MEANLINE (1D) DESIGN, OVERALL PERFORMANCE ANALYSIS AND CFD (3D) SIMULATION FOR A 600 kW CYCLE GAS ENGINE

Rubén A. Miranda Carrillo, ruben.miranda@unifei.edu.br
Marco A. R. Nascimento, marcoantonio@unifei.edu.br
Universidade Federal de Itajubá – UNIFEI, Instituto de Engenharia Mecânica
Grupo de Estudos em Tecnologias de Conversão de Energia - GETEC
CEP: 37500-903, Itajubá - MG, Brasil. 55-35-3629 1355

Abstract. *This paper presents the meanline simulation on the aerothermodynamic design and overall performance analysis of the nozzle and radial-inflow rotor for a 600 kW cycle gas engine using a one-dimensional computer FORTRAN code (OFC), aimed at computational and work time reduction. This program utilizes a one-dimensional solution of flow conditions through the turbine along the streamline. The meanline (1D) modeling allows designers to explore an immense design space, produces a basic flow path and explores their effects on the preliminary performance prediction in a short period of time. This procedure now involves a code allowing off-design predictions. Computational Fluid Dynamics (CFD) tools represent a significant source of improvement in the design process of turbomachines, aiming to better performances, lower costs and associated risks. In order to find the most promising design option, a CFD simulation has been used to study the performance, the aerothermodynamic design and the flow characteristics of the turbine components. The OFC results were compared with a computer program for the design and off-design analysis of radial-inflow turbines, CFD simulation and analytical results taken from specialized literature. The comparisons showed the results were in agreement.*

Keywords*: Aerothermodynamic design, overall performance analysis, CFD, numerical meanline simulation.*

1. INTRODUCTION

Small radial-inflow turbines have various applications in industry and are successfully being used as a major component of gas turbines and turbochargers. Although, the design of radial-inflow turbines has improved in the last few years, the detailed aerodynamic study of these components still needs considerable attention. Design of vanes and blades are essential for optimum aerodynamic and overall performance analysis of turbomachinery "Hamakhan and Korakianitis, (2009)". At that time, the design process mainly involved using fundamental flow equations, empirical relationships and engineering expertise to arrive at the best design "Qiu *et al.* (2010)".

This paper presents the meanline simulation on the aerothermodynamic design and overall performance analysis of the nozzle and radial-inflow rotor for a 600 kW cycle gas engine using a one-dimensional computer FORTRAN code (OFC), aimed at computational and work time reduction "Miranda, (2010)" utilizes a one-dimensional solution of flow conditions through the turbine along the streamline. The meanline (1D) modeling allows designers to explore an immense design space, produces a basic flow path and preliminary performance prediction and explores their effects on performance in a short period of time. This procedure now involves elaborated code allowing off-design predictions with the use of specific routines such as "Wasserbauer and Glassman, (1975)".

By "Binder *et al.* (2008)" a numbers of efficient procedures derived from one-dimension analysis have been proposed with that purpose, for example, by "Rohlik, (1968)", "Rogers, (1987)", "Whitfield and Baines, (1990)", "Moustapha *et al.* (2003)" and "Aungier, (2006)".

CFD tools represent a significant source of improvement in the design process of turbomachines, aiming to better performances, lower costs and associated risks. The design of individual gas turbine components using CFD is now commonplace "Lapworth and Shahpar, (2004)". In order to find the most promising design option, a CFD simulation has been used to study the performance, the aerothermodynamic design and the flow characteristics of the turbine components. Numerical models are constantly improved to account for real geometry and include effects of tip clearances, flow path alignment and multi stage simulations "Joubert and Quiniou (2000)". The OFC results were compared with a computer program for the design and off-design analysis of radial-inflow turbines, CFD simulation and analytical results taken from specialized literature. The comparisons showed the results were in agreement.

2. RADIAL-INFLOW TURBINE METHODOLOGY AND PRELIMINARY DESIGN

Preliminary studies of gas turbine power or propulsion systems requires the capability to rapidly produce conceptual designs of the turbines in order to determine geometry, design point performance, and off-design performance "Glassman, (1995)". The design starts with the thermal performance simulation for the design point of the 600 kW cycle gas engine at steady state condition using the GE Gate Cycle Enter software 5.51. Figure 1 describes the methodology design.

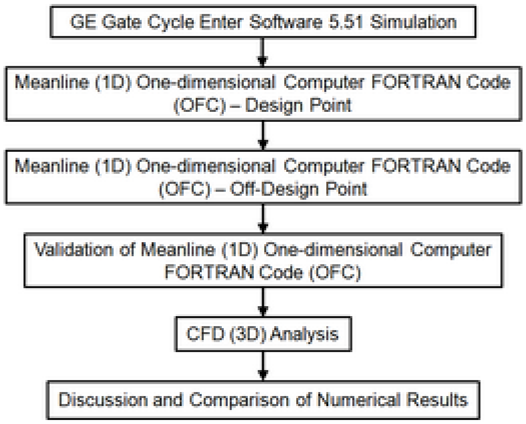

Figure 1. The methodology design

2.1. An integrated cycle gas engine simulation

The design input parameters based on current technologies for radial turbo machineries used in the Gate Cycle simulation are shown in Tab. 1 and the model created for a cycle is illustrate in Fig. 2 "Nascimento *et al.* (2008)". The pressure ratio was 4 and turbine inlet temperature of 1123 K was selected as it is the highest temperature put up by the material of the radial turbine, while maintaining the mechanical resistance and the useful life without any blade cooling. The values assumed for efficiencies, losses, etc., were based on a conservative technology at this phase of the project.

