Proceedings Paper:

Olaru, D., Cuciumita, C.F. and Vilag, V.A. (2017) Test bench configuration to facilitate gas turbine in-situ combustion experimentation. In: Anton, A., Balan, C., Balan, M.C., Bode, F., Bosschaerts, W., Croitoru, C., Inard, C., Meslem, A., Nastase, I. and Sandu, M., (eds.) Energy Procedia. Sustainable Solutions for Energy and Environment, EENVIRO 2016, 26-28 Oct 2016, Bucharest, Romania. Elsevier, pp. 306-313. ISSN: 1876-6102.

https://doi.org/10.1016/j.egypro.2017.03.1101

Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 112 (2017) 306 – 313

Energy Procedia

Sustainable Solutions for Energy and Environment, EENVIRO 2016, 26-28 October 2016, Bucharest, Romania

Test Bench Configuration to Facilitate Gas Turbine In-Situ Combustion Experimentation

Daniel Olaru[a]*, Cleopatra Florentina Cuciumita[a], Valeriu Alexandru Vilag[a]

[a]*Romanian Research and Development Institute for Gas Turbines COMOTI, 220 D Iuliu Maniu Bd., sector 6, Bucharest code 061126, Romania*

Abstract

The present technology in gas turbine engines is to burn fuel in the combustion chamber and occasionally in the postcombustion chamber. Current conventional developments of gas turbine aero-thermodynamics provide small efficiency and power increase, because with the present technology one reached an asymptotical convergence to the upper limit of the gas turbine performance. An interesting and almost unexploited possibility is to continue combustion in the turbine, option that, up until recently, has been considered undesirable for a number of reasons. A turbine-combustor is defined as a turbine in which fuel is injected and combusted. The process of combustion in the turbine is called in situ reheat. Thermodynamic cycle, (a hybrid between the Ericsson and Brayton cycles) analyses have demonstrated the benefits of using reheat in the turbine in order to increase specific power and thermal efficiency. [1]

Peer-review under responsibility of the organizing committee of the international conference on Sustainable Solutions for Energy and Environment 2016

Keywords: in-situ combustion; gas turbine; specific power increase

1. Introduction

The work presented in this paper is carried out at the Romanian Research and Development Institute for Gas Turbines COMOTI within the ongoing TURIST (Gas Turbine using in situ combustion –TURIST) project, aiming at developing the required dedicated test rig in order to perform combustion in a turbine (in-situ reheat), including the

* Corresponding author. Tel.: +4-021-434-0236; +4-021-4340198 / 450 fax: +4-021-434 02 41.
E-mail address: daniel.olaru@comoti.ro

Peer-review under responsibility of the organizing committee of the international conference on Sustainable Solutions for Energy and Environment 2016
doi:10.1016/j.egypro.2017.03.1101

adaptations required for the coupling with the detonation chamber inlet (integrated in COMOTI's gas turbine experimentation facility) and the adaptations to the turbine and instrumentation system.

The experimental program for the TURIST project takes into consideration, in the initial phase, two experimental campaigns. The first, integrates the experimentation of a turbine sector consisting in 2 stator blade passages and 3 rotor blade passages, in order assess the details from an intermediate burning in the turbine point of view and the second refers to the experimentation of the entire gas turbine in order to obtain the global image over the modifications occurred in turbine performance when subjected to this supplementary burning. The experiments focus on the first stage of the TV2-117 gas turbine engine's power turbine.

The rise in technological level of a turbine can be achieved by intensive experimental studies conducted in dedicated test benches. The turbine is the part of a gas turbine which has the most complex stresses from both thermal and mechanical points of view. For testing a turbine stage under conditions close to those encountered while functioning at nominal regime on an actual gas turbine, it requires very high inlet temperature and pressure, high mass flow rate and rotational speed. All these parameters are extremely difficult to be achieved simultaneously on a test bench, both from a technological and a financial point of view. Such a test bench would require an hot air / burnt gases at high pressure source, as well as a dynamometer, either hydrodynamic, electric or pneumatic, to act as a power consumer. At the same time, testing the rotor under rotating conditions makes it extremely difficult to properly instrument it in order to acquire relevant data for assessing the performances.

Only a select group of universities or research institutes worldwide have developed such experimental facilities, such as: AneCom AeroTest GmbH, Germany; Stuttgart University, Germany; Cambridge University, Great Britain; CONCEPTS NREC, SUA; NASA Gleen, SUA; Royal Institute of Technology, Sweden; MTU Aero Engines, Germany; National Aerospace Laboratory, Netherlands; National Research Council (NRC), Canada.

