NUMERICAL SIMULATIONS OF TECHNOLOGICAL EFFECTS ENCOUNTERED ON TURBOMACHINERY CONFIGURATIONS WITH THE CHIMERA TECHNIQUE

L.Castillon*, G. Billonnet*, S. Péron, C. Benoit****

***ONERA , Applied Aerodynamics Department,**
8 rue des Vertugadins, 92190 Meudon - France
****ONERA, CFD and Aeroacoustics Department,**
29 avenue de la Division Leclerc, 92320 Châtillon - France

Keywords: *CFD, turbomachinery, Chimera*

Abstract

This paper presents an overview of numerical simulations performed at ONERA on turbomachinery configurations which include technological effects, such as tip clearance, hub disk leakage, casing treatments, blade fillets and cooling holes. An overset grid approach (Chimera technique) is used to simulate these geometrical effects with ONERA's structured CFD solver elsA. Calculations performed on the different configurations enable to quantify the impact of these technological effects on the flow solution.

Nomenclature

CFD	Computational Fluid Dynamics
Pt	Absolute total pressure
Tt	Absolute total temperature
Ttr	Relative total temperature
Pio	Absolute far stream total pressure
Tio	Absolute far stream total temperature
$m_{leakage}$	Leakage mass-flow
m_{tot}	Total mass-flow
M	Mach number
η_{is}	Isentropic efficiency

1 Introduction

With the increase of computational resources available for numerical simulations, CFD is commonly used today for aeroengine design. Yet many efforts are still to be achieved to improve the results which suffer from simplifications made in: 1) flow modeling, 2) boundary treatment, 3) inaccurate modeling of the geometry. If a considerable amount of work has been devoted in the past to flow modeling (turbulence and transition modeling) or boundary treatment (development of specific boundary conditions for turbomachinery configurations), significant efforts remain to be achieved on the geometrical modeling. The present paper deals with this third point. Indeed, aeroengine designers wish to simulate more and more accurately the very complex geometries encountered on industrial turbomachines by integrating in the design process geometrical components such as casing treatments, grooves, blade slots, cooling holes or gaps separating fixed and rotating walls. Figures 1 and 2 represent an illustration of such geometrical components encountered on industrial compressor and turbine stages [1,2].

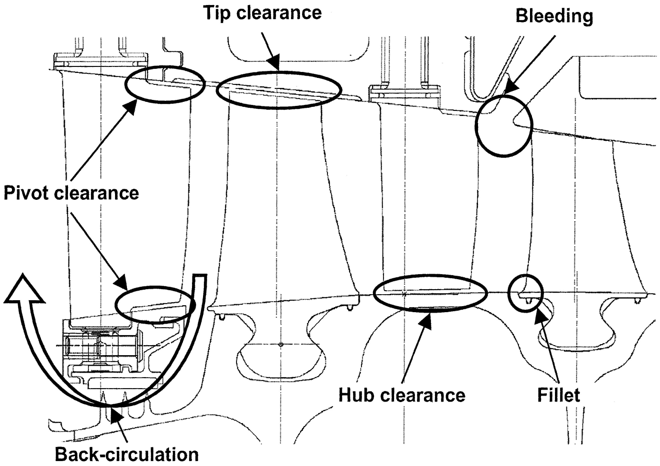

Fig . 1: Meridian view of a compressor stage with technological effects [1].

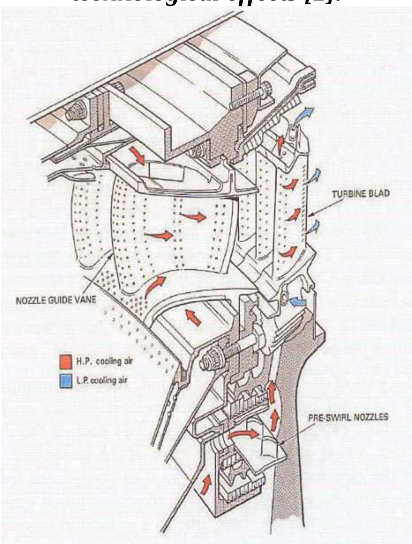

Fig. 2: Cooled gas turbine stage, (figure extracted from [2]).

Despite their small dimensions, these components, often mentioned as “technological effects”, can have a significant impact on the engine performances, and many experimental and numerical studies have shown in the past the importance of such geometrical details on the flow. One of the first technological effects to be taken into account in the design process was the tip clearance gap which is detrimental to the turbomachine performance in many aspects [3,4,5]. It is responsible for a significant part of losses and reduction of blade loadings. It also produces blockage of passage flow and distorts the flow angle distribution downstream of the rotor, interfering with flow condition for the downstream blade row. The impact of the size of the tip clearance gap on engine performances and on the downstream flow characteristics has also been widely investigated. After that some researchers studied the effect of hub disk leakage flow, as Shabbir & al [6], who investigated its effect on two high speed axial compressor rotors. They observed that adding the hub leakage flow to the primary passage flow reduced the pressure rise capability of the compressor, confirming experiments. Paniagua & al [7] experimentally studied the effect of hub endwall cavity flow on the flow field of a transonic high pressure turbine, showing that leakage flow can modify significantly the rotor relative incidence, secondary flows intensity and blade loadings. Another important geometrical component is the blade fillet, whose influence on aerodynamic performances has been studied through experimental and numerical studies by Zess and Thole [8], followed by Pieringer & Sanz [9]. Zess showed that leading edge fillets could reduce the horseshoe vortex, leading to a delay in the development of the passage vortex. It was also shown that when the profile close to the endwalls tends to induce flow separation, neglecting the fillet can result in considerable errors. Finally, some technological effects have been used to control the flow field, as casing treatments [10,11] for stall margin improvement, or film-cooling holes in order to protect turbine blades from thermal stress [12]. All these studies demonstrate that knowing which geometry detail significantly affects the flow solution and which does not is important to reduce the time and resources in order to find a sufficiently accurate solution to a flow problem.