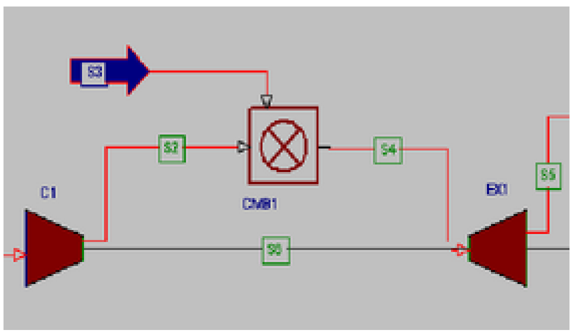

Figure 2. The 600 kW cycle gas engine scheme

Table 1. Input parameters for 600 kW cycle gas engine

Description	Values	Units
Ambient temperature	288	K
Ambient pressure	101.32	kPa
Turbine inlet temperature	1123	K
Pressure ratio	4	--
Compressor adiabatic efficiency	80	%
Combustion adiabatic efficiency	99	%
Turbine efficiency	85	%
Mechanical efficiency	98	%
Recuperator effectiveness	92	%
Combustion chamber pressure loss	2	%
Gas turbine power output	600	kW

2.2. A meanline (1D) modeling

The one-dimensional computer FORTRAN code (OFC) for the design and analysis of radial-inflow turbine components was developed. This program runs very fast; answers are obtained in seconds and many of the codes run on a PC in a matter of minutes or less. This analysis is based on the assumption that there is a mean streamline running through the machine and the conditions on this streamline are representative of the stations being considered "Qiu and Baines, (2007)" and includes consideration of nozzle and radial-inflow rotor trailing-edge blockage. The objective of a meanline analysis is to not reveal the full details of the flow state and velocity through the machine, but conversely, it is limited to determine the machine's overall performance or the combination of overall geometric parameters achieving maximum efficiency "Moustapha *et al.* (2003)". Figure 3 shows the meanline method. The blockage refers to the difference between the real area that circulates the fluid and the geometric area of the component as in Eq. (1). Figure 4 shows the effect of the blockage factor.

$$B = 1 - \frac{A_{eff}}{A_{geo}} \qquad 1)$$

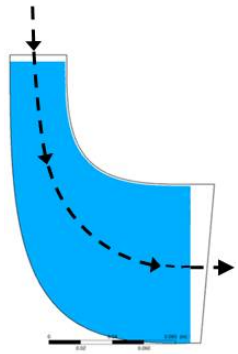

Figure 3. Meanline method

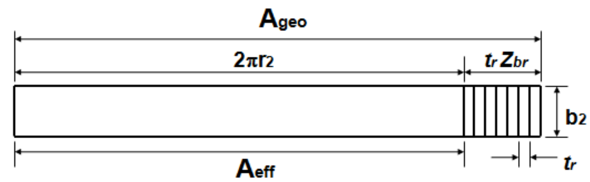

Figure 4. The blockage factor

This computer program initially calculates the velocity triangle and the Mach number at the rotor inlet, simultaneously with the thermodynamic relation between the temperatures and pressures at this point. The next step is the calculation of the velocity triangle at the discharge, aimed at finding the discharge end speed and the relative Mach number. The three dimensional variation of inlet and outlet flow angles is determined by streamline curvature calculations and they vary from hub to shroud. Once the velocity triangles are established, the OFC initiates the calculation of the non-dimensional performance parameters and the specific speed.

In this phase of the project the turbine itself was developed, finding the appropriate geometry, the required number and size of vanes and blades of the nozzle and rotor, respectively. Furthermore, the nozzle will be sized before the turbine as it plays an important role in the flow acceleration. Figure 5 shows the axial and radial clearance. This effect on the aerodynamic performance is taken into account and is assumed to act as an orifice.

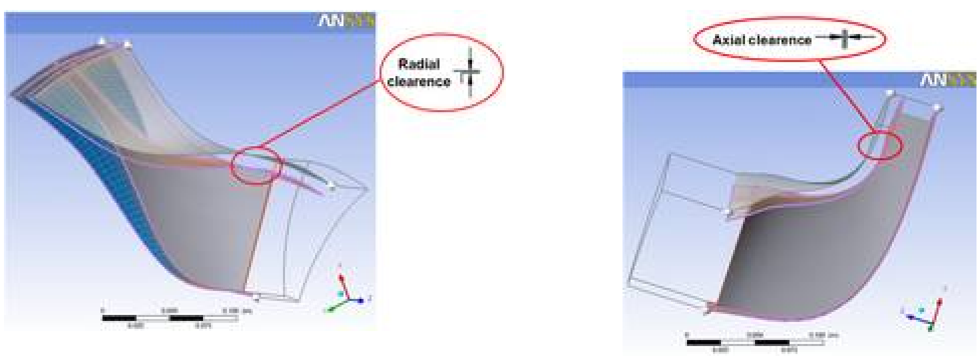

Figure 5. Axial and radial clearance

2.3. Radial-inflow turbine geometry

Turbine geometry was examined in detail for calculated points and near the curve of maximum static efficiency. Several design parameters were then plotted as functions of specific speed to illustrate the changes in optimum turbine shape. Table 2 describes the basic input values of the thermodynamic parameters of the 600 kW cycle gas engine.