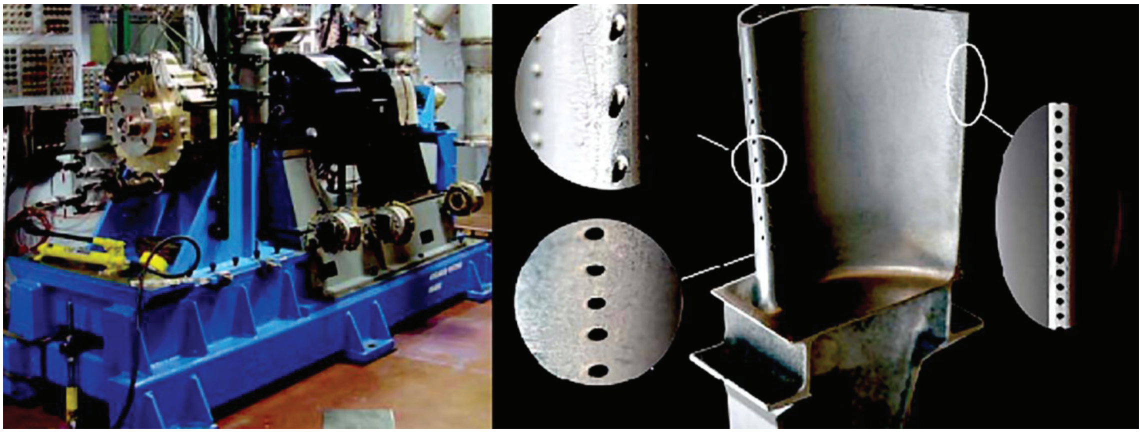

Fig. 1. Turbine testing facilities at CONCEPTS NREC, USA[2]

All the above, make a strong case for developing a methodology which to allow the testing of a rotor cascade under static conditions, while still reproducing the actual engine working conditions and allowing collecting relevant data of its performances under these conditions. Such a study has not been identified in the dedicated literature. At the same time, this testing methodology for a rotor cascade would be of outmost importance for the interested scientific community in turbines, as it would allow for less consuming tests, both in terms of financial and technical resources and would allow turbine applications to be developed by a less exclusive scientific community. In time, this can translate in a more rapid development of the turbine technology, as more studies are encouraged.

2. Experimental setup

The experimental facility integrates an existing tri-sonic air blowing station, equipped with 2,000 m^3 air tanks, capable to supply air up to a pressure of 12 bar and a mass flow up to 5 kg/s. The maximum value of the temperature of heated air depends on mass flow that passes through it. Thus, in accordance with data provided by the manufacturer's technical documentation, the air heater is capable of supplying air up to 0.4 kg/s in terms of mass flow and 900 K in terms of temperature, parameters in close range to the conditions at the first stage power turbine inlet (0.436 kg/s mass flow and 790 K temperature).

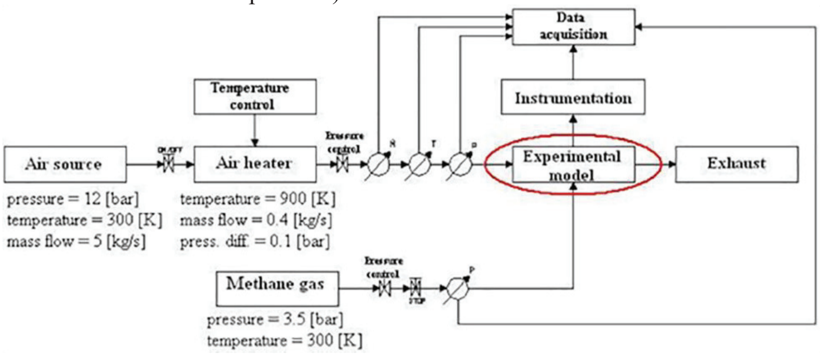

Fig. 2. Experimental facility operating scheme

This setup is dedicated to perform the experiments in case of the first campaign: a turbine sector consisting in 2 stator blade passages and 3 rotor blade passages. In order to provide optimum conditions for experimentation (ensuring an axial flow and minimum losses) the experimental model highlighted in fig. 2 features the following components (fig. 3):

- adaptation with the existing duct from the test bench facility (this adaptation represents a welded assembly integrating a two connection flanges) (1);
- adaptation ring (featuring profiled orifices corresponding to a stator sector formed by two blade passages) (2);
- cylindrical sector assembly (featuring stator guiding blades with a double role: ensuring an axial flow and ensuring a rigid assembly, acting as struts) (3);
- first stage of the power turbine (consisting in stator blades and static rotor blades at different angles) (4).