Nevertheless for turbomachinery applications, geometrical details such as tip clearance, cooling slots or fillets can be a real challenge for geometry modeling. Taking into account these technological effects in a CFD structured solver with a multi-block approach is indeed far away from being trivial. In order to simulate such effects with the ONERA structured multi-block CFD solver elsA [13,14] a Chimera technique has been developed. This overset grid method has been initially developed for simulations of bodies in relative motion (helicopter rotors) [15,16], and allows for a high flexibility in the grid generation. It has been improved and is applied here to the modeling of technological effects encountered on turbomachinery applications.

The aim of this paper is to give an overview of different numerical simulations performed at ONERA on representative turbomachinery applications including a wide range of technological effects. The first part of the paper describes the main characteristics of elsA CFD software and its implemented Chimera technique. The second part is devoted to the application of this technique to different technological effects encountered on industrial turbomachines. Five examples are presented, each simulating a turbomachinery configuration with a specific technological effect: axial slots, tip clearance, casing treatment, cooling holes, and blade fillets.

2 About *elsA* software features

The *elsA* solver, developed at ONERA since 1997, is a multi-application aerodynamic code based on a cell-centered finite volume method for structured meshes. Solving the compressible, three-dimensional Reynolds-averaged Navier-Stokes equations, *elsA* allows to simulate a wide range of aerospace configurations such as aircrafts, space launchers, missiles, helicopters and turbomachines [13,14]. Therefore a wide range of numerical tools and models are available.

Classical numerical tools are provided in *elsA* for turbomachinery applications such as a cell-centered space discretization scheme of Jameson with artificial viscosity, and a second-order accurate Roe scheme for transport equations of turbulence models.

Several time integration schemes are available in *elsA* to perform steady and unsteady computations. Explicit or implicit schemes, such as a pseudo-time approach (Dual Time Stepping) or the Gear Method, are available. Time integration can be solved either by an implicit residual smoothing phase with a 4-step Runge-Kutta technique or by an implicit LU scalar relaxation phase associated to a backward Euler scheme. For steady computations convergence acceleration techniques such as local time stepping and multi-grid method are available in order to reduce the global CPU time.

Suitable boundary conditions for turbomachinery configurations have been implemented in order to compute steady flow applications: coincident and non coincident matching conditions have been developed for the treatment of periodic condition frontiers and a steady multi-stage condition using pitch-averaging for the treatment of the rotor-stator interface; different types of inlet, outlet and wall conditions are also available.

A large variety of turbulence models are presently available from algebraic to non-Boussinesq models. Computations presented in this paper were performed with 1 or 2 transport equation turbulence models, such as Spalart-Allmaras or k-ε.

3 Chimera method description

For structured grid flow solvers, a major obstacle is the difficulty in building meshes around complex configurations. One way to alleviate this problem is to use the Chimera technique [17,18]. It is an overset grid approach for which grids are generated independently around each body.

A Chimera technique has been developed in *elsA* [15][16]. It is a matching condition between blocks which overlap. Any grid can overlap with an arbitrary number of other grids which may overlap themselves. The RANS equations are solved on each grid system, and transfers are then performed between overlapping grids, by interpolation of the conservative and turbulent variables, first at overlapping boundaries, and then around blanked mesh cells lying under solid bodies. A cell search procedure is accomplished thanks to an alternating digital tree research algorithm (ADT) [19] which determines the donor cell and the interpolation coefficient for a given target cell. Multiply defined walls are treated thanks to the algorithm of Schwarz [20] for interpolation coefficient calculations. Several hole-cutting techniques have been developed, including the Object X-Ray technique, originally developed by Meakin [21].

A major advantage of the Chimera method is that it significantly simplifies the process of mesh generation by using overlapping grids. A second advantage is that this technique is also very well adapted for parametric studies on the characteristics of the technological components. For example if one wishes to study the impact of the size or of the position of a technological effect on the flow field, one will only modify the grids associated to this geometrical component.

Figure 3 represents an example of Chimera grids applied to a jet in a cross-flow configuration (3-a). With a coincident matching grid using two blocks (3-b) one notices that the grid densification at the wall of the cavity propagates through the adjacent domains due to the coincident matching of the nodes at the join boundaries. The Chimera approach enables to alleviate this constraint either by using 2 overlapping blocks (3-c) or by adding a third buffer grid (3-d).

