Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 82 (2015) 194 – 200

Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

# Optimized aerodynamic design of axial turbines for waste energy recovery

Gaetano Morgese[a*], Marco Torresi[a], Bernardo Fortunato[a], Sergio Mario Camporeale[a]

[a]Polytechnic of Bari, Via E. Orabona 4, Bari 70126, Italy

**Abstract**

Industrial processes experience waste of energy when throttling valves are used for flow control. In such cases, the overall process efficiency can be improved by means of inline installations of well suited turbines in place of throttling valves, in order to recover useful power from the pressure drops, which otherwise result in a waste of energy. Energy recovery turbines may require an "ad hoc" design when their operating ranges are unsuitable for traditional design methods. For this purpose, a general methodology is proposed, which can be helpful in the aerodynamic design of these non-conventional turbines. The design procedure is iterative and begins with a one dimension (1-D) study for the definition of the main geometric parameters, once appropriate loss coefficients are considered. Then, the blade profiles are designed by means of panel method with "viscous/inviscid interaction". Finally, the actual values of the loss coefficients are evaluated by means of fully 3-D CFD simulations, and used for updating the loss coefficients used in 1-D calculations. The iterative design procedure has been automatized by means of a Matlab script for the 1-D study, a ModeFrontier project for the blade design optimization and a log file for the automated 3D mesh generation. In order to validate the proposed methodology and show its generality, the design of a first stage of a steam turbine whose reference data were available in the literature, is presented.


Peer-review under responsibility of the Scientific Committee of ATI 2015

*Keywords*: Aerodynamic design, axial turbines, CFD

## 1. Introduction

In many industrial process plants, the flow control isperformed by means of pressure drops. Sometime, a flow expansion is required by the process itself, for example, when the cooling effect due to expansion (Joule-Thomson expansion) is used in order to liquefy gases (e.g. Oxygen, Nitrogen, or Argon, used in

* Corresponding author. Tel.: +39-349-079-5685; fax: +0-000-000-0000 .
*E-mail address:* gaetano.morgese@poliba.it.

1876-6102 
Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.12.019

many cryogenic applications) **Errore. L'origine riferimento non è stata trovata.**, **Errore. L'origine riferimento non è stata trovata.**, or in the pressure reducing station of the natural gas transportation system [3]. These isenthalpic expansions are usually performed by allowing the gas to expand through anozzle or a valve. Actually, these irreversible expansions are accompanied by no work extraction. However, the same expansion could be performed by means of a turbine. The turbine determines an enthalpy reduction while extracted energy can be used for other plant necessities with the advantage of improving the overall plant efficiency.

In order to recover this wasted energy, the intent is to develop an axial turbine. This decision has been dictated by the choice to incorporate a "direct drive" electric generator into the turbine ("embedded" configuration) avoiding the use of the gearbox, obtaining a more compact, light and reliable solution. The studies carried out on this solution [1] have shown an efflux angle from the stator smaller than the ones considered in the traditional design methods [5], **Errore. L'origine riferimento non è stata trovata.**, so they result unsuitable for this application. For this purpose, an iterative methodology is proposed, which can help in the aerodynamic design of these non-conventional turbines. Its main advantages are its versatility and its high level of automation obtained with the use of proper software. In order to show the capability of the proposed methodology, a case studyhas been considered, which takes into account a low reaction steam turbine [6].

## 2. Methodology

The design process (see the flowchart in Fig 1) starts with the definition of the turbine inlet conditions and the initialization of the loss coefficients (for the sake of simplicity, it is possible to start with an ideal flow calculation, considering unitary loss coefficients). By means of the 1D calculation, considering appropriate loss coefficients, the turbine geometry can be obtained.

Then, a 2D calculation based on the “viscous/inviscid interaction” technique is performed in order to evaluate the airfoil shape, which provides the best performance under the assigned operating conditions.

Finally, by means of fully 3D CFD simulations, the resulting geometry can be studied and the actual values of the loss coefficients can be estimated and used for updating the 1D calculation.

