Comparison of Chimera and Sliding Mesh Techniques for Unsteady Simulations of Counter Rotating Open-Rotors

Benjamin FRANCOIS
Michel COSTES
Guillaume DUFOUR

1 Airbus Operations S.A.S., 306 route de Bayonne, 31000 TOULOUSE Cedex 9 FRANCE, benjamin.francois@airbus.com
2 ONERA, Applied Aerodynamics Department, Meudon FRANCE
3 CERFACS, CFD Team, TOULOUSE FRANCE

Abstract

Due to the growing interest from engines and airliners manufacturers for Counter Rotating Open-Rotors, lots of efforts are presently devoted in the development of reliable CFD methodologies. This paper compares two techniques, namely chimera and sliding mesh, enabling 360° moving propeller mesh into airframe fixed mesh used for unsteady CROR simulations. The assessment of these techniques is a major step in the development of dedicated numerical techniques and remains to be achieved.

Nomenclature

\( \rho \) density \([\text{kg/m}^3]\)
\( F_{T,Y,Z} \) thrust, lateral and lift force component \([\text{N}]\)
\( C_{T,Y,Z} \) thrust, lateral and lift force coefficient,
\( C_{T,Y,Z} = F_{T,Y,Z}/(\rho.n^2.D^4) \)
\( D \) propeller diameter \([\text{m}]\)
\( R \) propeller radius \([\text{m}]\)
\( n \) propeller rotation speed \([\text{1/s}]\)
\( U \) Free stream velocity \([\text{m/s}]\)
\( C_p \) power coefficient, \( C_p = P/(\rho.n^3.D^3) \)
\( \eta \) propeller efficiency, \( \eta = J.C_T/C_p \)
\( J \) advance ratio, \( J = U/(n.D) \)
\( P \) Power \([\text{kw}]\)
\( a \) velocity of sound \([\text{m/s}]\)
\( N_{FR} \) number of front blades
\( N_{AR} \) number of rear blades
\( f_{FR} \) Blade Passing Frequency of Front Rotor \([\text{Hz}]\)
\( f_{FR} = 2.n.N_{AR} \)
\( f_{AR} \) Blade Passing Frequency of Aft Rotor \([\text{Hz}]\)
\( f_{AR} = 2.n.N_{FR} \)
\( y^+ \) non-dimensional size of the first cell from the wall
\( \Delta x \) mesh size
\( t_e \) equivalent time step related to mesh quality
\( V_{opp} \) relative velocity of the opposite grid system

I. Introduction

In the context of increasing costs for fuel, the development of new aircraft concepts is mainly driven by the fuel burn reduction. To reach this end, new engines concept such as Counter Rotating Open-Rotors (CROR) appears to be one suitable option for the single aisle segment, currently dominated by the Airbus A320 and Boeing 737. This concept had been previously the focus of a large research effort led by NASA and US industry in the late 1970s and 1980s, motivated by the high fuel costs arising from the 1973 oil crisis (Hager [7], 1988). Significant advances were achieved but due to the decrease in oil prices, the interest in bringing those engines to market waned. Presently, CROR concept appears to be one promising option for powering new generation of short range aircraft.

CROR represents a large gap in the evolution of the modern aircraft architecture. This new concept raises major challenges in CFD such as the simulation of counter-rotating propellers in an external free stream environment, the impact of an installation...
system and the modelling of the aerodynamic interactions between front and rear rotors. These interactions have a significant impact on the aerodynamic performances or the radiated noise.

The high level of maturity of numerical methods in aerodynamics enabled CFD to play an important role in the research and design of CRORs to address these concerns. Consequently, lots of efforts are currently devoted to the development of reliable CFD methodologies.

In this paper, these two techniques are applied to an isolated CROR geometry (Airbus generic design used on European Research Platform CleanSky SFWA, [20]) shown in Fig. 1. URANS (Unsteady Reynolds Averaged Navier-Stokes) simulations at typical cruise conditions of M=0.73 and an altitude of 35000 ft were performed for both techniques without incidence.

The first part of the paper will describe and analyse the mesh used for the comparison. This part will also present the chimera and sliding mesh techniques. The second part will present the methodology followed to perform a relevant comparison. Then, aerodynamic performances and flow-fields are analysed and compared. At the end, overall CPU performances of both techniques are discussed.

II. Computational Strategy

II.A. Geometry

This rig is planned to be used to both test, validate and develop the numerical approach in term of CFD techniques and mesh requirements, and enhance the understanding of the complex aerodynamics of CROR at Airbus. The CROR geometry (Fig. 1) is an 11x9 bladed pusher configuration with a rotor diameter of D = 4.2672m (14ft). Inlet and exhaust are not modelled in the nacelle shape. The nacelle was designed to slightly accelerate the flow in order to have a homogeneous flow at M=0.75 just before the front rotor.

