U.P.B. Sci. Bull., Series D, Vol. 84, Iss. 3, 2022 ISSN 1454-2358

NUMERICAL STUDY OF AXIAL TURBINES PERFORMANCE ENHANCEMENT TECHNIQUE BY SPECIFIC FLUID INJECTION

Razvan NICOARA[1], Daniel CRUNTEANU [2], Valeriu VILAG[3]

Axial flow turbines, extensively used in modern gas turbine engines, have achieved high performances and are now capable of working under high loads and temperatures. One drawback of these systems is that high performances are reached under certain conditions, near the nominal point. When operating at different conditions the efficiency and power output can decrease significantly. This paper presents a method of performance enhancement for an axial turbine, operating at a partial regime, by specific fluid injection. To determine the effect of the injection method on the performance and flow through the turbine, a methodology is presented as well as the numerical studies conducted.

Keywords: Axial turbines, Injection, Numerical study, Methodology.

1. Introduction

Axial flow turbines are used in modern gas turbine engines, in industries such as: aviation, marine propulsion, energy generation, terrestrial propulsion etc., to convert the potential energy of the high pressures high temperatures gases into mechanical energy that is used to drive the compressor or to drive external consumers. The constant improving processes and the use of complex systems (i.e. cooling) and materials, lead to the complex geometries of the modern axial flow turbines that can withstand high temperature and mechanical loads achieving, at the same time, high efficiencies and power outputs. As the design of the turbines is completed for a set of input parameters and requirements, namely the ones from the nominal regime, the turbine will be designed to achieve the best performances at these conditions. By coupling the turbine with the compressor and other elements of the engine, or by coupling with the power consumer (in case of a free power turbine), the resulting working line of the turbine usually does not coincide with the maximum performance line. Thus, for regimes different than the nominal one, partial regimes, the performances of the turbine

[1] PhD student, University POLITEHNICA of Bucharest, Romania, e-mail: razvan.edmond@gmail.com
[2] Prof., University POLITEHNICA of Bucharest, Romania, e-mail: daniel.crunteanu@upb.ro
[3] PhD. Eng., Romanian Research and Development Institute for Gas Turbines COMOTI, Romania, e-mail: valeriu.vilag@comoti.ro

may decrease significantly. By considering these aspects, one can identify the need for a method of adapting the turbine flow regime to achieve maximum performances for different input parameters and power requirements. A possibility of adapting the turbine regime to different conditions is by introducing a control parameter in form of variable stator blades. A patent for this technology was issued as early as 1966 [1]. By adjusting the vane angle the flow velocity and angles at the outlet can be controlled, thus controlling the turbine performances. The main drawback of this technology is the use of mobile elements in a high temperature environment which affects significantly the overall turbine reliability.

The control method proposed in this paper use, instead of mobile elements, a fluid injected in specific parts of the turbine profiles in order to control the boundary layer and the flow inside the turbine. By specific injection it is aimed the control of critical section of the turbine stator and the flow angle at the stator outlet. For the injection fluid it is considered that bleed air from compressor is used, similar to the system used for turbine cooling.

A similar use of boundary layer control was tested for aircraft wings [2], [3], [4], [5] where a fluid is injected in the boundary layer in order to delay the flow separation delayed by the introduction of momentum into the boundary layer region. This will increase the energy of the boundary layer and keeps it attached to the profile, thus increasing the generated lift of the wings.

Christian Rohr and Zhiyin Yang [6] describe a method of turbine efficiency enhancement at partial load through fluid injection into boundary layer near the trailing edge of the stationary blades. By injecting fluid into the free stream, normal to the pressure surface, the passage between blades is effectively contracted causing increased acceleration through the throat formed by adjacent blades. The numerical study was conducted with STAR-CCM+ software on a Pak-B LPT blade, two configurations for the injection jet being tested. The first case simulated was with the injection jet normally to blade surface on the pressure side near the trailing edge. Another cases studied by these authors implies a 45° tilt of the injection jet relative to the blade pressure side. Tilting the jets yields considerable advantages in loss reduction, the previous best case of 14.3% total pressure loss for the right angled jet flap can be almost reduced to zero at the jet flap mass flow rate = 3.5%. [6]

Similar studies were conducted by Postl et al. [7], Galbraith et al. [8], Gross et al. [9], Balzer et al. [10] etc.

