IMPLEMENTATION OF ONE-DIMENSIONAL CENTRIFUGAL COMPRESSOR DESIGN CODE

Elkin I. Gutiérrez Velásquez, elking@unifei.edu.br
Marco A.R. Nascimento, marcoantonio@unifei.edu.br
Universidade Federal de Itajubá – UNIFEI 37500-903 Itajubá MG, Brasil.

Abstract. *Turbomachinery improvement processes include analytical methods for predicting the performance of the equipment by means of parametric studies in order to show the influence of geometrical changes on the performance under both design and off-design conditions. Nowadays three-dimensional Computational Fluid Dynamics (CFD) codes are used as a reliable turbomachinery design tools which enable savings on expensive experimental developments. However, the CFD assessments require a significant computational cost, both in hardware and in time, what demands a great effort in the approach on the final design of the different components of a turbomachinery.*
The main goal of this work is to present the results of a one-dimensional computer program as a support for tasks of design of a centrifugal compressor impeller with the finality of carrying out fast analysis which can be subsequently evaluated in CFD applications in order to save time and effort in design activities. Besides, it offers the possibility of an open code that permits to implement future improvements and to incorporate different calculation correlations.
The computational program was developed in FORTRAN code which permits the calculation of the main characteristics of a centrifugal compressor by means of the application of non-dimensional parameters, with a vast reduction of computational cost. In order to validate the results of the developed code, the results were compared with experimental measurement data.

Keywords*: CFD, centrifugal compressor design, gas turbines engines.*

1. INTRODUCTION

The implementation of new types of electric generation must consider several issues such as geographic distribution of electric energy production, reliability and flexibility of operation, availability and fuel prices, terms of installation and construction, financing conditions and environmental licensing, among others. Electricity generation must therefore be adapted to the needs of the energy market, while taking features of the electrical system into account, introducing efficiency, reliability and flexibility, while seeking to meet the challenges of ever increasing efficiency use of energy resources and, at the same time, minimizing environmental impacts of this process.

In this context, the distributed generation (DG) arises as a viable alternative, offering a range of benefits to the electricity sector such as: fast response to rising demand, increased reliability by the use of a source not subject to transmission failures, reduction of transmission losses, decrease of environmental impacts by the use of cleaner fuels and the open up of more market opportunities.

Gas turbines present some advantages over other DG technologies, such as reasonable capital costs, a wide range of power (15-300 kW) (Nascimento et al., 2008), operation possibility with different fuels at reasonable efficiency (30 - 33% with regenerator) and low levels of emissions (Lora and Nascimento, 2004).

In the development of turbo-machines analytical methods for predicting performance, by means of parametric studies, are used to demonstrate the influence of geometry changes on performance under design and off-design conditions. 3-dimensional Computational Fluid Dynamics (CFD) codes are used as reliable design tools for centrifugal compressors avoiding expensive experimental development. The impeller is one of the key components of the industrial centrifugal compressors and turbochargers. The impeller design is critical to the success of a compressor stage design. Basic sizing information to establish the initial parameters can save development time for the industrial centrifugal compressor. The diffuser also plays an important role in the compressor operating range once a poorly designed diffuser reduces the compressor operating range and stage efficiency.

This paper presents the results of the preliminary compressor design of a simple cycle gas turbine engine taken in consideration some of the main losses presented in both the impeller as in the diffuser, obtained with the use of a one-dimensional FORTRAN code, as a continuation of work exhibited in Gutiérrez et al. (2010). The differences among the previous and the currents results are shown. Comparisons are made with full three-dimensional CFD analysis and experimental measurement data to validate the results of the developed code. The purpose of this study is to demonstrate that the use of a simple code can produce fast and reliable results without requiring sophisticated computer equipments.

2. CODE VALIDATION

The objective of the developed centrifugal compressor design code is to find out the basic geometry and the main performance parameters of two important components, the impeller and the diffuser, of a centrifugal compressor from some initial parameters what are presented in Tab. 5.

In order to validate the results obtained with the developed code were made comparisons with one experimental impeller employed by Krain (1994). The geometry of the impeller was obtained from the technical data shown in table 1, which were reported by Krain. The differences obtained between the program and the experimental data are shown in Tab. 2. This table shows that these differences are reasonably small. The greatest differences show deviation below 8%. No information related to other parameters is presented once the author does not report any additional information.