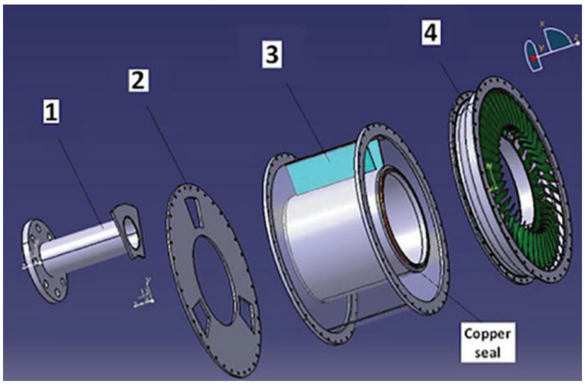

Fig. 3. Current state in the design process for the components of the test bench facility

The constructive solution considered for the injection of fuel in the stator consists in a series of circumferentially disposed small cavities machined on the stator casing. The presence of these cavities in the main flow will create recirculation regions where the fuel will mix with the Oxygen containing gas, and which will stabilize the flames. The solution is robust from the temperature resistance standpoint, as the flame is anchored on the casing, which is thicker and which can be easily cooled on the outside by secondary air streams from the compressor.

Due to the fact that the experimentations carried out on this turbine sector cannot be performed in the conditions provided by the gas turbine, namely the rotation effect of the rotor blades, the aim is to simulate the presence and the effect of the rotor by respecting, as close as possible, the velocity triangles. Thus, a static rotor blade position needs to be identified, position that can reach these velocity triangles. Numerical simulations at different rotor blade angles (in static position) have been conducted in order to achieve this goal.

3. Theoretical considerations

The static rotor blade position was first theoretically approximated. This approximation took into account the exit velocity angle, in relative frame, at nominal regime [3], based on the numerical simulations conducted as baseline and described in the next section. This angle had a mean value of -59.4^o. At the same time, the blade metallic angle at the exit at mean radius was measured [3], resulting a value of 60.25^o. The incidence and deviation angles were not taken into account [4]. Assuming that, at nominal regime, the flow is desired to be as close as possible to axial [5], it was concluded that a rotation of the blade with approximately 60^o as shown in figure 4 is required, where in red is depicted the original profile, while in blue the rotated one, to achieve axial flow with these airfoils while using them in a static mode. This value was, however, extremely roughly appreciated. In order to have more reliable results, the need for CFD numerical simulations was identified.

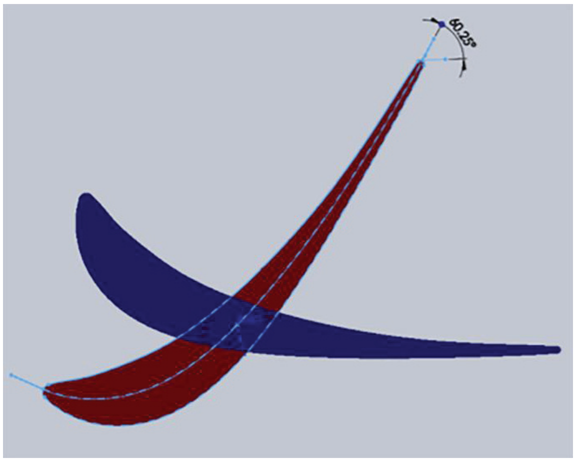

Fig. 4. Airfoil exit metallic angle and rotation

4. Numerical simulations

In order to properly identify the static rotor blade position that could satisfy the velocity triangles imposed by the rotation of the rotor blades the following geometry was subjected for analysis in ANSYS CFD environment:

- duct (90 mm outer diameter and 150 mm length) (1);
- adaptor sector (150 mm length) (2);
- cylindrical sector (150 mm length) (3);
- stator sector (2 blades) (4);
- rotor sector (consisting in 3 stationary blades with different rotation angles around the profiles overlapping axis) (5);
- exhaust sector (150 mm length) (6).