II.B. Mesh strategy

For the investigation presented in this paper, the techniques of moving meshes available in the elsA code were exploited. 360° meshes are used because they are suitable for simulations with incidence and installation effect.
The CROR geometry (Fig. 1) is composed of fixed parts (front and aft part of nacelle) and rotating parts (front rotor) and counter-rotating parts (aft rotor). Consequently, moving meshes techniques are required to compute the aerodynamic fields around the blades. This leads to split the mesh into three distinct domains as shown in Fig. 2: one background fixed mesh and two cylindrical rotating meshes (one for each rotor). Each rotor mesh is contained into a cylindrical domain. This enables to refine the mesh around the blades areas without any propagation of refined mesh into the far-field area. This mesh strategy is suitable for the modelling of installation effects on aircraft. The far-field mesh contains 5 millions of nodes. The front and aft rotor mesh contain respectively 26.5 and 21.5 millions of nodes. The total amount is 53 millions.

The periodicity in each rotor is exploited for the mesh: each rotor can be split into channels. For each channel, the blade is meshed by a C-block. H-blocks surround the C-block to complete the channel. 25 points are used to model the boundary layer on blades and nacelle for a y+ around 2.

Chimera and sliding mesh techniques follow this meshing architecture. However, these two techniques rely on different treatment at their interface. Consequently, meshes are slightly different at the boundaries between domains.

II.C. Moving meshes techniques

II.C.1 Chimera technique

The Chimera technique is a mesh strategy for assembling overlapping grid systems. This technique was first used by Steger et al. [15] in 1983. One purpose of his work was to ease the generation of structured mesh for complex shapes. Then, the Chimera approach has been used as a moving mesh technique. Many developments have been achieved and applied for helicopter applications (Ahmad and al. [1], 1994; Stangl et al. [12], 1996). Recently, this technique has been extended to various applications such as casing treatment in turbomachinery (Legras [9], 2009), forced motion of a spoiler on a aircraft (Blanc, 2009), etc. The first referenced application of Chimera on CROR was performed by Stuermer [14] in 2008 with an unstructured mesh using the TAU solver.
systems with complex geometry (Fig. 3) and/or with moving parts. Its flexibility relies on the fact that each part of the geometry can be meshed independently. The communication between meshes is done through the overlap areas. Inside overlap areas, several rows of cells neighbouring the domain border are interpolated by computed cells from the other domain. The interpolations are performed on the state variables at the center of the cells as shown in Fig. 4. However, these interpolations don't ensure the conservation of fluxes through the domains and may introduce numerical errors. This is why fine enough meshes and cells of similar size are required at the interface.

Fig. 4: Chimera interpolation on 1D domains

II.C.2. Sliding mesh technique

The sliding mesh technique is based on the use of non-conforming grid systems having different relative motion. A sliding surface can be defined as the boundary between the two non-conformed meshes. Non-conforming meshes have been first used with fixed meshes in order to optimize the size of structured mesh (Rai [10], 1986) and ease the generation of meshes for complex geometry (Fillola et al. [5], 2004). This technique has been extended for grids with different relative motion. A recent industrial application was the aerodynamic simulations of multi-stage compressor in turbo-machinery (Gourdain et al. [6], 2010).

The communication through the sliding surface is performed using a computation of fluxes through the non-matching interface. To achieve this, the cell faces of the two non-matching interfaces are split into subfaces which are conformed each other as shown in Fig. 6. This enables to perform the computation of flux. If sliding interfaces are planar, this ensures the conservation of flux at the interface. For curved interface, the conservativity of fluxes is no longer satisfied because normal vectors from each non-matching interface have different direction.

Fig. 5: Non-conforming mesh applied on a complete aircraft (Vassberg [19], 2009)

Fig. 6: Computation of fluxes through non-conforming interfaces
II.C CFD simulation settings

The computations were performed using the elsA code (Gazaix and Cambier [3], 2002). elsA is a structured finite-volume cell-center based CFD solver. It enables to solve URANS equations in fixed and mobile frame for compressible flows. For the spatial discretization, the centered Jameson scheme (Jameson et al. [8], 1983) with artificial viscosity is used. Time integration of the governing equations is based on a backward Euler scheme with dual time stepping. The turbulence is modelled by the Spalart-Allmaras model (Spalart and Allmaras [11], 1992). The simulations are performed for both Chimera and sliding techniques with a time step equivalent to a propeller rotation of 0.5°. To reach convergence of aerodynamic forces, 3 rotations were performed. 10 sub-iterations were used with the DTS scheme.

