Volume XXII 2019
ISSUE no.1
MBNA Publishing House Constanta 2019

Scientific Bulletin of Naval Academy

SBNA PAPER • OPEN ACCESS

Analysis of gas flows in ships turbines

To cite this article: V. Aftaniuk and B. Garagula, Scientific Bulletin of Naval Academy, Vol. XXII 2019, pg. 318-322.

ISSN: 2392-8956; ISSN-L: 1454-864X

doi: 10.21279/1454-864X-19-I1-046

Analysis of gas flows in ships turbines

V Aftaniuk[1] and B Garagula[2]

[1] Professor, National University «Odessa Maritime Academy», Odessa, Ukraine
[1] Docent, National University «Odessa Maritime Academy», Odessa, Ukraine

E-mail: valera2187@ukr.net

Abstract. In this paper comparison was made of an experimental study of gas flows and CFD simulation of turbine blades steam turbines for ships. For CFD simulation develop solid models, samples were taken from three blades similarly studied in the experiment. An analysis of experimental data and CFD simulation results of the gas flow near lattice the considered type shows the inappropriateness of their application at supersonic speeds. The calculations performed on the CFD model show that the use of CFD modelling methods allows, at the early stages of designing turbine blades, to select the most optimal (in terms of energy losses) forms of profiles for specific modes of operation of the ship's turbine. Further improvement of the profiles that have passed the preliminary calculation check is advisable to conduct on the basis of more detailed experimental tests to obtain dependences of losses in the lattice, the exit angle of the flow from the parameters of the lattice and the angle of gas flow.

1. Introduction

The process of development of ship power plants is characterized by a wide range of steam and gas turbines. Turbines more and more are used as main engines and actuators of auxiliary mechanisms, even in such vessels, where previously used only other types of engines [1]. This is achieved through the implementation of various measures, including rational inspection of blades, which is the responsible element of the flowing device in the turbine.

The science-research work of leading specialists in the field of aerodynamics of flow parts of turbines, aimed at increasing the reliability and efficiency of steam turbine operation, are carried out both by calculation and experimental research methods.

The most accessible, and therefore, the most widely used, is by now an experimental method of purging the models of the elements of turbines with air or steam. This research path also has another advantage. It allows simple analysis of the structure of the flow, as well as identify sources of losses and the mechanism of their occurrence. As a result analysis are planned to reduce them.

2. Exposition

The main element of the flow part of the turbine, the efficiency of which significantly depends on its efficiency, are turbine blades. The flow through the turbine blade apparatus (with the exception of the end sections) can be reproduced using a fixed lattice of profiles streamlined by a flat stream. Based on the study of such a flow, they work out the shapes of the profiles and determine the optimal lattice parameters and profile losses.

Air is used as a working substance in studies on fixed models. If the full-scale flow part works at subsonic and moderate supersonic speeds (which takes place in a significant part of the last stages of ship steam turbines), then this does not introduce tangible errors in the results, but it significantly simplifies the experiments.

The method of air blowing test both ring and flat grids of the blades. The layout of the package of the studied lattices of profiles in a wind tunnel is shown in Fig. 1 [2].

In studies [3, 4] we present the results of a comparative analysis of the calculated value of profile losses of energy in the flow of various forms of profiles of working blades of the last stages of turbines. Experimental studies of various forms of profiles are given in [4].

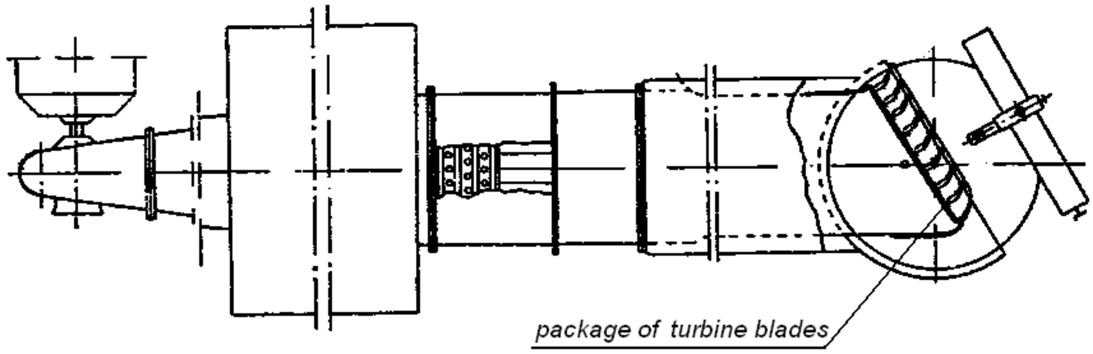


Fig. 1. Scheme of an aerodynamic stand for blasting turbine blades [2]

In studies [4], a series of studies on the flow in profiles of various modifications, was aimed at improving the design methods of the blades of the last steam turbine stages. The structure of the flow in the oblique section of the lattice profiles was investigated, taking into account the flow of two-dimensional nature. The effect of flow separation in the area of the output edges of the blades was studied by means of calculation and experimentally in a wide range of flow rates around profiles with different forms of bypass, various effective exit angles and other geometrical parameters depending on the Mach number.

