Propeller Blade Debris Kinematics: Blade debris trajectory computation with Aerodynamic effects using new FSI formulations

Roland Ortiz, Folco Casadei, Sylvain Mouton, Jean-François Sobry

► To cite this version:

Roland Ortiz, Folco Casadei, Sylvain Mouton, Jean-François Sobry. Propeller Blade Debris Kinematics: Blade debris trajectory computation with Aerodynamic effects using new FSI formulations. CEAS Aeronautical Journal, 2018, 9, pp.683-694. 10.1007/s13272-018-0313-4 . hal-01400250v2

HAL Id: hal-01400250

<https://hal.science/hal-01400250v2>

Submitted on 25 Jul 2018

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Propeller Blade Debris Kinematics

Blade debris trajectory computation with aerodynamic effects using new FSI formulations

R. Ortiz · F. Casadei · S. Mouton ·
J-F. Sobry

Received: date / Accepted: date

Abstract The paper presents numerical models and simulations performed at ONERA in order to investigate aerodynamic effects on the trajectory of an open rotor blade fragment released or lost during an engine burst event. The models are based on Finite Elements for the structural part and on Finite Volumes for the fluid part and are implemented in the EUROPLEXUS explicit code. Fluid-Structure Interaction (FSI) is taken into account by an embedded (or immersed) technique and precision is enhanced by refining the fluid grid locally using automatic mesh adaptivity. A new methodology has also been implemented in order to limit the air volume to be discretized in the numerical model, thus reducing the size of the fluid mesh and allowing fine-mesh (adaptive) computer simulations in a reasonable CPU time.

Keywords Fluid-Structure Interaction · mesh adaptivity · blade trajectory · CROR · EUROPLEXUS.

1 Introduction

For the aviation industry, reduction of fuel consumption, gas emissions and noise remains an important challenge [1]. The emerging Counter-Rotating

Roland Ortiz
ONERA, 5 boulevard Paul Painlevé, BP 21261, 59014 Lille Cedex
E-mail: roland.ortiz@onera.fr

Folco Casadei
Retired from European Commission, Joint Research Centre
E-mail: casadeifolco@gmail.com

Sylvain Mouton
ONERA, 8 rue des Vertugadins, 92190 Meudon
E-mail: sylvain.mouton@onera.fr

Jean Francois Sobry
ONERA, 5 boulevard Paul Painlevé, BP 21261, 59014 Lille Cedex
E-mail: jean-francois.sobry@onera.fr

Open Rotor (CROR) engine technology might bring substantial benefits in this respect and several test campaigns in wind tunnel installations are being conducted, often accompanied by numerical simulations.

A lot of work has been done during the initial development of CROR propulsion systems [2–6]. However, generally, the CFD codes used for aerodynamic analysis can only deal with a rigid-body representation of the structures involved. Recently, ONERA is studying the CROR propeller designed by AIRBUS [7] both experimentally and by numerical simulations, which require multidisciplinary tools and approaches.

In view of the future certification of a CROR-engine powered aircraft, manufacturers should address and assume the event of engine burst, including the release/loss of a fragment of an open rotor blade. To minimize the extent of shielding on the aircraft fuselage, the areas of possible impacts have to be precisely identified [8–10]. The simulation of the fragment trajectory requires accurate capturing of the aerodynamic forces.

The main contribution of the present study is to set up and validate a simulation methodology to predict debris trajectory and possible impact areas, by taking into account the deformability of the structural components and the aerodynamic effects induced by the surrounding fluid environment.

To this end, a detailed Finite Element/Finite Volume (FE/FV) model with Fluid-Structure Interaction (FSI) is developed in the EUROPLEXUS code (EPX) [11], an explicit computer program jointly developed by the French Commissariat à l’Energie Nucléaire et aux Energies Alternatives (CEA) and by the Joint Research Centre (JRC) of the European Commission.

The paper is organized as follows. Section 2 presents some key and innovative numerical techniques used in the present study: *i*) a FSI algorithm to couple the FV fluid model with an embedded (immersed) FE structural model; *ii*) a fully dynamic mesh adaptivity algorithm to automatically refine the fluid mesh only in the vicinity of the embedded structure, enhancing the accuracy of the FSI algorithm at a reasonable computational cost; and *iii*) a methodology to drastically reduce the extent of the fluid domain included in the numerical model, so as to further reduce the cost of the simulation without sacrificing the overall accuracy.

