Mathematical Modeling of Flow Structure of Micro Power Gas Turbine by CFD Investigation

A.L. Kartashev, S.D. Vaulin, M.A. Kartasheva, A.Yu. Nitskiy, A.A. Martynov, E.V. Safonov

Abstract–**The paper analyses CFD research into operation process of a stage of a turbine which is a component of a micro gas turbine power installation. Using mathematical simulation, the optimum geometric parameters of the stage have been determined, and its performance characteristic has been obtained. The design characteristics of the turbine stage obtained using two independent packages ANSYS CFX and SolidWorks FlowSimulation have been compared.**

Index Terms–**micro power gas turbine installation, turbine stage, mathematical modelling, CFD investigation, Ansys CFX, Solidworks FlowSimulation**

Nowadays micro gas turbine installations (MGTI) [1] are increasingly used for distributed power generation and combined generation of heat and electric power. Generally, a single stage of compressor and gas turbine is provided in MGTI design, such installations have electric power output in the range between 20 kW to 500 kW,

MGTI have some advantages over other small-scale power generation technologies, such as diesel or natural-gas fuelled reciprocating-engine power plant units, specifically: small number of moving parts, high reliability, small size, high efficiency and environmental friendliness, and the total power generation cost reduction.

Manuscript received July 10, 2015, revised July 27, 2015. The work was conducted with the financial support of the Ministry of Education and Science of the Russian Federation in the framework of the complex project "Establishing Production of New Generation Micro Turbine Units Range" according to the contract no. 02.G25.31.0078 d.d. 23.05.2013 between the Ministry of Education and Science of the Russian Federation and Joint stock company Special Design Bureau "Turbina" in cooperation with the head R&D works performer - Federal State Educational Institution of Higher Professional Education "South Ural State University" (National Research University).

A. L. Kartashev. Doctor of engineering science. Professor of Flying Machines and Automatic Apparatus Department of South Ural State University, Chelyabinsk, Russia. Tel.: +79193458556, e-mail: al_kartashev@mail.ru.

S. D. Vaulin. Doctor of engineering science. Prorector for scientific work. Head of Aircraft Engines Department of South Ural State University, Chelyabinsk, Russia. Tel.: +79028993852, e-mail: s.d.vaulin@susu.ac.ru.

M. A. Kartasheva. Candidate of engineering science, reader of Flying Machines and Automatic Apparatus Department of the South Ural State University, Chelyabinsk, Russia. Tel.: +7(351)2679461, e-mail: ma_kartasheva@mail.ru.

A.Yu. Nitskiy. Engineer of Super Computer Center of South Ural State University, Chelyabinsk. Tel.: +79080819528, e-mail: nitskiy.a@gmail.com.

A. A. Martynov. Post-graduate student of Flying Machines and Automatic Apparatus Department of South Ural State University, Chelyabinsk. Russia. Tel.: +79068918614, e-mail: andmartynov@list.ru.

E.V. Safonov. Candidate of engineering science. Dean of Aerospasce Faculty, reader of Aircraft Engines Department of South Ural State University, Chelyabinsk. Russia. Tel.: +73519049585, e-mail: e-safonov@yandex.ru.

TABLE I
MGTI-100 TURBINE SPECIFICATIONS

Symbol	Quantity	Value
G	Turbine working medium mass flow rate	1.017 kg/s
N	Turbine shaft speed	65000 rpm
P_{2tot}	Total pressure of working medium at outlet	109 kPa
T_{0tot}	Total temperature of working medium at nozzle diaphragm inlet	1170 K
P_{0tot}	Total pressure of working medium at nozzle diaphragm inlet	~446.8 kPa
π_{tot}	Total pressure reduction degree	4.1
R	Individual gas constant of the working medium	289.3 J/kg K
k	Specific heat ratio of the working medium	1.33138

In the modern context one of important performance parameters of any equipment is its economic efficiency, so designing of MGTI turbine stage suggests performing some turbine stage gas-dynamic calculations.