Table 2. Design point input parameters for a 600 kW cycle gas engine

Description	Values	Units
Mass flow rate	4.5	kg/s
Inlet turbine total temperature	1123	K
Inlet turbine total pressure	396	kPa
Total to static turbine pressure ratio	3.96	--
Total to static rotor efficiency	85	%
Total nozzle efficiency	90	%
Inlet relative flow angle	-25	º
Outlet relative flow angle	-60	º
Tip clearance	1	mm

2.4. Evaluating the nozzle preliminary design

The radius of the nozzle vane inlet and outlet at the design point were determined according to the turbine rotor diameter and the geometric constraint conditions. Table 3 shows the nozzle inlet and outlet aerothermodynamic model. The nozzle consists of 17 vanes with a radial-flow inlet from a low Mach number. The design inlet and the exit angle is 54.59° and 77.97°, respectively. Both the trailing-edge and the leading-edge thickness are constant and the blockage factor is 0.98. Figure 6 shows the nozzle geometry calculates by OFC computer program.

Table 3. Nozzle aerothermodynamics outputs.

Description	Inlet ($_0$) Values	Outlet ($_1$) Values	Units
Vane height	33.51	33.51	mm
Nozzle radius	278.48	227.67	mm
Thickness	1	1	mm
Number of vanes	17		--
Absolute flow angle	54.59	77.97	°
Ratio of nozzle inlet to rotor inlet diameter	1.25	--	--
Ratio of nozzle exit to rotor inlet diameter	--	1.04	--
Ratio to nozzle inlet to nozzle exit diameter	1.19		--
Total temperature	1123	1123	K
Static temperature	1117.05	981.95	K
Total pressure	396	374.31	kPa
Static pressure	388.70	234.00	kPa
Absolute velocity	109.30	532.30	m/s

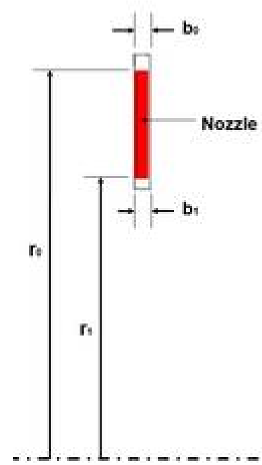

Figure 6. Nozzle geometry

Figure 7 presents the variation of nozzle exit absolute flow angle (α_1) as function of the specific speed. A continuous decrease from α_1 = 83° to 52° occurs as specific speed increases from 0.2 to 1.34. For the design point (PP), α_1 = 77.93 this value is 3.76% above the value obtained by "Rohlik, (1968)".

Figure 8 shows the minimum number of nozzle vanes (Z_{bb}) as function of nozzle exit absolute flow angle for different values of nozzle inlet absolute flow angle (α_0), nozzle vane radius ratios (r_0/r_1) and ideal nozzle efficiency (η_n = 100%). The ideal design point (PP_{id}) is positioned below the nozzle vane radius ratio curve from 1.2 and between nozzle inlet absolute flow angle curves range from 55° to 50°.

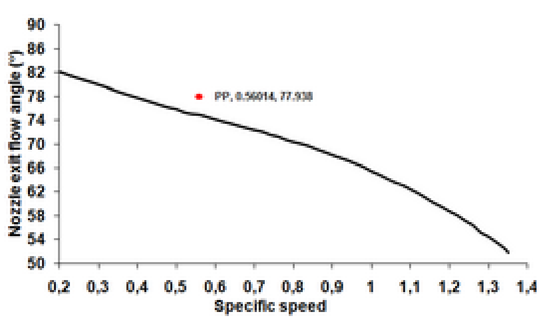

Figure 7. Variation in nozzle exit flow angle corresponding to maximum static efficiency with specific speed "Rohlik, (1968)".

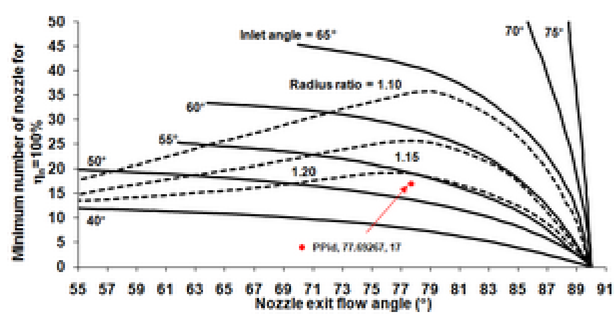

Figure 8. Predicted correlation of minimum nozzle vane number with vane inlet and exit angles and vane inlet/exit radius ratio "Moustapha *et al.* (2003)".

2.5. Evaluating the radial-inflow rotor preliminary design

The radial-inflow rotor consists of 15 blades with radial-flow inlet and axial-flow outlet. The trailing-edge, the leading-edge thickness and the rotor axial and radial clearance are constant whereas the blockage factor is 0.98. Table 4 presents the rotor geometric and thermodynamic output parameters at the design point calculate by OFC. Figure 9 shows the radial inflow rotor geometry calculates by OFC computer program.