III. Comparison of the results

III.A. Methodology for comparison

To perform relevantly a comparison, the meshes used with Chimera and sliding meshes techniques have to be as close as possible. However, one relies on overlapping domains and the other on non-matching interface. To limit the mesh difference between the two techniques, the chimera mesh was only modified at domain border to be turned into a sliding mesh. To achieve that, the overlapping area had been deleted to be turned into non-matching interface (see Fig. 7). Thus, the two meshes are quasi-identical and enable to compare strictly these two techniques without introducing mesh uncertainties.

All numerical settings were kept the same for the simulations.

### Table 1: comparison of aerodynamic performances

<table>
<thead>
<tr>
<th></th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Front rotor</td>
<td>$C_T &lt; 0.1%$</td>
</tr>
<tr>
<td></td>
<td>$C_F &lt; 0.1%$</td>
</tr>
<tr>
<td></td>
<td>$\eta &lt; 0.1%$</td>
</tr>
<tr>
<td>Aft rotor</td>
<td>$C_T + 0.2%$</td>
</tr>
<tr>
<td></td>
<td>$C_F + 0.1%$</td>
</tr>
<tr>
<td></td>
<td>$\eta &lt; 0.1%$</td>
</tr>
</tbody>
</table>

III.B. Aerodynamic analysis

### III.B.1 Aerodynamic performances

The Table 1 presents the comparison of the aerodynamic coefficients extracted from simulations with both techniques. This shows a very good agreement between them.

### III.B.2 Flow-fields analysis

The unsteady nature of the flow-field around a CROR geometry is mainly driven by the motion of the rotors. The deflection of the flow imposed by the blades generates an inhomogeneous distribution of pressure around it and creates thrust. Also, the flow-field is unsteady because each blade sees the other blades from the opposite rotor moving at a constant speed of $2 \times n$. Consequently, the flow-field exhibits strong unsteady phenomena.

![Fig. 7: conversion from a chimera mesh to a sliding mesh](image)
which are periodic in nature, linked to the rotational speed and the number of blades of the rotors. Each rotor interacts with each other through pressure waves and wakes. Thus, as the number of blades is different between the two rotors, the frequency of those phenomena are different in each rotor (\(f_{FR} = 231.5\) Hz in the front rotor, \(f_{AR} = 291.5\) Hz in the aft rotor). These interactions are well-known (Tyler and Sofrin [17], 1962) and drive the time step sizing. To capture them properly, the period of the signal is sampled by 25 points in order to capture properly the shape of the signal (Hirsch [8], 2009). This leads to 0.64° of rotation per time step. For practical reasons of post-processing, a time step equivalent to 0.5° of rotation is used in this paper.

Aerodynamic interactions are very relevant to study in the framework of this comparison because they represent all the unsteadiness going upstream and downstream the interface.

### III.B.3 Aerodynamic interactions

Aerodynamic interactions can be split into pressure waves and wakes.

**Pressure Waves**

In the vicinity of rotors, the flow remains subsonic. Thus, pressure waves can propagate in the upstream direction at speed U-a. Thus, the front rotor is directly impacted by the pressure fluctuations from the aft rotor.

Figure 8 shows the impact of these pressure waves on the thrust level on one blade of the front rotor chosen randomly. The fluctuations (peak to peak) of the signal represent 4% of the mean value of thrust level. Figure 9 highlights the spectrum of the front blade thrust. This spectrum contains two harmonics (\(f_{AR}, 2f_{AR}\)) whose amplitude of the second frequencies is ten times lower than the fundamental one. The sinusoidal shape of the signal pictured in Fig. 8 confirms that the impact of pressure waves can be reduced to one harmonic.

Chimera and sliding mesh show a very good matching on the prediction of thrust level, so do the spectrum of frequencies. Figure 8 and 9 assess that pressure waves propagate through a chimera or sliding mesh boundary with the same accuracy.

![Fig. 8: Non-dimensional thrust on a blade from one blade of the front rotor](image)

**Wakes**

The second component of aerodynamic interactions between rotors is the impact of the wakes from the front rotor on the aft rotor. In addition, pressure waves, propagating also in the downstream
direction, affect also the aerodynamic field around the aft blades.

Fig. 10: Non-dimensional thrust on a blade from rear rotor

Figures 10 and 11 show that Chimera and sliding mesh techniques offer a very good matching for the shape of the signal and for the spectrum of frequencies. The mean thrust level from the sliding mesh is slightly higher than the thrust obtained with the Chimera one. This may be due to the fact that sliding mesh technique is conservative on the plane interface between rotors whereas chimera is not.