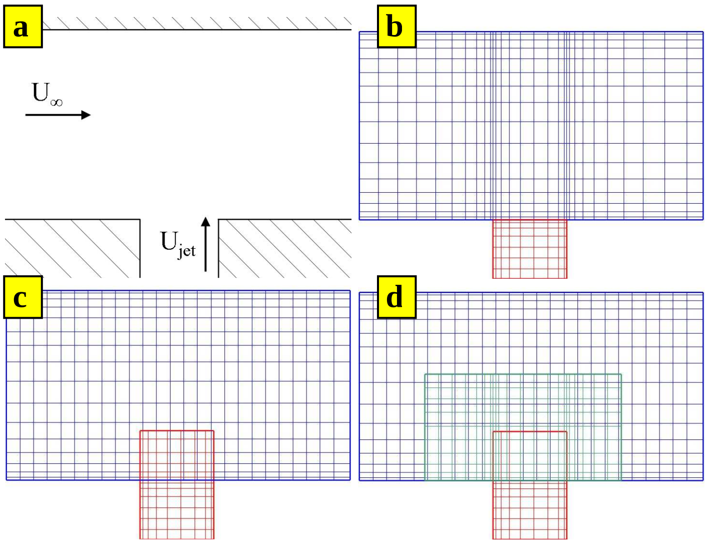

Fig. 3: Illustration of the Chimera approach on a jet in a cross-flow configuration; a) geometry to compute, b) coincident matching grid, c) chimera overset grids, d) chimera overset grids with buffer intermediate grid.

One important drawback of the Chimera method is that it induces conservation losses. Indeed, the interpolation of conservative variables during the Chimera transfer does not ensure flux conservations between domains. This means that the upstream and downstream mass-flow of computed configurations including overset grids can slightly differ. Since mass-flow is one of the key parameters in turbomachinery applications, it is highly important to check that these conservation losses are acceptable for the interpretation of results. In the presented applications, the allowed value of the conservation losses (quantified in terms of normalized difference between upstream and downstream mass-flow: $(m_{upstream}-m_{downstream})/m_{upstream}$) was set to be less than 0.1%. When it was found to be higher, the Chimera grids were densified in order to reduce these conservation losses. Generally, it is advised to ensure consitency of grid density on the overlap boundaries.

The following paragraphs of the paper present five examples performed on different turbomachinery applications including different technological effects.

4 Presentation of the computations

4.1 Transonic rotor with hub disk leakage

The first example, presented in figure 4, is a simulation of the NASA 37 rotor configuration taking into account the hub disk leakage flow emanating from the gap existing between the fixed and rotating parts of the hub upstream of the blade. This transonic rotor is a well known turbomachinery test-case which has been computed by numerous authors [22]. Experimental data were obtained at various measurement planes using both Laser Doppler Velocimetry and classical rake measurements of pitch-wise averaged total pressure and pitch-wise averaged total temperature [23].

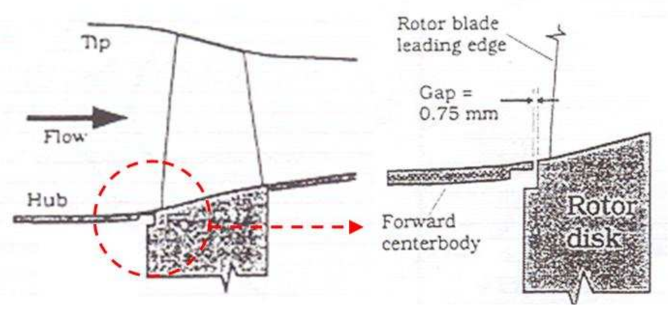

Fig. 4: Detailed view of NASA 37 forward center body and rotor disk interface [22].

Figure 4 presents the simulated technological effect: a 0.75 mm gap located upstream of the rotor between the stationary and rotating parts of the hub. It is modeled by additional grids overlapping the ones defining the rotor channel as presented in figure 5. The grid, including 1.5 10^6 points, is composed of 3 families of blocks:

- a multi-block O-H type coincident grid for the rotor channel, (blue & black), including the tip clearance gap at the tip of the rotor,

- one H-type block for the hub disk leakage gap (red) at the bottom of which is imposed a mass-flow injection condition (0.1% of the upstream total mass-flow),

- a “buffer” intermediate H-type block (green), overlapping the previous grids. The cell size of this grid is adapted to the gap near the cavity zone and gradually increases to match the size of the channel cells far away from the gap.

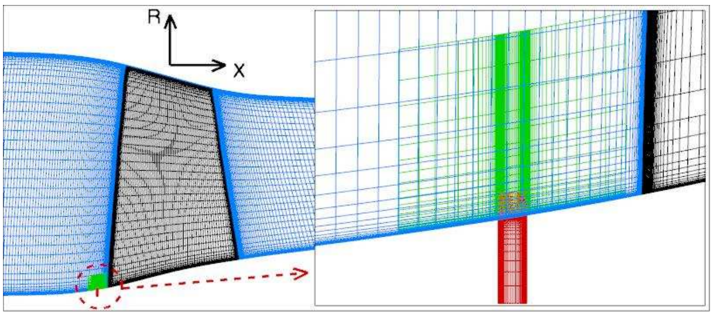

Fig. 5: View of the Chimera grids used for hub disk leakage simulation.

The use of these patch grids enables to obtain a good description of the detached flow at the exit of the hub disk leakage, as can be seen in figure 6 which represents a view of the relative Mach number in the interaction zone between the main flow and the cavity jet. One notices the flow separation induced by the interaction and also a good continuity of the flow variables at the Chimera boundaries in the overlapping zones, confirming that the node density in the buffer zone is satisfactory.