The subject of research considered in this paper is a single-stage radial-axial turbine of MGTI-100 installation, the rated design specifications of which are provided in Table 1.

The research object, discussed in this article, is a Francis turbine of MGTI-100 (Microturbine power plant with rated power 100 kW). As it is known, optimal total pressure ratio in compressor of microturbine power plant is in the range $\pi_{c\ opt}* \in 4{,}5 .. 5{,}5$ [2, 3]. With this in mind, total pressure ratio in turbine taking into account hydraulic losses in the combustion chamber, the heat exchanger, and the exhaust device is $\pi_t* > 3{,}5 .. 4{,}5$. Such a high value of total pressure ratio traditionally operated in two-stage turbines, but to simplify design, improve maintainability and reduce the cost it is necessary to design high-efficiency single-stage Francis turbine. Total pressure ratio of MGTI-100 turbine is $\pi_t* = 4{,}1$.

At the first stage of design the gas-dynamic calculation of turbine by mean parameters was performed according to the method given in [4]. As a result of the calculation basic dimensions of turbine were defined: the height of the blades and the diameter on which they are located. In these sizes, by the method described in [5].

The general view of geometric model of the turbine stage is shown in Fig. 1. The impeller blade profile has been designed by its bending along the parabolic arc using technique [4]. Each of the shapes of nozzle blade suction side and pressure side are formed by Bezier curve and straight-line trailing portion, leading and tailing edges are formed by circular arcs.

Fig. 1. General view of the turbine.

The gas-dynamic research into the operation process in the turbine was performed using CFD analysis packages Ansys CFX 14.5 and SolidWorks FlowSimulation .V2013.

The following problems were solved by means of the gas-dynamic calculation using CFD package ANSYS CFX:

1) determination of the optimum quantity of nozzle blades from the range of 23 to 25 in the rated operation mode and the height of nozzle blades and leading edge of impeller to obtain the rated operation point corresponding to parameters of Table 1.

2) determination of optimum quantity of impeller blades from the range of 13 to 14 in the rated operation mode

3) plotting the performance curve of the turbine for the selected quantity of nozzle diaphragm and impeller blades.

In addition, the calculation was performed in the rated mode using SolidWorks FlowSimulation to compare the calculation results with the data obtained in ANSYS CFX.

ANSYS CFX software system contains an extended toolbox for solving Turbo problems. The geometric model of blade passages was constructed using Design Modeller tool into which a solid model of nozzle diaphragm blade and profile is imported in Parasolid format.

Then, using Export Points tool, the geometry is transferred to a grid generator TurboGrid where a high-quality structured hexa grid is plotted [6]. As an example, Fig. 2 illustrates a structured grid obtained in TurboGrid for a nozzle diaphragm blade passage.

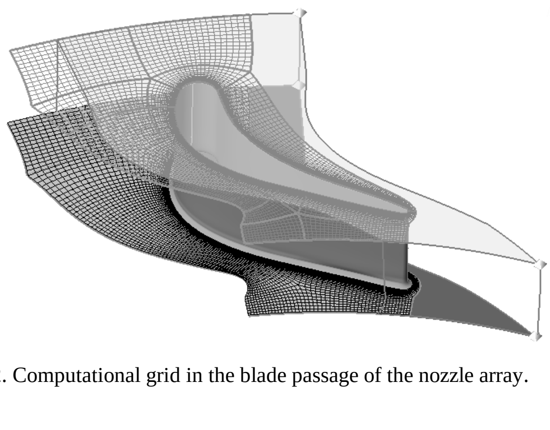

Fig. 2. Computational grid in the blade passage of the nozzle array.

The size of the computational grid amounted 0.7 mln. cells for 1 sector of the impeller, 0.5 mln. cells for 1 sector of the nozzle diaphragm.