2. Study methodology

In order to characterize the interaction between the injected fluid and the working fluid and to determine the optimum configuration of this system, a study methodology has been developed. The methodology, presented in figure 2, starts with defining a reference turbine, for which the nominal regime as well as a partial regime are defined in terms of performance and flow characterization. A simplified 2D model, representative for the mean radius of the turbine, is then developed as well as the numerical test matrix for a design of experiments (DOE) approach. Based on the results from the simplified model, an initial configuration for the 3D model injection system is proposed, as well as the test matrix for a similar, DOE, approach. Iterative processes are considered for high dimensions test matrices. This paper presents the studies conducted up to results from the simplified model in the methodology.

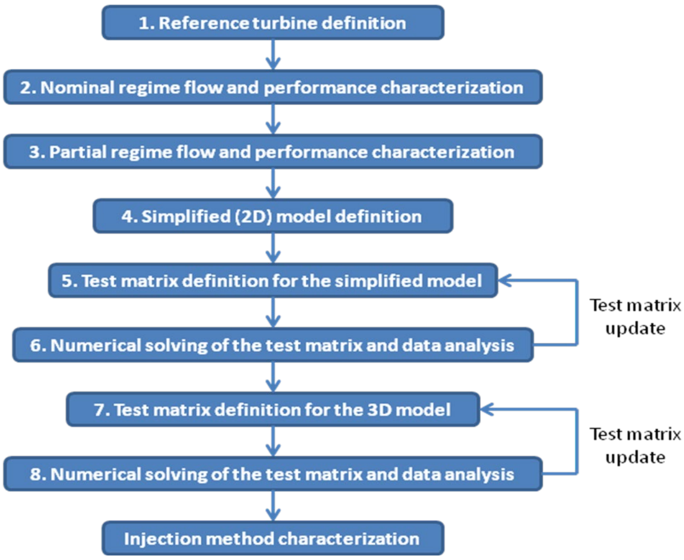


Fig. 2. Study methodology for injection method characterization

The reference turbine used in this study is a free power turbine with a mass flow at the nominal regime of 8 kg/s, an isentropic efficiency of 88% and an expansion ratio of 2.1. The single stage turbine, presented in figure 3, has a shroud diameter of approximately 381 mm for the vane and 470 mm for the rotor.

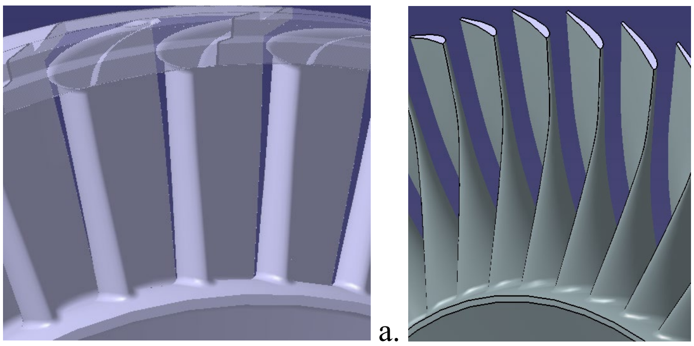

a. b.

Fig. 3. Reference turbine a. vane, b. rotor

Turbine performances and flow characterization was determined using CFD analysis and the commercial software ANSYS CFX. In this phase the flow through the turbine was considered uniform, thus a single channel for the vane, respectively for the rotor was used. For both elements, stator and rotor, the mesh used is unstructured with a general cell dimension of 1 mm and a finer distribution (lower cells dimensions) near the model walls. Special attention was placed for mesh distribution at the rotor tip in order to account for the tip clearance. Thus, a mesh of 602480 and 989206 elements was generated for stator and rotor respectively. The turbulence model selected was the standard k – ε turbulence model. The model is mostly used for turbulent flows conditions, being utilized in many industrial processes due to robustness, lower computational costs and reasonable accuracy. The good prediction of k – ε model for turbulent flows is of particular interest due to the turbulent nature of the interaction between fluids in case of the injection method. The results of this numerical simulation as well as the inlet parameters are presented in table 1.