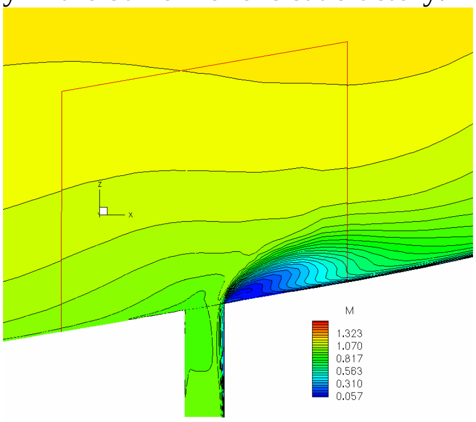

Fig. 6: View of the relative Mach number in the interaction zone between the primary flow and the cavity jet.

Figure 7 represents a comparison of the experimental and computed radial distributions of the total-to-total pressure and temperature ratios at the outlet of the rotor. One notices the improvement when taking into account the hub disk leakage flow (0.1% of the total mass-flow): simulating this gap reduces the total pressure overshoot which is observed on the lower part of the blade for the calculation without the gap, and also improves the prediction of the enthalpy rise. These results are in good agreement with those obtained by Shabbir et al. [6], who had shown that the leakage flow is responsible for a significant deficit of total pressure near the hub.

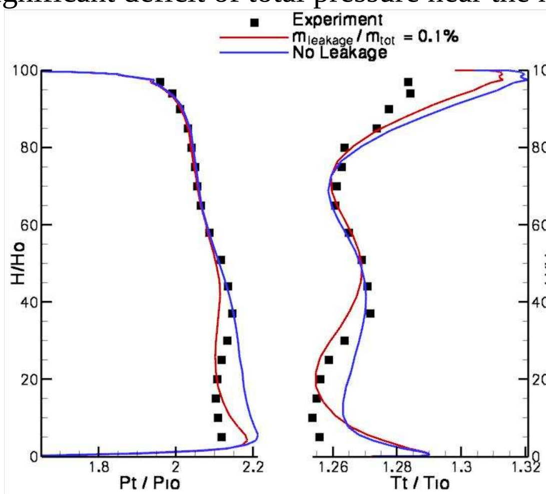

Fig. 7: Effect of hub disk leakage injection on the radial distribution of downstream pitch-wise averaged total pressure and temperature.

Figure 8 enables to understand the main flow mechanism induced by the hub disk leakage flow. One can observe that the streamlines emanating from the hub disk leakage gap tend to merge with the streamlines coming from the hub boundary layer. The pressure gradient across the blade passage makes the streamlines migrate on the suction side of the blade, amplifying the corner stall, thereby increasing aerodynamic losses and reducing the compression rate.

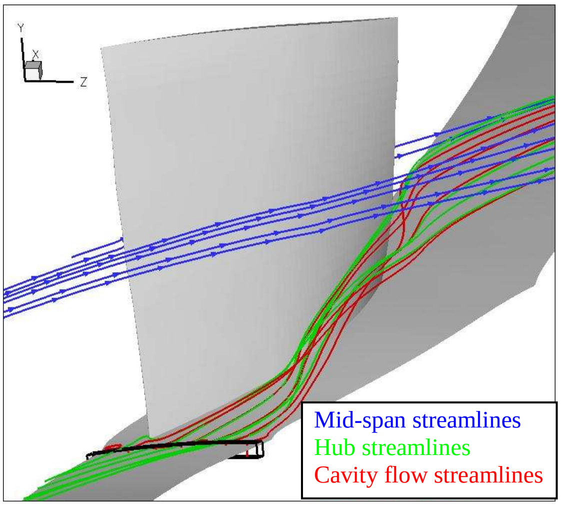

Fig. 8: Streamlines emanating from the cavity.

4.2 Transonic rotor with tip leakage simulated with the Chimera technique

The second technological effect modeled is the tip clearance gap, which is a key geometrical effect to take into account in turbomachinery applications. In the previous computation the clearance located at the tip of the blade was meshed with an O-H coincident grid. Nevertheless it can be interesting to use overset grids in the tip clearance zone to simulate this gap. To take into account the tip clearance gap with the Chimera approach the mesh strategy proposed by Gerolymos [24] is applied. The 1.7 10^6 point mesh, represented in figure 9, is composed of two families of blocks:

- a multi-block O-H type grid for the rotor channel (in blue) without the tip clearance. This grid is only densified in the radial direction near the endwalls (hub and casing), but does not present any radial densification at the blade tip,

- two additional patch O grids (green and red), representing the tip clearance gap, which overset the previous grids.

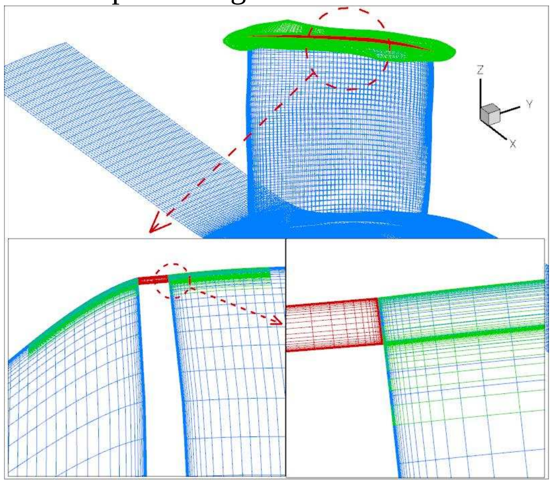

Fig. 9: View of patch grid approach for tip leakage simulation.

