THE USE AND COMPARISON OF AVAILABLE DESIGN TOOLS FOR A 3-STAGE AXIAL-FLOW COMPRESSOR: MEANLINE, STREAMLINE CURVATURE AND CFD

Jesuino Takachi Tomita, jtakachi@ita.br
João Roberto Barbosa, barbosa@ita.br
Instituto Tecnológico de Aeronáutica, ITA
Praça Marechal Eduardo Gomes, 50 - Vila das Acácias
CEP 12.228-900 - São José dos Campos – SP – Brasil

Oleg Dubitsky, obd@conceptsnrec.com
Concepts NREC
217 Billings Farm Road
White River Jct., VT 05001 USA

Abstract. *In the past, researchers on the gas turbine components design had developed several methodologies and techniques, incorporating their empirical data and expertise, for the design and analysis of the components and of the turbomachines of which they were made. Industries like GE, Pratt-Whitney, Rolls-Royce and Snecma, to name just a few, developed high technological content computational codes for their compressors and turbines design. However, these technologies are proprietary and may not be available for any users. Moreover, to understand the design process, expertise is needed on the theory of turbomachine flows, loss mechanisms and design parameters, without which it is difficult to improve their efficiencies, namely the compressor's or turbine's. A solid knowledge and very well trained human resources become necessary to complete the design process. In this work, it is shown how to design a 3-stage axial flow compressor using different design tools, in-house codes and commercial softwares. A design procedure for an axial flow compressor is described and comparison of the results obtained from different computer programs is shown.*

Keywords*: gas turbine, axial compressor, internal flow, turbomachinery, CFD*

1. INTRODUCTION

The design of high performance turbomachines requires, in addition to very specialized manpower, powerful software special machines, materials and fabrication processes. It is a time consuming and very expensive activity, so that just a few companies are in this business. In the last decades, with the advent of new computational technologies, it became possible to use robust numerical methods to solve the complex systems of equations of fluid mechanics for the machine design and analysis.

Modern multistage turbomachines are the result of a highly complex design process based on simulations, design and manufacturing expertise, supported by numerous and very powerful computational resources and numerical methods. Despite all theoretical developments, experimental tests are still required to match the ever increasing performance demand. Due to the complex design process, the technology involved in turbomachinery design is multidisciplinary (Kau, 1998). Several areas like, not necessarily in this order, thermodynamics, transport phenomena, heat transfer, pure and applied mathematics, numerical methods, manufacturing, material sciences, vibration and structural analysis, acoustic, gas dynamics must be associated with the designer expertise to make any project viable. This work is an example of the many that must be carried out the turbomachine design. It is presented bearing in mind that all turbomachinery components are affected by requirements on aerodynamic and structure to bear the loads involved.

Gas turbine components are designed by repeatedly adjusting geometrical parts until finding a suitable machine that combines acceptable aero-thermodynamic performance with low stress levels, and it is practical and economic to manufacture and to be maintained.

This work deals with a design of a 3-stage axial flow compressor using different tools to calculate the main compressor geometry, flow properties and to study the flowfield behavior in the compressor. The study is based on softwares developed at the Center for Reference on Gas Turbines (Streamline Curvature – SLC) at ITA, detailed in Barbosa (1987, 2001) and a commercial package developed by Concepts ETI, Inc. (AXIAL™[1] and AxCent®[2] with the Pushbutton CFD®[3] module), from now on referenced as commercial software. The results are presented, discussed and compared.

[1] AXIAL is a trademark of Concepts ETI, Inc.
[2] AxCent is a registered trademark of Concepts ETI, Inc.
[3] Pushbutton CFD is a registered trademark of Concepts ETI, Inc.

2. DESCRIPTION OF THE DESIGN TOOLS

The requisites for the axial compressor design were taken from thermodynamic cycle requirements, from which the air mass-flow, pressure ratio and efficiency are obtained for the input parameters to start the compressor design.

The first step is the preliminary design of the compressor. The mean-line based computational code Axial Flow Compressor Code (AFCC), developed by Tomita (2003), has been used to generate the first approximation of the compressor geometrical parameters and performance maps (Tomita and Barbosa, 2003a, 2003b). Variable Inlet Guide Vanes (VIGV), Variable Stator Vanes (VSV) and Bleed-off-Valve (BOV) were considered using that computational code, whose loss models and implementation issues were described by Tomita (2003) and optimization techniques by Sousa *et al.* (2005). The optimized compressor for the gas turbine was studied calculating the engine running line using the computational code developed by Bringhenti (2003), whose details can be seen in Bringhenti *et al.* (2006). Silva *et al.* (2007) studied the gas turbine transient performance using the compressor variable geometry operation maps calculated using AFCC.

The following step was the use of the output from the AFCC code to generate the input for the streamline curvature program. The streamline code used in this work is the development of the one developed by Barbosa (1987), from which it is possible to obtain the entire compressor channel and blading data, on several streamlines from hub to tip.

Streamline curvature (SLC) is a technique well suited for the design of axial flow compressors, mainly because it allows quick access to vital flow properties at the blade edges, from which actions may be taken to improve its performance. Such technique has been successfully used for analysis, from which more realistic compressor maps can be synthesized early in the design process, even with variable geometry.

The calculations are standard for each row, starting with the integration of the complete radial equilibrium equation at the blade leading edge based on assumed axial velocity at the hub. The integration of the radial equilibrium equation results in a spanwise axial velocity distribution, from which the total mass flow at the blade inlet can be calculated. The exact velocity distribution is obtained forcing the calculated mass flow to be equal to the required one, acting on the assumed hub axial velocity. At this point, new radial positions for the streamlines are calculated, based on the specified mass flow for each streamtube and by inverse interpolation of mass flows as function of radii.

