AERODYNAMIC STUDIES IN HIGH-SPEED COMPRESSORS DEDICATED TO AERONAUTICAL APPLICATIONS

F.Leboeuf, I.Trébinjac, X.Ottavy*, N.Gourdain**
*** Fluid Mechanics and Acoustic Lab. LMFA (UMR CNRS 5509 ECL, INSAL, UCBL University of Lyon), Ecole Centrale Lyon, Ecully, 69130, France,**
****Computational Fluid Dynamics Team, CERFACS, Toulouse, 31057, France**

Keywords: *High-speed, compressors, unsteady, aerodynamics, rotor-stator*

Abstract

This paper concerns flows that develop in axial and centrifugal high-speed compressors. The availability within the LMFA of two test rigs of strong power for the studies of an axial multistage compressor CREATE and a transonic centrifugal compressor named TM justifies coordinated projects to deeply exploit the results for the scientific and industrial communities. The present paper focuses on the unsteady effects induced by rotor-stator interactions. Two complementary approaches are considered to increase data reliability and investigation capacity. First the flow is computed by the mean of a numerical approach, considering a 3D unsteady RANS flow solver. Then experimental data are used to validate and enhance the database. Results show that a good estimation of the mean flow features is obtained with the numerical model, even if some discrepancies are also observed, especially when regarding the transport of information along the meridional direction on a long distance across three stages. Very good agreement is also obtained for a high-speed centrifugal compressor for both global performance and local detailed time evolution of flow quantities. First results for surge signal are also analysed in detail for the centrifugal compressor.

1 General Introduction

The global trend in the engine design points towards more compactness, higher efficiency and larger operability. An increase in performances of the critical components like the compressor is thus a necessary step. This can be done only if we get a good understanding of the aerodynamic unsteadiness and of the aerodynamic instability growth. For instance, a compact design means indeed a shorter axial distance between blades rows, which implies stronger flow unsteadiness induced by stronger rotor-stator interactions. Also, it is well established that overall performances of a compressor are strongly dependent on the flow behaviour near end-walls [5]. Indeed, a better description of the flow that develops near the casing in a multistage compressor can give valuable information to design high efficiency and more stable compressors. For high-speed centrifugal compressors, the technical constraints (low weight, small size, high performances…) lead to increase the compressor loading and specific speed. Therefore, the blade passage frequency increases and the flow emerging from the impeller is even more distorted because of the loading increase. Consequently the unsteady flow structures play an important role from an energetic point of view [11]. Impeller–diffuser interaction in centrifugal compressors with vaned diffuser plays an important role in the compression process. The vaned diffuser has to tolerate the distorted upstream flow due to jet-wake structure coming from the impeller, whereas the impeller is submitted to the potential effect of the vaned diffuser. Beside the contribution of the impeller-diffuser interaction to the overall performance, this impeller–diffuser interaction may impact the growth of instabilities.

The present paper intends to present recent unsteady aerodynamic studies at LMFA on a three-stage axial flow compressor and a high-loaded centrifugal compressor ([1], [2], [3], [9], [12], [13], [15]). We aim at presenting the importance of unsteady phenomena from the point of view of global performances. Information on the local flow behaviour is also given, with respect to the unsteady effects and the possible instability growth.

Today two approaches are mainly available to obtain valuable data for studying unsteady effects in compressors. In one hand, the numerical simulation allows detailed investigation possibilities for a reasonable cost. But the main drawback of this approach is still its poor predictive capacity for very complex flows, especially in a multistage environment. In the other hand, experimental investigations are very useful to validate numerical models and provide reliable data, but this approach induces a higher cost than numerical methods and is not well suited to study aerodynamic instabilities in a high-pressure compressor. Indeed, the results in this paper combine numerical simulation and experiments.

2 Description of the experimental facilities

2.1 Instrumentation

The experimental results presented in this paper have been obtained on the tests rigs of 2 MW and 1 MW dedicated to the exploration of the flow in high-speed compressors. The 2MW test rig (Fig.1) accommodates an axial multistage compressor named CREATE (Research Compressor for the Study of the Aerodynamic and Technological Effects) while the 1MW test rig is used for a transonic centrifugal compressor named TM.

These test rigs are instrumented for the global performances or monitoring: 236 and 67 measurements (pressure, temperature, accelerometers…) are acquired for the 2 MW and the 1 MW test rigs respectively. The specific measurements realized on these rigs are of steady type (directional pressure probes with 4 taps coupled with a temperature sensor) and of unsteady type (laser anemometry, high frequency response pressure sensors, strain gages) performed in synchronization with the rotation of the machine.

Concerning the laser measurements, backscatter 2D Laser Doppler Anemometer (LDA), designed and built by Dantec, is used. With the triggering signal, the flow field can be described either inside a single blade passage, or within several blade passages covering the circumferential periodicity of the whole machine. The data reduction process filters the random time scales of the turbulent flow. Thus, the unsteadiness captured only relates to phenomena clocked with the rotor passing frequency.