This mesh approach using Chimera grids presents two main interests. The first advantage is that in full coincident grid taking into account tip leakage, the grid presents a radial densification at the tip of the blade. Because of the node coincidence at the join boundaries, this grid densification propagates far away upstream and downstream of the blade, in zones where the densification becomes useless, which generates a waste of grid points. Therefore the use of Chimera grids enables a local densification in zones of interest, thus optimizing the mesh distribution. The second interest of such an approach is that it enables to perform easily parametric studies on the gap size, which only requires to modify the patch grid corresponding to the tip clearance, without changing the rotor channel grid.

Comparisons of computed and measured performances for the NASA 37 rotor are shown in figure 10. The total-to-total pressure ratio and the isentropic efficiency are plotted as a function of the mass-flow. Results obtained with the Chimera approach is compared with those obtained on a full coincident grid. The validation against experimental data and against results obtained with a full coincident grid shows satisfactory agreement, validating the Chimera approach.

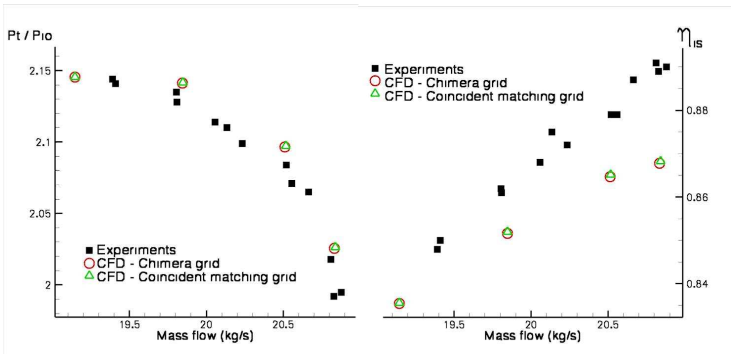

Fig. 10: Compression rate and isentropic efficiency as a function of the mass-flow.

The Chimera grids enable to capture the complex flow phenomena which occur in the tip leakage zone, as can be seen on the flow field views in the tip zone (figure 11). The interaction between the tip leakage vortex and the oblique shock wave, which generates a low velocity zone, is well reproduced in the CFD.

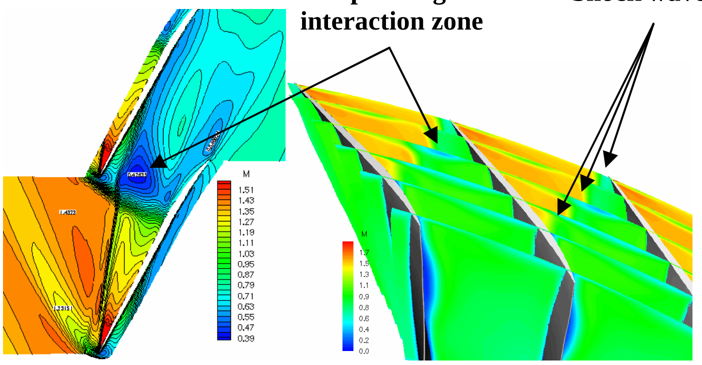

Fig. 11: Radial (95% span height) and axial slices of relative Mach number contour in the tip zone.

4.3 Transonic rotor with tip leakage simulated with the Chimera technique

The third example presented in this paper is a numerical simulation on NASA 37 configuration with five circumferential grooves placed at the casing over the blade tip and equally distributed in the axial direction. These devices, called casing treatments, are known to enable stall margin improvements [10][11].

Figure 12 represents a view of the 1.4 10^6 point grid, composed of two families of blocks:

- a multi-block full coincident O-H type grid for the “smooth wall” configuration (without slots, blue & black), including the rotor tip clearance,

- five H-type blocks representing each a circumferential slot, overlapping the previous grids.

The Chimera approach can be very useful for such configuration in order to optimize casing treatments by performing a parametric study on the geometry of the slots (position, depth, number).

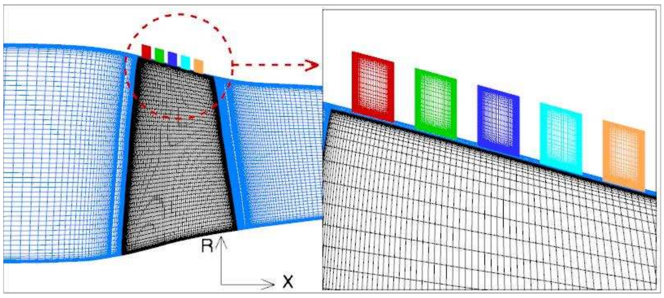

Fig. 12: Meridian view of the grid used for the casing treatments configuration.

Figure 13 represents the impact of the casing treatments on the compressor map. One can notice the increase of the surge margin in terms of mass-flow obtained thanks to the casing treatments, corresponding approximately to 3% of the nominal mass-flow, which is comparable to results obtained in the literature surveys [10][11]. For the nominal operating point (~20.5kg/s) the casing treatment slightly deteriorates the compression rate. For operating points near stall, casing treatments generates a stabilizing effect; according to the computations, the numerical stall occurs for lower mass-flow rates (18.36 kg/s with casing treatments versus 18.85 kg/s for the smooth wall test case).