The calculations proceed to the blade trailing edge using a loss model to predict the pressures and temperatures at the outlet nodes. The user can choose one among several loss models, based on his experience and/or on hints available from the program. The calculations proceed at the trailing edge similarly to what is done at inlet, except that, for the sake of better convergence, the axial velocity guess is at the mean streamline. After the calculation at a specified row is finished and the streamlines repositioned at the trailing edge, the losses at the bladeless space are calculated and the conditions at the next blade leading edge are obtained. Blade after blade the calculations are repeated until the end of the compressor. At this stage the blockage factor due to the endwall boundary layer is calculated for each row, replacing the existing values. If the streamlines have been repositioned to new positions that differ from the previous more than a suggested fixed limit (for instance, one micrometer is used in this work), an iterative procedure redoes all the calculations from the first blade row. Otherwise, a converged solution is achieved. At this point all blades geometry and flow properties are known at the blade's edges. Efficiency, calculated based on mass-averaged enthalpy distribution, is checked against the target value.

The commercial software is widely used both by industries and universities around the world. In particular, at ITA it has been used as one of the data sources for verification and validation of the indigenous softwares.

The results of the SLC program were transferred to the commercial software for the sake of comparison of results and generation of the 3D compressor geometry. After the 3D geometry was calculated, the commercial CFD module was activated to generate a mesh for the numerical calculation of the flow using the Navier-Stokes equations.

The commercial design tool, which is meanline-based software, with a reduced through-flow capability to produce hub, mid-height and tip dimensions and flow properties at these positions, from which full compressor geometry is built. The process is iterative, allowing the designer expertise to be used aiming at improvement of the compressor operational stability and efficiency. The flow thermodynamic properties are finally made available at any position at the grid. It is also possible to predict the compressor performance, choke and stall, as described by Dubitsky (2003). Empirical models may also be used to predict noise and stress levels.

All turbomachinery design tools described above are based on inviscid formulation. The viscous effects are accounted for by semi-empirical correlations. These correlations depend on the airfoil profile, related Reynolds and Mach numbers, type of machine (if axial, radial or diagonal types) and so on. The success of each model is linked to the way the losses are incorporated in it.

There are many factors influencing losses in a compressor (Lakshiminarayana, 1996). A good assessment depends on the knowledge of the mechanisms through which they act. Due to the tremendous complexity of these mechanisms through which the losses are generated, and to their interactions, it is unlikely that they act independently. Nevertheless, different types of losses are considered as independently generated. Correction factors are then applied to overcome these approximations. The calibration and modification of these models and their coefficients are complex tasks, only recommended for specialists.

Since the early stages of compressor design, an extensive evaluation of those blade profiles best suited to axial compressors has been carried out at many research centers. Most of the results are published and available to the public but key information is retained as proprietary. Company-based in-house expertise in the particular machine types and operating conditions is the key to good design because it allows the incorporation of correction factors which bring the model closer to reality. Therefore, any attempts to accurately predict the behavior of a particular compressor are likely to be unsuccessful if that expertise is not available.

Fortunately, some researchers have incorporated their expertise into correlations of parameters which describe the flow in blade passages. Such correlations are an attempt to synthesize the results of many tests into simpler formulae or sets of curves. They are generally averages of test results or their statistical curve fits. Hence, they are not expected to represent each individual compressor; that is they cannot guarantee good results every time they are applied. In other words, a general model for all compressors is unrealistic: the model can do well for some compressors and not quite so well for others.

2. THE 3-STAGE AXIAL FLOW COMPRESSOR

In this work, the 3-stage axial flow compressor has the following characteristics:

- Mass Flow = 16.4 kg/s
- Inlet Total Pressure = 101325 Pa
- Inlet Total Temperature = 288 K
- Tip speed = 433 m/s
- Inlet Guide Vane (IGV)
- Rotors and Stators Blade Profiles: Double-Circular Arc (DCA)

The SLC and the commercial softwares are used to calculate the flow properties at blade leading (LE) and trailing (TE) edges. Both programs are based on the inviscid formulation modified by semi-empirical loss models to account for losses.

Eleven streamlines were used to calculate the flow properties from hub to tip of the blades using the SLC program. The user can choose the loss models, which can be seen in details in Barbosa (1987) that best fits his needs.

For the commercial program, the flow properties are calculated at three positions (hub, mid-height and tip). Several loss models are also available. The experience and expertise of the program developers are used to select the combination of the loss models.

Since each loss model is developed to account for a specific design environment, there is no general loss model. In this work, the same compressor geometrical data were used for all the studies reported herein.

2.1. Comparison of the results of SLC and AXIAL

The figures below show some of the calculated values using the chosen software. Data are plotted for the blade span rotor inlet. R1 stands for the first rotor, and so on. Figures 1 to 6 show the total-to-total and static-to-static pressure ratios. Qualitatively the results are correct and quantitatively in reasonable agreement. The differences may be caused by the differences in the calculation methodology (meanline vs streamline curvature), implementation of loss, deviation correlations and end-wall blockage models. The incidence and deviation angles depend on the loss correlation used by the designer. Figures 7 to 9 show the Mach number distribution at the rotor inlet and outlet. The results from both programs are in good agreement mainly at the third rotor outlet.

The diffusion factor is a measure of flow deceleration, which is associated to the losses: high values may indicate stall. Figures 10 to 12 show the diffusion factor distribution along the blade height, from which may be concluded that all results are within acceptable range, indicating that, at that design moment, the compressor geometry would be considered acceptable and adequate for further and more sophisticated analysis.

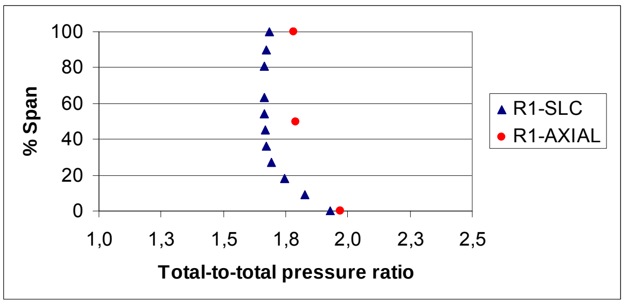

Figure 1. Comparison of the total-to-total pressure ratio at the first rotor.