The convergence criterion is based on the loss coefficients. The procedure ends when the loss coefficients differ less than 1% from their previous values. In the next paragraphs, each step of the methodology is analyzed in depth.

### *2.1. 1D Calculation*

The 1D calculation actually consists in the application of both the First Law of Thermodynamics and mass conservation law **Errore. L'origine riferimento non è stata trovata.**, [5]. In order to complete the set of governing equations, the equation of state for the operating gas and the velocity diagrams are considered.

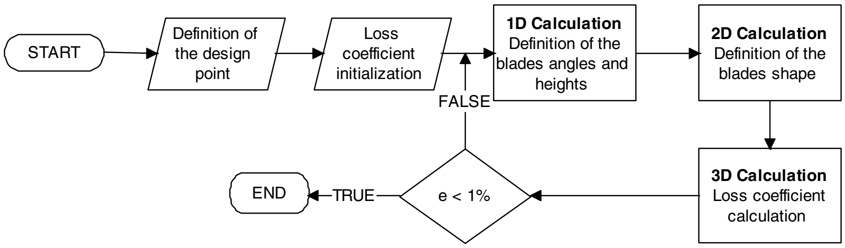


Fig. 1. Flow chart of the design process

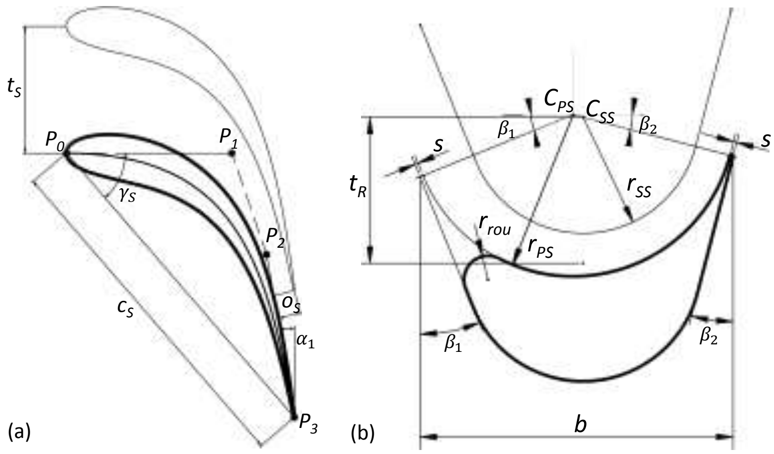

Fig. 2. (a) Stator blade geometry; (b) Rotor blade geometry

The stator is designed in order to elaborate a fraction of the assigned pressure drop (according to the required reaction ratio), accelerating and deflecting the flow. Distributed losses are accounted for by means of a loss coefficient, $\varphi_S$, i.e. the ratio between the outlet speed under real conditions and under ideal conditions.

The rotor is the actual stage where the total enthalpy is converted into work, and the pressure drop is completed. Similarly to the stator, the losses in the rotor are taken into account by means of a loss coefficient, $\varphi_R$, defined as the ratio between the outlet relative speed under real flow conditions and under ideal flow conditions.

The starting point of this calculation is the definition of the boundary conditions in terms of mass flow rate, thermodynamic inlet condition, the pressure drop to be developed and the rotational speed, as in [6]. As result, the geometry of both stator and rotor blade (in terms of inlet and outlet blade angles and heights), and the main performance of the turbine are obtained.

### 2.2. 2D Calculation

The stator blade design is performed according to Thévenin and Janiga [7]. Its camber line is designed according to a third order Bezier curve and the blade thickness according to a symmetric NACA four digit profile. Actually, the blade profile is a function of the number of stator blades, $N_S$, the chord length, $c_S$, the stagger angle, $\gamma_S$, the opening, $o_S$, the inlet and outlet angle, $\alpha_0$ and $\alpha_1$, and the four Bezier control point, $P_0$, $P_1$, $P_2$ and $P_3$ of the camber line (Fig 2a). Among these, the only three independent parameters are the number of blades, $N_S$, the chord length, $c_S$, and the stagger angle, $\gamma_S$.