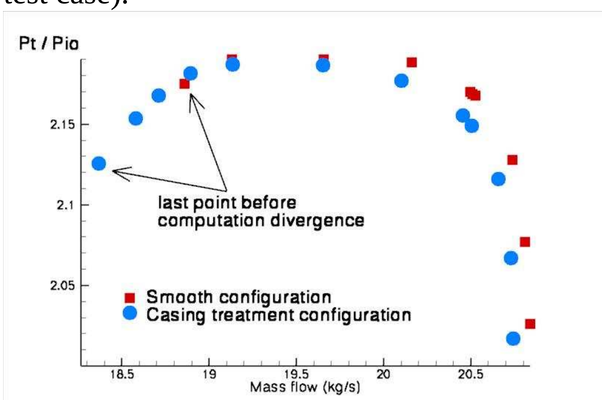

Fig. 13: Impact of casing treatments on the compressor map.

The analysis of the flow field at the interface between the main flow and the flow within the grooves highlights mechanisms leading to performance enhancements. The slots generate a radial transport of low energy fluid from casing boundary layer to the grooves, as can be seen on the radial velocity distribution presented in figure 14. It highlights the important flow recirculations at the tip of the blade. According to the computation the flow leaving the grooves to the passage tends to disturb the flow within the tip clearance and limits the extension of the leakage vortex. One notices that the flow recirculation mainly occurs for the first slots. Analysis of the computed results has also shown that the casing treatments lead to a reduced loading of the blade tip.

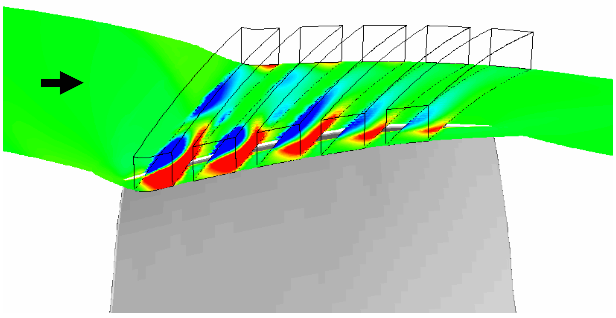

Fig. 14: Radial velocity at blade tip (red : Vr > 0, blue : Vr <0).

4.4 Film-cooling test case

Another important technological effect to simulate is the film-cooling holes for turbine blades, which are geometrical components difficult to include in a full coincident multi-block structured grid, because of their small dimensions. Therefore an overset grid approach can be interesting for such a configuration. In order to quantify the interest of the Chimera approach for film-cooling configuration a test has been performed on the Aachen turbine rotor [3][25] with 4 cooling holes placed on the blade, 2 on each part of the blades. The 2.10^6 point grid presented in figure 15 is composed of 3 families of blocks:

- a multi-block O-H type grid for the channel (green and blue),

- O type blocks for cooling holes (red), overlapping the previous grid,

- a buffer intermediate grid located at the exit of the holes (in the interaction zone) ensuring the connection between the cooling holes and the channel flow. The size of the cells within this buffer grid is adapted to the cooling holes at the hole exit, and gradually increases to match the size of the channel grids at the boundaries.

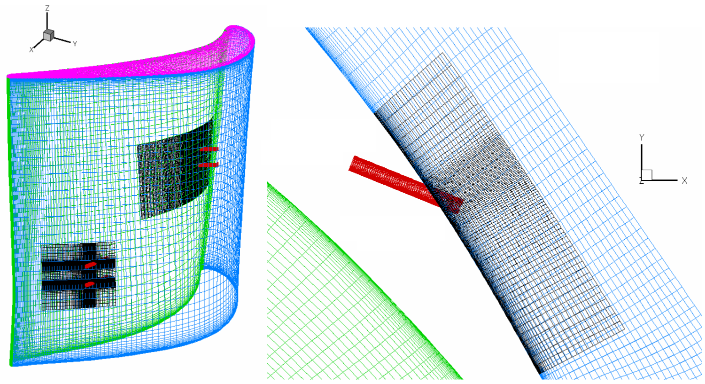

Fig. 15: View of the grid in the cooling zone.

The use of specific grids for cooling holes linked to a densified grid at the exit of the holes enables to capture the transport of the cold flow emanating from the cooling holes, and to simulate its convection in the passage (figure 16-right). The buffer grid also enables to simulate the interaction effect between the cooling flow and the main flow, allowing the capture of small flow structures, as the counter-rotating vortices generated at the exit of the film-cooling holes (figure 16-right).

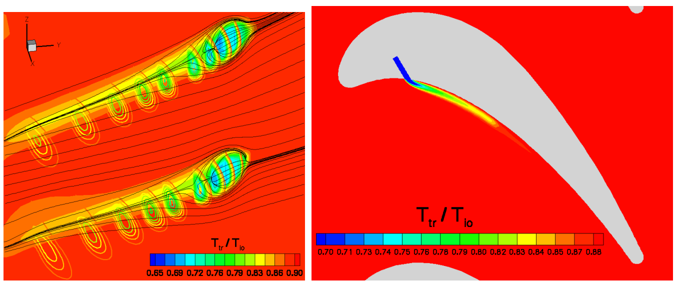

Fig. 16: View of the relative total temperature contours.

This test case represents a first validation of the Chimera approach for such configurations, with only 4 holes simulated, which is of course not representative of realistic geometries, which can present hundreds of holes For efficient film-cooling. Nevertheless this first computation demonstrates the potential capacity of the Chimera technique to compute a film-cooling configuration. The next calculations will be performed on a turbine blade including a more realistic number of holes.