Section 3 shows preliminary aerodynamic CFD simulations including only the fluid. The structure (blade) is assumed to be rigid and fixed in space, and is simply represented by a rigid boundary (a solid wall in CFD terms). The scope is to validate EPX’s fluid model and aerodynamic force computation against the dedicated purely CFD code elsA developed at ONERA.

Finally, Section 4 presents an example of fully coupled FSI simulation where all the above mentioned numerical models are employed to accurately predict the blade trajectory and to identify the zones of impact against the fuselage. Since all structural components are treated as deformable (damageable) bodies in this study, such a type of simulations might also include the detailed study of the mechanical effects of the impact on the fuselage.

2 Numerical models

EPX is a general computer code for fast transient dynamic analysis of Fluid-Structure systems. The explicit central difference time integration scheme is used to advance the solution in time. This facilitates treatment of geometric non-linearities such as large motions and large strains, and of material non-linearities such as plasticity and damage.

The structural sub-domain is discretized by FE such as continuum, shells, bars etc. and the governing equation is the conservation of momentum (equilibrium). In the fluid sub-domain, the Euler inviscid compressible equations expressing conservation of mass, momentum and energy are discretized by either FE or, like in the present work, by a cell-centred Finite Volume (CCFV) method.

2.1 Fluid-Structure Interaction

The EPX code offers a wide panoply of FSI algorithms, see [12] for an overview. The optimal algorithm for each application depends upon the chosen fluid formulation (FE or FV) and on the amount of expected nonlinearity in the structure, namely the possibility of large rotations, failure, fragmentation.

Figs. 1(a) and 1(b) compare the FE and the CCFV classical approaches in the simplest possible FSI case, namely for conforming F-S meshes.

Fig. 1 FE and FV discretizations with a nodally conforming F-S mesh.

In the FE formulation the discrete variables are defined at the fluid nodes, which therefore must be kept separated from the structure nodes as shown in Fig. 1(a). In the FV formulation, instead, the variables are discretized at the volume (or cell) centroids, so that fluid and structure nodes may be merged together, see Fig. 1(b).

If a FE description of the fluid domain is adopted, FSI conditions are typically imposed by enforcing equal fluid and structure velocities at the nodes along the F-S interface, projected along the normal to the interface (while tangential velocity components are left free). In EPX this is typically obtained by a strong formulation, based on Lagrange multipliers.

However, in general the FV method is more accurate than FE for the fluid domain, and results in better pressure values for a given fluid mesh size. Therefore, CFD computer codes for purely fluid applications, such as e.g. the elsA code developed at ONERA, generally use the FV method. In the CCFV formulation used in this work, FSI is achieved by updating the fluid pressure at the cell centroid at each time step and then by applying it directly to the adjacent structure (if present) through equivalent nodal forces. By blocking the numerical fluxes of mass and energy between neighboring fluid cells separated by a solid shell wall and by using the relative (fluid minus structure) velocity in the transport terms of the ALE equations, one ensures a (weak) feedback mechanism by which the fluid “sees” the presence of the structure.

The classical FSI algorithms based on conforming F-S meshes are quite accurate but they are limited to cases of structures which undergo neither failure (with fragmentation) nor large deformations, and in particular also no large rotations which, however, are of primary interest here. The difficulties typically arise from fluid mesh rezoning algorithms being unable to avoid mesh entanglement.

For such extreme conditions a different class of FSI algorithms has been recently implemented in EPX, based upon the concept of immersed (or embedded) structure. The structure and fluid meshes are completely independent at the topological level. The structure is arbitrarily discretized using a Lagrangian description and then it is embedded (superposed) into a (background) fluid mesh. The fluid grid is typically regular (structured) and uses a Eulerian description, thus avoiding by construction the fluid mesh rezoning difficulties typical of ALE formulations.

FSI coupling is realized by searching the fluid entities (fluid nodes in FE, or fluid cell interfaces in CCFV) which are located sufficiently close to the structure, and then writing suitable coupling conditions, in a similar manner to the case of conforming meshes described previously. The candidates for coupling are the fluid entities located within a thin fluid layer (influence domain) around the structure, as shown in Fig. 2. In the case of CCFV fluid formulation used in the simulations of the present work, the embedded coupling algorithm is denoted as FLSW model in EPX, and is further detailed in [13, 14].