Besides, to locate precisely the LDA measurement volume in each compressor, we use a 6 axes robot for CREATE compressor and a 3 axes robot for the centrifugal compressor. This system prevents the optical assembly of the anemometer being sensitive to machine vibrations. Due to the axial thrust and thermal dilatation of the machine, a procedure has been adopted to position accurately the LDA control volume during compressor operation. The uncertainty of the location of any measurement point is then estimated to ±0.15 mm.

The compressors are seeded with a polydisperse aerosol of paraffin oil. The size of the particles at the outlet of the seeding generator is measured and its mean value is smaller than 1μm. Seeding is performed upstream of the settling chamber. In such a flow configuration [11] proved that this technique was reliable.

The spatial and temporal discretizations for these measurements are chosen in agreement with the methodology applied in [13] to minimize the interpolation errors in rotor/stator interaction analysis.

Once all the quantifiable uncertainties have been taken into account, the velocity component measurement error is about ±1.5 %. The velocity angles presented in this paper are calculated from the velocity components.

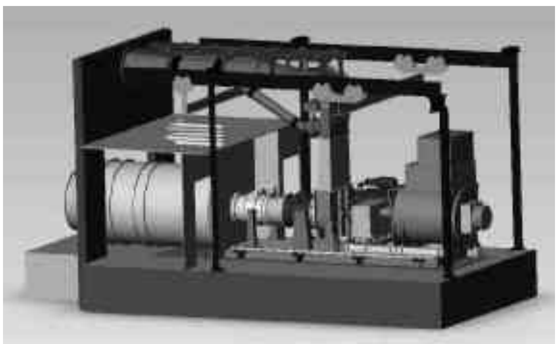

Fig.1: 2 MW Test Rig at LMFA

More specifically for measurements dedicated to the unsteady pressure, the axial compressor CREATE is equipped with 36 fast pressure piezo-resistive sensors permanently located in inter-rows sections. A dozen of sensors can be also located on the casing of rotors. The centrifugal compressor TM is equipped with 15 fast pressure sensors on the casing at the exit of the centrifugal rotor and in diffuser passage. These measurements are obtained by means of an analogical-digital conversion system with 48 entries, coupled with in-house developed conditioners that are used to amplify and to filter the pressure signal with the aim of acquiring some data up to 150 kHz, the sampling frequency being 500 kHz [3].

2.2 The axial flow compressor CREATE

The project involves detailed experimental and numerical studies performed within the framework of cooperation between LMFA, SNECMA and CERFACS.

The CREATE compressor allows to realize researches within 3.5 stages of a high-pressure axial flow compressor representative of median-rear blocks of modern turbojet engine.

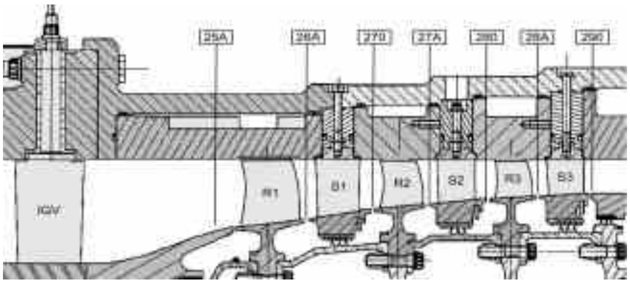

Fig.2: Meridional View of the CREATE Compressor

The number of stages is chosen in order to have a magnitude of the three-dimensional flow effects similar to a real compressor, and to be within the rig torque power limitation. In order to perform probe traverses between blade and vane rows, the axial gap is slightly increased compared to current compressors and an outer-case moving-rings technology is implemented to perform probe measurements in the circumferential direction at constant radius location. The circumferential periodicity of the whole machine (obviously 2π in general case with primary blade numbers) is reduced to 2π/16 on the CREATE compressor, choosing the number of blades of each rotor and stator (Inlet Guide Vane -IGV- included) as a multiple of 16 (see Table1). Consequently, measurements carried out over a sector of only 2π/16 (namely 22.5 degrees) contain all the spatial information (in the case of stabilized operating points) and are very useful when devoted to detailed studies, such as rotor-stator interaction analysis. The compressor and the inter-row measurement sections are presented in Fig.2.

The design speed of the compressor is 11,543 rpm. At this rotational speed, the Mach number at the tip of the first stage is 0.92. Indeed the flow is slightly transonic in the first stage and fully subsonic in the two last ones.

Row	IGV	R1	S1	R2	S2	R3	S3
Number of blades per row (for 2π)	32	64	96	80	112	80	128

Table 1: Number of Blades of the CREATE Rows

2.2 The centrifugal compressor TM

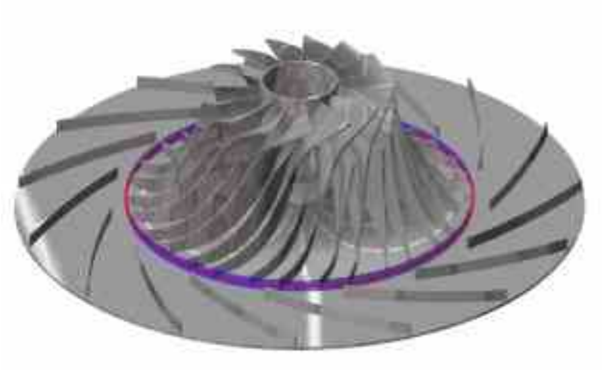
Fig.3: 1 3D View of the TM Compressor

The centrifugal compressor stage TM was designed and built by Turbomeca. It is composed of a backswept splittered unshrouded impeller (with 2 N_R blades) and a vaned diffuser (composed of N_S vanes). A 3D sketch of the stage is given in Fig 3.