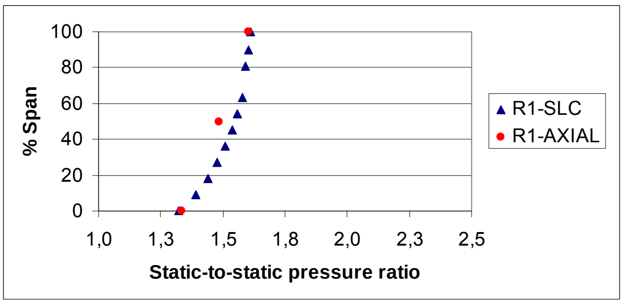

Figure 2. Comparison of the static-to-static pressure ratio at the first rotor.

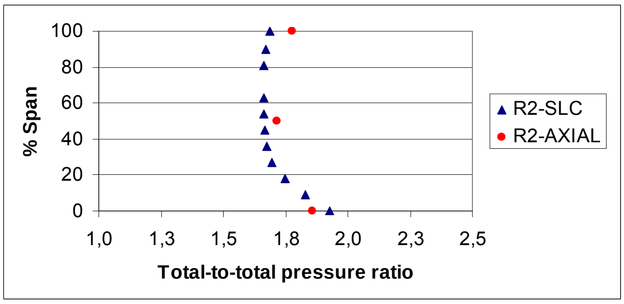

Figure 3. Comparison of the total-to-total pressure ratio at the second rotor.

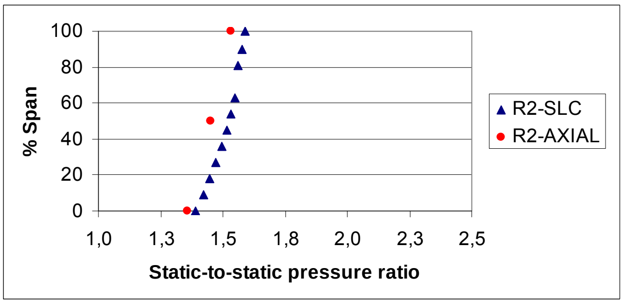

Figure 3. Comparison of the static-to-static pressure ratio at the second rotor.

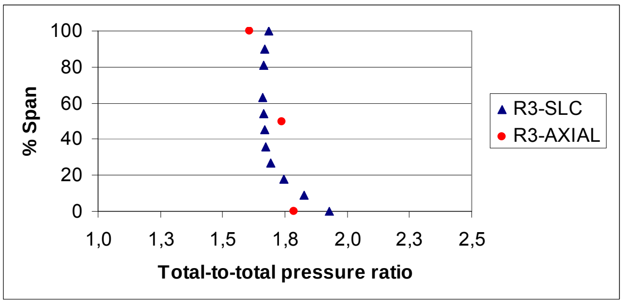

Figure 5. Comparison of the total-to-total pressure ratio at the third rotor.

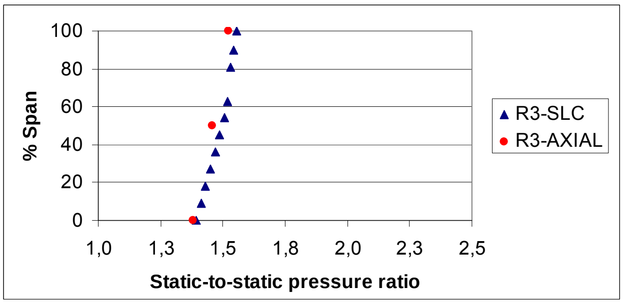

Figure 6. Comparison of the static-to-static pressure ratio at the third rotor.

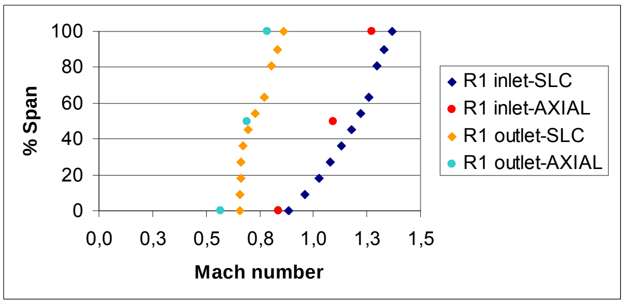

Figure 7. Comparison of the Mach number distribution at the first rotor.

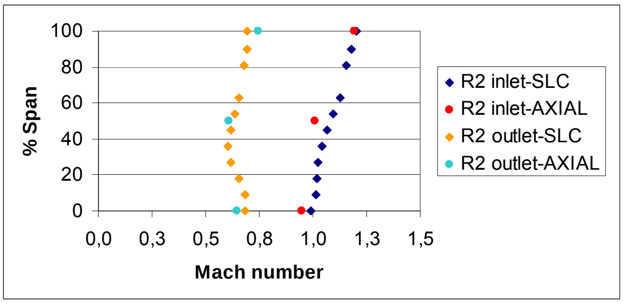

Figure 8. Comparison of the Mach number distribution at the second rotor.

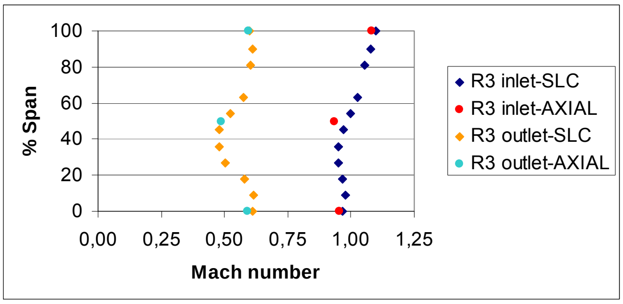

Figure 9. Comparison of the Mach number distribution at the third rotor.

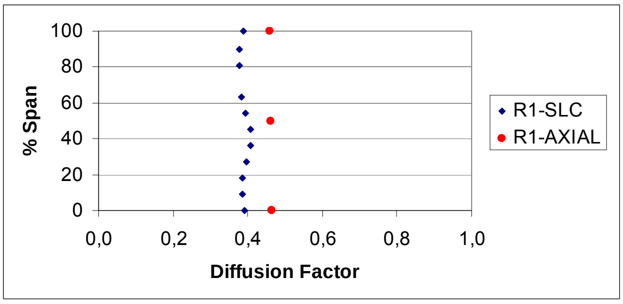

Figure 10. Diffusion factor distribution at the first rotor along the blade height.