2.2 Fluid mesh adaptivity

When using an embedded FSI algorithm such as FLSW, the precision of F-S coupling depends very much on the use of a sufficiently fine fluid mesh, typically much finer than the structure mesh if this is discretized by shells. Since uniform refinement of the entire fluid mesh is out of question due to

Fig. 2 FLSW : a FSI algorithm for a fully non-conforming mesh (embedded structure).

computational cost, the fluid mesh should be refined at least in the vicinity of the (moving and deforming) structure. This is precisely the scope of adaptivity: to refine the fluid mesh only where it is needed, in this case close to the structure (FSI-driven adaptivity). The general strategy adopted in EPX to adaptively refine and un-refine the computational mesh is described in [15].

FSI-driven adaptivity is best used in conjunction with an embedded FSI algorithm such as FLSW, see [16]. The interacting fluid and structure are discretized in a completely independent way at the topological level. Typically, the fluid is represented by a uniform and regular (even structured) mesh fixed in space (Eulerian description) used as a “background” mesh. The structure is meshed independently and then it is “embedded” or “immersed” in the fluid mesh. The two meshes are therefore simply superposed, see Fig. 2.

Adaptivity in the fluid is activated by adding the ADAP keyword in the FSI directive (FLSW) of the EPX input file. The FLSW directive with adaptivity has been used in the last blade trajectory simulation using CCFV and FSI presented in Section 4. The ADAP sub-directive introduces adaptivity-related data for the concerned FSI model. The only mandatory parameter is $L_{\max}$, the desired maximum fluid mesh refinement level near the structure.

2.2.1 Choosing the influence domain thickness

In the FLSW model for FSI, the structure is coupled with the fluid cell interfaces which are found to be currently located within the structural influence domain. This domain is formed by circles and quadrilaterals in 2D, by spheres, prisms and hexahedra in 3D. The circles or spheres are centred on the structural nodes while the other geometric shapes are built by connecting the circles or spheres, see Fig. 2.

Thus, the thickness of the influence domain at a structural node is the diameter of the associated circle or sphere. The domain thickness at a point of the structure not coinciding with a node is interpolated from the nodal values around it. The user can either specify the thickness by imposing a uniform sphere radius R via the keyword R, or let the code automatically determine R based on the local size of the fluid mesh. In any case, a sphere radius is

finally associated with each structural node, so that the local thickness of the influence domain is $D = 2R$.

In order to ensure tightness, i.e. to avoid spurious fluid passage across a solid structure, the local thickness of the structure influence domain (i.e. the sphere diameter $D = 2R$) must be greater than the diagonal of the local fluid mesh cell, see Fig. 3. If the fluid mesh is formed by squares in 2D or cubes in 3D, of size h , then in order to ensure tightness it must be:

$$D > \sqrt{d}h \quad (1)$$

where d is the space dimension (2 or 3).

Fig. 3 Structural influence domain ensuring tightness.

This condition ensures that a continuous layer of fluid entities is coupled with the structure, even in the worst possible case that the structure is located exactly in the middle between two fluid entities, and has an oblique direction (not aligned with the global axes). Using a value of D much larger than the one given by eq. (1) is not advisable, because too much fluid would be “attached” to the structure. In each practical situation, the minimum value ensuring tightness is the best one.

When mesh adaptivity is adopted in the fluid domain, the size of the fluid mesh varies according to the level of refinement. A binary rule is adopted, whereby the size of each level is half that of the previous level. By convention, the base (ancestor or unrefined) fluid elements are placed at level 1.

Let us therefore denote h_1 the size of the base fluid mesh. At any other level of refinement $L > 1$, the size of the fluid mesh will be:

$$h_L = h_1/2^{L-1} \quad (2)$$

i.e., 1/2, 1/4, 1/8 etc. of the base size.

The practical convention is adopted that the user always specifies the FLSW data referred to the base fluid mesh, also in the case of an adaptive calculation. Then, if adaptive FSI is desired, the ADAP keyword is added to FLSW and the LMAX keyword is used to introduce the desired maximum refinement level ($L_{\max}$) of the fluid mesh near the structure. In this way, various levels of refinement can be tried out by changing only the $L_{\max}$ value.

2.2.2 Adapting the fluid mesh

To obtain a progressively refined fluid mesh near the (moving) structure, from level 1 (base mesh) to level $L_{\max}$ (the chosen maximum), we proceed as illustrated in Fig. 4 where just one structural element is considered for simplicity. Each used fluid element (base or descendent element, in adaptivity) is uniquely identified by the position of its centroid, i.e. the average position of its nodes.