Table 1

Nominal regime parameters

No.	Parameter	Value
1	Total Inlet Pressure [barA]	2.55
2	Total Inlet Temperature [K]	977
3	Mass flow [kg/s]	8
4	Rotational speed [rpm]	22000
5	Expansion ratio	2.12
6	Isentropic efficiency [%]	87.8
7	Generated power [KW]	1351

As the reference turbine is a free power turbine the working regime is determined by the power requirements of the consumer. For this study a partial regime of 90% of the nominal speed and a low power requirement was selected. The performances of the partial regime were obtained through numerical simulation of the turbine using input parameters determined from the turbine performance map for selected power and rotational speed. Turbine efficiency at this regime is still high but the generated power is low and can be enhanced. The results of this numerical simulation as well as the inlet parameters are presented in table 2.

Table 2

Partial regime parameters

No.	Parameter	Value
1	Total Inlet Pressure [barA]	2.34
2	Total Inlet Temperature [K]	868
3	Mass flow [kg/s]	7

4	Rotational speed [rpm]	20000
5	Expansion ratio	1.37
6	Isentropic efficiency [%]	87.2
7	Generated power [KW]	466.6

Due to a high number of parameters that could influence the performances of the injection method and due to the complex nature of the flow through the turbine stage, a simplified model is needed to better assess the influence of the interest parameters. Thus, a 2D model, that is representative for the mean radius of the turbine, has been developed. As the primary object of the injection process is to increase the speed and modify the flow angle at the vane outlet the simplified model will represent only the turbine vane. In order to minimize the numerical errors that could appear as a result of the use of periodic conditions, the 2D model, presented in figure 4, incorporates 4 stator blades that form 4 channels.

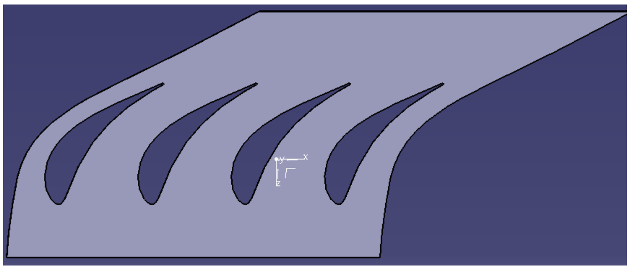


Fig. 4. Simplified model

The flow through the simplified model was determined using numerical simulation with same boundary conditions and turbulence model. The mesh used for this model is also similar, with an unstructured mesh type, similar dimension and fine distribution near the profiles walls, resulting in a number of 287500 elements. The results of this simulation, presented in table 3, represent the reference values for which the injection method will be characterized for this initial phase. Also, the results of this simulation were compared with the mean radius results of the 3D simulation, validating the 2D model.

Table 3

Simplified model performances

No.	Parameter	Nominal regime	Partial regime
1	Outlet velocity [m/s]	430	314
2	Outlet Mach number	0.72	0.546
3	Maximum velocity [m/s]	472	347
4	Maximum Mach number	0.85	0.609
5	Outlet flow angle	65.6	65.4
6	Pressure loss coefficient	0.986	0.991

For the design of experiments approach input parameters need to be defined. At this stage of study it was considered that the inlet parameters of the injection fluid (i.e. total pressure and temperature) will be maintained constant, thus determining the influence of injection fluid mass flow as well as the influence of the geometrical parameters.

For the simplified model the geometrical parameters of interest were considered the following: axial distance, injection angle and injection orifice. Axial distance represents the distance between the injection orifice center and the place of minimum section in the stator vane (a) divided by the length of the minimum section (b). The axial distance will be varied between 0.1 and 0.5. It is expected that the injection orifice will have greater influence near the minimum section of the vanes, thus values greater than 0.5 will have a small impact on the performances.

