Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 82 (2015) 252 – 257
Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

On the Development of an Efficient Regenerative Compressor

D. Griffini[1], S. Salvadori[1,*], M. Carnevale[2],
A. Cappelletti[1], L. Ottanelli[1], F. Martelli[1]

[1]*Department of Industrial Engineering, University of Florence, via di S. Marta 3, 50139 Florence, Italy*
[2]*Centre of Vibration Engineering, Imperial College of London, SW7 2BX London, UK*

Abstract

Regenerative compressors are attractive machines used in several industrial processes. Their main characteristic is the highly three-dimensional development of the flow. Consequently, usual approach for axial or centrifugal compressors design are not an affordable strategy. The analysis of the rotor/stator coupling is the main issue in the design of regenerative compressors because of the vane-less nature of the stator and the characteristic trajectory of the flow. This paper describes the design of an efficient regenerative compressor based on a highly detailed Reynolds Averaged Navier-Stokes (RANS) analysis. The targets of the activity are defined in terms of expected mass-flow, pressure rise and compressor efficiency, and then a preliminary design is performed using an in-house mono-dimensional tool based on simplified assumptions for the nominal operating conditions. Once the model provided the most promising geometrical characteristics for the target operating point, three-dimensional steady RANS analyses are performed to evaluate the actual performance of the compressor for a wide range of mass-flow values. Special attention has been paid to the generation of the computational mesh and a specific solution for the rotor row has been developed. Compressibility effects are non-negligible since the flow Mach number is higher than 0.5 in several compressor sections, including the leakage zone regions where the losses are higher. The rotor and the full compressor efficiencies are evaluated and discussed to underline the importance of the rotor/volute coupling. The flow behaviour inside of the volute as well as the distribution of losses is also discussed and some guidelines for the efficient design of regenerative compressors are presented.

Peer-review under responsibility of the Scientific Committee of ATI 2015

Keywords: Regenerative Compressors, Efficiency, CFD, Grid Generation

* Corresponding author. Tel.: +39 055 2758779
E-mail address: simone.salvadori@unifi.it

1876-6102
Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.12.030

Nomenclature

C	Absolute velocity [m/s]
U	Peripheral velocity [m/s]
W	Relative velocity [m/s]
Subscripts	
1	Impeller inlet
2	Impeller outlet
r	Radial direction
ref	Reference
t	Tangential direction
Greek	
β	Impeller relative velocity angle [deg]
γ	Stagger angle [deg]
η	Efficiency [-]

1. Introduction

Regenerative Flow pumps and Compressors (RFC) are tangential flow turbomachines characterized by low specific speed, high head and low flow rates. Such characteristic, together with the advantage of not being subject to stall or surge instability, makes them attractive for several areas of application as for example chemical, petroleum and pharmaceutical industries. The most relevant limit consists in the typical low efficiency, which is usually below 50%. Various theoretical models have been proposed in order to estimate their performance and it is possible to group them into two main classes based on their main assumptions. The first model has been presented in the works of Iversen [1] and Baljé [2] and considers the rotor as a "super-rough" surface. The work exchange is therefore a consequence of the viscous and turbulent friction between the rotor and the fluid. This simplification allows to evaluate the performance of the regenerative machines without considering details of their flow behaviour, but its drawbacks have been described by Borsati *et al.* [3]. Wright [4] pointed out the greater tangential velocity of the fluid inside of the blade passage with respect to fluid in the open channel, and highlighted the centrifugal action pushing the fluid outside of the rotor. As a consequence, angular momentum of the fluid is increased across the impeller and a flow circulation is induced. This theory has been experimentally confirmed by Burton [5] and improved by Wilson *et al.* [6], who underlined the relation between the tangential pressure rise and the continuous decrease in the angular momentum of the circulatory flow. The outcome is a helicoidally flow pattern that can be considered as a conventional stage of compression. Based on that principle Sixsmith and Altman [7] and Song [8] developed compressible flow theories for RFCs with aerofoil section blades designed to reduce friction forces thus increasing the efficiency. Present paper deals with design and analysis of RFCs with aerofoil blading. A preliminary design is performed by means of an in-house mono-dimensional tool. Once the geometry of the impeller is obtained for a selected operating point, a numerical analysis of the machine is performed by means of 3D RANS simulations with compressible and incompressible water vapour phase in order to estimate the performance and identify critical components to be improved. The objective of this work is to validate a procedure to be used during the design of RFCs.