The measurements and calculations are performed at a rotation speed Ω_R=0.927.Ω_{nom}.

At that speed, the rotor runs with subsonic inlet conditions all over the span whereas the absolute Mach number at the vaned diffuser inlet is supersonic.

The aerodynamic studies in the centrifugal compressor TM of high pressure rate (~ 9) are the subject, since about fifteen years, of experimental and numerical studies within the framework of a cooperation LMFA-Turbomeca / ONERA. The works realized aimed at understanding:
a) the three-dimensional flow structures,
b) the unsteady effects of mechanisms linked to the rotor-stator interactions,
c) the consequences of the unsteadiness on the flow structure in the bladed diffuser,
d) the mechanisms at the origin of the severe instabilities such the surge.

3 CFD tool

The flow solver used is the *elsA* software that solves the RANS equations using a cell centred approach on multi-block structured meshes [4]. To ensure a good precision, convective fluxes are computed with a third order Roe scheme considering a minimal Harten entropic correction [14] and diffusive fluxes are calculated with a second order centred scheme. A second order Dual Time Stepping (DTS) method is applied for the time integration [10]. To reach a converged state, the number of sub-iterations for the inner loop is defined to obtain at least two orders of reduction for the residuals magnitude.

Three sets of simulations are performed. For both compressors steady simulations are first performed using a mixing plane method for the flow communication between the two adjacent blade rows. In this inter-blade plane, a circumferential average is applied to the conservative flow quantities but preserving radial variations. Then unsteady simulations are a bit different for the multistage and the centrifugal compressors. Classical unsteady RANS calculations are performed over a reduced domain for the compressor CREATE, whereas a phase-lagged approach is used for the simulation of the single centrifugal stage.

In the case of the multistage compressor CREATE, unsteady RANS calculations are carried out over reduced spatial and temporal periodicities equalled to $2\pi/16$ and $2\pi/16\Omega$ respectively, using the common multiple 16 in the blade number of the rows and considering 3200 iterations to discretize one rotation ($2\pi/16$) of the compressor at the design speed. The turbulent viscosity is computed with the two equations k-omega formulation [16] and the flow is assumed to be fully turbulent since the Reynolds number based on the chord is around 10^6.

Considering the simulation in the CREATE compressor, for practical reasons of computation time, the whole experimental domain cannot be simulated. First the IGV is not represented but is taken into account using experimental data as inflow boundary conditions, as described in [9]. Second, it is assumed that a good description of the deterministic stresses is the most important parameter to compute correctly the flow at stable conditions, meaning the duct length has no impact on the compressor performances and stability [8]. Indeed only a part of the inlet and outlet ducts is modelled but a sufficient distance is considered between the boundary conditions and the blade rows to avoid numerical reflection problems (more than one rotor chord upstream and two rotor chords downstream). The natural periodicity of the compressor is used to consider only a $2\pi/16$ sector periodicity (22.5 degrees), representing all the rotor-stator interaction effects. Of course, the main limitation is that no tangential wavelength greater than a sixteenth of the circumference can develop. Indeed, the numerical model is not able to compute a realistic unstable phenomenon such as rotating stall or surge. Finally, the secondary flow induced by the hub gap between rotating and non-rotating parts is not taken into account, mainly because blown and injected mass flows are not yet correctly estimated at these locations.

The flow domain is discretized with a multi-block approach, using an O-H meshing strategy for each passage of the compressor. The typical dimensions of a blade passage mesh are 85, 33 and 57 points, respectively in the axial, tangential and radial directions. An O-H mesh with 13 points in the radial direction is used to discretize the radial tip gap. To obtain a good balance between computational cost and precision, a wall law approach is applied [7] with a fixed wall cell size that corresponds to a mean normalized wall distance y^+ of 20. This meshing strategy leads to a total nodes number of 8.4 millions to represent the three stages. A standard condition of spatial periodicity is considered for the lateral boundaries and a sliding mesh condition with non-matching points is applied at the rotor-stator interface [6]. The main advantage of this method is to be conservative in the case of plane interfaces (which is roughly the case here). To model the outlet duct, a throttle condition is coupled with a simplified radial equilibrium. Then the characteristic of the compressor is described from the choked point to the stall inception point, simply by increasing the value of the throttle parameter. Finally, all these unsteady flow calculations have been performed with four processors of a NEC-SX8 supercomputer. The physical time needed to reach a periodic state evolves from 100h at nominal operating conditions (corresponding to one rotation of the compressor) to 400h at near stall conditions. Up to four rotations are simulated to estimate the stability of an operating point. The computed point is assumed to be stable only if a periodic state is reached at the end of this time.