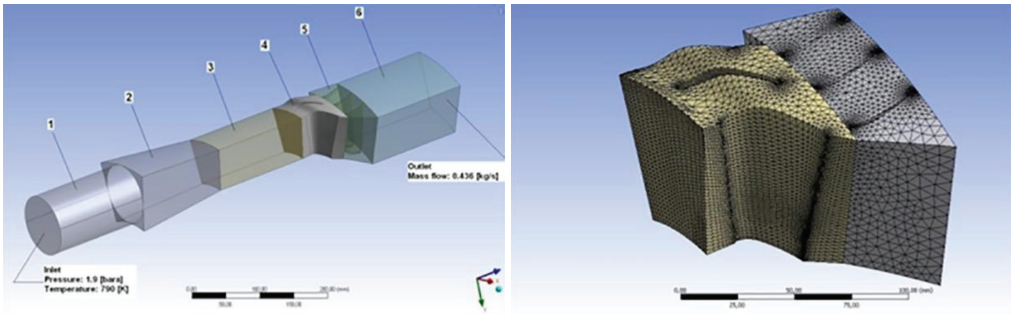

Fig. 5. Geometry modeled (left) and stator/rotor passages mesh (right)

The mesh used for the geometry is unstructured, consisting in tetrahedrons, adding up to approximately 1.75 million elements. In fig. 5 (right) the mesh for the stator/rotor passages is highlighted. Mesh independence studies performed in order to validate the gas turbine performance using boundary layer conditions around the blades revealed a 3% improvement in terms of accuracy in comparison with the gas turbine exploitation manual (as shown further on). Correlated with a major time saving factor, with minimum user involvement in creating the mesh, and the fact that no boundary layer around the blades is imposed, as no studies of flow separations or loss predictions studies are performed, these settings for the mesh are preferred. Moreover, the used software has proven its efficiency in turbomachinery (centrifugal compressors [6], axial turbines [7]). As boundary conditions, the following are imposed:

- inlet pressure: 1.9 [bar];
- inlet temperature: 790 [K];
- outlet mass flow: 0.436 [kg/s]

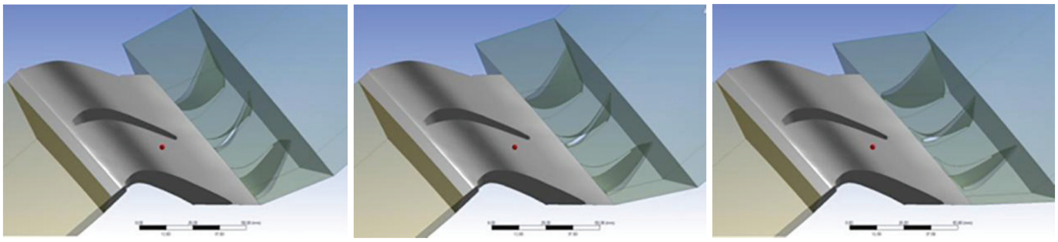

Fig. 6. Rotor blade angles around there own construction axis

Using the tools available in ANSYS Workbench, a parametric case in steady state was developed having as working points the rotational angles of the rotor blade around there own construction axis: 40^{o} (fig. 6 left), 50^{o} (fig. 6 centre) and 60^{o} (fig. 6 right) off their initial rotor blades position. The work fluid is considered air ideal gas and, as turbulence model, the SST (Shear Stress Transport) model was selected due to the fact that is widely tested, offering significant advantages for non-equilibrium turbulent boundary layer flows and heat transfer predictions. The SST model offers high fidelity, providing excellent answers on a wide range of flows and near-wall mesh conditions [8].