Application of the boundary conditions according to parameters of Table 1 has been performed using Turbo template in NSYS CFX Pre, the problem was solved in a stationary statement. To account for compressibility, the working medium is presented as an ideal gas using the viscosity-temperature relationship according to Sutherland's formula. The turbulence was taken into consideration using SST turbulence model, which has shown a high degree of correspondence to experimental data in [7].

Fig. 3 illustrates the turbine stage design model, which includes the inlet (IN), nozzle diaphragm (SA), impeller (R1) and outlet (OUT). To obtain the ratios of areas of boundary interfaces (STAGE interface type) between the nozzle diaphragm and impeller close to 1.0, 3 nozzle diaphragm blades and 2 impeller blades were used.

R1 impeller area is tied to a movable system of reference rotating at speed equal to the turbine rotor speed. All other design domains are located in a fixed system of reference.

The calculations were performed be supercomputer of South Ural State University ‘‘Tornado’’ [8].

The post-processing of the calculation results was performed using templates of Turbo reports (turbine stage report of ANSYS CFD Post).

At the last stage of design a parametric study of flow in Francis turbine stage was performed. Variable parameters in this case were nozzle blade angle α_{bl} (by substituting in calculation model corresponding optimized nozzle blade) and diameter of shroud at outlet of impeller d_{2s}. When building a computational model, it is necessary to fulfill the condition of approximate equality of areas at the exit of the stator part of the computational domain and at the entrance to the rotary part of the computational domain. For this purpose 3 interscapular nozzle channels and 2 interscapular impeller channels were included in calculation model (figure 3). The total number of computational cells was ≈ 2 500 000.

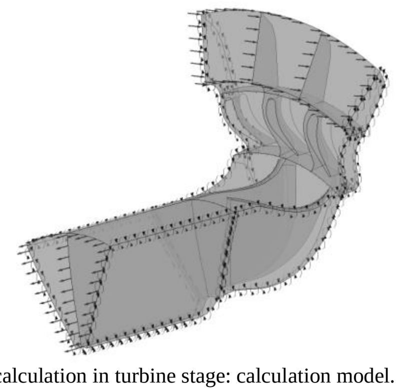

Fig. 3. Flow calculation in turbine stage: calculation model.

The combination of boundary conditions was total temperature T_0* and mass flow G_{gas} at the inlet of nozzle and static pressure p_2 at the outlet of impeller. The optimization criterion was the efficiency of the turbine at total pressure ratio η*. The results of calculation of stage with α_{bl} = 12 degrees and d_{2s} = 116 mm are presented in figure 4 in the form of the velocity field on the middle section of the turbine stage.

To obtain the rated parameters of the turbine and determine the optimum quantity and height of nozzle blades, a calculation series was performed, the calculation results are shown in Table 2.

ISBN: 978-988-19253-6-7
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

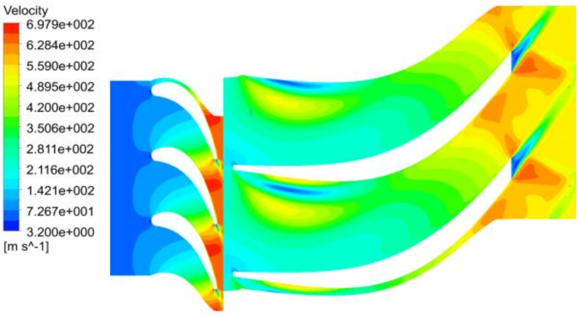

Fig. 4. Flow calculation in turbine stage: velocity in blade-to-blade view.

During calculations, the rated turbine parameters were achieved by selecting a height of the nozzle blade for each set of nozzle blades and the height of the impeller blade leading edge corresponding to it, taking into consideration 0.4 mm end clearance.

The maximum stage efficiency was obtained during calculation with parameter set No. 1, therefore it was selected as a working version of the stage.