In the unsteady RANS simulation of the TM centrifugal compressor, the phase-lagged approach is used. In this approach, the computation domain is limited to a single blade passage for each row. At a stable operating point and assuming uniform inlet conditions, the unsteady effects are only due to the impeller-diffuser interaction. Then, the flow is time-periodic in the frame of reference of the rows, $T_S=2\pi/\Omega_R N_R$ being the period in the diffuser frame and $T_R=2\pi/\Omega_R N_S$ being the period in the impeller frame (N_S and N_R are respectively the number of stator and rotor blades). As a consequence of the time-periodicity in each frame, a phase-lag exists between two adjacent blade passages. For each row, this phase-lag is the time taken by a blade of the next row to cover the pitch of the row, modulo the time-period of the row. Basically, the phase-lagged technique consists in storing the flow values on the periodic boundaries and on the impeller-diffuser interface boundaries in order to deal with the phase-lag existing between adjacent blade passages. In order to manage the stored data, the constant time-step used by the solver, Δt is conveniently defined such as $T_S = N_S N_q \Delta t$ and $T_R = N_R N_q \Delta t$, with N_q an integer intended to satisfy the CFL stability criterion. When N_S and N_R are prime numbers, the number of iterations to describe a thorough revolution of the impeller is then equal to $N_S N_R N_q$.

4 Unsteady effects in the 3-stage axial flow compressor CREATE

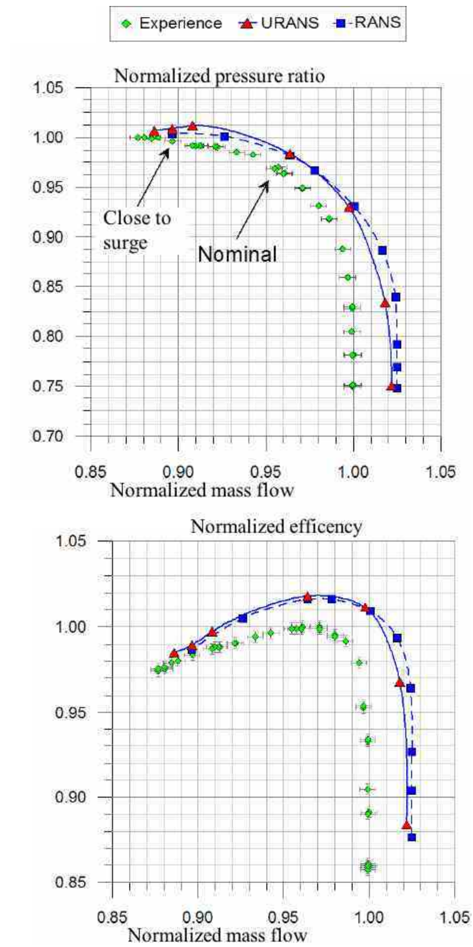

Fig.4: Performance map of the CREATE compressor

The overall compressor map of the version under investigation of the CREATE compressor is presented in Fig. 4. The predicted performances are slightly better than the measured ones. It is of interest to observe that the unsteady effects do not modify strongly the overall performances, when comparing unsteady RANS and steady RANS results. One explanation of this behaviour can be given when considering the local flow behaviour.

a) Downstream rotor 1

b) Downstream rotor 2

c) Downstream rotor 3

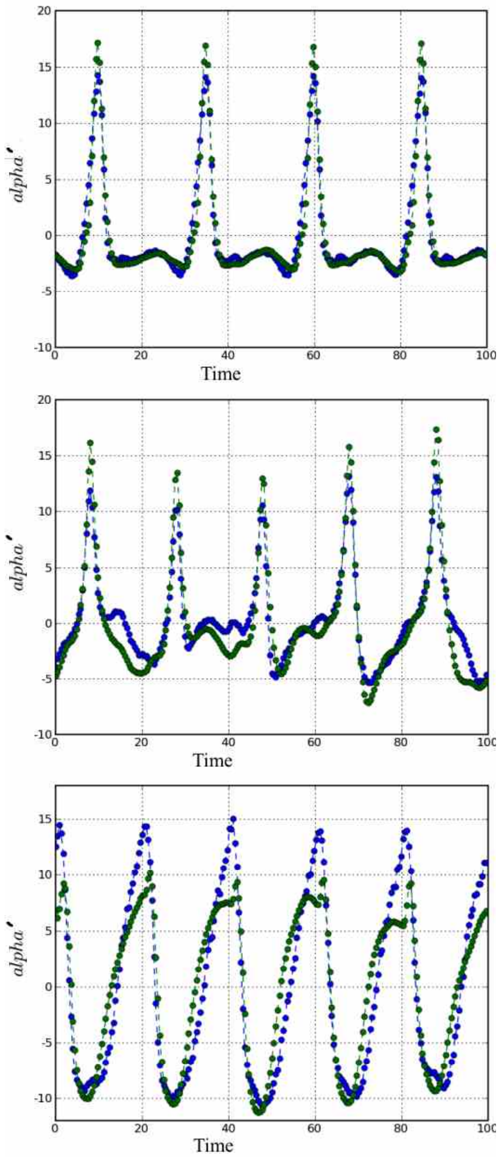

Fig.5: Temporal Fluctuations of the Absolute Velocity Angle for a Fixed Circumferential Position (CREATE Compressor, at 83% blade height, blue = measurements, green = unsteady RANS)