2. Preliminary design

A typical RFC with aerofoil blading (shown in Fig.1a from the work of Song [8] and in Fig.1b from the proposed calculation) is composed by an impeller, a core, an inlet and an outlet port, the casing and the stripper. The core is used to drive the fluid in its regenerative path across the flow channel with useful effects both for specific power and losses. The stripper (Fig.1c) is used to divide the high-pressure discharge port to the low-pressure inlet port. The blockage effect is obtained by means of a local reduction of the open channel that acts like a seal. The pressure drop across the stripper clearance results in a leakage mass-flow that affects the machine performance. In this work the target RFC has an annular flow channel with a constant passage area along the tangential direction: as a consequence, an internal (core) and an external (casing) radius are necessary to define its geometry. Impeller blades are generated using symmetrical profiles composed by two circular arcs.

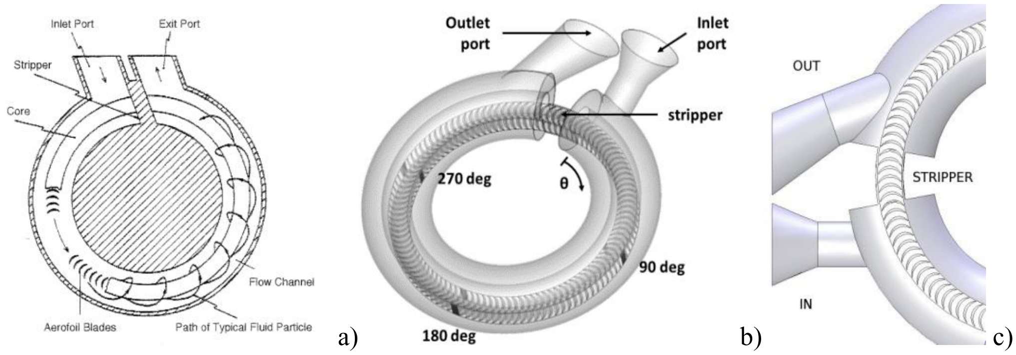


Fig. 1. (a) Axial view of an RFC with airfoil blading [8]; (b) Fluid model of the present RFC; (c) Detail view of the stripper.

The adopted tool for the preliminary design is a mono-dimensional model based on geometrical and physical relations that are representative of the targeted RFC. Present approach mainly refers to the work of Sixsmith and Altman [7]. The objective of the model is to design a RFC for the Best Operating Point (BOP) of the coupled rotor/volute system, without having regard of the stripper and of inlet/outlet sections. The choice of the inlet and outlet parameters is based upon design needs when selecting operating conditions and is strongly related to designers. Mean impeller diameter, rotational speed, blade chord, pitch and height, design pressure rise are typical input parameters. Among the output parameters it is worth mentioning the channel area and mean radius, rotor inlet and outlet relative velocity angle, blade number and compressor mass-flow. The proposed approach is based on steady analyses of incompressible flow. The rotor is considered as an infinite cascade where flow angles are equal to metal angles. The pressure gradient is considered constant along the tangential direction. The operating principle is based on the velocity triangle in Fig 2a. The tangential velocities of the rotor are defined U_1 and U_2 where $U_2 > U_1$ due to the different radial position of inlet and rotor outlet. Moreover it can be defined the mean velocity $U = (U_1+U_2)/2$. The optimal working point is characterized by a purely radial absolute velocity ($C_1 = C_{r1}$) while the rotor provides a tangential velocity to the flow $C_t = 2U$ in each passage. The maximum thickness and the camber angle of the rotor blades have been selected allowing calculating the stagger angle γ in order to match the relative velocity angles previously calculated (Fig. 2b). The flow slows down in the volute during his regenerative path, increasing its pressure at the same time. The tangential pressure gradient along the volute is correlated to the length of the flow streamline projected on a plane normal to the blade motion. Such a characteristic streamline is representative of the flow moving along the helical path on the flow channel. It can be evaluated as a function of relative inlet velocity, density and pressure gradient in the channel, as suggested in [7]. Once the projection length of the streamline is found it is possible to calculate the geometrical dimensions of the flow channel (core and casing diameters). Euler and Bernoulli equations are then used respectively for the impeller flow and the channel flow of each helical path in order to evaluate the expected local and global machine performance. The friction factor used to evaluate pressure losses along the flow channel is evaluated by means of the Darcy formulation based on the mean Reynolds number and the relative roughness. Impeller mass-flow, channel mass-flow and number of helical paths (corresponding to the number of recompression stages) are then calculated for the selected pressure rise and geometrical parameters.