Fig. 4 FSI-driven fluid mesh adaptivity.

Then, a hierarchy of $L_{\max}$ structural influence domains, similar to the one used to detect FSI, are built in order to adapt the mesh. Each influence domain is similar to, but has half the thickness as, the previous one in the hierarchy. The first (coarsest) influence domain (for $L = 1$) coincides exactly with the influence domain that would be used by FLSW in the absence of adaptivity i.e. it has the thickness D_1 declared in the input data set. For example, if a uniform sphere radius R_1 has been specified via the R keyword, then $D_1 = 2R_1$.

The last (finest) influence domain (for $L = L_{\max}$) has thickness $D_{L_{\max}} = D_1/2^{L_{\max}-1}$. This last domain is the one automatically used to detect FSI.

Therefore, when ADAP LMAX $l_{\max}$ [11] is specified the radius actually used for FSI is not R_1 , but $R_{L_{\max}} = R_1/2^{L_{\max}-1}$.

In the simple illustrative example of Fig. 4 we consider refinement up to level 3, i.e. ADAP LMAX 3. We assume that the user has declared in the input file an influence domain radius $R = \sqrt{2}h/2$ (or slightly larger), with h the size, assumed here uniform for simplicity, of the base fluid mesh. We indicate with $D_1 = 2R = \sqrt{2}h$ the diameter (thickness) of the base influence domain. This is used to find the base fluid elements that need to be refined, as shown in a). These are all the fluid elements whose centroid falls within the influence domain, and are indicated by a cross. In b) we can see the fluid mesh refined to level 2. Then a scaled-down structure influence domain of thickness $D_2 = D_1/2$ is used to identify the fluid elements which need to be further refined. In c) we can see the refined fluid mesh at level 3, which is the final one in this case. Finally, the code uses the finest influence domain $D_3 = D_1/4$ to locate the fluid entities which have to be coupled with the structure, as shown in d). In the example of Fig. 4 these entities are the fluid nodes (as is typical with the FE algorithm). In case of FLSW used in the present simulations, the fluid entities considered are rather the cell interfaces.

The complete refinement and un-refinement procedure is as follows. Two loops are performed at the beginning of each time step to adapt the fluid mesh. In the first loop, “coarse” fluid elements which now find themselves “near” the structure are (progressively) refined, while in the second loop “fine” fluid elements which now find themselves “far” from the structure are (progressively) unrefined. Each loop proceeds by examining one level at a time. The refinement loop does this in increasing level order (from level 1 to level $L_{\max} - 1$), while the unrefinement loop does this in decreasing level order, from level $L_{\max} - 1$ to level 1). Note that elements in level $L_{\max}$ are never examined directly. In fact, they need no refinement since they are already at the maximum refinement level.

2.3 The ALE GLOB methodology

The simulation of CROR debris trajectory and impact typically involves a huge fluid domain, from the engine to the impact point on the fuselage and possibly beyond. Discretizing this entire domain, even with the help of fluid mesh adaptivity described previously, might be computationally impractical and extremely CPU expensive.

Therefore, during the Clean Sky SFWA project [17–19], ONERA developed into EPX a new methodology (ALE GLOB command) to reduce the air volume actually discretized in the numerical model. Only a relatively small region (a box) of fluid around the moving blade debris is discretized. Thanks to ALE formulation used in EPX, the fluid mesh can be moved arbitrarily. So, the fluid box is set to translate in space by constantly following the center of gravity (CG) of the blade, which thus remains constantly embedded in it. Appropriate inlet and outlet boundary conditions are set on the walls of the fluid box.

Fig. 5 shows an example of this methodology using a relatively coarse fluid mesh in the moving box. The center of the fluid box constantly tracks the CG of the structure. Thus one is able to simulate large translation and a complete rotation of the blade within the moving fluid box at a relatively low computational cost, much lower than by discretizing the entire fluid domain along the trajectory of the blade.

Fig. 5 ALE GLOB approach for a blade embedded into a moving fluid volume (with FSI).

Of course, in order to improve the accuracy of aerodynamic forces computed on the structure (which in turn controls the accuracy of the trajectory) it is advisable to activate FSI-driven mesh adaptivity (FLSW ADAP) in the fluid box. This is feasible in practice, thanks to the small extent of the meshed fluid domain. An example of application of this methodology to the CROR problematic is given in Section 4.