Fig 5 presents the time evolutions of the flow velocity angle fluctuations α' with respect to the meridional direction, downstream of the three rotors at 83% of the blade span; this is a challenging location for the numerical flow solver since strong interactions occur at this place between upstream wakes, potential effects and the tip leakage flow. We observe that the local angle evolutions, generated here by the upstream wakes mainly, are correctly reproduced for all measuring planes. However, if the agreement is excellent in section 26A downstream of rotor 1 (Fig. 5a) (see Fig.2 for locations of the inter-row sections), the agreement deteriorates along the flow path particularly in section 28A downstream of rotor 3 (Fig. 5c).

Experimental observations in section 28A with respect to the section 26A show a mean increase of 60% in the flow unsteadiness, while numerical simulation predicts no or only a small increase. As in the section 26A, the peak of the flow angle is associated to a deficit of the relative velocity in the rotor wakes, but the relative velocity deficit is stronger in section 28A because it combines the wakes and the tip gap flow of the rotor 3. In the experimental case, the tip vortex flow and the wakes are clocking in a way that they superimpose their influences. That leads to higher levels of fluctuations and slightly increases the thickness of the zone associated to the wakes. In the CFD case, the convective transport suffers from a numerical model too much dissipative, having for consequence to smooth the wakes too quickly in the downstream stages. So the rotor / stator interactions, concerning here the interaction between the wakes of the stator 2 and the tip gap flow of rotor 3, are underestimated and lead to a wrong prediction of the trajectory of the tip gap flow. As a consequence the merge of the tip gap flow with the wakes is a bit miss-predicted. The numerical dissipation induced also that the interactions are largely under-estimated when they do not concern neighbouring wheels. This constitutes an important axis of improvement for the flow simulation in multistage machines with significant axial distances. It is then possible than the underestimation of the time evolution of the wakes downstream of rotor 3 are partly responsible of the optimist unsteady RANS prediction of the performance map shown in Fig.4.

Despite the too big numerical dissipation observed over a long distance in the unsteady RANS calculation, it gives however a correct description of the deterministic flow unsteadiness generated by neighbouring rows. This is a strong improvement in comparison with the steady calculations using mixing plan between rotor and stator. The important time variations of the flow angle in the rotor wakes (Fig.5), lead to strong fluctuations in the incidence angle on downstream blades. These temporal angle fluctuations modify then important specific flow structures as in the tip clearance gap, itself suspected of being at the origin of the instabilities for such a compressor.

5 Unsteady effects in the centrifugal compressor TM

The overall pressure ratio for the TM compressor is presented in Fig.6. While, there is large difference between the steady numerical results and the experimental results, we have a very good prediction with the unsteady simulation. In this TM compressor, we have then a strong unsteady effect induced by the interaction between the rotor and the bladed diffuser. The following analysis is performed at the nominal point of operation.

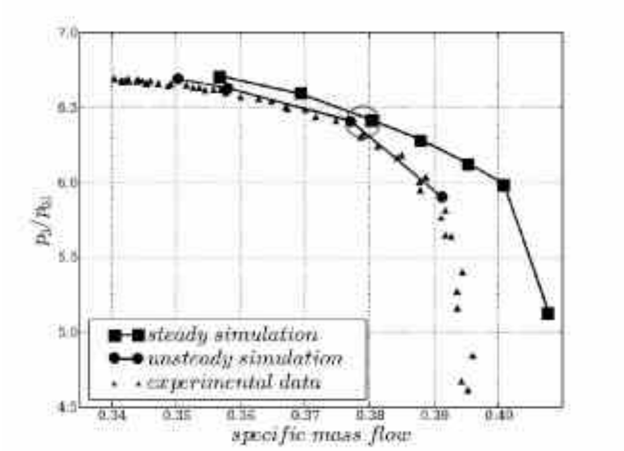

Fig.6: Performance Map of the TM Compressor

We were able to qualify and to quantify the sources of unsteadiness, to explain the differences observed between the results of the steady numerical simulations and the unsteady ones and to propose a model of correction for steady simulation. The obtained results are original with regard to the state of the knowledge because the studied configuration is supersonic, what is inherent to the new geometries of compact machines. Whereas it is often known that the unsteady part of the rotor-stator interaction can have beneficial effects on the global performances of the subsonic compressors, we explain the degradation observed in a supersonic centrifugal configuration by the interaction of the jet - wake structure (known in centrifugal compressor) with the shock wave detached from the leading edge of the diffuser blades (Fig.7).