Table 1. Input data used for the validation

Parameter	Value	Units
Impeller discharge angle	15.2	deg
Backswept impeller	-30	deg
Isentropic stage efficiency	0.83%	-
Hub/shroud radius ratio	0.386	-
Impeller radius ratio	0.459	-
Mass flow rate	4.0	[kg/s]
Pressure ratio	4.7	-
Absolute inlet pressure	101.32	kPa
Absolute inlet temperature	288	K

Table 2. Impeller compressor validation results

Parameter	Experimental	Code	Deviation
Hub radius [mm]	35.4	37.99	7.32%
Shroud radius [mm]	91.7	98.43	7.34%
Exit radius [mm]	200	214.67	7.33%
Exit width [mm]	14.7	14.16	3.67%
Rotation [rpm]	22360	22847	2.18%
Exit absolute Mach	0.96	0.97	1.04%

2. DESIGN POINT MODELING

The GE Gate Cycle Enter software 5.51 was used to predict the steady state condition design performance of a simple cycle gas turbine. The aim of this simulation was to obtain the air conditions at the compressor entrance. The simulation model used in the simple cycle is shown in Fig. 1.

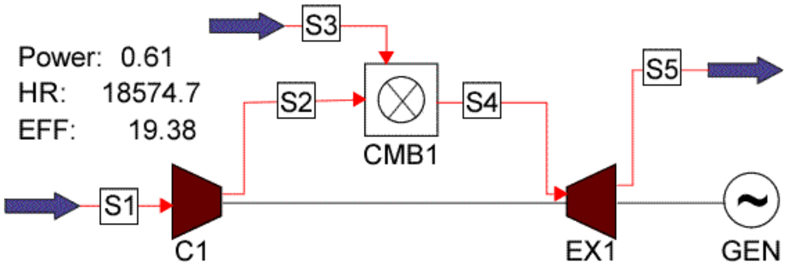

Figure 1. Simple gas turbine cycle scheme used for the thermal simulation

The input design parameters used in the thermal simulation are shown in Tab. 3. These data are based on current technologies for radial turbo-machine. The turbine inlet temperature of 1123 K was chosen as this is the maximum temperature put up by the materials for the manufacture of radial turbines, maintaining the mechanical resistance and the useful life without blade cooling (Oliveira et al., 2010).

Table 3. Input data used for the thermal simulation of the design point

Parameter	Value	Units
Ambient temperature	288	K
Ambient pressure	101.32	kPa
Turbine inlet temperature	1123	K
Fuel temperature	288	K
Pressure ratio	4	--
Compressor adiabatic efficiency	80	%
Combustion adiabatic efficiency	99	%
Turbine efficiency	85	%
Mechanical efficiency	98	%
Air humidity	60	%
Gas turbine power output	600	kW

The establishment of the design point was obtained by a preliminary simulation of the thermal cycle. The output results of the thermal simulation are shown in Tab. 4, according to Fig. 1.

Table 4. Thermal simulation results

Stream	Temperature [K]	Pressure [kPa]	Mass Flow [kg/s]
S1	288.0	101.32	4.28
S2	461.4	405.284	4.28
S3	288.0	500	0.07
S4	1123.0	397.17	4.35
S5	846.2	101.32	4.35

Accord with the information presented in Tab. 3, the purpose of this to design a 600 kW simple cycle gas turbine. This work was thought to be used with an annular combustion chamber, so the designed compressor does not present a volute.

A vaned diffuser model with fixed blades in the shape of a circle arc was selected. The mathematical model applied for the calculation was developed by Japikse and Baines (1998), what allows determining the recovery pressure that can be attained, and the ideal length of the blade, the number and radius of the curvature thereof.

3. ONE-DIMENSIONAL CALCULATION

For the realization of the calculates of a centrifugal compressor it is necessary take into account a set of features and/or limitations imposed by the aerodynamic condition, as well as the characteristics of the materials used for its construction.

The initial parameters required by the code, so as its respective values are shown in Tab. 5, this values were chosen taking in account the results obtained by means of the thermal simulation previously presented. The slip factor of 0.85 is proposed by Dixon (1977) as an usual value for centrifugal compressors.