For the rotor blade geometry (Fig 2b) a modified Curtis's construction has been considered in order to take into account the low reaction experienced by the steam turbine stage under investigation. The resulting blade consists of a circular shape (centered in the point $C_{PS}$) for the blade pressure side, and a suction side made up by a circular arc (centered in the point $C_{SS}$) merged with two segments; the suction side radius is $k$-time the pressure side radius (with $0<k<1$), and $s$ is the thickness of the trailing edge, with the blade pitch, $t_R$, depending on the number of blade $N_R$. The profile leading edge has been rounded with an arc tangent to the suction and the pressure sides. With this construction, the rotor blade profile has only three independent parameters: the number of blades, $N_R$, the pressure side vs. suction side radius ratio, $k$, and the leading edge radius, $r_{rou}$.

In order to evaluate the independent blade parameters, an optimization algorithm has been implemented using ModeFrontier® aiming to minimize the primary and secondary losses of the blades, and, only for the rotor, maximize the lifting coefficient. To perform this multi-objective optimization, the genetic algorithm NSGA-II has been applied. Since in this step a great number of different solutions have to be evaluated, the choice of the one-way coupled methodology has proven to be successful in order to save computational time.

In order to evaluate the profile force coefficients, a one-way coupled "viscous/inviscid interaction" technique is used [8]. The inviscid flow is computed with a vortex based panel method [9], which provides the lift coefficient, whilst the boundary layer is computed using an integral formulation [8]; finally, the drag coefficient is computed using the formulation used in Ramirez and N. Manzanares Filho [11]. This technique has been preferred with respect to other similar ones since its fast and robust computation of the performance of the profile.

The secondary losses have been accounted for by means of the empirical model presented by Aungier [12], which essentially relates secondary losses to the blade aspect ratio $h/c$.

### *2.3. 3D Calculation*

Finally, the 3D CFD calculation is performed in order to evaluate the actual value of the loss coefficients. In this work, the CFD code is ANSYS Fluent®. Turbulence is modeled by means of the SST k-ω model (Shear-Stress Transport). The fluid (steam, in the current case) has been considered as a real fluid according to the Aungier-Redlick-Kwongmodel.

The computational domain includes the entire machine with a settling chamber considered in order to avoid convergence problems related to eventual reverse flows. Each part of the computational domain is meshed separately, using hexa multiblocks meshes; a total of about three million cells were used.

The rotor/stator interaction has been accounted for by means of the multiple reference frame model, due to its low computational cost.

The CFD model has been validated comparing the global performance with the results of Yangyozov and Willinger [6].

## 3. Results

The turbine stage under investigation is the one analyzed by Yangyozov and Willinger [6], a low reaction stage (reaction ratio = 20%), which delivers a power of 4.984 MW with an isentropic efficiency of 91.1%.

The basic idea is to redesign the stage applying the proposed methodology in order to increase its performance, keeping constant the main characteristics of the machine, i.e. the input parameters of the 1D calculation. Under these assumptions, the redesigned stage actually delivers a power of 5.333 MW with an isentropic efficiency of 95.2%; this performance is obtained with 43 stator blades and 124 rotor blades. In order to reach this result, only three iterations were necessary. In Table 1 the main data obtained during the iterative procedure are summarized. The variation of the loss coefficients during the iterative process is reflected by the change of the geometry of the machine, in particular by the shape of both stator and rotor blades; both stator and rotor airfoils undergo only a small change in thickness, while the chord is almost constant.