Table 4. Rotor aerothermodynamic outputs

Description	Inlet (2) Values	Outlet (3) Values		Units
Blade height	33.51	99.67		mm
Radius	227.94	Shroud ($_{3s}$)	131.06	mm
		Hub ($_{3h}$)	31.39	
Axial length	125.15	--		mm
Thickness	1	1		mm
Number of blades	15			--
Relative flow angle	-25	Shroud ($_{3s}$)	-22.52	°
		Hub ($_{3h}$)	-60	
Absolute flow angle	77.50	0		°
Tip clearance	1	1		mm
Total temperature	1123	812.66		K
Static temperature	975.45	793.09		K
Total pressure	373.19	108.90		kPa
Static pressure	227.94	100		kPa
Absolute velocity	544.42	200.68		m/s
Blade speed	586.468	Shroud ($_{3s}$)	83.24	m/s
		Hub ($_{3h}$)	347.58	
Relative velocity	130.01	Shroud ($_{3s}$)	217.26	m/s
		Hub ($_{3h}$)	401.36	

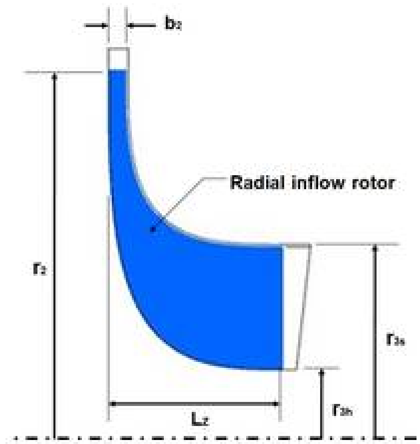

Figure 9. Radial-inflow rotor geometry

Figure 10 shows the variation in total to total efficiency (η_{tt}) as function of specific speed for a selection of nozzle exit flow angles. For each value of (α_1) a hatched area is drawn, inside of which the various diameter ratios are varied. The envelope of maximum (η_{tt}) is bounded by the constraints for $N_{ss} \geq 0.58$ in these hatched regions. This envelope is the optimum geometry curve and has a peak (η_{tt}) of 0.87 at N_{ss} = 0.58. For the design point (PP), α_1 = 77.93 this value is 5.70% above the value obtained by "Rohlik, (1968)".

Figure 11 shows the variation of total to static efficiency as function of specific speed for different reasons rotor diameters (D_{3m}/D_2) for a constant nozzle exit flow angle. The design point (PP) is positioned between the curves of the rotor diameter ratio of 0.55 and 0.49. For this condition, "Rohlik, (1968)" recommends values of D_{3m}/D_2 = 0.39.

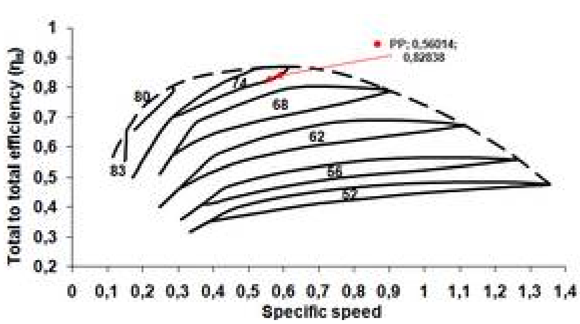

Figure 10. Calculated performance "Rohlik, (1968)".

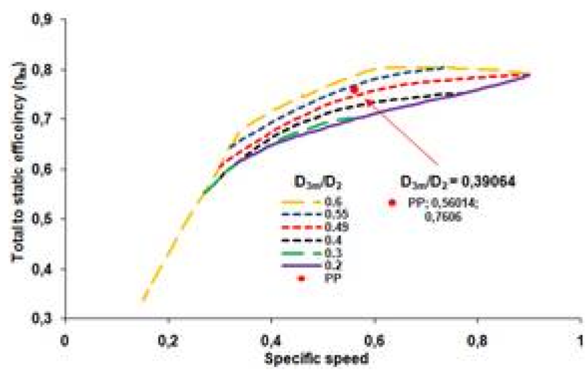

Figure 11. Calculated performance for a constant nozzle exit flow angle "Rohlik, (1968)".

3. TURBINE PERFORMANCE MAPS

The overall turbine performance analysis characteristics are presented in the form of performance maps. Figure 12 presents the mass flow parameter ($\dot{m}\sqrt{T_{02}}/P_{02}$) plotted against total to static pressure ratio (P_{00}/P_3) for lines of constant speed. The design point (PP) was 3.80 and at each speed the mass flow increases with total to static pressure ratio until it reaches a maximum when the turbine is to be in choked and further increases in total to static pressure ratio do not result in any greater mass flow rate. All of the curves in Fig. 12 collapse into a single curve occur in narrow band of total to static pressure ratio 5 to 6.5 due the nozzle chokes first, the choked mass flow rate is not a function of the rotational speed because the nozzle do not rotate “Miranda, (2010)”.

Figure 13 presents the mass flow parameters as a function of total to static pressure ratio for a nominal rotational speed, generated by CFD and OFC for the design point. The both programs have the same behavior in quite narrow band of pressure ratio 2.3 to 4.7 and the mass flow parameter obtained by OFC was 3.80. This value is 1.14% lower than of the value obtained by CFD “Miranda, (2010)”.

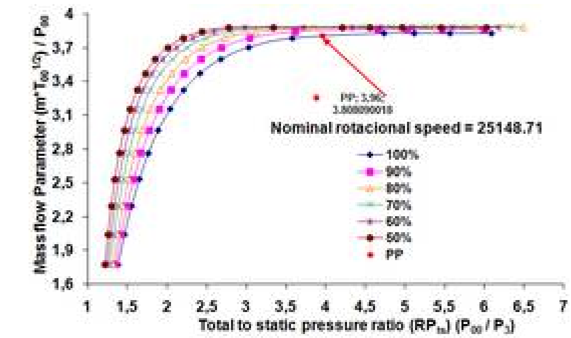

Figure 12. Mass flow parameter vs. total to static pressure ratio “Miranda, (2010)”.