Table 5. Input parameters required for the calculation code

Parameter	Value	Units
Pressure ratio (PR)	4.0	--
Mass flow rate ($\dot{m}$)	4.28	kg/s
Inlet stagnation pressure	101.32	kPa
Inlet stagnation temperature	288	K
Slip factor (μ)	0.85	--
Impeller efficiency (η_i)	84%	--
Stage efficiency (η_s)	80%	--
Inlet blade angle (β_1)	-60	deg
Discharge blade angle (β_2)	-25	deg
Inlet absolute flow angle (α_1)	0	deg
Discharge abs. flow angle (α_2)	65	deg

Hub/shroud radius ratio (υ)	0.28	--
Impeller radius ratio (r_{1s}/r_2)	0.5	--
Diffuser diameter ratio (D_4/D_6)	1.35	--
Exit diffuser mach number (M_6)	0.33	--

From these parameters, the code determines the velocity triangles as a function of the inlet Mach number, and the thermodynamic relations to the entry, after that it determines the velocity triangles in the discharge, and then sets the relative Mach number in the discharge.

Once the velocity triangles at the inlet and output are determined, the performance dimensionless parameters calculation is carry out. Finally the basic dimensions of both the impeller and the diffuser, and the rotational speed are computed.

3.1. Losses calculation

The code developed take in account some of the main losses presented in a centrifugal compressor, this losses and their calculation method are presented.

3.1.1. Inlet guide vane loss

The inlet vane loss is computed by mean of the Eq. (1) found in Galvas (1972) subject to the assumptions about bade geometry and boundary layer characteristics made in Rohlik (1968). The inlet vane loss is calculated as a fraction of the ideal kinetic energy at the rms diameter.

$$\Delta h_{IGV} = e_s \frac{C^2}{2} \tag{1}$$

Where e_s is an inlet vane loss coefficient.

3.1.2. Inducer incidence loss

The enthalpy loss due to incidence is calculated assuming that the relative velocity component normal to the optimum incidence angle is lost, and then the incidence loss is determined by de Eq. (2)

$$\Delta h_{INC} = \frac{V_L^2}{2C_p} \tag{2}$$

Where

$$V_L = V_{rms} \sin\left(\beta_{opt} - \beta_{rms}\right) \tag{3}$$

The optimum inducer angle was determined according with Stanitz (1953).

3.1.3. Blade loading loss

The equation for calculating the diffusion factor of the impeller based on a uniform loading along the blade chord was presented in Coppage et al. (1956), with the diffusion factor calculated of this way the blade loading loss can be expressed as in Eq. (4)

$$\Delta h_{BL} = 0.05 D_f^2 u_2^2 \tag{4}$$

3.1.4. Skin friction loss

In addition to the losses resulting from the aerodynamic loading of the impeller blades, the impeller incurs in losses due to skin friction of the impeller and shroud wetted areas. In Coppage et al. (1956) was developed an equation for this loss based on developed turbulent pipe flow. Equation (5) is used for calculate de skin friction loss.

$$\Delta h_{SF} = K_{SF} C_f \frac{L}{D_{HYD}} \left(\frac{V}{u_2}\right)_{av}^2 u_2^2 \tag{5}$$

Where K_{SF} = 5.6 for conventional impellers and K_{SF} = 7.0 for impellers with tandem blades.

3.1.5. Disk friction loss

The specific loss due to windage on the compressor back face is calculated using the Eq. (6), which is a form of the disk friction power loss presented in Shepherd (1956).

$$\Delta h_{DF} = 0.01356 \frac{\rho_2}{w \mathrm{Re}^{0.2}} u_2^3 D_2^2 \tag{6}$$

3.1.6. Recirculatio loss

Losses resulting from work done on the working fluid due de backflow into the impeller are expressed in Eq. (7), which is a modification of the equation proposed in Coppage et al. (1956)

$$\Delta h_{RC} = 0.02 \sqrt{\tan \alpha_2} u_2^2 D_f^2 \tag{7}$$

3.1.7. Vanelesss diffuser loss

The flow angle and Mach number variation whit the radius in the vaneless space are determined by numerical solution of the differential equations for vaneless space flow developed in Stanitz (1952). The equations are simplified through the assumptions of adiabatic flow and constant geometric depth passage. The radial total pressure distribution in the vaneless space is calculated using the equation derived from Coppage et al. (1956). When the fluid state and flow properties are determined at the vaned diffuser leading edge radius by the methods described previously, the vaned diffuser loss are calculated from the Eq. (8)

$$\Delta h_{VLD} = C_p T_2^{'} \left[\left(\frac{p_4}{p_4^{'}}\right)^{\frac{\gamma-1}{\gamma}} - \left(\frac{p_4}{p_2^{'}}\right)^{\frac{\gamma-1}{\gamma}} \right] \tag{8}$$