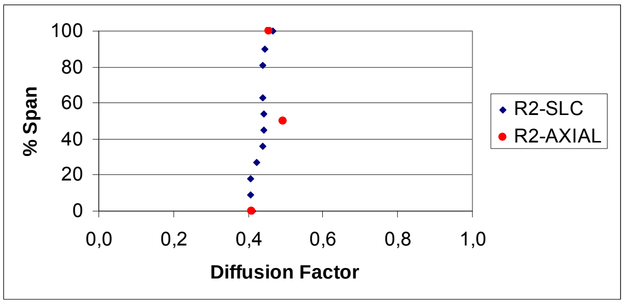

Figure 11. Diffusion factor distribution at the second rotor along the blade height.

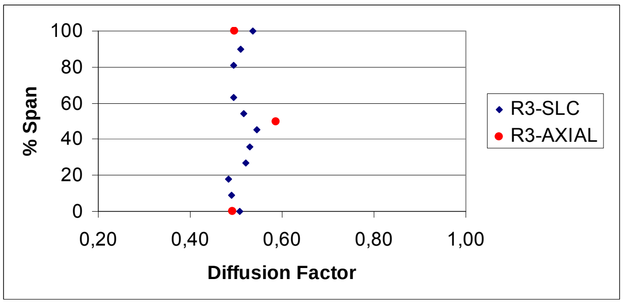

Figure 12. Diffusion factor distribution at the third rotor along the blade height.

Figure 13 shows the normalized static and total pressures and the Fig. 14 the normalized static and total temperatures both at the compressor outlet. Very good agreement between the calculated static temperature profiles by both computer programs.

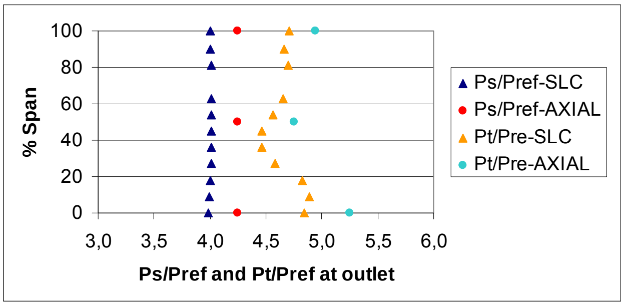

Figure 13. Static and total pressures distribution at the compressor outlet.

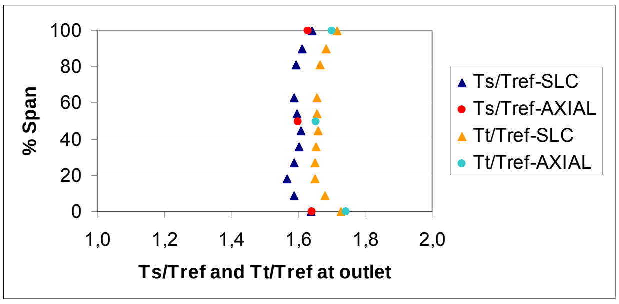

Figure 14. Static and total temperatures distribution at the compressor outlet.

Figure 15 also shows good agreement between the calculated compressor outlet Mach number profiles.

Figure 16 shows meridional views of the variable mean diameter 3-stage axial compressor reported in this work. The IGV blade is colored green; the rotors are colored red and the stators blue. Details of the smoothness of the axial channel can be observed from the figure. A smooth channel is a must to avoid losses (flow separation).

It is important to emphasize that the results of the calculations depend highly on the calibration and settings of the loss models, what is interactively achieved using the designer expertise. It cannot be claimed that any one of both computer programs may produce similar results without some understanding of axial compressor physics and modeling.

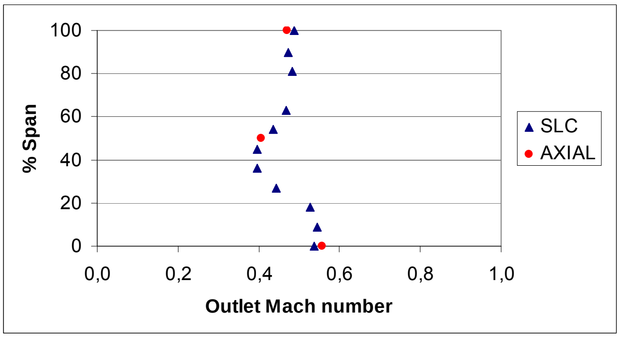

Figure 15. Mach number distribution at the compressor outlet.

Figure 17, which represents the compressor solid drawing, gives an idea of the actual compressor 3D geometry.

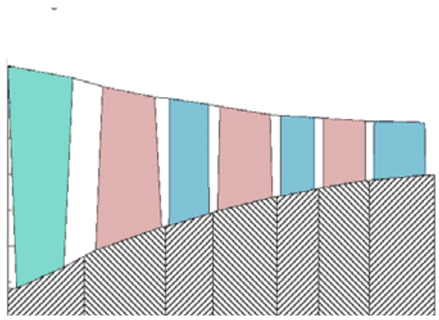

Figure 16. Meridional view of the compressor.

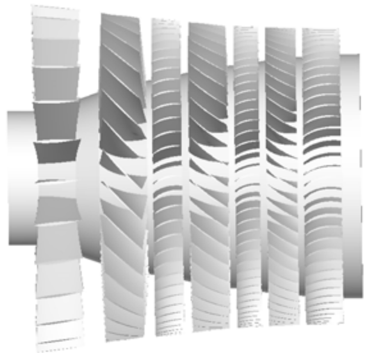

Figure 17. 3D view of the blade profiles.

Further steps are still needed to conclude the design, among which a detailed CFD analysis to search for possible sources of loss and undesirable flow properties. This must be carried out at both design and representative off-design conditions, although not included in this work.