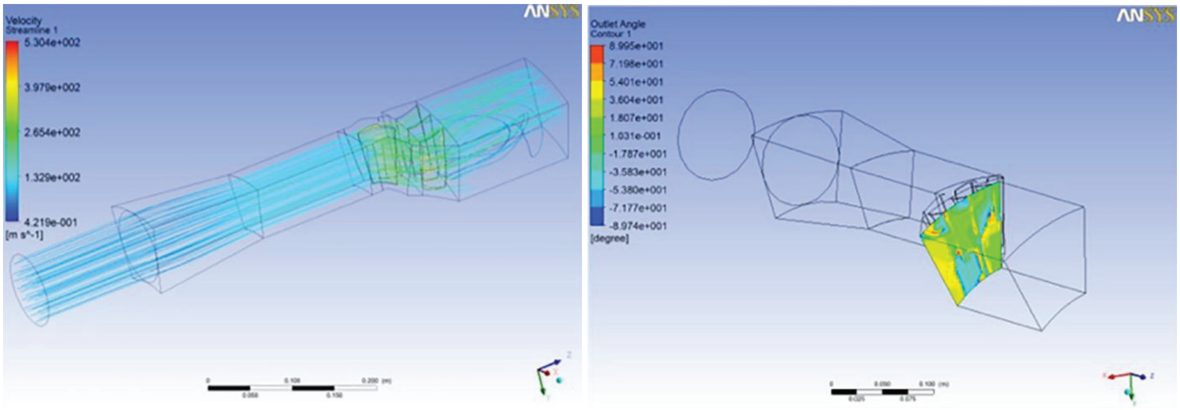

Fig. 7. Velocity streamlines (left) and flow angle contours (right) at 60º static rotor blades angle

Fig. 7 (left) indicates vertices at the rotor outlet, vertices that lead to a non-uniform distribution of the flow angle. In fig. 7 (right) the effect of the vertices over the flow angle contours located at the outlet of the static rotor domain can be visualized. As it reaches the domain of the static rotor blades, the mass flow travels into a passage of increased section than the one of the stator blades. This is the reason why vertices occur and, as a consequence of this physical phenomenon, the simulation software sets wall boundaries at the outlet of the exhaust domain.

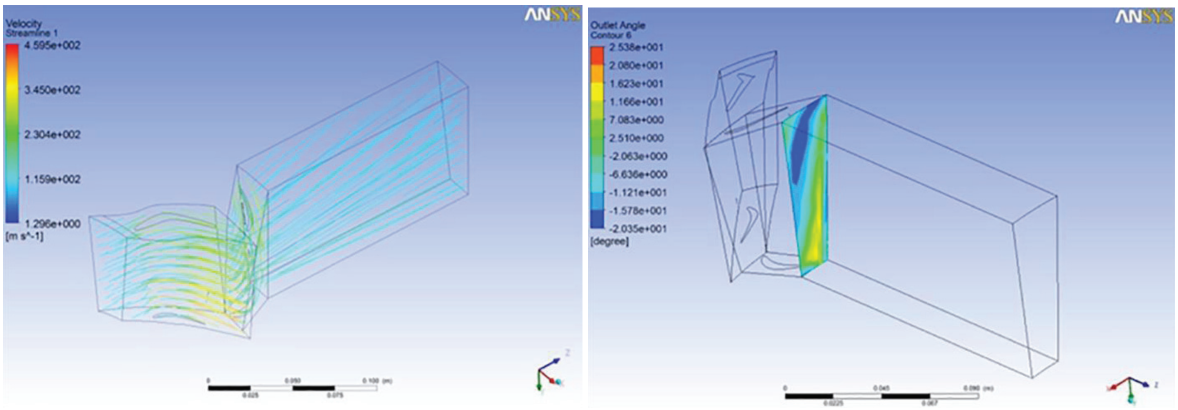

Fig. 8. Velocity streamlines (left) and flow angle contours (right) for the first stage of the power turbine working in nominal regime

The results obtained were compared with a previous case consisting in one stator/rotor blade passage and an exhaust sector. Due to the complexity of the entire axial turbine stage, the model was reduced to these passages using periodic boundaries. It has to be mentioned that the numerical simulations performed using the simplified model validated the gas turbine performance (the results obtained confirmed the values of the gas turbine performance parameters from the gas turbine exploitation manual with an error of 3%). In order to obtain a steady flow, with no transient effects frozen rotor interfaces are used between the stator and rotor. The rotation speed considered was close to the nominal regime for that axial turbine: 11 500 rpm.

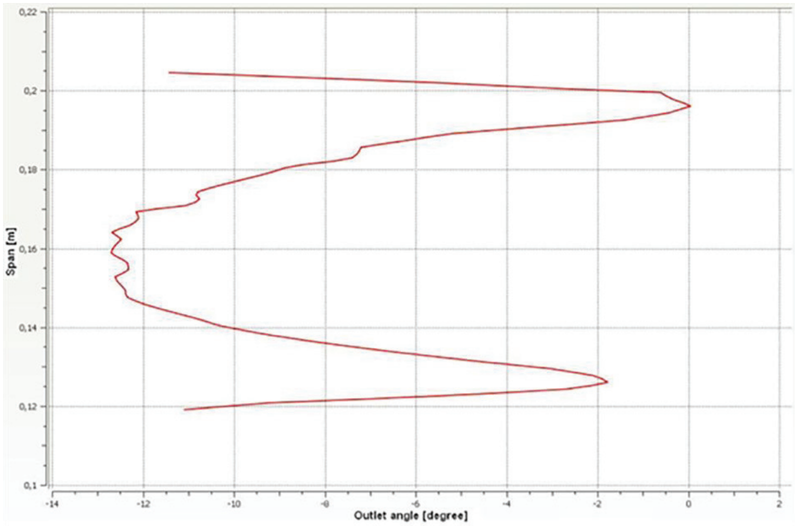

Fig. 9 Flow angle variation at the rotor partial cascade outlet location versus span