3.1.8. Vaned diffuser loss

The vaned diffuser performance is predicted by the use of test data reported in Runstander (1969). Lines of maximum pressure coefficient recovery at given area ratio were estimated from the performance maps reported for a single plane divergence diffusers with square throats. The value of static pressure recovery coefficient corresponding to the vaned diffuser geometric area ratio, inlet Mach number, and aerodynamic blockage is extrapolated from the test data

The vaned diffuser exit critical velocity ratio is calculated using one-dimensional continuity. The vaned diffuser loss is then calculated using the Eq. (9)

$$\Delta h_{VD} = C_p T_2^{'} \left[\left(\frac{p_6}{p_6^{'}}\right)^{\frac{\gamma-1}{\gamma}} - \left(\frac{p_6}{p_5^{'}}\right)^{\frac{\gamma-1}{\gamma}} \right] \tag{9}$$

3.1.9. Efficieny decrements

Decrements in efficiency caused by the individual losses are obtained from the Eq. 10.

$$\Delta \eta_x = \frac{\Delta h_x}{\Delta h_{act}} \tag{10}$$

Where the subscript x is the individual loss, and Δh_{act} is defined in Eq. 11.

$$\Delta h_{act} = C_p T_0^{'} \left(\frac{T_2^{'}}{T_0^{'}} - 1 \right) + \Delta h_{RC} + \Delta h_{DF} \tag{11}$$

The values obtained for the different losses are shown in Tab. 6.

Table 6. Efficiency decrements caused by individual losses

$\Delta \eta_{IGV}$	0.000
$\Delta \eta_{BL}$	0.020
$\Delta \eta_{SF}$	0.038
$\Delta \eta_{DF}$	0.026
$\Delta \eta_{RC}$	0.012
$\Delta \eta_{INC}$	0.000
$\Delta \eta_{VLD}$	0.012
$\Delta \eta_{VD}$	0.070

4. CFD SIMULATION

A numerical CFD modeling was performed using ANSYS CFX®, this applies the Finite Volume Method (FVM) technique to solve the Navier-Stokes equations. The CFD modeling was performed based on the values presented in Tab. 5.

4.1. Boundary conditions

The whole physic domain was conformed for two sub-domains; one for the impeller (rotating) and another for diffuser (stationary). The reference pressure was setting to 0 kPa, the boundary conditions for the inlet and outlet of the passage are shown in Tab. 7, and a schematic representation of the boundary conditions applied to the whole passage are shown in the fig. 2.

Table 7. Boundary conditions at the inlet and the outlet of the passage

Inlet	
Sub-domain	Impeller
Flow direction	Normal to boundary condition
Flow regime	Subsonic
Heat transfer	Stationary frame total temperature (288 K)
Mass and momentum	Stationary frame total pressure (101.32 kPa)
Turbulence:	Medium intensity and eddy viscosity ratio
Outlet	
Sub-domain	Diffuser
Flow regime	Subsonic
Mass and momentum	Average static pressure (274.52 kPa)
Pressure profile blend	0.05

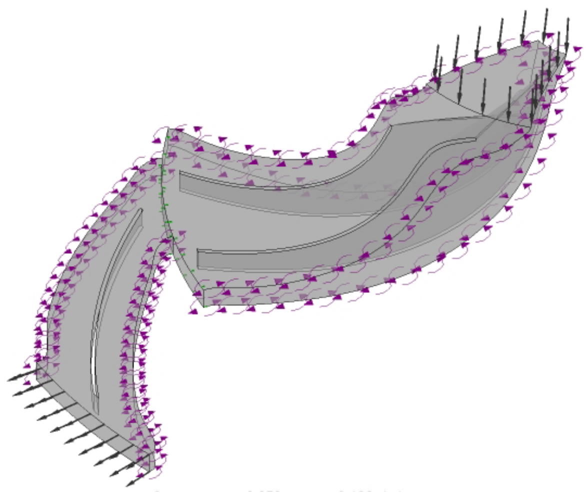

Figure 2. Boundary conditions applied on the passage

4.2. Mesh generation

The mesh generation was made from geometrical information obtained from developed code, this dimensions are presented in the Tab. 8 for the impeller, and in Tab. 9, for the diffuser.

The modeled domain of the whole stage was dissected by a structured multi-block grid topology. H-topology was used for both sub-domains. This topology provides good resolution at the leading and trailing edge. The grids are refined at the near-wall, end wall, leading edge, and trailing edge.