2.2. Comparison of results from SLC and commercial softwares

As indicated, two different tools were used for the design and analysis of a 3-stage axial compressor, which gave rise to discrepancies in the calculated parameters, attributed to the different numerical methods and loss correlations. Despite proprietary information that may be included in the computer programs of each research center, the results may be comparable and in good agreement.

For the 3D flow calculation, a 71x61x421-node grid was generated using the commercial CFD module, resulting in 1,823,351 nodes in the computational domain. Tip clearances ($5x10^{-4}$m, $3.5x10^{-4}$m and $2.5x10^{-4}$m for each rotor blade) were included in the study, with allowance of four cells in the spanwise direction.

The boundary conditions are set at the compressor inlet and outlet; periodic boundary condition at the blade-to-blade domain; stagnation pressure and temperature at inlet; static pressure at hub outlet, and using radial equilibrium, from extended from hub to tip. Between rows, flow properties are transferred by the mixing-plane technique. The effects of turbulence are calculated using the Spalart and Allmaras (1992) one-equation turbulence model.

Spatial discretization is based on Liou's (1996) third order Advection Upstream Splitting Method ($AUSM^+$), which uses a cell interface Mach number based on characteristic speeds from the neighboring cells. Upwind extrapolation for the convective inviscid fluxes is based on the interface Mach number. Another type of splitting is used for the pressure terms. An explicit four-step second order Runge-Kutta scheme is used for the time integration, with Courant number of 1.5.

As expected, the convergence was highly dependent on the flow initialization. If the initialization was in a condition near instability, no convergence was achieved, due to the flow transient phenomena involved, as reported by Tomita (2009). Thus, selection of good initial condition is a must to achieve convergence. When using the 3D CFD code developed by Tomita (2009) the initial flow field was calculated using the SLC program, while the commercial CFD module derives the initial condition from the design results.

- **Comparison between SLC and Pushbutton CFD**

For the calculations a DELL laptop with 2.0 GHz INTEL Core 2 Due CPU with 4GB RAM was used. The solution was obtained after 65h20min of CPU and 2500 iterations. The numerical residual history is shown in Fig. 18. Other parameters, like mass flow, were monitored to check convergence.

The pressure ratio of 4.66 and the isentropic efficiency of 82.8% were calculated from CFD results. These values are in agreement with the values used for the compressor design. Figure 19 show the static pressure contours at the compressor mean diameter. The y^+ calculated at compressor hub and blades are 35.28 and 5.17, respectively.

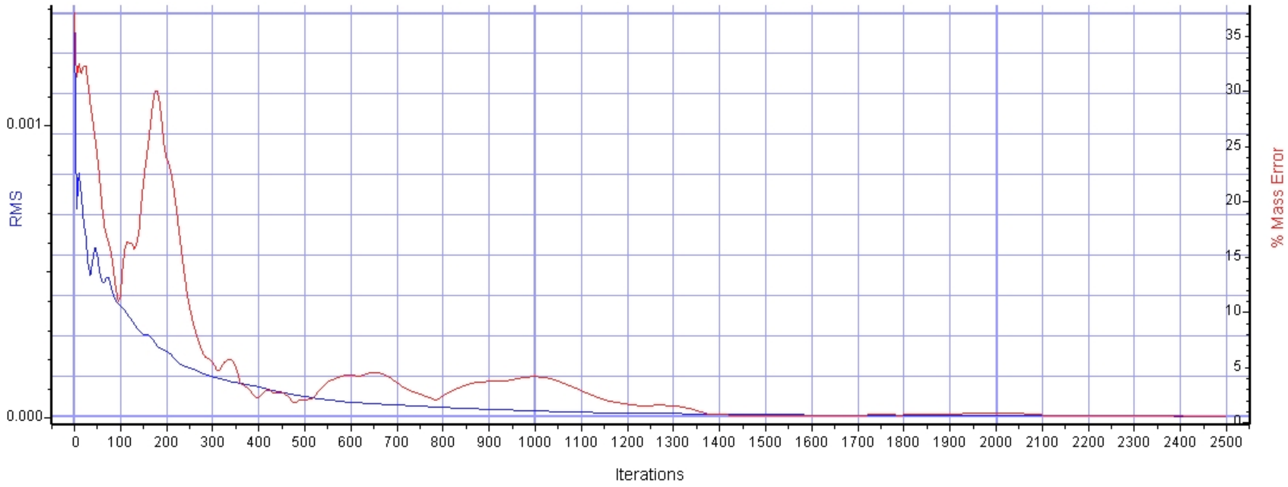

Figure 18. RMS residual history.

Figure 20 shows the velocity vectors at the clearances above the rotor tip. The study of the flow in this region may suggest ways to improve the compressor efficiency, since high tip clearances may cause an unacceptable drop in efficiency, as demonstrated by Tomita and Barbosa (2007). A reverse flow was detected at the suction side of the first rotor, indicating that modification in the rotor blade profile design would be necessary.

The Mach number and static pressure profiles obtained from SLC and CFD spanwise calculations are analyzed and compared in Fig. 21 and 23, showing the Mach number distributions at all the rotors' leading edges. Figures 24 to 26 show the static pressure distribution, indicating considerable differences close to the hub and in the gaps at the tips. This is because the turbomachinery design tools account for both viscous effects and tip clearance using semi-empirical loss correlations. Despite these differences, the main characteristics are similar for both SLC and CFD calculations.