4.5 Blade fillet modelling

The final example presented in this paper is the simulation of a rotor blade with a fillet radius. Fillets linking the blades with the endwalls are usually not considered in numerical simulations, since the grid generation with a full coincident structured approach is complex. The Chimera technique enables a very simple meshing strategy presented in figure 17. The 1.4 10^6 point mesh is once again composed of two families of blocks:

- a multi-block coincident O-H type grid for the channel with a smooth hub (no fillet, green),
- an O-block type patch grid representing the fillet of 2mm radius, connecting both the hub and the blade walls.

The Chimera technique is once again very useful for parametric studies (for example on the radius value of the blade fillet), since it just requires to modify the fillet grid, without changing the channel grid.

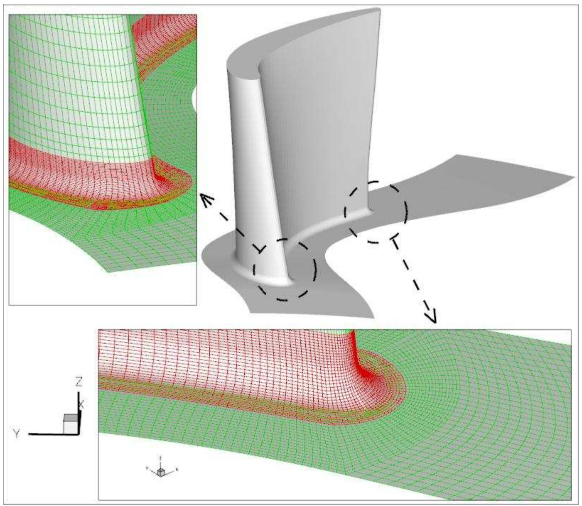

Fig. 17: View of the computational grid.

Two calculations have been accomplished: with and without hub fillet. The isentropic efficiency is equal to 0.947 on the smooth wall configuration, and drops very slightly to 0.946 with the fillet. Figure 18 represents a comparison of the entropy value as a function of the pitch position, half a chord downstream of the rotor for different span heights. Results show that considering the fillet radius in the CFD leads to differences on the bottom part of the blade (0-10%). From 20% to the casing, the effect is negligible. To conclude the impact of the blade fillet on this configuration is small. Nevertheless when the profile close to the endwalls tend to induce flow separation, neglecting the fillet can result in considerable errors [8][9].

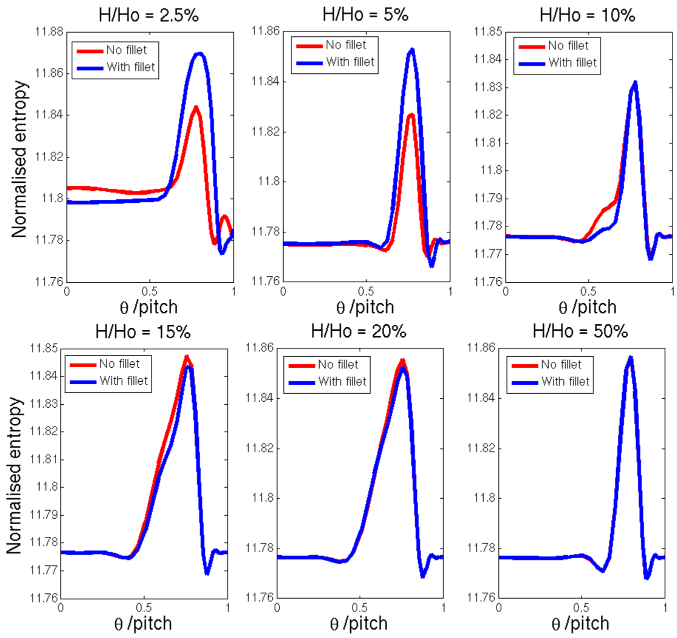

Fig. 18: Impact of the blade fillet on the entropy distribution downstream of the rotor.

5 Conclusion

In this paper, different numerical simulations including technological effects have been simulated with the Chimera technique. The channel including the blades and the technological effects are meshed separately in a standard multi-block approach, and are assembled in an overset grid system, which eases the modeling of these technological effects.

The Chimera technique is an efficient and flexible method to take into account technological effects. A wide range of geometrical components can be treated. It allows grid refinement in the zones where the technological effects are located. The method has proved to be robust. It allows to quantify the impact of each technological effect on the flow performances: some can lead to minor differences, others to significant modifications of the flow field. This tool can be useful for aeroengine designers to indicate which effect to model depending on the problem. Parametric studies are also eased with this technique.

Despite its simplicity, some difficulties must not be neglected, first of all concerning conservation losses, which must remain small, and therefore requires attention on the grid density. A second difficulty remains the knowledge of boundary conditions close to technological effects, as for hub disk leakage flows. Finally this paper presents a first step in the validation of the Chimera approach for turbomachinery applications. Efforts will be pursued to deal with more complicated configurations, such as centrifugal compressors, pivot clearances, or realistic film-cooled turbine blades. The Chimera approach will also be used for unsteady applications, such as non circumferential casing treatment simulation.

References

[1] J. Lépine: *"Fan and low-pressure compressor design: general considerations Fan aerodynamic design Low-pressure compressor aerodynamic design"*. VKI Lecture series: Aeroengine design: a state of the art April 7-11, 2003.