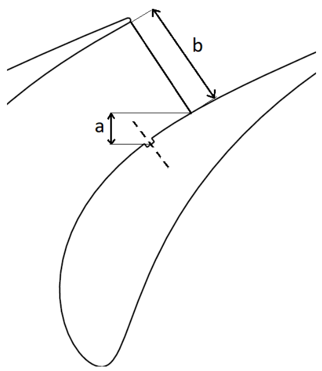
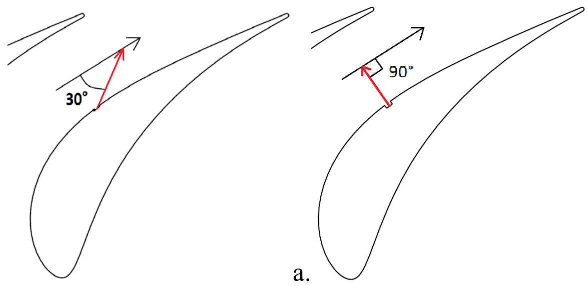


Fig. 5. Axial distance definition

Injection angle (α_{inj}) represents the angle between the injection velocity vector and the velocity vector of the working fluid at the place of injection. Definition of the injection angle can be determined in figure 6. The injection angle will be varied between 30° and 90°, lower values will not be physically feasible to manufacture while values greater than 90° will result in high turbulence and greater pressure losses.



a. b.

Fig. 6. Injection angle definition a. 30°, b. 90°

Diameter of the orifice will be varied between 0.4 and 0.8 mm. It is expected that smaller diameters to have a greater influence for the same injection mass flow due to higher injection speed. The mass flow injected was limited at 5% of the total mass flow through the turbine for the respective engine regime in order to avoid affecting the overall engine performance. For this study it was considered that values of the orifice diameter smaller that 0.4 mm requires complex manufacturing techniques, thus they are not considered.

3. Results

Simulation work was carried out using parametrical design option available in ANSYS CFX resulting in a number of 65 numerical cases. For each case the same mass flow was imposed at the model outlet, thus when the injection mass flow is increased the working fluid is decreased accordingly. The injection orifices geometry (diameter, angle and position) was created with parametrical variables in order to automatically update the geometry from one design point to another. From data analysis it was determined that downstream of the injection orifice, in the near vicinity, due to the interaction of injection fluid with the working fluid, a low pressure zone was formed. The bubble formed as a result of this interaction leads to a shrinking of the section through the vanes thus to an acceleration of the working fluid. An example of the low pressure bubble is presented in figure 7.

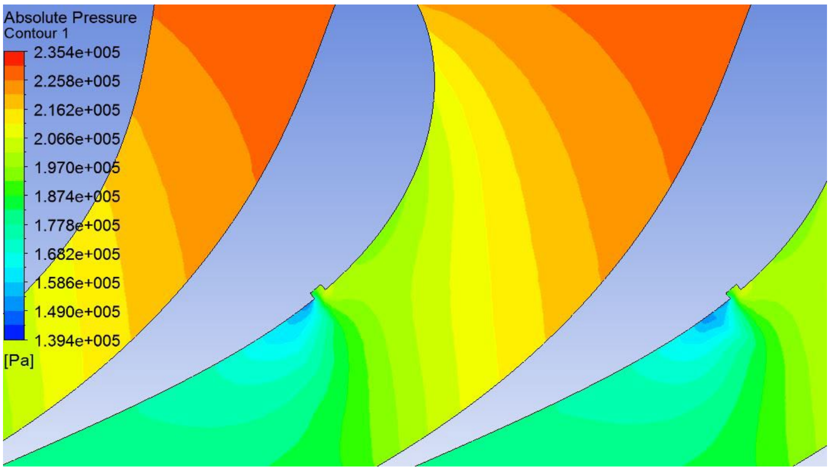


Fig. 7. Low pressure zone formed as a result of fluid injection

The injection angle has a direct influence on the creation and scale of the low pressure zone. For a low value of injection angle (i.e. α_{inj} =30°) the bubble is not formed, due to a rapid attaching of the injection fluid to the suction side of the profile. With the increase of this angle the bubble grow in size reaching a maximum size (for the simulated cases) at 90° as can be determined from figure 8.