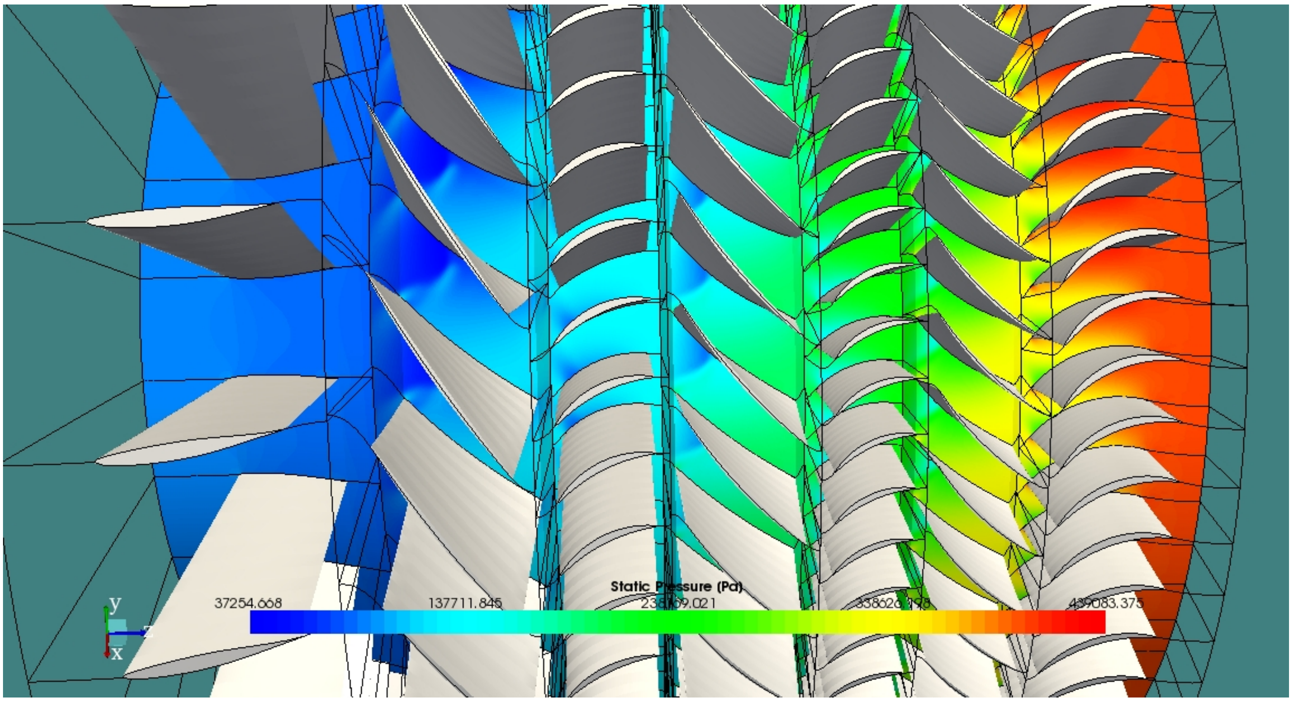

Figure 19. Static pressure contours at the compressor mean diameter.

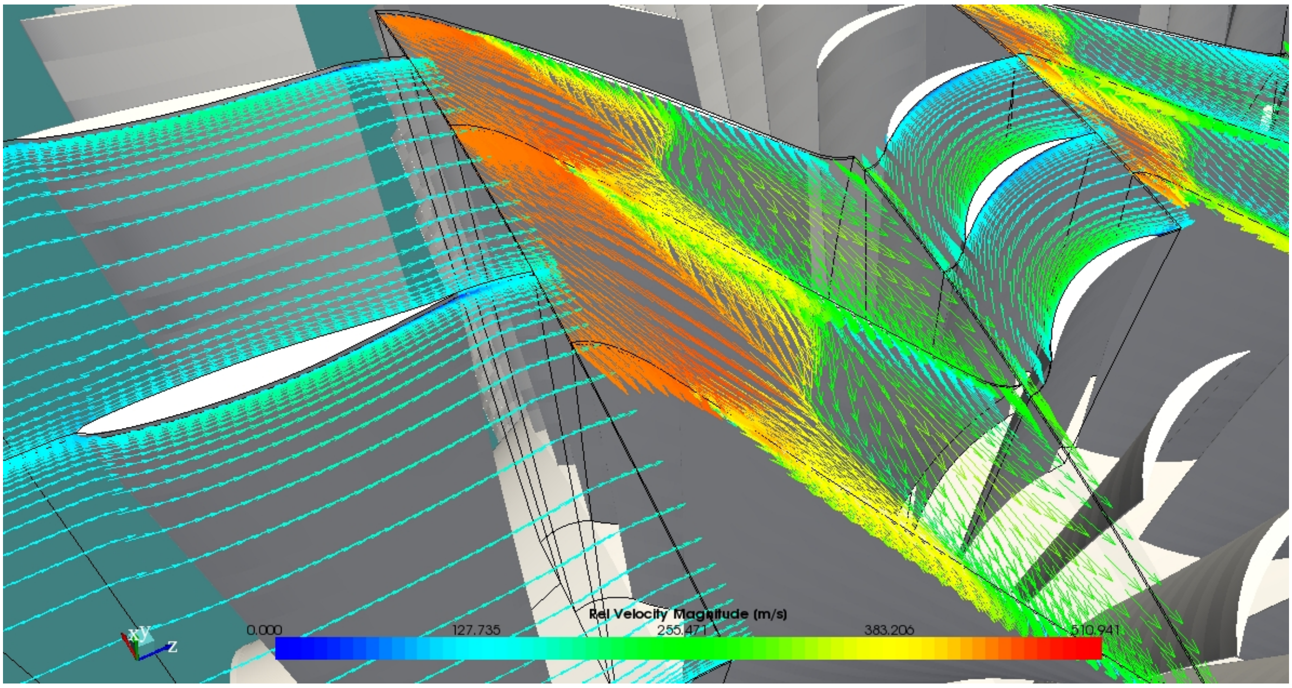

Figure 20. Velocity vectors close to compressor tip.

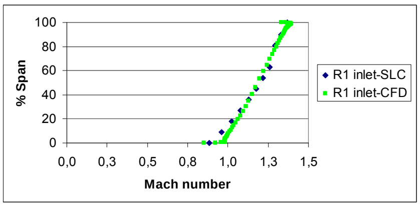

Figure 21. Mach number distribution at the first rotor – SLC vs CFD.

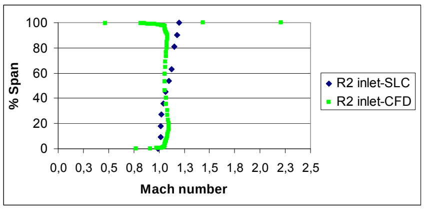

Figure 22. Mach number distribution at the second rotor – SLC vs CFD.