Experimental studies of interprofile channels [4] used various methods for measuring flow parameters: static pressures were measured on the canal walls, static pressure was measured using a special probe in the “throat” of the canal (across the canal from flat to curved surface); measured in the same area with the help of pressure probes of complete deceleration P_0, and the direction of the velocity vector; with the help of the Tepler device, the flow in the “throat” area was investigated. The combination of these methods yielded reliable experimental results.

In Fig. 2 shows (top right) a typical lattice of the last stage of the low-pressure power unit of a powerful ship steam turbine and the dependence of the profile energy loss during its flow on various Mach numbers M_{1t}. Complicated character ξ_p. = f (M_{1t}) and high level ξ_p are explained due to special the flow around at $M_{1t} > 1.0$.

The purpose of this study is to compare the results of an experimental study of turbine profiles of model lattice with modern computational computer techniques [5, 6], which allow the selection of the most aerodynamically perfect forms of turbine blades profiles.

In this paper, computational fluid dynamics (CFD) modeling software complex is used for numerical study of aerodynamics blades ship turbine [6].

The adequacy and accuracy of the mathematical model was determined by a set of factors taken into account and accepted assumptions.

The mathematical model of gas flows in the channels between turbine blades is based on the Navier-Stokes equations, the equation of continuity, and the law of conservation of energy.

Numerical methods used in [6] allow you to calculate three-dimensional problems in the calculated areas, precisely reproducing the geometry of the object of the study, with fairly accurate installation of all input parameters.

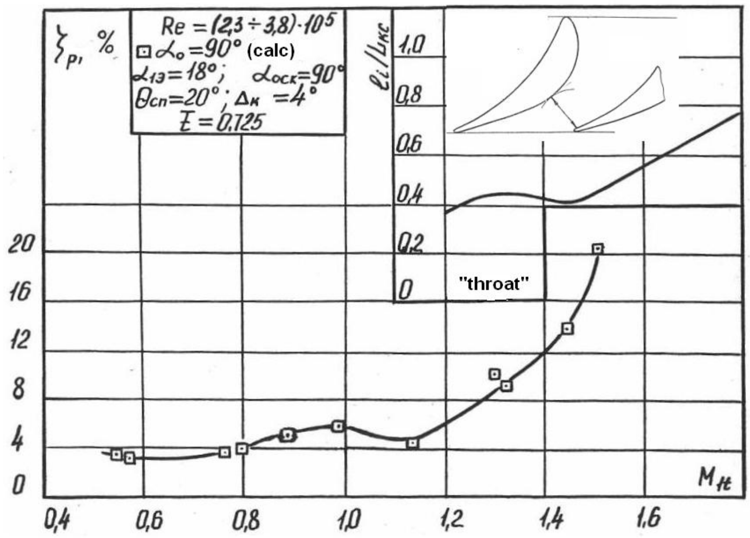


Fig. 2. Ratio $\xi_p = f(M_{1t})$

To develop solid models, samples were taken from three blades (similarly studied in the experiment [4]). Material of blades - alloy steel.

The cycle of work includes the flow of gas from the inlet plane through the channels of blades to the exit. The calculations were carried out as an "external aerodynamic task".

As a result of the task, it is necessary to find the distribution of gas velocities in the channels between the turbine blades at different operating modes, and to determine the zone of intrusive motion and flow sealing.

The geometric model of a set of blades of a ship turbine (Fig. 3) was built as a solid model (as an “assembly”) and consists of three objects (blades).

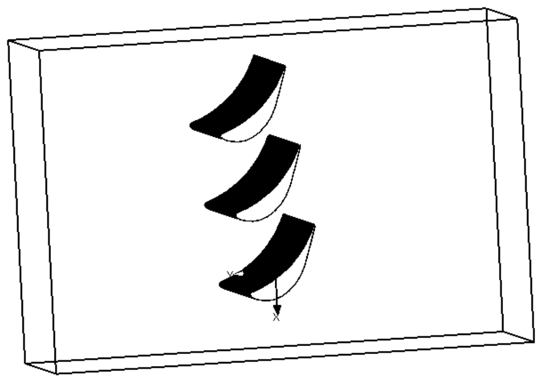


Fig. 3. Solid-state model of turbine blades in the calculated area

The model was implanted into the CFD software complex [6], which includes a wide range of technical data, assignment of materials, input parameters, boundary conditions, setting goals for the study, saving the project for analysis and others.