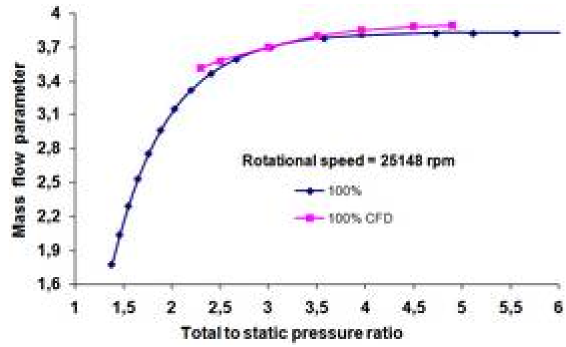

Figure 13. Mass flow parameter by CFD and comparison with OFC “Miranda, (2010)”.

Figure 14 shows the total to static efficiency plotted against total to static pressure ratio for lines of constant speed. The maximum efficiency occurs in quite narrow band of total to static pressure ratio 2.5 to 3.5, particularly at speeds below the design point (rotational speed = 90%) “Miranda, (2010)”.

Figure 15 presents the total to static efficiency as function of total to static pressure ratio at the nominal rotational speed, generated by CFD and OFC for the design point. The both programs have the same behavior in quite narrow band of pressure ratio 3.0 to 4.8 “Miranda, (2010)”.

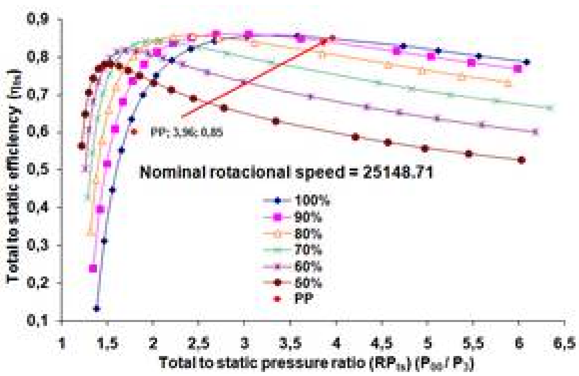

Figure 14 Total to static efficiency vs. total to static pressure ratio “Miranda, (2010)”.

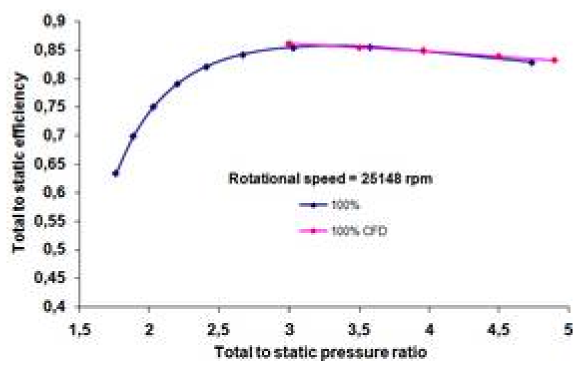

Figure 15. Total to static efficiency by CFD and comparison with OFC “Miranda, (2010)”.

4. A MEANLINE (1D) AND CFD (3D) SIMULATIONS REGIONS

The general requirement for the analytical procedure is to predict the component discharge conditions from the known inlet conditions and component geometry. The computed discharge conditions then become known inlet conditions for the next component. In the OFC method and CFD numerical simulation, the modeling of flow passage in the radial-inflow turbine is divided into several regions. Figure 16 shows the imaginary inlet duct, nozzle (0-1), interspaces (1-2), rotor (2-3) and imaginary outlet duct.

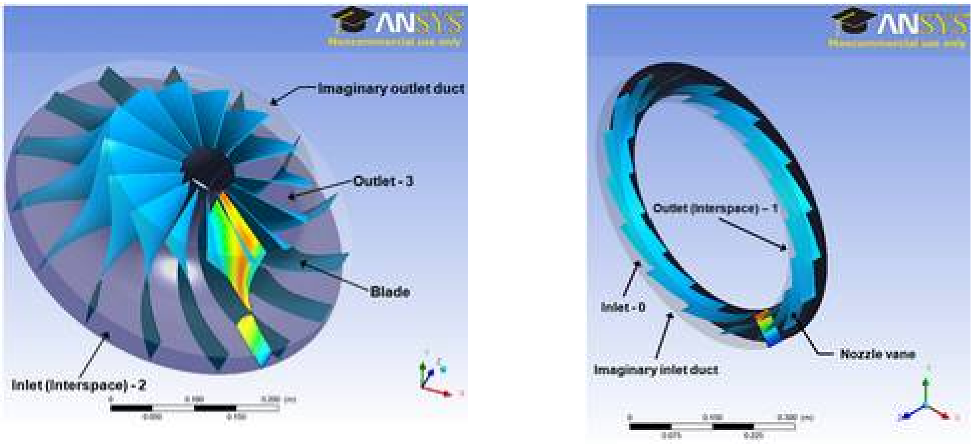

Figure 16. Geometry modeling and simulation regions

From the meanline results, an initial 3D geometry model can be constructed. This geometry model serves as the core of the design system on which further aerodynamic design, such as full 3D CFD analysis, can be subsequently performed. Once the important geometric parameters of the turbine stage have been defined, such parameters are transferred to the blade generator using ANSYS BladeGen so that the 3D geometry of the blades, for both nozzle and rotor, can be developed. The geometry, the axial and radial clearance of the nozzle vanes and rotor blades are internally defined using the ANSYS TurboGrid 12.0. The numerical simulation models were produced using the commercial package ANSYS CFX 12.0® codes. The computational domain of the whole stage was dissected by the hexahedron structural multi-block grid topology. In this case an H grid was used, reaching good resolution at the leading and trailing edge. Figures 17 and 18 shows the meshes used around the nozzle vanes and the rotor blades.