3 Wind tunnel tests and numerical simulations

The objective of the wind tunnel test campaign was to build up a static, sub-sonic aerodynamic database over a large range of angles of attack α and side slip angles β . This database should then be used to validate the numerical simulations. The L2 wind tunnel (ONERA facility) was used, following successful recent testing on light aircrafts [20]. A 1:1.2 scale model of a blade was built and mounted on a support strut fixed on a six-component balance measuring the forces. Pressure was also measured on the scaled model, equipped with

40 pressure transducers. The repeatability analysis concerning lift and drag coefficients was considered satisfactory.

3.1 Configuration and flow conditions

This section sets the definitions and notations concerning the static aerodynamic forces exerted on CROR blade fragments. The geometry of the rear and front blades is shown in Fig. 6. Only the front blade is used for the present study.

Fig. 6 Rear and front blades.

Some tests in wind tunnel are performed and then numerically simulated. Due to confidential aspects only the numerical results (aerodynamic simulations) are presented. We shall consider the frame shown in Fig. 7(a). The angle of attack and the side-slip will be defined regarding the blade frame # 1, with the standard definitions for an aircraft. All dimensions in this part of the paper are given at the scale 1. Naturally, for the tests in the wind tunnel, the surfaces and the chosen reference lengths are scaled like the model (1/1.2).

In the present paper, only the PORBR 100 (no-root blade, Fig. 6) is studied to compare numerical methods. The reference surfaces and lengths are defined as follows (see Fig. 7(b)):

- The reference surface S_{ref} is the projection of the area of the full blade in the X_{B1} - Y_{B1} plane, that is 0.543974 m^2 at full scale.
- The reference length L_{ref} is the radius of the front propeller: 2.1336 m .
- The moment is calculated at the point O_{B1} .

With the definitions of Fig. 8, the normalized vector coordinates which give the direction of the wind are, in the blade referential B_1 : $(\cos \alpha \cos \beta, \sin \beta, \sin \alpha \cos \beta)$ (see also [21]). The wind referential A (O_{B1}, X_A, Y_A, Z_A) is defined

Fig. 7 Definition of the standard and Airbus frame and of the blade referential B_1 .

Fig. 8 Definition of the angle of attack and of the side-slip angle.

Table 1 Forces and moments notation.

Referential	$\vec{F}$	$\vec{M}$
Wind referential A	$F_{X_A}, F_{Y_A}, F_{Z_A}$	$-M_{X_A}, +M_{Y_A}, -M_{Z_A}$
Blade referential B_1	$F_{X_{B1}}, F_{Y_{B1}}, F_{Z_{B1}}$	$-M_{X_{B1}}, +M_{Y_{B1}}, M_{Z_{B1}}$

from the blade referential B_1 with a rotation of α (angle of attack) around the Y axis, and a rotation of β (side slip angle) around the Z axis, so that the direction X_A gives the direction of the wind.

The incident flow, whose far-field velocity and density are V_∞ and ρ_∞ , produces a force $\vec{F}$ and a moment $\vec{M}$ around the point O_{B1} on the blade fragment. The components of $\vec{F}$ and $\vec{M}$ are noted as indicated in Table 1.

Table 2 Force coefficients notation.

Referential	$\vec{C}_F$	$\vec{C}_M$
Wind referential A	C_{XA}, C_{YA}, C_{ZA}	C_{LA}, C_{MA}, C_{NA}
Blade referential B_1	$C_{XB1}, C_{YB1}, C_{ZB1}$	$C_{LB1}, C_{MB1}, C_{NB1}$

Table 3 Flight configurations for comparison between tests, elsA and EPX.

Mach M_∞	Reynolds Re	Density ρ_∞ [kg/m ³]	Temperature T_∞ [K]	Static pressure p_∞ [Pa]	Dynamic viscosity μ_∞ [Pa·s]	Sound speed a_∞ [m/s]	Velocity V_∞ [m/s]
0.490	4.4×10^6	1.164	303.2	101 315	18.7×10^{-6}	349	171
0.853	2.8×10^6	0.380	218.8	23 843	14.4×10^{-6}	297	253

The minus signs on the rolling and yawing moments allow to retrieve the standard aircraft conventions: the rolling moment is positive when the right wing goes down, the pitching moment is positive when the nose goes up, and the yawing moment is positive when the nose goes to the right. The force coefficients are defined in Table 2.