Comparison and analysis of experimental data and the results obtained in CFD modeling made it possible to more fully investigate and justify the nature of the flows in turbine blades.

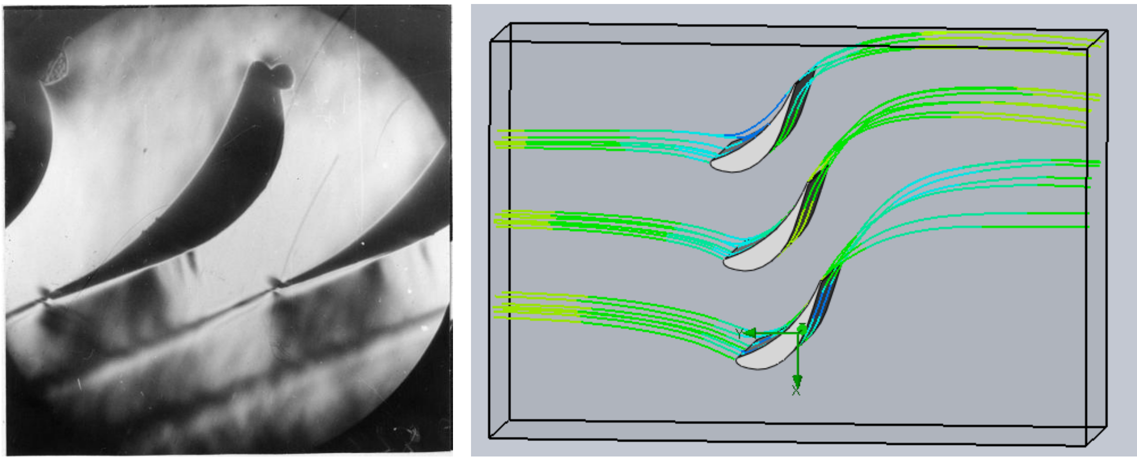
Experimental studies [4] of this lattice in the field of the Toepler optical device (Fig. 4, *a* - 6, *a*) showed that, at M_{1t} = 0,836 (Fig. 4,*a*), on the “back” (convex side) of the profile, an oblique cut appears "Chain" of relatively weak λ - shaped shock waves caused by a turbulized boundary layer (zone 1).

There is no disconnect of the flow that flows through the "back" of the profile. But the loss of $\xi_{p.}$ increased (Fig. 2). Here $M_{1t} = C_{1t}$, where C_{1t} is the absolute speed at the exit from the nozzle grating, and M_{1t} is the local velocity of sound.

This corresponds to the calculation picture of the flow (Fig. 4,*b*), from which it is clear that the lines of current are completely adjacent to the convex surface of the profile, that is, the flow is un separated.

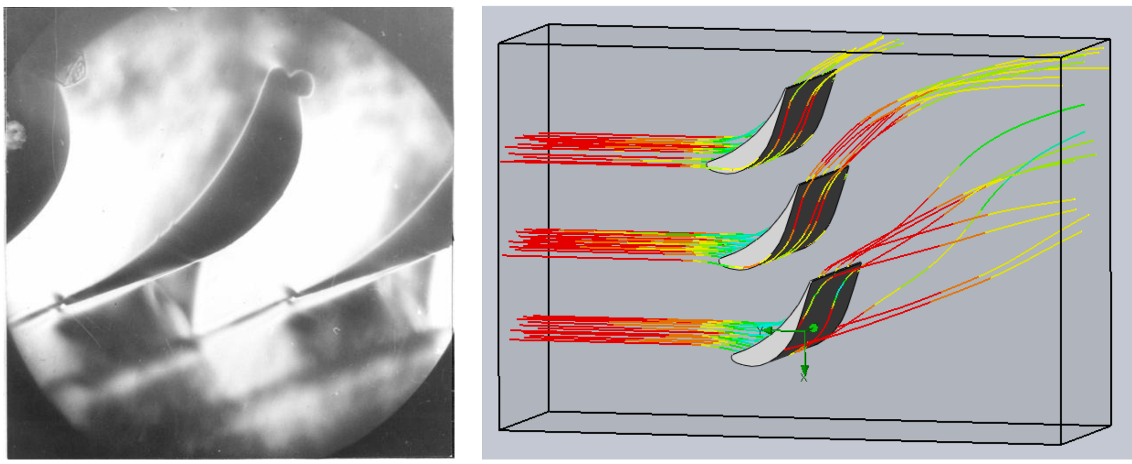
At M_{1t} = 0,98, the "chain" disappears, there is an λ-like jump of sealing, which causes development (spreads over the lattice, in flow) separation (Fig. 5.*a*). Prolonged separation zone 2 adjoins the "back" of the profile; ξ_p somewhat increase (Fig. 2). This phenomenon can be seen in Fig. 5,*b*, zone 2.