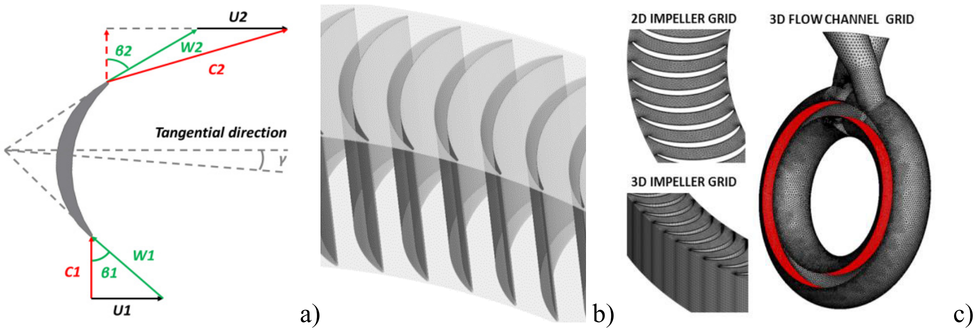


Fig. 2. (a) Velocity triangles at the inlet and outlet of the impeller blade; (b) Impeller geometrical model; (c) RFC grids: 2D impeller, 3D impeller and flow channel.

3. Computational Fluid Dynamics

Once the preliminary design has been accomplished the model has been analysed by means of Computational Fluid Dynamics (CFD). The objective of this part of the work is to evaluate the machine performance and to provide a strategy to perform such kind of analysis. A geometrical model composed by a fluid domain only has been realized from the preliminary design results both for the impeller and the flow channel. The obtained domain has been then discretized with the commercial software Centaur™ obtaining a 6M elements grid (see grids on Fig. 2c). The impeller grid has been realized in 2D and then extruded using an in-house code to match the interface regions highlighted in red in the flow channel grid (Fig. 2c). The 2D grid is composed of 10 layers of prismatic elements along the blades in order to allow for a better boundary layer reconstruction using a *k-ω* SST and tetrahedral cells in the impeller blades passages.