Figure 17. Computational nozzle vane mesh

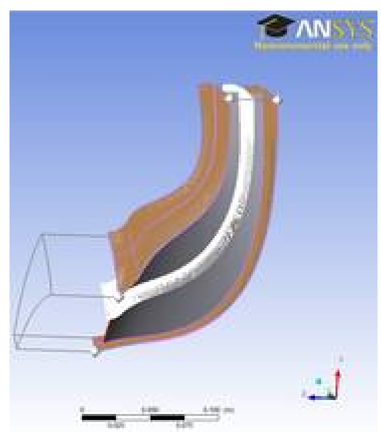

Figure 18. Computational rotor blade mesh

The CFD simulations were carried out on the improved designs using grids of the order of 81138 nodes per nozzle passage and 105556 nodes per single rotor passage (Tab. 5), using a mixing plane model to connect the stationary and rotating domains. The boundary conditions have been projected from the leading to trailing edge as show in Tab. 6.

Table 5. Nozzle and rotor mesh information

Domain		Nodes	Elements
Domain I: Stationary	Nozzle	81138	67952
Domain II: Rotating.	Rotor	105556	89248

Table 6. Boundary conditions

Average total pressure imposed at the inlet area
Average static pressure imposed at the outlet
Heat transfer model = Total energy
Turbulence model = SST

5. VALIDATION OF NUMERICAL MODELS

The NASA TN D-8164 report, by "Glassman, (1976)", contains the FORTRAN computer program, developed for the design analysis of radial inflow turbines. This report provides sufficient information and comprehensive geometric design data for the validation case test. Table 7 presents the design point input parameter used in the one-dimensional computer FORTRAN code (OFC) for NASA TN D-8164.

Table 7. Design point input parameters for NASA TN D-8164 and OFC

Description	Values	Units
Mass flow rate	0.2771	kg/s
Inlet turbine total temperature	1083.329	K
Inlet turbine total pressure	91.0110	kPa
Total to static turbine pressure ratio	1.6125	--
Total to static rotor efficiency	89.1500	%
Total nozzle efficiency	81.7688	%
Inlet relative flow angle	-31.5000	º
Outlet relative flow angle	-70.6500	º
Rotor exit hub to shroud radius ratio	0.3493	--
Specific heat ratio	1.6670	J/kgK
Tip clearance	0.0059	mm
Relative velocity ratio	2.3698	--
Rotor blade thickness	0.2884	mm
Nozzle vane thickness	0.2884	mm

The results obtained using the OFC method were compared with theoretical results provided by "Glassman, (1976)", and CFD validation simulation (Fig. 19). The discrepancies observed for each computer program are because "Glassman, (1976)", considered the nozzle and rotor loss coefficient calculated by Reynolds number equal to 1. Table 7 describes the nozzle comparisons between the NASA TN D-8164 and OFC. Figure 19 presents the total pressure distribution on meridional plane.

Table 7. Nozzle numerical model

Description	Units	OFC	NASA TN D-8164 [13]	Variation (%)
Nozzle inlet				
Diameter	mm	199.24	195.55	1.89
Total temperature	K	1083.32	1083.33	0.00
Total pressure	kPa	91.01	91.01	0.00
Static temperature	K	1076.34	1072.02	0.40
Static pressure	kPa	89.55	88.65	1.02
Abs. flow angle	°	51.45	55.60	-7.46
Absolute velocity	m/s	105.73	108.49	-2.54
Number of vanes	--	17	16	6.25
Nozzle outlet				
Diameter	mm	161.04	158.75	1.44
Total temperature	K	1083.32	1083.33	0.00
Total pressure	kPa	87.27	89.95	-2.97
Static temperature	K	1008.19	1013.94	-0.57
Static pressure	kPa	72.92	76.23	-4.33
Abs. flow angle	°	74.38	72.00	3.32
Absolute velocity	m/s	279.56	268.60	4.06

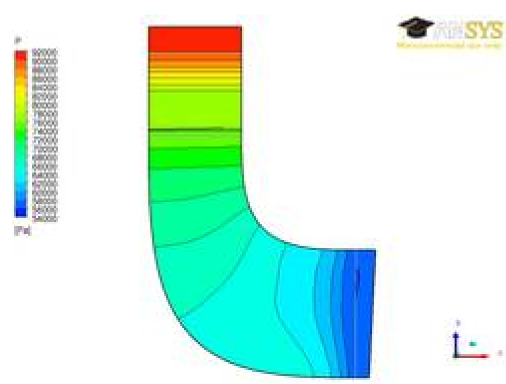

Figure 19. Total pressure distribution on meridional plane

Table 8 describes the rotor comparisons between the NASA TN D-8164, OFC and CFD simulations. Figure 20 presents the velocity vectors at 50% span.