Table 8. Geometrical dimensions used for impeller mesh generation

Parameter	Value	Units
Shroud radius	95.82	mm
Hub radius	26.82	mm
Discharge radius	191.64	mm
Blade height	10.31	mm
Axial length	95.82	mm
Blade Thickness	2	mm
Beta Shroud	60	deg
Beta Hub	25.87	deg
Backswept	25	deg
Blades number	20	----
Rotation	25149	rpm

Table 9. Geometrical dimensions used for diffuser mesh generation

Parameter	Value	Units
Inlet radius	210.8	mm
Discharge radius	284.59	mm
Blade length	144.79	mm
Positioning angle	60.47	mm
Blades number	19	----
Blade Thickness	2	mm

Preliminary simulations showed that the impeller inlet area could to restrict the required air flow, so it was build the geometry of the impeller with 10 passages; each passage of the rotor is compostect for a main blade and a splitter. The splitter has a 30% reduction by the leading edge. The computational mesh for the rotor was composed of 94725 nodes and 84864 elements.

For the diffuser a single blade passage was modeled. The blade was located in the center of the domain and nodes are buried in the width of the blade to improve resolution of the leading and trailing edges, and to reduce grid skewness in these regions. The computational mesh consists of 95732 nodes and 79848 elements.

5. ASSESSMENT OF RESULTS

The code developed permits the calculation of the basic dimensions, the velocity triangles at the inlet and outlet, and some performance parameters of the two components of a centrifugal compressor, the impeller and the diffuser. The set of results obtained by this code were compared with CFD simulation for two cases, with and without losses consideration. The comparisons of the code against CFD simulations results are presented in Tab. 10 are shown the results without losses consideration and taking in account the losses presented in the impeller and in the diffuser.

Table 10. Code results vs. CFD results

		Parameter	without losses			with losses		
			CFD	**Code**	**Dev**	**CFD**	**Code**	**Dev**
Stage Performance		Mass Flow	4.11	4.28	4.14	4.26	4.28	0.38%
		Input Power	780.4	752.2	3.61	724.0	731.2	0.99%
		TPR	4.08	4.23	3.68	4.08	4.00	1.96%
		TTR	1.69	1.61	4.73	1.64	1.59	3.05%
		η_t	74.58	80	7.27	0.81	0.82	0.02
Impeller	Inlet	Static Pressure	89.84	89.84	0.00	89.41	90.13	0.81%
		Total Pressure	101.3	101.3	0.00	101.36	101.32	0.04%
		Static Temperature	278.3	278.3	0.00	277.62	278.53	0.33%
		Total Temperature	288.1	288	0.03	288.02	288.00	0.01%
		Absolute Mach	0.41	0.42	0.01	0.42	0.41	0.01
		Absolute Velocity	138.5	144.8	4.55	141.94	142.797	0.60%
	Outlet	Static Pressure	280	232.4	17.00	290.78	291.37	0.20%
		Total Pressure	512.4	429	16.28	521.05	475.30	8.78%
		Static Temperature	402.7	388.6	3.50	398.95	385.79	3.30%
		Total Temperature	486.6	463	4.85	471.25	458.07	2.80%
		Absolute Mach	0.93	0.98	0.05	0.95	0.97	0.02
		Relative Mach	0.58	0.56	0.02	0.49	0.56	0.07
		Absolute Velocity	377.2	386.5	2.47	380.11	381.07	0.25%
Diffuser	Inlet	Static Pressure	281.36	298.07	5.94	292.53	291.37	0.40%
		Total Pressure	506.76	428.95	15.35	512.74	475.30	7.30%
		Absolute Mach	0.94	0.88	0.06	0.93	0.87	0.06
		Absolute Velocity	380.6	349.5	8.17	373.39	341.04	8.66%
		Flow Angle	26.36	24.32	7.74	71.87	67.02	6.74%
	Outlet	Static Pressure	375.8	375.8	0.00	355.66	375.84	5.67%
		Total Pressure	413.6	405.3	2.01	413.82	405.28	2.06%
		Static Temperature	467.3	453.1	3.04	446.69	448.31	0.36%
		Total Temperature	486.1	463	4.75	473.78	458.07	3.32%
		Absolute Mach	0.33	0.33	0.00	0.43	0.33	0.10

Results show that the discrepancies obtained with losses consideration are significantly lower than the differences obtained without considerer losses as was expected, being those lesser than 4% when compared with the results obtained by CFD simulation. The greatest differences are found in the output impeller total pressure, which differs around 9% which is a percentage appreciably lower that the presented in Gutiérrez et al. (2010). These discrepancies are presented specifically in the area near at the interface impeller-diffuser, which is a highly unstable area. The results in this area show considerable differences with the CFD results fundamentally because the code developed ignores the effect of impeller-diffuser interface.