Table 1. Convergence history of the loss coefficients in the iterative procedure

| | 1st guess | 1st iteration (error) | 2nd iteration (error) | 3rd iteration (error) |
|---|---|---|---|---|
| $\varphi_S$ | 1 | 0.9572 (-4.28 %) | 0.9670 (+1.03 %) | 0.9677 (+0.07 %) |
| $\varphi_R$ | 1 | 0.9572 (-7.86 %) | 0.9145 (-0.75 %) | 0.9131 (-0.13 %) |

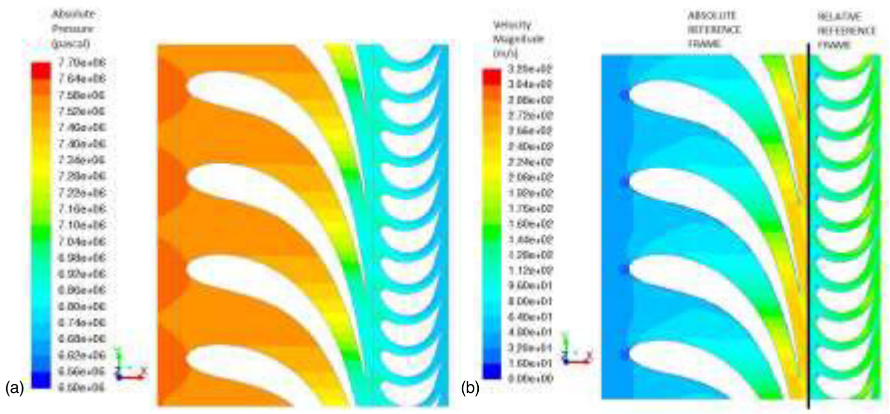


Fig. 3. (a) Contours of abs. pressure at midspan; (b) Contours of abs. (stator) and rel. (rotor) velocity magnitudes at midspan

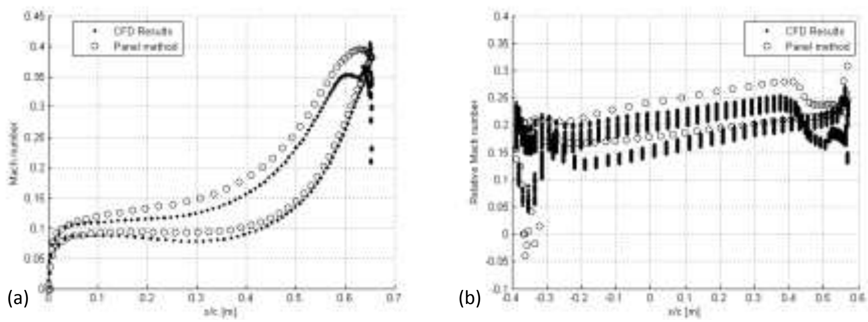


Fig. 4. (a) Mach number distribution around the stator blade; (b) Relative Mach number distribution around the rotor blade

In Fig. 3a the contours of the absolute pressure are shown at midspan. Almost all the pressure drop occurs across the stator, while across the rotor the flow experiences only a small pressure drop, according to the low reaction value. In Fig. 3b the velocity fields are shown according to their own reference frames (absolute and relative reference frames for stator and rotor, respectively). As expected, the flux experiences the majority of the acceleration in the stator, where the majority of the pressure drop occurs.

In Fig. 4, the Mach number distributions at midspan of each blade of the cascade obtained from the CFD simulation and the results obtained with the panel method are superimposed. Both in the stator and the rotor cases, the general behaviors of the Mach number distributions are preserved. Particularly, when the stator is considered (Fig. 4a), the main differences are at the trailing edge, where the presence of the rotor downstream alters the outflow ;this phenomenon cannot be captured by the panel method. When the rotor is considered (Fig. 4b), both methods capture the Mach number increase from the leading edge toward the trailing edge. However each rotor blade shows a different behavior because of the moving reference frame model applied for the stator/rotor interaction in the CFD simulation: actually the relative rotor position with respect to the stator is frozen, which means that each rotor blade actually experience different inlet flow conditions.

The method here proposed has been applied for the design of a single stage impulse axial turbine, commissioned for recover at least 1 kW of power from an assigned pressure drop of 0.3 bar with a mass flow rate of air equal to 0.27 kg/s. Again, the methodology has required only three iterations to converge and the obtained turbine develop a power of 1.8 kW (80% more than the minimum required). The results are shown in terms of contours of the absolute pressure and velocity at midspan in Fig. 5.