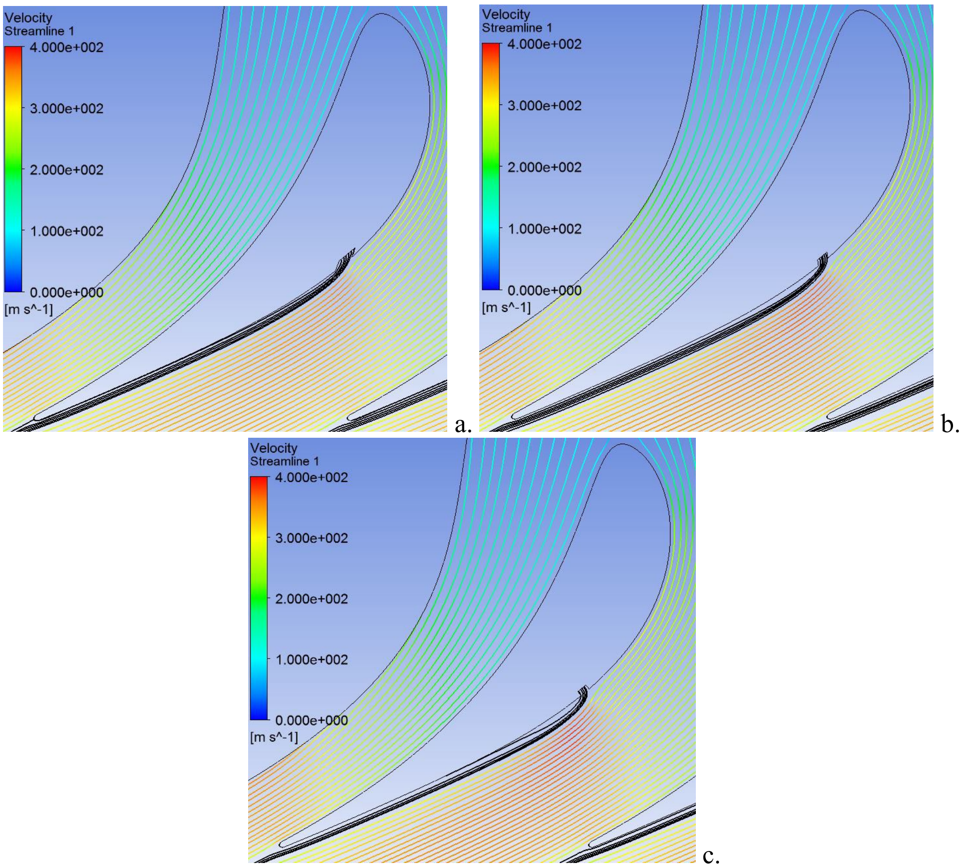


Fig. 8. Resulting velocity streamlines for an injection angle of: a. 30°, b. 60°, c. 90°

The low pressure zone dimensions directly impacted the flow throw the model, thus the best results were obtained for high injection angles, in terms of increases of flow velocity at the model outlet, approximately 4%. Similar evolution has been observed in terms of maximum speed through the model, were a maximum increase of approximately 12% was reached.

The pressure loss increases, as expected, for higher injection angles and mass flows. This is explained by higher turbulence generation as the injection velocity increase with the mass flow or by the higher differences between the direction of injection and the velocity direction of the working fluid. The pressure loss coefficient reached a value of 0.970, compared with 0.991 for no injection, for the best results obtained in terms of performance increases.