Table 8. Rotor numerical model

Description	Units	OFC	NASA TN D-8164 [13]	Variation (%)
Rotor inlet				
Diameter	mm	158.83	155.39	2.21
Total temperature	K	1083.33	1083.33	0.00
Total pressure	kPa	87.16	89.95	-3.10
Static temperature	K	1005.98	1010.85	-0.48
Static pressure	kPa	72.43	75.65	-4.25
Abs. flow angle	°	74.25	71.92	3.24
Absolute velocity	m/s	283.64	274.59	3.30
Blade speed	m/s	320.18	313.24	2.22
Relative flow angle	°	-31.50	-31.50	0.00
Shaft power output	kW	24.22	22.37	8.27
Number of blades	--	13	12	8.33
Rotor outlet				
Shroud diameter	mm	34.98	38.72	-9.64
Hub diameter	mm	100.16	110.84	-9.63
Total temperature	K	915.27	926.12	-1.17
Total pressure	kPa	57.18	57.35	-0.28
Static temperature	K	910.46	920.20	-1.06
Static pressure	kPa	56.44	56.44	0.00
Abs. flow angle	°	0	0	--
Absolute velocity	m/s	70.90	78.48	9.65
Blade speed	m/s	201.91	223.44	9.64
Relative velocity	m/s	213.99	236.82	9.64

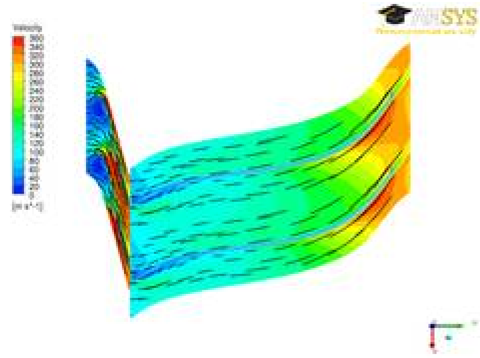

Figure 20. Velocity vectors at 50% span

6. NUMERICAL SIMULATION

The machine overall performance and the average mixed out properties at the inlet and outlet boundary of the CFD were also calculated. Table 9 describes the comparison between the CFD simulation and the OFC for nozzle turbine and shows discrepancies lower than 7% between the values calculated by each simulation. Figure 21 presents the nozzle total pressure distribution on meridional plane.

Table 9. Nozzle numerical simulation

Description	Units	ANSYS CFX 12.0	OFC	Variation (%)
Mass flow rate	kg/s	4.55	4.50	1.16
Nozzle inlet				
Static pressure	kPa	393.02	395.78	-0.69
Total pressure	kPa	396.00	396.00	0.002
Static temperature	K	1119.48	1122.82	-0.29
Total temperature	K	1123.00	1123.00	0.00
Abs. flow angle	°	-0.11	54.51	--
Nozzle Outlet				
Static pressure	kPa	212.35	233.83	-9.18
Total pressure	kPa	369.29	374.28	-1.33
Static temperature	K	959.85	981.77	-2.23
Total temperature	K	1122.48	1123.00	-0.04
Absolute velocity	m/s	565.68	532.39	6.251
Abs. flow angle	°	77.17	77.72	-0.71

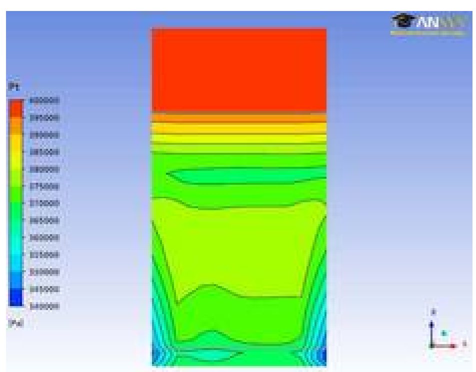

Figure 21. Nozzle total pressure distribution on meridional plane

Table 10 describes the comparison between the CFD simulation and the OFC for rotor turbine and shows discrepancies lower than 13% between the values calculated by each simulation. Figure 22 presents the relative Mach number on meridional plane.

Table 10. Rotor numerical simulation

Description	Units	ANSYS CFX 12.0	OFC	Variation (%)
Shaft speed	rpm	25148.64	25148.44	0.001
Mass flow rate	kg/s	4.55	4.50	1.16
Rotor inlet				
Static pressure	kPa	212.38	227.94	-6.82
Total pressure	kPa	361.83	373.19	-3.04
Static temperature	K	961.57	975.45	-1.42
Total temperature	K	1123.47	1123.00	0.04
Abs. Mach number	--	0.93	0.87	7.09
Blade speed	m/s	589.91	586.48	0.58
Absolute velocity	m/s	569.11	544.43	4.53
Abs. flow angle	°	77.28	77.50	-0.27
Relative velocity	m/s	132.07	130.02	1.58
Rotor outlet				
Static pressure	kPa	99.97	100.00	-0.02
Total pressure	kPa	111.35	108.90	2.24
Static temperature	K	788.76	793.09	-0.54
Total temperature	K	813.34	812.66	0.08
Abs. Mach number	--	0.39	0.35	9.95
Blade speed	m/s	242.64	215.42	12.63
Absolute velocity	m/s	218.53	200.68	8.89
Abs. flow angle	°	15.02	0.00	--
Relative velocity	m/s	281.87	294.41	-4.26