$$C_{XA} = \frac{2F_{XA}}{\rho_\infty V_\infty^2 S_{\text{ref}}} \quad C_{YA} = \frac{2F_{YA}}{\rho_\infty V_\infty^2 S_{\text{ref}}} \quad C_{ZA} = \frac{2F_{ZA}}{\rho_\infty V_\infty^2 S_{\text{ref}}} \quad (3)$$

$$C_{LA} = \frac{2M_{XA}}{\rho_\infty V_\infty^2 S_{\text{ref}} L_{\text{ref}}} \quad C_{MA} = \frac{2M_{YA}}{\rho_\infty V_\infty^2 S_{\text{ref}} L_{\text{ref}}} \quad C_{NA} = \frac{2M_{ZA}}{\rho_\infty V_\infty^2 S_{\text{ref}} L_{\text{ref}}} \quad (4)$$

Similar notations are used also for C_{XB1} , C_{YB1} etc. The variation of force coefficients is due to the non-dimensional numbers which characterize the aerodynamic conditions:

- Angle of attack α and side-slip angle β .
- Far-field Mach number $M_\infty = V_\infty/a_\infty$ with V_∞ the incoming fluid speed and a_∞ the incoming sound speed.
- Reynolds number $Re = \rho_\infty V_\infty L_{\text{Re}}/\mu_\infty$ where L_{Re} is a reference length and ρ_∞ , μ_∞ are the incoming fluid density and viscosity, respectively. For this study, the length L_{Re} is the chord value, so $L_{\text{Re}} = 0.417$ m.

The quantities S_{ref} and L_{ref} appearing in eqs. (3) and (4) are a reference surface and length, respectively, previously defined in Section 3.1. Sets of elsA simulations are performed at $M_\infty = 0.490$ and $M_\infty = 0.853$, respectively, by varying α and β and are then compared with the experimental results in [21].

3.2 Numerical models for wind tunnel simulations

Fig. 9 shows the computational model used for the wind tunnel simulation. In the elsA CFD simulations, only the fluid was included and therefore the presence of the structure was treated as a rigid boundary. The EPX simulations,

instead, included both the fluid and the structure, treated as a deformable body. Brick volumes (FV) are used to represent the air and 4-node shells (FE) for the structure (in EPX). The structure material characteristics are very close to AIRBUS data (mass, position of the center of gravity, modal analysis). Due to confidential aspects these data are not included in this paper. The following conditions are used at fluid boundaries:

- At the inlet, flux is defined by imposing the velocity measured in the wind tunnel. Density and energy are imposed as constants, while continuity is imposed for pressure.
- At the outlet, continuity is assumed for all variables except pressure, which is imposed. This results in an absorbing boundary condition, which avoids (or greatly reduces) spurious pressure wave reflections at the outlet. A value for sound speed and a typical relaxation length are provided, which must be greater than the largest wave length of interest.
- The lateral walls are treated as infinite-volume boundaries.

The correctness of flow conditions around the structure (no perturbation at the fluid boundaries) is checked *a posteriori* against the experimental observations.

A regular uniform fluid mesh of 50 mm cell size is used for the air domain. For the structure used in the EPX model, the element sizes are between 50 mm and 80 mm. Recall in fact that the FLSW model used for FSI in these EPX simulations requires a fluid mesh finer than the structure mesh. No adaptivity is used in the EPX simulations of this experiment.

Fig. 9 Numerical model for the wind tunnel simulation.

3.3 Numerical results and comparison between EPX and elsA

For different angles of attack α and a chosen side-slip angle ($\beta = 0$), so-called polar curves are obtained. A comparison of force *vs.* angle of attack at fixed side-slip angle β and Mach number M (polar curves) as computed by the elsA and EPX codes is presented in Figs. 11 to 13. For confidentiality reasons the numerical values of force coefficients are not shown.

The elsA code was validated in [22,23], where it was shown that the elsA simulations are accurate in predicting force and pressure within 10 % of the experimental data (Fig. 10), and even better at low and high angles of attack. Overall, good accuracy is obtained from the EPX simulations compared with elsA simulations.

Fig. 10 PORBR 100 (no-root blade) model during wind tunnel tests.

Fig. 11 Comparison between EPX and elsA for $\beta = 0$, $M = 0.490$.

Fig. 12 Comparison between EPX and elsA for $\beta = 0$, $M = 0.853$.