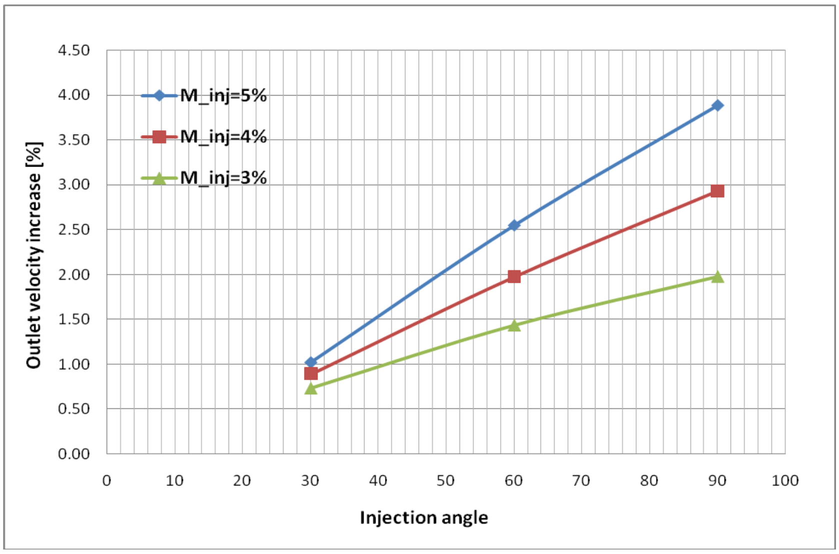


Fig. 9. Outlet velocity increase for different mass flows and angle of injection

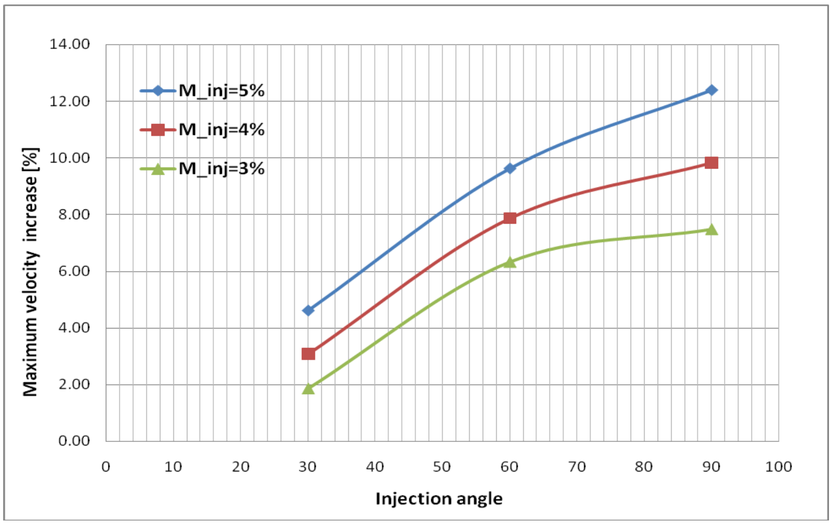


Fig. 10. Maximum velocity increase for different mass flows and angle of injection

Injection placement on the profile, described by the axial distance parameter, was found to be most effective in the near vicinity of the minimal section of the vanes. The influence of the axial distance on the outlet velocity is presented in figure 11. Fluid injection near the minimum section leads to development of the low pressure bubble at the minimum section, thus reducing the area which in turn leads to an increase acceleration of the working fluid. It can be determined that the maximum influence of the injection placement is at a 0.27 value of the axial distance. Lower values results in poorer performances which can be explain be the formation of the bubble downstream of the minimal section.