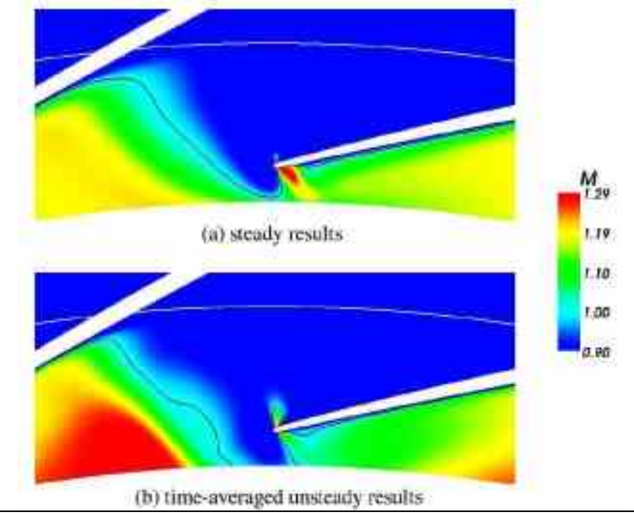

Fig.7: Map of Absolute Mach Number for the TM Compressor in the Rotor-Stator Interaction Region

For steady flow Fig.7a, the shock wave generated in the neighbourhood of the diffuser leading edge is constrained into the stator domain, and its influence is not transferred into the rotor domain. On the contrary for the unsteady simulation (Fig. 7b), the shock wave is felt into the rotor domain and strongly interacts with the jet-wake structure that develops in the rotor.

The flow interaction between the rotor with the shock wave detached from the leading edge of the diffuser blades leads to the generation of pressure waves, which propagate in the bladed diffuser. This is observed in Fig.8.

This figure presents, superimposed on a map of static pressure gradient, two types of waves: the α+ wave which propagates downstream; their interaction with the boundary layers on the diffuser vane surfaces generates alternately opposite and favourable pressure gradients, which drive the boundary layers to periodic separation. The α- waves propagate upstream up to the shock wave. Convected by the flow, the wake dives towards the hub whereas the jet migrates to the casing. The combined effects of the pressure waves and of the wake migration lead to an accumulation of weak energy fluid in the corner formed by the hub and the pressure side of the diffuser channel. This zone becomes then an area of an aerodynamic blockage, which can lead to the surge when the aerodynamic blade load increases.

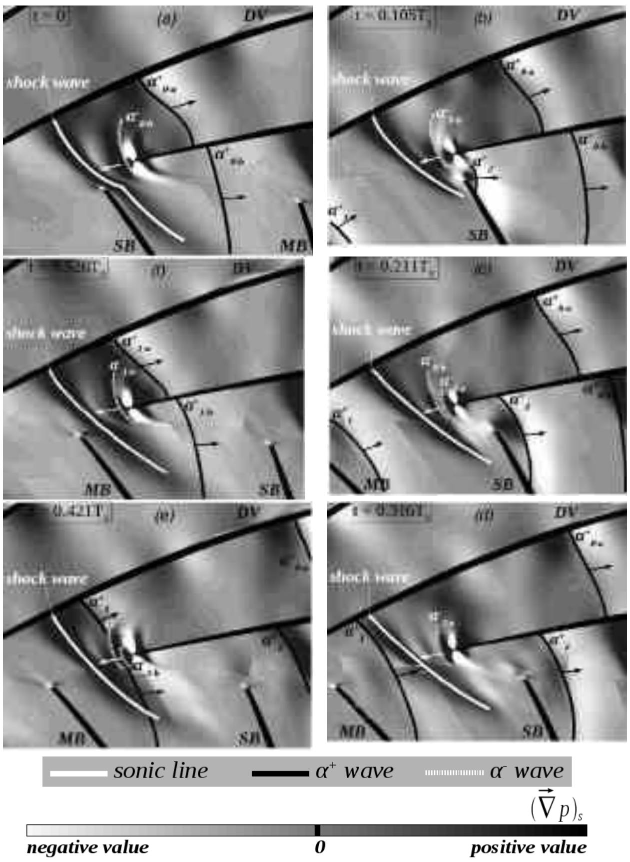

Fig. 8: Generation of computed Pressure Waves at Six Time Steps for the Compressor TM

For three locations in the diffuser passage (Fig. 10), we present in Fig.9 a comparison between static pressure traces measured and computed with the unsteady RANS simulation. The agreement is very good for the three traces both in amplitude and frequencies, although the experimental seem to have a wider spectrum.

