponds to that of the low-pressure part last stage stator of a real turbine. The test section is made of 4 blades, which gives 3 complete blade-to-blade channels. The blade axial chord is 173.97 mm, the blade pitch – 91.74 mm and the test section width – 110 mm. The geometrical details and arrangement of the static pressure measuring points on the surface of the blades are presented in [18].

The inlet parameters, i.e. pressure and total temperature, for the presented measuring series (cf. Table 2) correspond to about 60 ÷ 70% of the nominal load of a 200 MWe steam turbine, i.e. to 120 ÷ 140 MWe. Under such conditions, the condensation process in this turbine type begins in the stator of the low-pressure part last stage, whereas for the nominal load – the process usually commences in the penultimate stage.

| Case                  | $p_0$, Pa(a) | $t_0$, °C   | $\Delta t$, °C | $p_1$, Pa(a) | $Ma_{1,x}$ |
|-----------------------|--------------|-------------|----------------|--------------|------------|
| CS1 (cascade, superheated steam) | 103000\textsuperscript{1250} | 106\textsuperscript{10.25} | 5.6 | 42000\textsuperscript{1250} | 1.29 |
| CW1 (cascade, wet steam)     | 114000\textsuperscript{1250} | 99\textsuperscript{10.25} | -4.3 | 60000\textsuperscript{1250} | 1.09 |

In the static pressure distribution (Fig. 6), at the distance of about 0.12 m, a slight increase in pressure can be noticed on the blade suction side, which corresponds to the separation area. The next rise in pressure is related to the reflection of the shock wave formed downstream the blade trailing edge. The shock wave location in the wet steam flow is strongly dependent on the condensation process, which is triggered a little further downstream the sonic flow region. Good agreement between numerical and experimental data can be seen in Fig. 6 in the separation area for both measuring series, but it is only in the in-house TraCoFlow code that the shock wave reflection is modelled correctly. The shock waves generated past the blade trailing edge can also be seen very well in the flow field visualization using the Schlieren image (Fig. 7). It can be seen here that the shock waves have a similar nature, both for the experiment and for the numerical computations. However, in the case of the Ansys CFX v16.2 analysis they are too weak to get reflected off the neighbouring blade suction surface.
Figure 6. Static pressure distribution on the blade surface – Case CS1 (left) and Case CW1 (right)

Figure 7. Schlieren distribution – Case CS1 (top), Case CW1 (bottom) – Experiment (left), TraCoFlow (centre) and Ansys CFX (right)

The following figures present the moisture degree and the liquid phase average diameter distributions (Fig. 8 and Fig. 9, respectively) obtained from numerical calculations for the blade cascade outlet part. For the CS1 measuring series, the distributions of both \( y \) and \( r \) have a similar nature in the two codes, but – like in the case of the nozzle – the values of \( r \) are smaller. By contrast, for the CW1 measuring series the results presented below are completely different for the two
numerical codes. The differences are probably due to the way in which parameters are assumed at the blade channel inlet below the saturation line ($\Delta t_{CW1} = -4.3^\circ C$). In the TraCoFlow code, the “local” equation of state is used to determine steam parameters, which enables extrapolation of the surface approximating the steam properties into the wet steam region [14]. In the Ansys CFX v16.2 code, on the other hand, steam parameters are taken as table-format values of the IAPWS'IF97 standard.

3.4. Determination of losses
The quantitative assessment of losses arising in nozzles and blade channels is made herein using the entropy loss coefficient. The form of the entropy loss coefficient is derived directly from the definition of isentropic efficiency, which is defined for static parameters upstream and downstream the blade ring, where the difference between the values of real and isentropic enthalpy at the end of the expansion process is replaced with an increment in entropy, according to the second law of thermodynamics.
The coefficient is based on the theorem that the only reliable representation of losses occurring in the flow where an adiabatic process takes place is the increase in entropy. This increase may have several sources. One of them is internal friction resulting from the flow of any viscous fluid. Apart from that, an increment in entropy may be caused by heat transfer at a finite temperature difference and also by the unbalanced nature of some processes.

The loss coefficients calculated in this manner for individual measuring series in the de Laval nozzle are listed in Table 3 together with numerically determined enthalpy and entropy values and the gaseous phase temperature in characteristic points. Additionally, Fig. 10 presents numerically determined expansion lines in the nozzle centre for the NS1 measuring series. Although the expansion lines differ slightly from each other, especially in the nozzle outlet part, the differences are so small that they have a very small impact on losses generated in the flow. Therefore, the values of $\zeta_S$ presented in Table 3 do not differ for the same measuring series. A strange thing, however, is the increment in entropy beyond the condensation region for the results obtained using the Ansys CFX code, which has no physical justification.