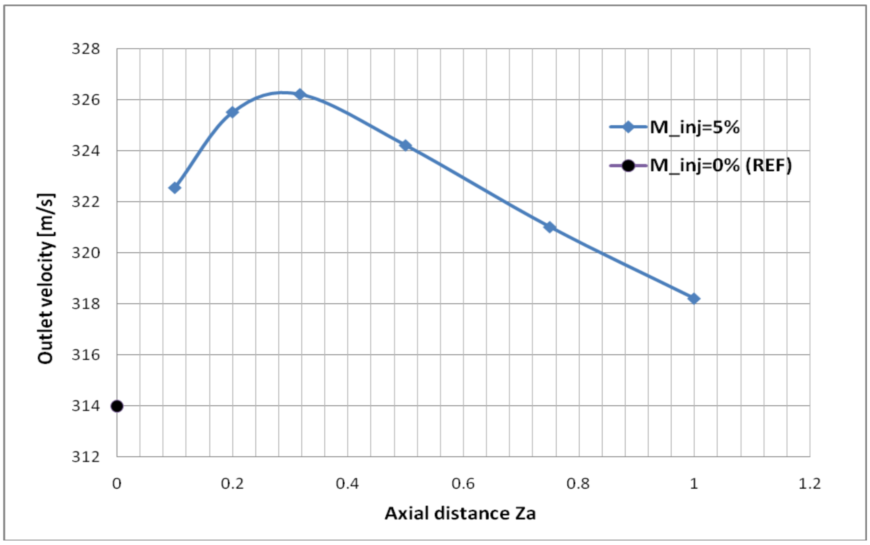


Fig. 11. Velocity variation for different axial distances

The pressure losses variation with the axial distance was found to be minimal, with higher values when injection is closer to the minimal section due to higher velocities of the working fluid which leads to high turbulence generation.

The increasing value of the injection mass flow has a direct influence on the performances, as can be determined from figure 9 and 10, thus it is anticipated that the injection velocity has a direct influence on the flow through the model. As the mass flow was limited at 5%, in order to increase the velocity of the injection fluid, the orifices diameter was decreased. The results of the injection velocity increase as a result of orifice diameter modification is presented in table 4.

Table 4

Injection fluid parameters for different orifices diameter

No.	Orifice diameter	Injection velocity [m/s]	Injection Mach number
1	0.8	197	0.338
2	0.6	262	0.452
3	0.5	305	0.531
4	0.4	374	0.66

With the reduction of orifice diameter and the resulting injection velocity rise, the low pressure bubble developed downstream of the injection has grown, as can be determined from figure 12, leading this way to an increase of the model performances. In terms of maximum velocity through the model the velocity improved by 25% for an orifice diameter of 0.4 mm compared to 12% for an injection diameter of 0.8 mm.

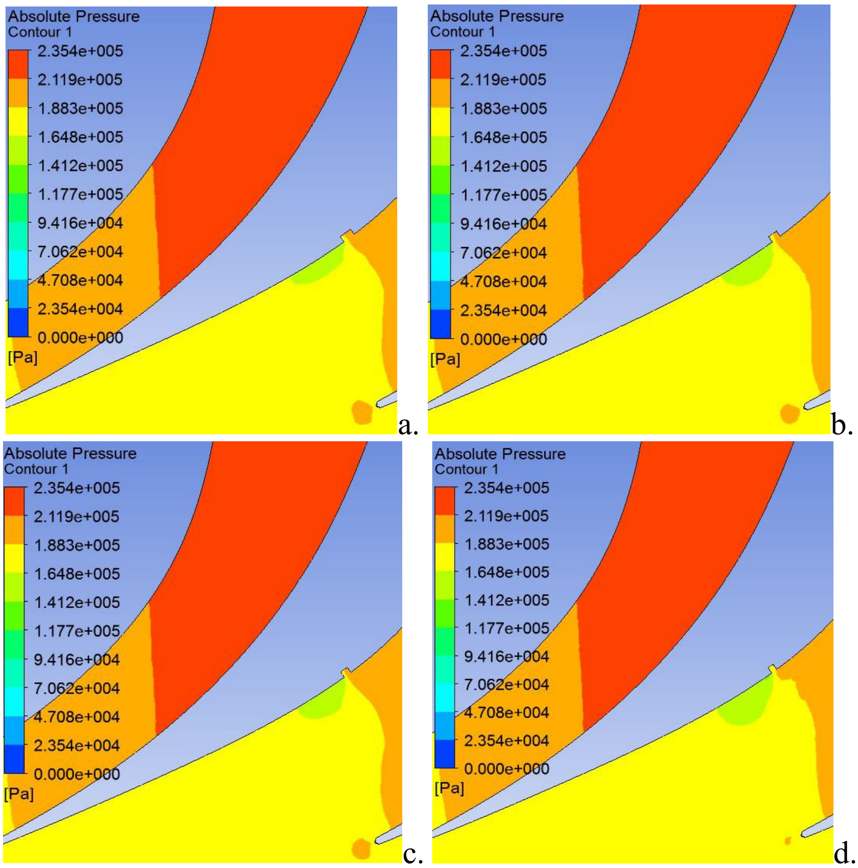


Fig. 12. Absolute pressure distribution through the model for a. 0.8 mm, b. 0.6 mm, c. 0.5 mm and d. 0.4 mm injection orifice

6. Conclusions

This paper presents a method of turbine performance enhancement at partial loads by specific fluid injection. The study methodology was presented starting with the reference turbine and its characterization at nominal and partial regime, followed by the definition of a simplified, 2D model. The numerical study was conducted in order to determine the influence of different parameters on the model performances.