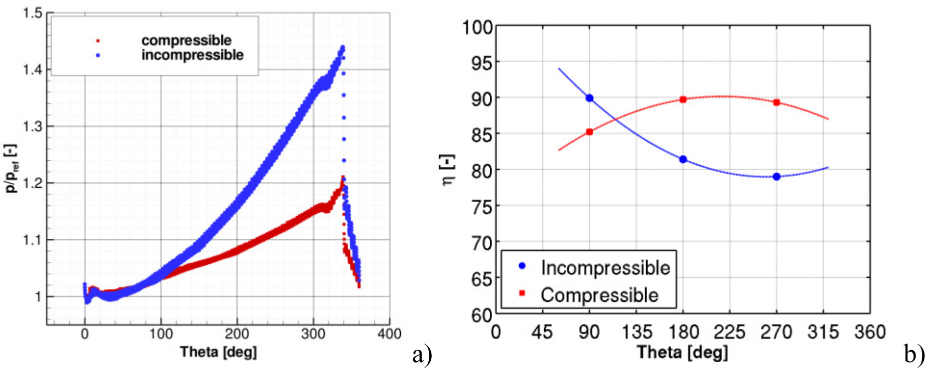


Fig. 3. (a) Tangential pressure rise distribution at mid-section of the impeller for the BOP; (b) Tangential blade efficiency at mid-section of the impeller for the BOP.

3D RANS simulations have been performed with the commercial code ANSYS Fluent® using the frozen rotor approach for the rotor/volute interaction model. Simulations have been performed varying the mass-flow for both compressible and incompressible cases. The working fluid is water vapour of density 0.5977kg/m^3 and has been considered as an ideal gas for the compressible simulations. Results from the 3D compressible and incompressible RANS simulations at BOP are compared in Fig. 3 and Fig. 4. It is necessary to underline that BOP is here referred to the operating point satisfying the prescribed pressure rise and specific power together with an efficiency higher than a minimum reference threshold. The non-dimensional pressure rise along the tangential direction, reported with respect to a reference pressure and evaluated at the mid-section of the impeller are highly sensible to compressibility effect. The reason for such behaviour is shown in Fig.4a and Fig.4b, where pressure iso-surfaces are shown. It is possible to observe how the incompressible case shows in an increased number of helical paths along the tangential direction, resulting in a higher number of compression stages, although with lower blades efficiency (Fig. 3b). It can also be observed that the first tangential sector of the impeller is necessary to reach a helical regime (Fig 3a, Fig.4a and Fig.4b). In fact, at least 70° are necessary before such a regime is reached. The evolution of impeller efficiency (after the recirculating region) is shown for the best operating condition in Fig. 3b considering three stations highlighted in Fig. 1b. It can be observed that blade efficiency increases in the compressible case and the majority of the machine works within 85% to 90%. Inverse trend is shown for the incompressible case working within 77% and 90% for the regime tangential points. Since the analysis of the Mach number has shown Mach peaks of about 0.6 downstream of the impeller (see Fig. 4c) it has been concluded that the different trend of variation is caused by compressibility effects. Therefore, compressible simulations have been used to evaluate the whole machine performance.

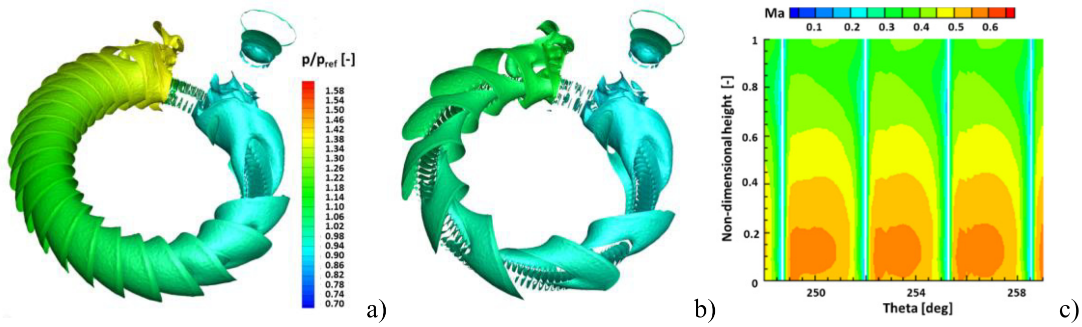


Fig.4. (a) Pressure iso-surfaces of the incompressible 3D RANS simulation at BOP; (b) Pressure iso-surfaces of the compressible 3D RANS simulation at BOP; (c) Mach number at the impeller exit section for the BOP simulation.