3. CONCLUSIONS

The developed one-dimensional code constitutes a practical and reliable preliminary design tool for determining the basic configuration of centrifugal compressors.

The code was validated by means of values obtained through experimental measurements and through simulations based on CFD techniques and the results have shown good approximation.

The main contribution of this paper is to demonstrate that by the use of a simple code it is feasible to obtain fairly close results in comparison with those which can be obtained by laborious iterative processes such as those developed through the analysis using CFD techniques.

4. NOMENCLATURE

C	absolute velocity, m/s
C_f	skin friction coefficient
C_p	specific heat, J/(kg K)
D	diameter, m
D_f	diffusion factor
D_{HYD}	mean hydraulic diameter
M	mach number
p	pressure, kPa
P	power, kW
r	radius, mm
rms	root mean square
T	temperature, K
TPR	total pressure ratio
TTR	total temperature ratio
u	blade speed, m/s
U	tangential velocity, m/s
V	relative velocity, m/s

4.1 Greek symbols

α	flow angle
β	blade angle
η	efficiency
γ	specific heat ratio
μ	slip factor
ρ	density
υ	hub/Shroud Radius Ratio

4.2. Subscripts and superscripts

0	Station just upstream of inlet guide vanes
1	Impeller inlet
2	Impeller discharge
4	Vaned diffuser leading edge
5	Vaned diffuser Throat

6 Vaned diffuser exit
av average
h hub
i impeller
m meridian component
s shroud
' Absolute total condition

5. ACKNOWLEDGEMENTS

The authors wish to thank the Coordination of Improvement of Higher Education (CAPES), the National Council of Technological and Scientific Development (CNPq), and the Foundation for Research Support of Minas Gerais (FAPEMIG) for their collaboration and support in the development of this work.

5. REFERENCES

Coppage, J.E. et al., 1956. Study of supersonic radial compressors for refrigeration and pressurization systems. Garrett Corp Los Angeles Ca Airesearch Mfg.
Dixon, S.L., 1977. Fluid mechanics, thermodynamics of turbomachinery. International Journal of Heat and Mass Transfer, 20(2): 185-185.
Galvas, M.R., 1972. Analytical correlation of centrifugal compressor design geometry for maximum efficiency with spefific speed. NASA technical note TN D-6729.
Gutiérrez , E., Nascimento, M.A.R., Miranda, R. and Moura, N., 2010. Centrifugal Compressor One-Dimensional Design Calculation and Results Validation. In: E. 2010 (Editor), 23rd International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems, Lausanne, Switzerland.
Japikse, D. and Baines, N., 1998. Diffuser Design Technology. Concepts ETI Inc, Vermont, USA.
Krain, H., 1994. Test Case 2: Centrifugal Impeller. In: S.a.W.o.D.T.F.P. II (Editor), ERCOFACT Turbonaninery Special Interest Group. DLR, Cologne, Val d'Isère, France.
Lora, E.E.S. and Nascimento, M.A.R., 2004. Geração Termelétrica: Planejamento, Projeto e Operação. [Thermoelectric Power Generation: Design, Project and Operation]. Editora Interciencia, 231-434 pp.
Nascimento, M.A.R. et al., 2008. Cycle Selection and Compressor Design of 600kW Simple Cycle Gas Turbine Engine. ASME Conference Proceedings, 2008(43116): 917-923.
Oliveira, L., Nascimento, M.A.R. and Menon, G., 2010. The Thermal Impact of Using Syngas as Fuel in the Regenerator of Regenerative Gas Turbine Engine. Journal of Engineering for Gas Turbines and Power, 132(6): 062301.
Rohlik, H.E., 1968. Analytical Determination of Radial Inflow Turbine Design Geometry for Maximum Efficiency. NASA Tech. Memo., 4384.
Shepherd, D.G., 1956. Principles of Turbomachinery, New York.
Stanitz, J.D., 1952. One-dimensional compressible flow in vaneless diffusers of radial- and mixed-flow centrifugal compressors, including effects of friction, heat transfer and area change. naca-tn-2610.
Stanitz, J.D., 1953. Effect of blade-thickness taper on axial- velocity distribution at the leading edge of an entrance rotor blade row with axial inlet, and the influence of this distribution on aliniament of rotor blade for zero angle attack. NACA TN-2986.

7. RESPONSIBILITY NOTICE