TABLE II
STAGE PERFORMANCE RESULTS FOR SET OF GEOMETRIC OPTIONS

Quantity	Value set 1	Value set 2	Value set 3	Value set 4
Geometric parameters of impeller and nozzle diaphragm				
Quantity of nozzle blades	23	23	24	25
Nozzle blade height, mm	13.8	13.8	14.2	14.7
Impeller leading edge height, mm	14.2	14.2	14.6	15.1
Quantity of impeller blades	13	14	14	14
Calculation results				
Shaft power, kW	343.8	342.3	343.0	342.6
Intake volume flow rate, m^3/s	0.74	0.77	0.76	0.77
Intake flow coefficient	0.42	0.41	0.40	0.39
Power coefficient	0.39	0.39	0.39	0.39
Reaction	1.14	1.15	1.15	1.15
Total pressure reduction coefficient	4.12	4.12	4.13	4.13
Total temperature reduction coefficient	1.34	1.33	1.34	1.33
Polytropic head, kJ/kg	404.1	404.0	405.3	405.0
Polytropic head coefficient	4.07	4.04	4.05	4.05
Polytropic efficiency of stage, %	**84.72**	84.33	84.23	84.18
Coefficient of ND losses	2.82	2.75	3.26	3.78
ND efficiency, %	93.83	93.88	93.24	92.63

The turbine design model plotted in SolidWorks FlowSimulation software system contains the full 360-degree geometric models of impeller and nozzle diaphragm (Fig. 5), as distinct from ANSYS CFX. In the initial model all holes in the housing are plugged, an inlet and outlet caps are provided additionally. They seal the inner space and enable setting inlet and outlet boundary conditions.

One of the main features of CFD package FlowSimulation is the use of adaptive orthogonal grid, which is generated automatically when the necessary conditions in the geometric model are met.

The design model has been modified by adding a solid embracing the impeller (solid 1, Fig. 5) - for setting the rotating zone (type: "local rotating zone") and a solid crossing the nozzle diaphragm (solid 2, Fig. 5) - for refinement of the grid in it ("local initial grid").

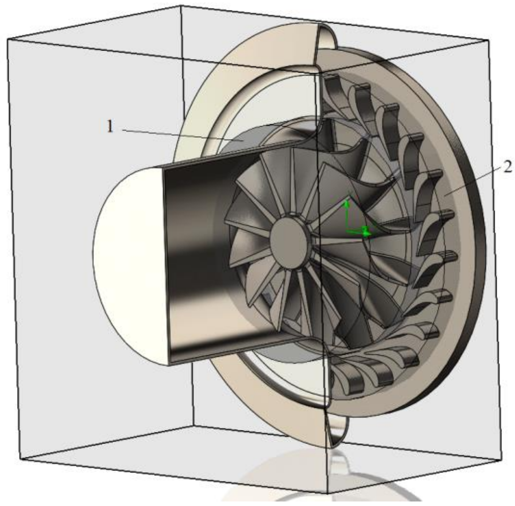

Fig. 5. Turbine design model in FlowSimulation.

When the grid was plotted for adequate resolution of the turbine geometry and radial clearance, parameters "minimum clearance" and "minimum wall thickness" were set to 0.4 mm and 0.5 mm respectively. The minimum quantity of cells in cross-section of the nozzle diaphragm (the property was applied to solid 2, Fig. 5) was set to 10 (Fig. 6.), impeller (the property was applied to solid 1, Fig. 5) to 6, as when the quantity of cells in the blade passage is less, the solution accuracy is not guaranteed.

The total quantity of cells in the design model amounted ≈ 2,000,000.

The problem was solved in stationary statement, with averaging of parameters of the flow on rotor/stator interface. The following combination was set as boundary conditions: flow rate and total temperature at the inlet - static pressure at the outlet, which was selected in such a way so that total outlet pressure would amount ≈ 109 kPa. The global and superficial (on surfaces of inlet and outlet to/from the design zone) values of the total and static pressure, temperature, speed, mass flow rate, and torque at the impeller blades are assumed to be the convergence criteria.