Table 1. Comparison of the flow angles between the two cases

	Nominal regime case (rotation speed: 11500 rpm)	Static rotor (60^o rotation around its construction axis)	Static rotor (50^o rotation around its construction axis)	Static rotor (40^o rotation around its construction axis)
Average flow angle at the rotor outlet location	-6.83	-6.44	-4.8	-2.4

Fig. 9 indicates the variation of the flow angle versus span at the rotor outlet location. We can observe variations at the blade's tip and base. The same variations can be observed in fig. 8 (right), expressed in the flow contours. The flow angle defined here represents average values of the angles versus the axial direction at the outlet location of rotor domain, in relation to the mass flow. The average obtained was compared to the average values from the static rotor blades case (see table 1). The value obtained at the 60^o static rotor case is close to the value obtained at the nominal regime case, indicating the fact that a simulation of the rotor's presence could be provided by these static rotor blades orientated at 60^o around their own construction axes.

5. Conclusions

In order to configure the experimental test bench to facilitate in situ combustion, adaptation parts for the inlet sector have been designed in order to ensure the passage from the circular section to the annular section of the first stage power turbine. Copper gaskets and high temperature silicon are considered to be adequate candidates in order to resolve the sealing issues.

For the experimental campaign considered, it has been ascertained that for the solution of rotating the rotor blades around their own construction axis, the average value of flow angle at the rotor outlet is relatively close to the value of the flow angle in case of the nominal working regime of the first stage power turbine, although the distribution is non-uniform. This distribution is the result of the mass flow travelling into a passage of increased section than the one of the stator blades. Previous work, consisting in simulations on simplified stator/rotor passages and an exhaust domain, using periodic boundaries condition, confirmed the performance parameters of the gas turbine engine with an error of 3%. The solid results obtained were considered a validation key for the numerical simulation performed in this paper. The final validation should be provided by the actual experiments conducted in the future within the TURIST project.

The work within the TURIST project continues with the configuration and positioning of the gas injector and instrumentation systems. For the instrumentation, total pressure probes existing within the Research and Development Institute for Gas Turbines COMOTI will be used and mounted with the help of a solution developed in-house.

Acknowledgements

The work presented herein is being supported through the PN-II-PT-PCCA-2013-4 research project *Gas Turbine with In Situ Combustion*, TURIST, contract no. 286/2016.

References

[1] Funding Application for Joint Applied Research Projects PN-II-PT-PCCA-2013-4. Gas Turbine using in situ combustion –TURIST

[2] http://www.conceptsnrec.com/Resources/Photo-Gallery/

[3] David Japikse & Nicholas C. Baines.Introduction to Turbomachinery. Concepts ETI, Inc. And Oxford University Press, Vermont, USA, 1994, ISBN: 0-933283-10-5

[4] Hany Moustapha, Mark F. Zelesky, Nicholas C. Baines & David Japikse - " Axial and Radial Turbines", Concepts ETI, Inc. D.B.A. Concepts NREC, Vermont, USA, 2003, ISBN: 0-933283-12-0

[5] Ainley, D. G. and Mathieson, G. C. R.. A Method of Performance Estimation for Axial-Flow Turbines. Aero Res Council Reports and Memoranda no. 2974, 1951

[6] Bogdan Gherman, Cristina Silivestru, Marian Draghici. Aerodynamic Geometry of a Centrifugal Blower. U.P.B. Sci. Bull., Series D, Vol. 74, Iss. 1, 2012 ISSN 1454-2358

[7] D. Olaru, V. Vilag, C. Cuciumita, J. Popescu, R. Mihalache. Axial Turbine Gasodynamic Study. INAS Conference Sinaia, High Performance Computer Engineering Solutions, 19 – 21 May 2011; ISBN 978-973-0-11025-8

[8]http://resource.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+CFX/Features/Turbulence+Modeling