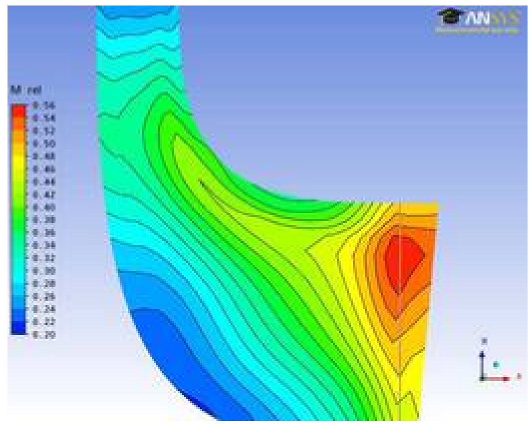

Figure 22. Relative Mach number on meridional plane

7. CONCLUSIONS

The numerical meanline simulation and investigations on the aerothermodynamic design, overall performance prediction and CFD analysis for a 600 kW cycle gas engine are conducted in this paper:

Comparing the one-dimensional computer FORTRAN code (OFC) results and the NASA TN D-8164 by "Glassman, (1976)" showing discrepancies lower than 10% between the values calculated by each computer program and when comparing the one-dimensional computer FORTRAN code (OFC) and the ANSYS CFX 12.0 simulations results, with discrepancies lower than 13% between the values calculated for each simulations program.

In spite of the sophistication and improvement of 3D CFD simulation, one-dimensional meanline analysis still has its irreplaceable role in the turbomachinery design process. First, the meanline program is the starting point of a new design. With only the design requirements known at the preliminary design stage, the CFD software would have no geometry to work with. The meanline program can come up with the initial geometry and the flow path. Second, the fast meanline (1D) solver allows the exploration of a large design space. It can run hundreds of parametric studies in seconds, while the same task would be prohibitively time-consuming for a 3D CFD solver.

The role of CFD in the design procedure is fast growing. New tools have been developed allowing the treatment of numerous difficult problems as close as possible to the reality: tip clearance and multistage effects.

8. ACKNOWLEDGEMENTS

The authors wish to thank the Petrobras Research and Development Center (CENPES), 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.

9. REFERENCES

Hamid, S. and Elder, R.L., 1992, Radial Inflow Turbine Study, Cranfield Institute of Technology.

Hamakhan, I.A. and Korakianitis, T., 2009. "Aerodynamic performance effects of leading-edge geometry in gas-turbine blades". Applied Energy 87 (2010), Elsevier Ltd., doi:10.1016/j.apenergy.2009.09.017, pp. 1591–1601.

Qiu, X., Anderson, M. and Japikse, D., 2010. "An Integrated Design System for Turbomachinery", Proceedings of the 9th International Conference on Hydrodynamics, Shanghai, China, pp. 358-364.

MIRANDA, R.A.C., 2010. "Design and Performance Analysis of Radial Inflow Turbine", MSc. Dissertation - Instituto de Engenharia Mecânica, Universidade Federal de Itajubá, Itajubá, Brazil pp 224.

Wasserbauer, C.A. and Glassman, A.J., 1975, "FORTRAN program for predicting off-design performance of radial-inflow turbines," Tech. Rep. TN D-8063, NASA, Washington, DC, USA.

Binder, N., Carbonneau, X., and Chassaing, P., 2008. "Off-design considerations through the properties of some pressure-ratio line of radial inflow turbines". Hindawi Publishing Corporation, International Journal of Rotating Machinery, Vol. 2008, Article ID 273296, doi:10.1155/2008/273296, pp 8.

Rohlik, H.E., 1968, "Analytical determination of radial inflow turbine design geometry for maximum efficiency," Tech. Rep. TN D-4384, NASA, Washington, DC, USA.

Rogers, C., 1987, "Mainline Performance Prediction for Radial Inflow Turbine in Small High Pressure Ratio Turbine," VKI, Lecture Series 1987-07.

Whitfield, A. and Baines, N., 1990, Design of Radial Turbomachines, JohnWiley & Sons, New York, NY, USA.

Moustapha, H., Zeleski, M. F., Baines, N. and Japikse, D., 2003, Axial and Radial Turbines, Concepts NREC, White River, Junction, Vt, USA.

Aungier, H., 2006, Turbine Aerodynamics: Axial-Flow and Radial- Flow Turbine Design and Analysis, ASME Press, New York, NY, USA.

Lapworth, L. and Shahpar, S., 2004, "Design Of Gas Turbine Engines Using CFD", Proceedings of the European Congress on Computational Methods in Applied Sciences and Engineering, ECCOMAS 2004, Jyväskylä, Finland, pp. 21.

Joubert, H. and Quiniou, H., 2000, "Turbomachinery Design Used Intensive CFD", Proceedings of the ICAS 2000 Congress, pp. 12.

Glassman, A. J., 1995, "Design Geometry And Design/Off-Design Performance Computer Codes For Compressors And Turbines" NASA-CR-198433, Lewis Research Center, National Aeronautics and Space Administration, Cleveland, Ohio, USA.

Nascimento, M.A., Venturini, O. J., Lora, E., Sierra, G., Rodrigues, L., Carvalho, H. and Moura, N., 2008, "Cycle selection and compressor design of 600 kW simple cycle gas turbine engine", ASME TURBOEXPO 2008, Berlin, ASME GT2008-51523.

Qiu, X., and Baines, N., 2007, "Performance prediction for high pressure-ratio radial inflow turbines", ASME TURBOEXPO 2007, Montreal, ASME GT2007-27057.

Glassman, A.J., 1976, "Computer Program for Design Analysis of Radial-Inflow Turbines", NASA TN D-8164, Report N° E-8394, Lewis Research Center, National Aeronautics and Space Administration, Cleveland, Ohio, USA.

10. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.