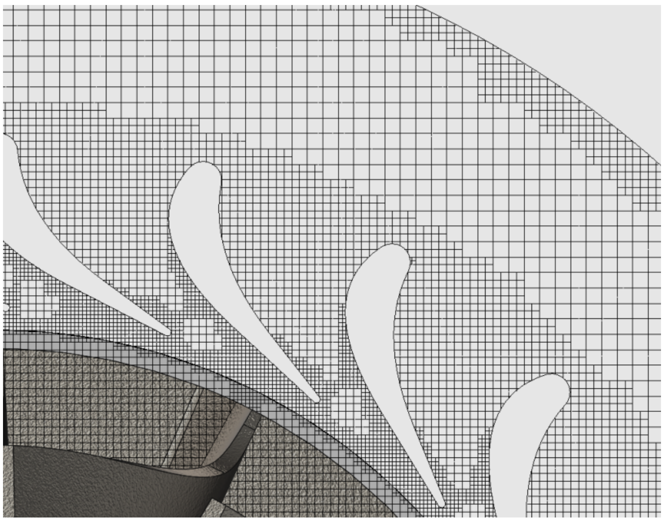

Fig. 6. Computational grid in the nozzle diaphragm.

Fig. 7 and Fig. 8 show the field of pressure and speed in the nozzle diaphragm computed in ANSYS CFX (left) and SolidWorks Flow Simulation (right) for rated working medium flow rate mode.

The analysis of physical quantities distribution allows to assert that the results of computation in ANSYS CFX and SolidWorks Flow Simulation are characterized by similar fields of speed and pressure.

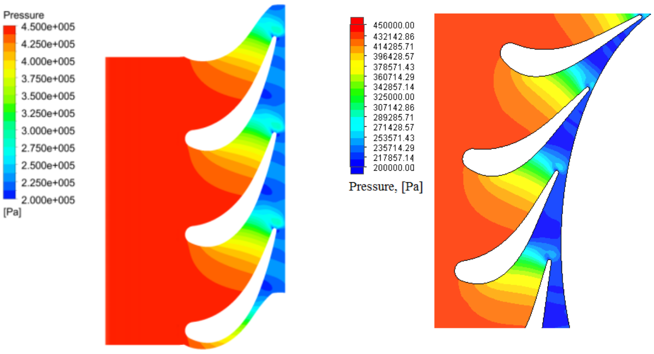

Fig. 7. Field of pressure in the nozzle diaphragm:
left - ANSYS CFX, right - SolidWorks FlowSimulation.

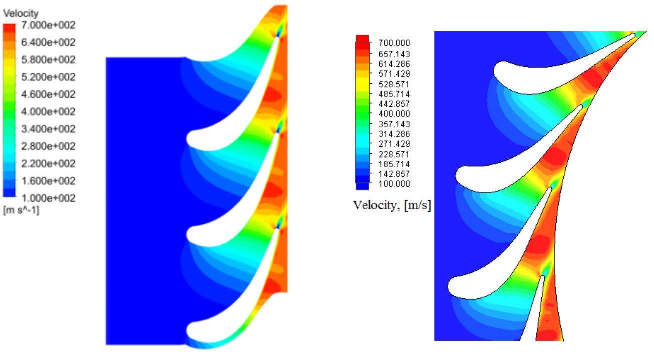

Fig. 8. Field of speed in the nozzle diaphragm:
left - ANSYS CFX, right - SolidWorks FlowSimulation.

Table 3 provides comparison of integral characteristics of the stage obtained using the two software systems. As it follows from the Table, the software systems under consideration demonstrate close results differing by less than 2.5%, hence, re-calculation of the problem in one of them can be considered as a method of checking of the built design model. Comparing peculiarities of computation in ANSYS CFX and SolidWorks Flow Simulation, we can note that in Flow Simulation the design model development is less labour-consuming than in ANSYS CFX, but to achieve convergence of the main gas-dynamic parameters, a greater computation time is required.