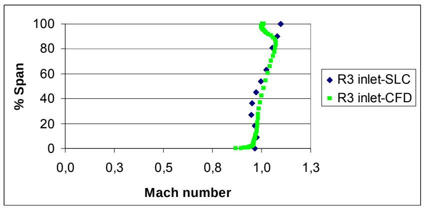

Figure 23. Mach number distribution at the third rotor – SLC vs CFD.

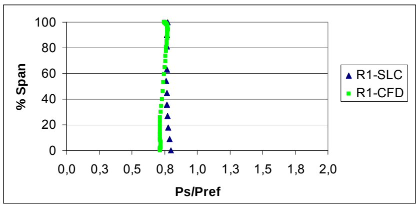

Figure 24. Static pressure distribution at first rotor – SLC vs CFD.

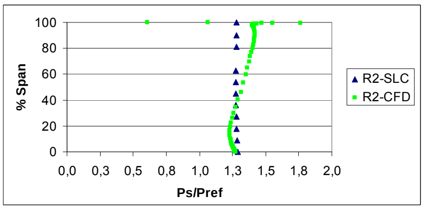

Figure 25. Static pressure distribution at second rotor – SLC vs CFD.

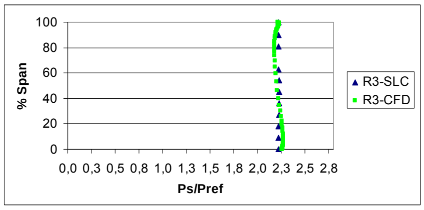

Figure 26. Static pressure distribution at third rotor – SLC vs CFD.

- **Comparison between AXIAL and Pushbutton CFD**

Initial trials were made using previous CFD converged solutions to analyze the influence of the outlet static pressure boundary condition, but no converged solutions were achieved due to numerical instabilities. Converged solutions after circa 2300 iterations were only achieved when initialization was generated based on the SLC calculations. Thus, based on the CFD calculations, the pressure ratio was 4.87 and the compressor efficiency 81.5%.

The reverse flow in the third rotor and last stator rows, around 90% of the blade spanwise is also observed, as can be seen in Figure 27. It may be argued that the design point is near the stall limit.

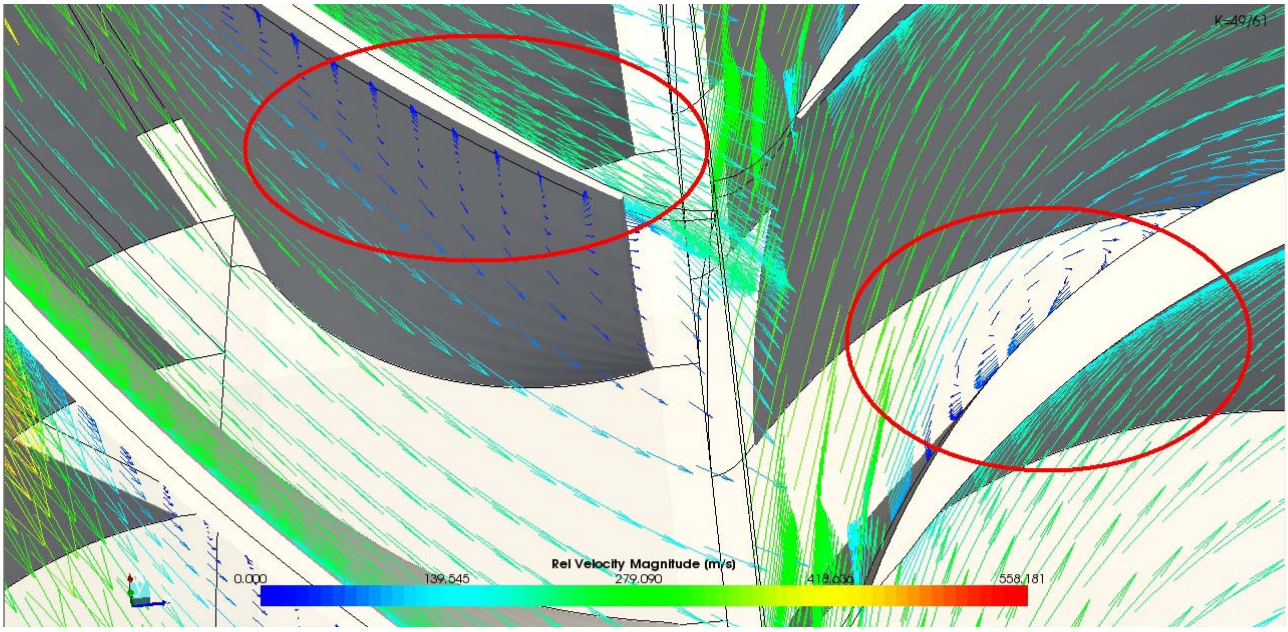

Figure 27. Velocity vectors at the compressor mean diameter.

The spanwise Mach number and static pressure profile from the commercial software can be analyzed and compared using Figures 28 to 30 that show the Mach number distribution at the three rotors leading edges and Figures 31 to 33 that show the static pressure distribution at the same positions. It is possible to say that the main calculated characteristics are in very good agreement with the commercial software.

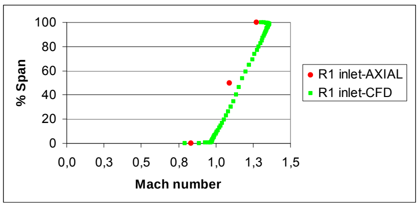

Figure 28. Mach number distribution at the first rotor – AXIAL vs CFD.

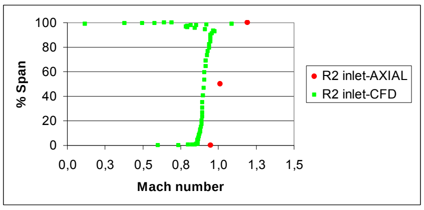

Figure 29. Mach number distribution at the second rotor – AXIAL vs CFD.