Form the results analysis it was determined that the injection angle has a direct influence on the method performance, higher angles of injection conducting to higher dimensions of the low pressure zone, developed downstream of the injection, which lead to higher acceleration of the working fluid. Similar effect was obtained for the variation of the injected mass flow.

The injection orifice placement on the profile, which was expressed by the axial distance parameter described in this paper, was found to have an influence of the injection method, achieving a maximum at a value of approximately 0.27. This was explained by the developing of the low pressure bubble at the minimum section when the axial distance reached this value.

A velocity improvement at the model outlet of 4% and a maximum velocity increase of 12% were obtained. In order to further improve the performances of the model, the injection velocity was increased by reducing the injection orifice diameter. This method was chose in order to limit the mass flow injected at 5% of the working fluid and to keep constant the input parameters of the injection fluid. The enhanced injection speed had a direct impact on the model performances, the velocity increase reaching 25%., when compared with the reference case (the case without injection).

The injection method is thus proving to be suitable for performance increase at partial regimes, future studies being needed in order to fully characterize the method on the simplified model, and on a more complex 3D model. Furthermore, the numerical results must be validated by experimental data.

R E F E R E N C E S

[1]. *Clarence Edgar Le Bell*, *Alvin Taub*, U.S. Patent No. 3237918. (1966). Washington, DC: U.S. Patent and Trademark Office.

[2]. *Gecheng Zha, Craig Paxton,* A Novel Airfoil Circulation Augment Flow Control Method Using Co-Flow Jet, AIAA Paper, DOI: 10.2514/6.2004-2208, 2004.

[3]. *Adam Wells, Clark Conley, Bruce Carroll, Craig Paxton, Gecheng Zha,* Velocity Field for an Airfoil with Co-Flow Jet Flow Control, 44th AIAA Aerospace Sciences Meeting and Exhibit, DOI: 10.2514/6.2006, 2006.

[4]. *Yang Wang,* Study of Power Minimization of Co-Flow Jet Active Flow Control with Fixed Airfoil Geometry, University of Miami, Mechanical Engineering, 2019.

[5]. *Shima Yazdani, Erfan Salimipour, Ayoob Salimipour, Mikhail Sheremet, Mohammad Ghalambaz*, Effects of CoFlow-Jet Active Flow Control on Airfoil Stall, 2021.

[6]. *Christian Rohr, Zhiyin Yang,* A numerical study of active flow control for low pressure turbine blades, Proceedings of 4th International Symposium on Jet Propulsion and Power Engineering, ISJPPE-2012-H0020, 2012.

[7]. *Dieter Postl, Andreas Gross, H. Fasel,* Numerical Investigation of Active Flow Control for Low-Pressure Turbine Blade Separation. DOI: 10.2514/6.2004-7, 2004.

[8]. *Marshall Galbraith, Amit Kasliwal, Kirti Ghia, Urmila Ghia,* Numerical Simulation of a Low Pressure Turbine Blade Employing Active Flow Control, 45th AIAA Aerospace Sciences Meeting and Exhibit, DOI: 10.2514/6.2007-650, 2007.

[9]. *Andreas Gross, H. Fasel,* Simulation of Active Flow Control for a Low Pressure Turbine Blade Cascade, 43rd AIAA Aerospace Sciences Meeting and Exhibit, DOI: 10.2514/6.2005-869, 2005.

[10]. *Wolfgang Balzer, Andreas Gross, H. Fasel, (2007).* Active Flow Control of Low-Pressure Turbine Separation, DoD High Performance Computing Modernization Program Users Group Conference,. DOI: 10.1109/HPCMP-UGC.2007.8, 2007.