TABLE III
COMPARISON OF COMPUTATION RESULTS

Quantity	ANSYS CFX	SolidWorks Flow Simulation	Δ, %
Total gas pressure at the turbine stage inlet, kPa	448.8	438.3	2.3
Average gas temperature at the turbine stage outlet, K	843.4	847.06	0.4
Gas speed at the nozzle diaphragm outlet, m/s	635.5	627.4	1.3
Torque at the impeller blades, N*m	50.5	50.1	0.8

CONCLUSION

1. The CFD research of operation mode of a turbine stage, which is a component of a micro gas turbine power installation, has been conducted.
2. Subsequent to the results of parametric calculations, the optimum geometric characteristics of the turbine stage and its performance characteristics have been determined.
3. The comparison of the turbine stage characteristics obtained using two independent software packages, ANSYS CFX and SolidWorks Flow Simulation, has shown their close correlation (with up to 2.5% error), which enables using both software systems for determination of the turbine stage performance characteristics, including that for mutual validation of the obtained results.
4. Mathematical modeling of flow in stage of turbine using CFD investigation allows to find the optimal geometrical configuration of Francis turbine stage within the existing technological limitations. In the next phase of the study verification of results obtained will be performed on the test bench. This will allow us to judge the correctness of the computational model and optimization algorithm.

References

[1] Kartashev A.L. Research of Optimal Operating Regimes of Power Micro Gas Turbine Installation with Heat Recuperation / A.L. Kartashev, S.D. Vaulin, B.G. Degtyar, M.A. Kartasheva, E.V. Safonov, A.A. Martynov. – Lecture Notes in Engineering and Computer Science. World Congress on Engineering and Computer Science 2014, WCECS 2014; San Francisco; United States; 22 October 2014 through 24 October 2014. Volume 1, 2014. Pages 272–275.

[2] Degtyar B.G., Kartashev A.L., Martynov A.A. Investigation of modes of micro gas turbine power plant, *Bulletin of the South Ural State University. Series «Mechanical engineering», 2013, no. 2, issue 13, pp. 132-135.*

[3] Baturin O.V., Yu. D. Smirnova. Selection of optimal scheme and workflow options autonomous mikrogazoturbinnoy installation, *Bulletin of the Samara State Aerospace University, 2012, no. 3, pp. 257-262.*

[4] Mitrokhin V.T. Selection of parameters and calculation of the centripetal turbine at steady state and transient conditions. Moscow, Mashinostroyeniye, 1974, 228 p.

[5] Sherstyuk A.N. Low power Francis turbines. Moscow, Mashinostroyeniye, 1976, 208 p.

[6] Dmitriyeva I.B., Shabliy L.S. Avtomatizatsiya sozdaniya ob#yomnoy modeli pera lopatki v ANSYS TurboGrid na baze traditsionnogo predstavleniya ego geometrii [The blade airfoil portion 3D model development in ANSYS TurboGrid based on the traditional presentation of the f a 3D model of the blade airfoil portion geometry] // Vestnik Samarskogo gosudarstvennogo aerocosmicheskogo universiteta imeni akademika S.P. Koroleva, Samara, 2011, vol. 3, no. 3(27), pp. 106-111. (in Russian)

[7] Umesh Javiya, CFD analysis of flow and heat transfer in a direct transfer pre-swirlsystem / Umesh Javiya, John Chew, Nick Hills/ Proceedings of ASME Turbo Expo 2010 / GT2010-22964.

[8] Internet page of Tornado Supercomputer, South Ural State University, Available: http://supercomputer.susu.ru/en/computers/tornado/

ISBN: 978-988-19253-6-7
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)