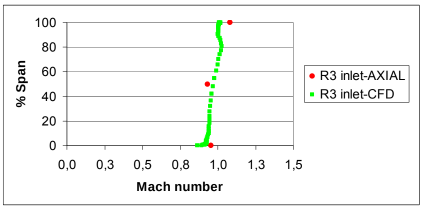

Figure 30. Mach number distribution at the third rotor – AXIAL vs CFD.

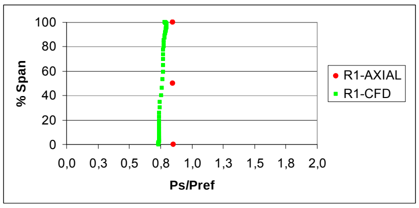

Figure 31. Static pressure distribution at the first rotor – AXIAL vs CFD.

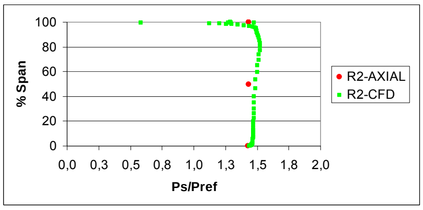

Figure 32. Static pressure distribution at the second rotor – AXIAL vs CFD.

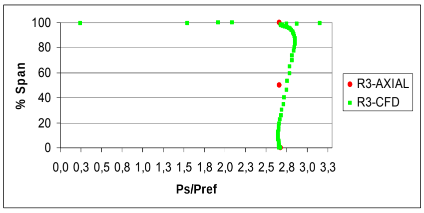

Figure 33. Static pressure distribution at the third rotor – AXIAL vs CFD.

3. CONCLUSIONS

The compressor design tools and their value for the design of high performance axial compressors were described and vastly used in this study. Those programs showed that it is possible to design high performance compressors based on the methodologies implemented in both programs. It was also shown that interactive use of those programs for design and analysis is necessary to arrive at adequate compressor geometry. The technique described in this work will reduce substantially the number of experimental tests, since it makes it possible to start with a high performance compressor prototype.

The CFD results, nevertheless, show that modifications to the blade profile may be necessary to avoid separation in the first rotor. It was also shown that off-design performance can also be predicted, based on the results of increased outlet static pressure.

Good agreement between the calculations of the SLC and AXIAL programs and between those results and the ones from the CFD calculations indicates that a good procedure for the compressor design was defined. It was also shown that the calculation initialization is of much importance and that both design computer programs can be used indifferently. It was also indicated that the design process requires highly skilled people.

4. ACKNOWLEDGMENTS

This research was carried out at the Center for Reference on Gas Turbines at ITA and was only possible because of the support from iTSix Tecnologia and Concepts ETI, Inc.

5. REFERENCES

Barbosa, J.R., 1987, "A Streamline Curvature Computer Programme for Performance Prediction of Axial Flow Compressors", PhD Thesis, Cranfield Institute of Technology, 1987.

Barbosa, J.R., "The aerodyonamic design of a multi-stage, high performance axial flow compressor", 16th Brazilian Congress of Mechanical Engineering COBEM2001, Paper TRB0320, Uberlandia-mg, 26-30 November.

Bringhenti, C., 2003, "Variable Geometry Gas Turbine Performance Analysis", Doctor of Science-DSc - Instituto Tecnológico de Aeronáutica - ITA, São José dos Campos-SP-Brazil, 2003.

Bringhenti, C. and Sousa, F. and Barbosa, J.R. and Tomita, J.T., 2006, "Gas Turbine Performance Simulation Using an Optimized Axial Flow Compressor", ASME Turbo Expo 2006, Proceedings of ASME, GT2006-91225.

Kau, H., 1998, "The Multidisciplinary Design Process. In: RTO AVT LECTURE SERIES", Integrated Multidisciplinary Design of High Pressure Multistage Compression Systems, 1998, p. 1--16, RTO Lecture series, 211.

Lakshiminarayana, B., 1996, "Fluid Dynamics and Heat Transfer of Turbomachinery", John Wiley & Sons, 1996.

Liou, M., 1996, "The A Sequel to AUSM: AUSM^{+}", J. Computational Physics, vol. 129, 1996, pp. 364-382.

Dubitsky, O., 2003, "The Reduced Order Through-Flow Modeling of Axial Turbomachinery", IGTC03-ABS-028.

Silva, F. J. and Barbosa, J.R. and Tomita, J.T., 2007, "Gas Turbines Transient Performance Study for Axial Compressor Operation Characteristics", COBEM 2007, paper COBEM2007-2158.

Sousa, F. and Manzanares, N. and Barbosa, J. R. and Tomita, J. T., 2005, "Design point efficiency optimization of a multi-stage axial-flow compressor for aero application, applying a specially developed computer code", COBEM 2005, paper COBEM2005-1802.

Spalart, P. and Allmaras, S., 1992, "A One-Equation Turbulence Model for Aerodynamic Flows" Aerospace Sciences Meeting & Exhibit, AIAA 1992, p. 1-22, AIAA-92-0439.

Tomita, J.T, 2003a, "Numerical Simulation of Axial Flow Compressors", Master of Science-MSc - Instituto Tecnológico de Aeronáutica - ITA, São José dos Campos-SP-Brazil, 2003.

Tomita, J.T. and Barbosa, J.R., 2003b, "A Model for Numerical Simulation of Variable Stator Axial Flow Compressors", COBEM 2003, paper COB01-0239.

Tomita, J.T. and Bringhenti, C. and Barbosa, J.R., 2003, "Study of the Air Bleed Influence in the Industrial Gas Turbine Performance", COBEM 2005, paper COBEM2005-1344.

Tomita, J.T, 2009, "Three-Dimensional Flow Calculations of Axial Compressors and Turbines Using CFD Techniques", Doctor of Science-DSc - Instituto Tecnológico de Aeronáutica - ITA, São José dos Campos-SP-Brazil, 2009.

Tomita, J.T. and Barbosa, J.R., 2007, "A Study of Tip Clearance influence on Axial Flow Compressor Performance", COBEM 2007, paper COBEM2007-1988.

6. RESPONSIBILITY NOTICE

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