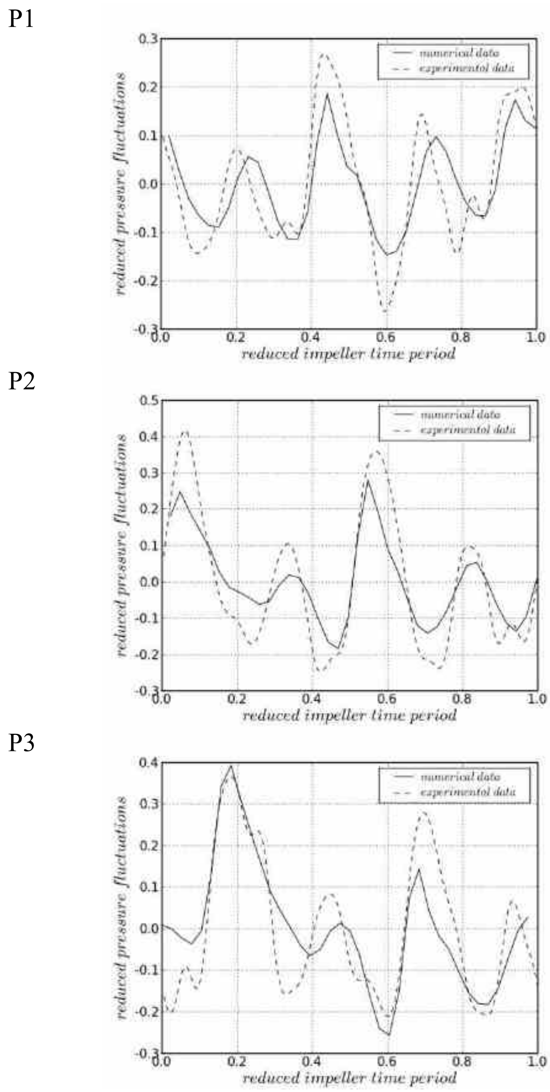

Fig. 9: Fluctuations of Reduced Pressure over a Rotor Time Period at Three Sensor Locations within the Diffuser Vane Passage.

Raw pressure measurements when entering into surge are low pass filtered with $f_{1/3}$ = 145 kHz and are presented in Fig.11. This pressure signal is obtained with the pressure sensor located in the vaneless diffuser and slightly ahead of the diffuser blade leading edge. It is normalized by the mean temporal value obtained during the stabilized operating point just before the surge. The x-axis represents the number of impeller revolutions, which is limited from a zero value arbitrarily chosen as 1400 in order to see the first surge cycle with its duration of about 850 revolutions. In this figure, as in the following ones, the white curves are for the averages over one rotor revolution. During the first rotor revolutions in the surge, the level of the pressure spikes can reach about four times the mean value before entering into surge. Note that at this operating point the pressure ratio is 6.7, as plotted in Fig. 6. These measurements highlight the damaging effect that the surge could lead regarding the instrumentation and the machine.

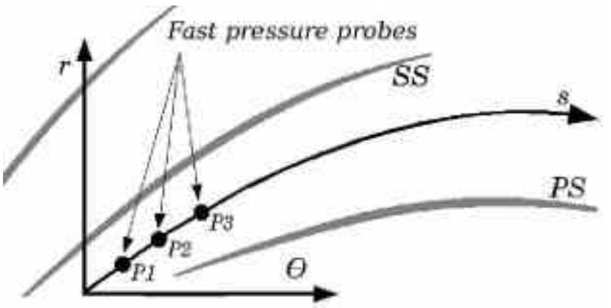

Fig. 10: Location of Fast Pressure Sensors in the Diffuser Passage

In Fig. 11, the link between the massive pressure fluctuations and the unsteady impeller-diffuser interactions is highlighted. Here the pressure signals have been recorded by the sensor during 6 different entrances into surge and superimposed on the same figure with the same time reference. The three plots have an abscissa, which represents the time expressed as a number of rotor blade (main or splitter) pitches. The y-axis is the same for the 3 plots and is the non-dimensional pressure, as defined previously. In the x-axis-zoom-plot at the bottom of the figure, one can note the extremely good repetition of the pressure signals, entirely triggered by the rotor blade passages. These results show how suddenly (in less than 20 main blade passages) and with the same process the spike-type surge occurs.

6 Conclusion

The local flow behaviour in high-speed compressor is strongly dependant on unsteadiness. We have shown that the coupled use of detailed flow measurement in a realistic machine with a fully unsteady RANS simulation can provide a better understanding of the flow physics.

The use of unsteady RANS simulation provides an improvement in the prediction of the global performances of a compressor compared to the steady flow simulation. This is particularly true for the high-speed centrifugal machine for which the interaction between the impeller exit and the vaned diffuser has been underlined. We underline the fact that the shock deformation and the backward and forward waves are responsible on the strong increase in the local losses.

The global impact of unsteadiness on the performance of the three-stage axial flow compressor CREATE is smaller than in the centrifugal compressor TM case. Although strong local effects exist in CREATE, they have only weak influence on the global performance, because they counterbalance each other.