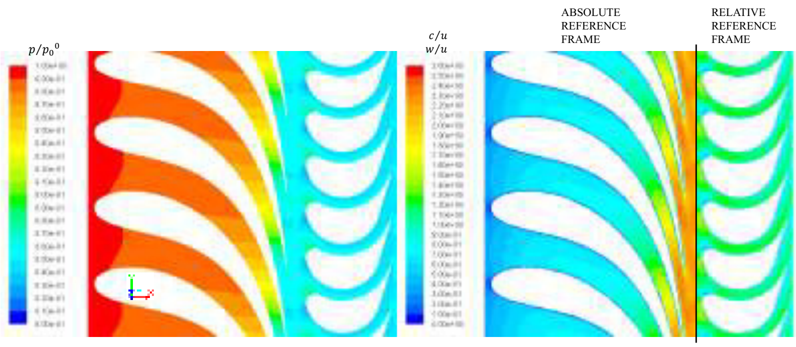


Fig. 5. (a) Contours of abs. pressure at midspan; (b) Contours of abs. (stator) and rel. (rotor) velocity magnitudes at midspan

## 4. Conclusions

In this work, a general automated design methodology for axial turbines has been proposed and validated against a case study found in the literature. The methodology consists in: (a)an iterative procedure involving a 1D calculation for the definition of the main parameters of the turbine; (b)an optimized 2D calculation based on a panel method for the definition of the blade shape, and (c) a fully 3D CFD simulation for the calculation of the loss coefficients. When applied to the case study an increase in the extracted power by 7% and an increase of isenthalpic efficiency by 4.1% was achieved showing the potentiality of the proposed approach. The method was then applied to a practical case.

## References

[1] Krishnasing R, Mariotti G, Byrne K. Turboexpander-Compressor Technology for Ethylene Plants. *Spring National Meeting*, 2010

[2] Verma R, Sam AA, Ghos P. CFD Analysis of Turbo Expander for Cryogenic Refrigeration and Liquefaction CyclesOriginal, *Physics Procedia* 2015;**67:** 373-8.

[3] Iancu A, Tudorache V, Tarean C, Toma N. Recovery of Wasted Mechanical Energy from the Reduction of Natural Gas Pressure. *Procedia Engineering* 2014; **69**: 986-90.

[4] Morgese G, Torresi M, Fortunato B, Camporeale SM. Design of an Axial Impulse Turbine for Enthalpy Drop Recovery, In: *Proceedings of ASME Turbo Expo 2014*: Power for Land, Sea and Air - GT2014-25284 - Düsseldorf, Germany, June 16-20, 2014

[5] Saravanamuttoo HIH, Rogers GFC, Cohen H. *Gas Turbine Theory*, Pearson Education, 1995

[6] Yangyozov A, Willinger R. Calculation of flow calculation characteristics in heat turbomachinery turbine stage with different three dimensional shape of the stator blad with Ansys CFX software.

[7] Thévenin D, Janiga G. *Optimization and Computational Fluid Dynamics*, Springer Science & Business Media, 2008

[8] Coenen GM, Veldman AEP, Patrianakos G. Viscous-inviscid interaction method for wing calculations. *European Congress on Computational Methods in Applied Sciences and Engineering*, 2000

[9] Lewis RI. *Vortex element methods for fluid dynamic analysis of engineering systems*, Cambridge University Press, 1991.

[10] Moran, J., *An introduction to theoretical and computational aerodynamics*, John Wiley & Sons, 1984

[11] Ramirez R.G., Manzanares-Filho N., "A Source Wake Model for Cascades of Axial Flow Turbomachines", Journal of the Brazilian Society of Mechanical Sciences and Engineering, Vol. XXVII-No 3, pp. 288-299, 2005

[12] Aungier, R.H., Turbine Aerodynamics: Axial-flow and Radial-inflow Turbine Design and Analysis, ASME Press, 2006