Table 3. Numerically calculated thermodynamic properties and the entropy loss coefficient for measuring series in the de Laval nozzle

| Case | Numerical code | $h_0$, kJ/kg | $s_0$, kJ/kg·K | $h_{1,v}$, kJ/kg | $s_{1,v}$, kJ/kg·K | $t_{1,v}$, °C | $\gamma$, % | $\zeta_S$ |
|------|---------------|---------------|----------------|-----------------|-----------------|--------------|----------|----------|
| NS1  | TraCoFlow     | 2697.6        | 7.40           | 2455.9          | 7.41            | 61.0         | 6.5      | 0.01     |
|      | Ansys CFX     | 2697.5        | 7.40           | 2457.9          | 7.41            | 60.4         | 6.4      | 0.01     |
| NS2  | TraCoFlow     | 2697.6        | 7.40           | 2664.8          | 7.42            | 94.6         | 0.1      | 0.18     |
|      | Ansys CFX     | 2697.5        | 7.40           | 2664.6          | 7.42            | 94.6         | 0.1      | 0.18     |

Table 4 presents loss coefficients and the values of $h$, $s$ and $t_v$ needed to calculate them for individual measuring series in the blade channel. It can be noticed that the losses calculated by means of the TraCoFlow code are slightly higher compared to the results of the Ansys CFX program. This is the effect of the differences in the values of entropy (Fig. 11 and Fig. 12) and of enthalpy and the gaseous phase temperature calculated for individual numerical codes (Table 4).
4. Summary and conclusions

Comparing the calculation results obtained using the in-house code and the Ansys CFX software with the results of the experiment, it can be noticed that both TraCoFlow and Ansys CFX v16.2 give qualitatively similar results. Among others, they model the location of the condensation wave occurrence correctly. The biggest differences can be observed, both for the de Laval nozzle and for the blade channel, in the distributions of the moisture degree, the liquid phase droplet average diameter and the gaseous phase temperature. However, quantitative differences can also be noticed between the results despite the fact that for both codes identical numerical meshes were used and the same flow and condensation models were applied. Moreover, the same IAPWS’IF97 standard was used for both numerical codes to describe the properties of steam. In order to make a quantitative assessment of the calculation results, distributions of the wet steam flow main parameters were compared and a comparison was also made between the results obtained from the experiments and the calculations. The good agreement between the numerical and experimental data justifies the conclusion that both numerical codes model the steam transonic condensing flow correctly. It should be emphasized, however, that the results of the in-house TraCoFlow code are closer to those obtained from the experiment.
Acknowledgements
The presented work was supported by the Polish National Science Centre with funds from Project UMO-2014/15/B/ST8/00203.

References
[1] Bakhtar F and Mohammadi Tochai M T 1980 An Investigation of Two-Dimensional Flows of Nucleating and Wet Steam by the Time-Marching Method International Journal of Heat and Fluid Flow 2(1) pp. 5-18
[2] Simpson D A and White A J 2005 Viscous and unsteady flow calculations of condensing steam in nozzles International Journal of Heat and Fluid Flow 26(1) pp. 71-79
[3] White A J and Hounslow M J 2000 Modelling polydispersed droplet spectra in nucleating steam flows AIP Conf. Proc. 534 (Rolla, Missouri)
[4] Bakhtar F 2006 Recent Advances in Steam Turbine Research Proceedings of the International Conference on Energy and Environment (Malaysia)
[5] Crane R I 2004 Droplet Deposition in Steam Turbines, Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science 218(8) pp. 859-870
[6] Gerber A G, Sigg R, Völker L, Casey M V and Sürken N 2007 Predictions of non-equilibrium phase transition in a model low-pressure steam turbine, Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 221(6) pp. 825-835
[7] Gerber A G and Mousavi A 2007 Application of quadrature method of moments to the polydispersed droplet spectrum in transonic steam flows with primary and secondary nucleation Applied Mathematical Modelling 31(8) pp. 1518-1533.
[8] White A J 2003 A comparison of modelling methods for polydispersed wet-steam flow International Journal for Numerical Methods in Engineering 57(6) pp. 819-834
[9] Hill P G 1966 Condensation of water vapour during supersonic expansion in nozzles Journal of Fluid Mechanics 25(3) pp. 593-620
[10] Beck J C and Watkins A P 2003 The droplet number moments approach to spray modelling: The development of heat and mass transfer sub-models International Journal of Heat and Fluid Flow 24(2) pp. 242-259
[11] Zywotia†agin A F, Kaliniczenko S D, Kroszilin A E and Kroszilin W E 2000 Wysokotocnyj metod cisliennowo modelirowania wolnowowo istiecenia dwuchfaznych sriend, Teploenergetika 5
[12] Wróblewski W, Dykas S, Gardzilewicz A and Kolovratnik M 2009 Numerical and Experimental Investigations of Steam Condensation in LP Part of a Large Power Turbine Journal of Fluids Engineering 131(4)
[13] Wróblewski W, Dykas S and Gepert A 2009 Steam Condensing Flow Modelling in Turbine Channels International Journal of Multiphase Flow 35(6) pp. 498-506
[14] Dykas S and Wróblewski W 2011 Single- and two-fluid models for steam condensing flow modelling International Journal of Multiphase Flow 37(9) pp. 1245-1253
[15] Dykas S, Majkut M, Strozik M and Smolka K 2015 Experimental study of condensing steam flow in nozzles and linear blade cascade International Journal of Heat and Mass Transfer 80 pp. 50-57
[16] Frenkel J 1955 Kinetic Theory of Liquids (New York: Dover Publications Inc)
[17] Gyarmathy G 1960 Grunlagen einer Theorie der Nassdampfturbine (Zürich: Juris Verlag)
[18] Dykas S, Majkut M, Strozik M and Smolka K 2015 Losses Estimation in Transonic Wet Steam Flow through Linear Blade Cascade Journal of Thermal Science 24(2) pp. 109-116