[2] The Jet Engine. Rolls Royce, 4th Edition, 1986.

[3] R. Heider, J.M. Duboue, B. Petot, G. Billonnet, V. Couaillier, *"Three-dimensional Analysis of Turbine Rotor Flow including Tip Clearance"*, ASME 93-GT111. Gas Turbine and Aeroengine Congress and Exposition, Cincinnati, USA, May 24-27, 1993.

[4] M. Inoue, M. Furukawa: *"Physics of Tip Clearance Flow in Turbomachinery"*. FEDSM2002-31184, ASME 2002 Fluids Engineering Meeting, Montreal, Canada, July 14-18, 2002.

[5] M. Inoue, M. Kuroumaru, M. Furukawa: *"Effect of Tip Clearance on Stall Evolution Process in a Low-Speed Axial Compressor Stage"*. ASME Turbo Expo 2004, GT2004-53354, June 14-17, Vienna, Austria.

[6] A. Shabbir, M.L. Celestina, J.J. Adamczyk, A.J. Strazisar : *"The effect of hub leakage flow on two high speed axial flow compressor rotors."*, ASME Turbo Expo, ASME97-GT346, Orlando, Florida, June2-5, 1997.

[7] G. Paniagua, R. Dénos, S. Almeida, *"Effect of the hub Endwall Cavity flow-field of a transonic high pressure turbine"*, GT2004-53458, ASME Turbo Expo 2004, June 14-17, 2004, Vienna, Austria.

[8] G.A. Zess, K.A. Thole: *"Computational Design and Experimental Evaluation of using a Leading Edge Fillet on a Gas Turbine Vane"*, ASME journal of turbomachinery, 2002, Vol 124, pp167-175.

[9] P. Pieringer, W. Sanz, *"Influence of the fillet between blade and casing on the aerodynamic performance of a transonic turbine vane."*, GT2004-53119, ASME Turbo Expo 2004, June 14-17, 2004, Vienna, Austria.

[10] A. Shabbir, J.J. Adamczyk, *« Flow mechanism for stall margin improvement due to circumferential casing grooves on axial compressors.»*, ASME Turbo Expo 2004, June 14-17 Austria, GT2004-53903.

[11] G. Brignole, F. Danner, H. P. Kau *"Time Resolved Simulation and Experimental Validation of the flow in axial slot casing treatments for transonic axial compressor"*. ASME2008-50593, june 9-13, Berlin, Germany.

[12] B. Tartinville, Ch. Hirsch, *"Modeling of Film-Cooling for Turbine Blade Design"*, ASME 2008-50316, june 9-13, Berlin, Germany.

[13] L. Cambier, J.P. Veuillot, *"Status of the elsA CFD software for flow simulation and multidisciplinary applications"*, 48th AIAA Aerospace Science Meeting and Exhibit, 2008.

[14] L. Cambier, M. Gazaix, *"elsA: an efficient object-oriented solution to CFD complexity"*. 40th AIAA Aerospace Science Meeting & Exhibit, Reno, USA, 2002.

[15] C. Benoit, G. Jeanfaivre, E. Canonne, *"Synthesis of ONERA Chimera method developed in the frame of CHANCE program"*. 31st European Rotorcraft Forum, Florence, 2005.

[16] Jeanfaivre G. Benoit C., Lepape M.C., *"Improvement of the robustness of the Chimera method"*, 32nd AIAA Fluid Dynamics Conference, 2002.

[17] J.A. Benek, J.L. Steger, F.C. Dougherty. *"A flexible grid embedding technique with application to the Euler Equations"*. AIAA Paper 83-1944, January 1983.

[18] J.A. Benek, J.L. Steger, F.C. Dougherty. *"A Chimera grid scheme"*, ASME mini-symposium on advances in grid generation, Houston, 1993.

[19] J. Bonet, J. Peraire: *"An Alternating Digital Tree (ADT) Algorithm for 3DGeometric searching and intersection Problems"*, International Journal for Numerical Methods in Engineering, Vol. 31, pp-1-17, 1991.

[20] Th. Schwarz, *"Development of a Wall Treatment for Navier-Stokes Computations using overset Grid technique"*. Proceedings of the 26th European Rotorcraft Forum, The Hague, September 2000.

[21] R.L. Meakin, *« Object X-Rays for cutting holes in composite overset structured grids"*, AIAA 2001-2537, 2001

[22] Agard-AR-355, *"CFD Validation for Propulsion System Components"*, May 1998.

[23] A.J., Strazisar, *"Data Report and Data Diskette for NASA Transonic Compressor Rotor37"*, NASA Lewis Research Center, 1994.

[24] G.A. Gerolymos, G. Tsanga, I. Vallet: *"Near-wall k-ε computation of transonic turbomachinery flows with tip clearance"*, AIAA journal, vol. 36, no10, pp. 1769-1777, 1998.

[25] Autogrid manual version and training course CD.

[26] J. Martegoutte, "Modélisation numérique des effets technologiques en turbomachines par la technique chimère du code elsA". Rapport de stage de fin d'étude, ONERA technical report- 2005.

Acknowledgements

French Ministry of Transport (DPAC) and French Ministry of Defense (SPAé) are acknowledged for the financial support to the numerical studies presented in this paper.

We are also grateful to J. Martegoutte for his work accomplished at ONERA on the validation of the Chimera approach [26].

Contact Author Email address

Lionel.Castillon@onera.fr

Copyright Statement