With a further increase in the Mach number to M_{1t} = 1,24, the amplification of external shock waves in an oblique cut (Fig. 6, *a*). As a result, the area on the "back" occupied by the separation increases sharply: the point at which the separation begins shifts against the flow to the "throat" of the grid (Fig. 6, *a*, zone 3). At the same time sharply increase and $\xi_{p.}$ (Fig. 2). In fig. 6, b, one can observe the same process obtained by calculation (zone 3) in CFD modeling.



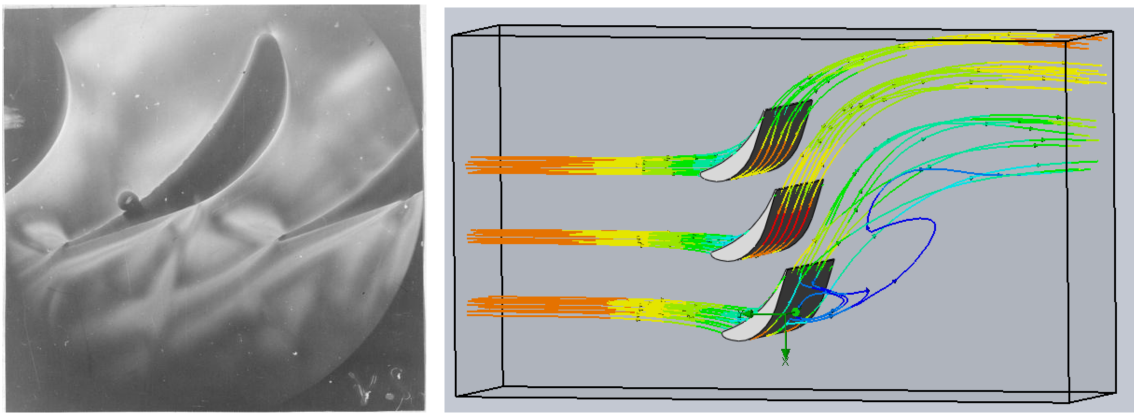
a *b*

Fig. 4. Visualization of currents of gas on blades at M_{1t} = 0,836
a - experimental; *b* - CFD modeling



a *b*

Fig. 5. Visualization of currents of gas on blades at M1t = 0,98
a - experimental; *b* - CFD modeling



a *b*

Fig. 6. Visualization of currents of gas on blades at M_{1t} = 1,24
a - experimental; *b* - CFD modeling

3. Conclusion

An analysis of experimental data and CFD simulation results of the gas flow near lattice the considered type shows the inappropriateness of their application at supersonic speeds.

The calculations performed on the CFD model show that the use of CFD modeling methods allows, at the early stages of designing turbine blades, to select the most optimal (in terms of energy losses) forms of profiles for specific modes of operation of the ship's turbine.

Further improvement of the profiles that have passed the preliminary calculation check is advisable to conduct on the basis of more detailed experimental tests to obtain dependences of losses in the lattice, the exit angle of the flow from the parameters of the lattice and the angle of gas flow.

References

[1] Shatrovsky D Advanced high-power power plants for ships 2015 (Shipbuilding vol 5) pp 33-37

[2] Matveev G, Kamenev G, Markov N and Elizarov V 1961 Aerodynamics of the flow part of ship turbines (Leningrad: Sudpromgiz) p 363

[3] Garagulya B, Kulikov V, Mayorsky E 1982 Investigation of transonic flow in the region of the "throat" of the lattice profiles of the last stages of the low-pressure cylinder of steam turbines Proceedings of the Moscow Energy Institute vol 583 p 78

[4] Garagulya B 1985 Optimization of the geometric parameters of supersonic nozzle and working grids of the last turbine stages (Moscow Energy Institute) 19 p.

[5] Aftaniuk V, Spinov V 2018 Modeling of scrubber for cleaning industrial gases Journal of the Technical University of Gabrovo. Volume 57'. pp 35-38

[6] Alyamovsky A, Sobachkin A, Odintsov E, Kharitonovich A and Ponomarev N 2005 SolidWorks. Computer modeling in engineering practice (Petersburg: BHV) 800 p