Computations with lower time steps (not shown here) have proven that up to eight harmonics can be captured.

This highlights that wakes contain a larger number of harmonics than pressure waves. Consequently, wakes cannot be fully captured with the current mesh and the current time step. Either chimera or sliding mesh interface act as low-frequency filters. The cut-off frequency of this filter can be estimated as the maximum of two parameters:
- implicit time step
- equivalent time step $t_m$ related to mesh refinement

The first one is the sampling frequency calculated from the implicit time step used ($0.5^\circ$ of rotation per time step): this samples the unsteady signal transmitted through the interface.

The second parameter is related to the maximum mesh size of each moving grid domain. To enable an accurate exchange of data through interface of moving grid domains, a minimum number of cells is required in the direction of the grid motion. If cells are too large, unsteady data are transmitted for consecutive time steps to the same cell and consequently, some unsteady variations are lost. To some extent, the unsteady signal is sampled on the mesh cells in the direction of grid motion which leads to the definition of the second parameter. This parameter can be expressed as a time step $t_m$ equal to the largest size of mesh cell divided by the relative motion speed of the opposite grid system:

$$t_m = \frac{\Delta_{\text{max}}}{V_{\text{opp}}}$$

For the present mesh, this time step $t_m$ is equivalent to $0.79^\circ$ of rotation. This means that the unsteady signal going through moving interfaces is filtered due to the mesh refinement.
Figure 12: ratio $\mu_t/\mu$ of the turbulent and laminar viscosity coefficients at $r/R = 0.75$

Figure 12 highlights the filter applied by the interface for both techniques on the ratio $\mu_t/\mu$ of the turbulent and laminar viscosity coefficients field projected on a cylindrical cut at $r/R = 0.75$. Chimera and sliding mesh techniques present both the same wake deficit after passing the interface between the front rotor and aft rotor mesh and then between the aft rotor and the far-field mesh.

This analysis leads us to some guidelines for unsteady computations with moving grids. First, mesh quality, regarding the equivalent time step, and implicit time step have to be consistent. Second, to capture wakes accurately, the unsteady signal needs to be less filtered and consequently, a more refined mesh with a smaller and consistent time step would be required. However, this leads to very expensive simulations.

### III.C. Overall CPU performances

The previous parts proved that the chimera and the sliding mesh technique can produce reliable CFD results with the same accuracy. The next step of this work is to compare their CPU performances.

The simulations performed on 360° meshes require large CPU resources during a large amount of time. Reducing the CPU time for CROR simulations is a key point for the improvement of reliable CFD techniques for industrial purposes and applications.

<table>
<thead>
<tr>
<th>Technique</th>
<th>CPU time for one rotation (128 cores)</th>
<th>Memory per core (Gb)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chimera</td>
<td>3.5 days</td>
<td>17.5</td>
</tr>
<tr>
<td>Sliding Mesh</td>
<td>3 days</td>
<td>10</td>
</tr>
</tbody>
</table>

Table 2: comparison of CPU performances

Table 2 summarizes the overall CPU performances of both techniques. These results state that sliding mesh is slightly faster than Chimera. Moreover, the sliding mesh technique uses nearly twice less memory than the chimera technique. Indeed, chimera is based on volume interpolations whereas sliding mesh techniques use surface interpolations. This mainly contributes to the reduction of memory used with the sliding mesh.

The benefits from the sliding mesh technique are very promising. The use of the sliding mesh technique on 360° CROR external application is quite new and doesn't benefit yet from the solid and wide experience gathered on internal turbomachinery applications. Consequently, large benefits on CPU performances can be expected. Thus, developments in the elsA code are currently ongoing to improve the overall CPU
performances for 360° CROR unsteady simulations.

Conclusions

In this paper, a comparison between the chimera and the sliding mesh techniques has been made. This paper has shown that the Chimera and the sliding mesh technique are able to produce reliable CFD results with an equivalent accuracy. In term of CPU performances, the sliding mesh technique offers a slight improvement on the CPU time and a large reduction of the memory used. New developments are ongoing and large potential gains in CPU are expected.

Acknowledgments

This work is part of a thesis research funded by Airbus and done in close collaboration with CERFACS and ONERA. The authors wish to thank all the people of the Airbus Aerodynamic Methods and Tools team in Toulouse, people from the CFD Team in CERFACS for their fruitful help on the sliding mesh technique and ONERA for their helpful suggestions.

References