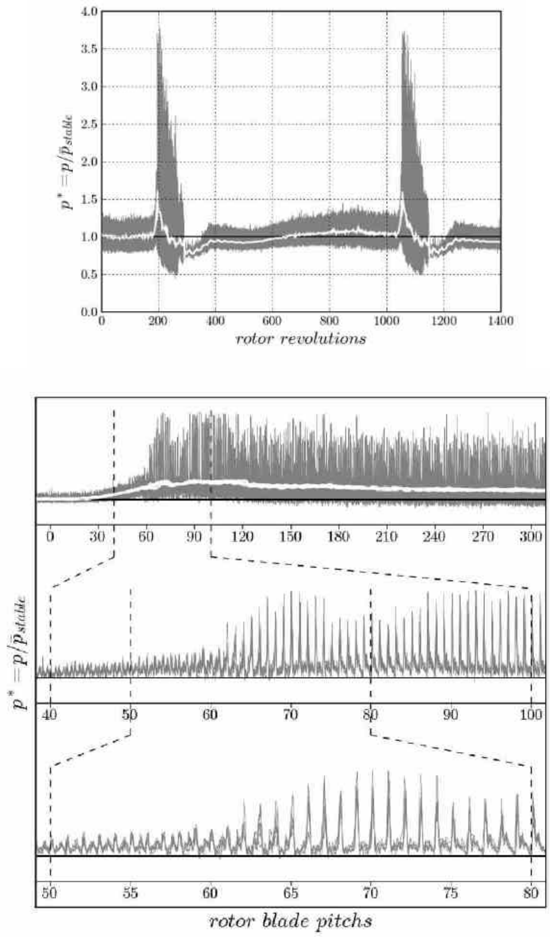

Fig. 11: Onset of surge instabilities – pressure signals of the B10 sensor, superimposed for 6 different entrances into surge (Top: global view; bottom: progressive enlarged views with time)

The quality of the unsteady RANS prediction deteriorates from stage to stage, owing to an excessive numerical dissipation that tends to minimize the unsteady impacts.

Finally, development of instabilities as surge may be extremely brutal in the centrifugal machine, a fact that needs further studies in order to expect to control their growth and then extend the surge margin.

References

[1] Bulot N and Trebinjac I. Effect of the unsteadiness on the diffuser flow in a transonic centrifugal compressor stage, *International Journal of Rotating Machinery*, vol. 2009, Article ID 932593, 11 pages, 2009.

[2] Bulot N, Trebinjac I, Ottavy O, Kulisa P, Halter G, Paoletti B and Krikorian P. Experimental and numerical investigation of the flow field in a high-pressure centrifugal compressor impeller near surge, *IMechE Journal of Power and Energy*, Volume 223, No. 6, pp.657-666, 2009.

[3] Bulot N, Ottavy X and Trebinjac I. Unsteady pressure measurements in a high-speed centrifugal compressor, *Journal of Thermal Science*, Science Press, Vol.19, No.1, pp.1-8, 2010.

[4] Cambier L, and Veuillot J P. Status of the elsA CFD software for flow simulation and multidisciplinary applications, *46th AIAA Aerospace Science Meeting and Exhibit, AIAA* 2008-664, Reno, USA, 2008.

[5] Domercq O, and Escuret J F. Tip clearance effect on high-pressure compressor stage matching, *J. of Power and Energy*, Vol. 221, pp. 759-767, 2007.

[6] Fillola G, Le Pape M C, and Montagnac M. Numerical simulations around wing control surfaces, *24th international congress of the aeronautical sciences ICAS*, Yokohama, Japan, 2004.

[7] Goncalves E, and Houdeville R. Reassessment of the wall functions approach for RANS computations, *Aerospace Science and Technology*, Vol. 5, pp. 1-14, 2005.

[8] Gourdain N, Burguburu S and Leboeuf F. Rotating stall simulation and analysis in an axial compressor, *17th International Symposium on Air Breathing Engine, ISABE* paper 2005-1138, Munich, Germany, 2005.

[9] Gourdain N, Ottavy X and Vouillarmet A. Experimental and numerical investigation of unsteady flows in a high speed three stages compressor, *Proceedings of the 8th European Conference on Turbomachinery*, Graz, Austria, p. 247-266, 2009.

[10] Jameson A. Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings, *AIAA Journal*, AIAA-91-1596, 1991.

[11] Krain H and Hah C. Numerical and experimental investigation of the unsteady flow field in a transonic centrifugal compressor. *Proceedings of the International Gas Turbine Congress, IGTC*, Tokyo, Japan, 2003.

[12] Ottavy X, Trébinjac I and Vouillarmet A. Analysis of the inter-row flow field within a transonic axial compressor: part 2 - unsteady flow analysis, *45th International Gas Turbine & Aeroengine Technical Congress*, ASME Paper 2000-GT-0497, *Journal of Turbomachinery* Vol. 123 No. 1, pp.57-63., Münich, Jan. 2001.

[13] Ottavy X, Trébinjac I, Vouillarmet A and Arnaud D. Laser measurements in high speed compressors for rotor-stator interaction analysis, *Proceedings of ISAIF 6th*, Shanghaï, April 2003, *International Journal of Thermal and Fluid Sciences*, Nov. 2003.

[14] Roe P L. Approximate riemann solvers, parameter vectors and difference schemes, *J.of Computational Physics*, Vol. 43, pp. 357-372, 1981.

[15] Trebinjac I, Kulisa P, Bulot N and Rochuon N. Effect of the unsteadiness on the performance of a transonic centrifugal compressor stage, *J. Turbomach*inery, Volume 131, Issue 4, 041011 (9 pages), October 2009.

[16] Wilcox D C. Reassessment of the scale-determining equation for advanced turbulence models, *AIAA Journal*, Vol. 26, pp. 1299-1310, 1988.

Acknowledgments

The authors wish to thanks SNECMA, TURBOMECA and CERFACS for allowing the publications of these results.

Contact Authors Email Address

francis.leboeuf@ec-lyon.fr
isabelle.trebinjac@ec-lyon.fr
xavier.ottavy@ec-lyon.fr
nicolas.gourdain@cerfacs.fr

Copyright Statement