Non-dimensional pressure rise and efficiency are reported in Table 1 with respect of inlet and outlet ports and are compared with the data (referred here as "channel") extracted from the sections immediately downstream of the inlet port and upstream of the outlet one, in order to evaluate and separate the effects of these two components.

Table 1. RFC performance

mass-flow [-]	Δ p in/out [-]	Δ p channel [-]	η in/out [-]	η channel [-]	specific power [-]
0.6	3.05	3.12	37.1	47.2	3.00
0.8	2.04	2.18	35.4	51.7	1.67
1.0	1.00	1.56	25.3	49.8	1.00
1.2	0.08	1.11	17.0	35.6	0.62

As it is possible to observe a strong reduction in machine performance is due to the effects of inlet and discharge ports both in terms of pressure rise, with a peak of 93% reduction at the higher mass-flow rate, and global efficiency, ranging from 37% at 0.6 to 17% at 1.2. These results highlight the opportunity for a performance increase optimizing such components. Particularly, flow expansion and a strong separation are pointed out at the inlet region and should be reduced in order to obtain benefits both for pressure rise and efficiency. The non-dimensional target operating point was set to 1.56 in terms of pressure rise at best operating condition; as it is possible to observe such target has been reached if losses connected with inlet and outlet ports are neglected.

4. Concluding remarks

The design procedure of a regenerative flow compressor has been proposed. The most promising geometry is designed using a design tool that put into relation the geometrical parameters to the target best operating point by means of a simplified incompressible approach. Once the model provided the impeller and flow channel geometry a numerical campaign has been performed in order to better evaluate the obtained performance and to find critical components to be improved. Compressibility effects of the working fluid have been evaluated by means of 3D RANS simulations, the results show a non-negligible effect on both pressure rise and efficiency along the impeller. Since the Mach number downstream of the rotor has shown peaks of 0.6 the compressible approach has been used to evaluate global performance. Strong negative effects have been found for the machine inlet section, driving the flow to a high expansion and a massive flow separation bubble in the inlet region thus lowering the performance obtained along the impeller. Moreover, a strategy is necessary to increase the number of helical paths along the vane and in order to allow the fluid to earlier reach a helical regime condition.

References

[1] Iversen HW. Performance of the periphery pumps. *Transactions of ASME* 1955; Vol. 77, pp. 19-22.

[2] Baljé OE. Drag-Turbine Performance. *Transactions of ASME* 1957; Vol. 79, pp. 1291-1302.

[3] Borsati L, Cravero C, Dell'Orco G, Massardo A, Satta A, Schepis G. Rilevamenti sperimentali a confronto con il calcolo delle prestazioni di un compressore rigenerativo. *53° Congresso Nazionale ATI*; Firenze; 1998.

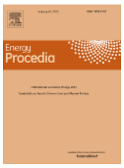
[4] Wright AM. Discussion of reference [1].

[5] Burton JD. The prediction and improvement of regenerative turbo-machine performance. *presented at the British Hydromechanics Research Association*; Cranfield, Bedford, U.K.

[6] Wilson WA, Santalo MA, Oelrich JA. A theory of the fluid-dynamic mechanism of regenerative pumps. *Transactions of ASME* 1955; Vol. 77,pp. 1303-1311.

[7] Sixsmith H, Altman H. A regenerative compressor. *Transaction of ASME* 1977; Ser.B, J. Eng. Ind.,Vol 99, 637.

[8] Song JW Raheel M. Engeda A. A compressible flow theory for regenerative compressors with aerofoil blades. *Proc. Instn Mech. Engrs, Part C: J. Mechanical Engineerign Science* 2003; Vol. 217, 1241-1257.



Biography

Dr Simone Salvadori is research fellow at the Department of Industrial Engineering of the University of Florence. He is currently Assistant Professor of Fluid Dynamics of Turbomachinery at the School of Engineering. He is expert in the field of numerical modelling of unsteady heat transfer in gas turbine components.