3.4 Influence of the fluid mesh size

Fig. 13 compares simulations performed by varying the (uniform) fluid mesh size in EPX (50 mm or 15 mm cell size) against the elsA solution as a reference.

We can observe that a refinement of the fluid mesh is necessary to obtain sufficiently accurate results. Unfortunately, in many practical applications it is not realistic to uniformly refine the mesh over the entire fluid domain. Precisely for this reason, adaptivity was introduced into EPX to automatically refine the fluid mesh only where necessary, i.e. near the structure, as detailed in Section 2.2.

Fig. 13 Influence of EPX fluid mesh refinement on the force coefficients: $\beta = 0$, $M = 0.853$.

4 Blade trajectory simulation

Some sample EPX blade trajectory simulations are finally performed by combining all the key models (FSI, fluid mesh adaptivity, ALE GLOB) recently implemented in the code and presented in Section 2. We assume that:

- The blade loss is due to an instantaneous rupture near the blade root.
- The blade loss occurs after a stable thrust of the CROR (in good agreement with AIRBUS data) has been reached.
- Only the front PORBR 100 blade is released for the cruise and take off configurations, which are shown in Fig. 14.

For the following discussion of results we consider the reference frame shown in Fig. 14: the positive X axis points from the engine towards the fuselage, the positive Y axis is directed vertically upwards and the positive Z

Fig. 14 CROS model: different pitch angles for the cruise and take off configurations.

axis is pointing forwards, i.e. in the direction of the flight. The simulations are performed up to 2.5 m of CG X-translational displacement.

Fig. 15 shows the computed longitudinal (Z) displacement of the blade center of gravity (CG) *vs.* the lateral (X) displacement of the CG as computed for the cruise and take off configurations, without and with fluid mesh adaptivity.

Fig. 15 Blade CG longitudinal *vs.* lateral displacement.

One would expect that, immediately after being detached from the root, the blade debris initially tends to move forward (positive Z displacement) with respect to the engine and the more so, the higher is the engine thrust. Only somewhat later would the blade be “overtaken” by the advancing aircraft (negative Z displacement).

We saw in Section 3.3 that the largest inaccuracy in the EPX polar curves for uniform and relatively coarse fluid mesh (Figs. 11 and 12) is essentially observed in the first instants of the blade-off event, in particular for the cruise con-

dition, due to the initial pitch angle (small angle of attack α). If activated, the automatic fluid mesh refinement provides more accurate aerodynamic forces and it should better describe the initial motion of the blade debris just after release.

In fact, in Fig. 15 we can observe that the predicted Z-motion of the blade debris CG is inaccurate (initially negative dZ/dX derivative), if the aerodynamic forces are not sufficiently precise (too coarse fluid mesh). The pink curve CG_FLSW_CRUISE is obtained for a coarse uniform fluid mesh of 50 mm. The black curve CG_ADAP_CRUISE corresponds to a mesh with adaptivity up to a minimum fluid mesh size of about 2 mm near the structure.

The same kind of curves is plotted also for the take-off configuration. One can see that with the important engine thrust typical of take off, an initial forward displacement (along the positive Z-axis) is indeed predicted, in agreement with physical intuition, but this phenomenon is not caught by the coarse mesh. The same effect is observed for the cruise configuration, albeit to a lesser extent due to the lower engine thrust characterizing this configuration.

These simulations have been carried out to study possible impacts in the fuselage during take off and cruise. See for example Fig. 16 where, however, no values are shown due to confidentiality of results. This type of calculations opens the possibility of studying shielding solutions, thanks to an accurate knowledge of possible impact locations on the fuselage and to the possibility of modelling mechanical effects (deformation, damage, failure) on the impacted structure with EPX in a fully coupled FSI simulation.

Fig. 16 Envelope trajectory with upper and lower possible impact on fuselage for two different configurations.

5 Conclusions

The paper focused on some key computational aspects to obtain accurate yet computationally efficient numerical simulations of blade debris trajectory after a postulated CROR engine failure, up to impact against the aircraft fuselage. An embedded-type FSI algorithm (FLSW) was used to couple the structure with the surrounding fluid and to compute the resulting aerodynamic forces. Automatic refinement (adaptivity) of the fluid mesh only near the structure (ADAP) was used to increase FSI accuracy. Finally, a technique to limit the extent of the actually discretized fluid domain to a box constantly following the embedded blade debris (ALE GLOB) drastically reduced the CPU cost.

The first results obtained show clearly that these approaches are well suited to the class of problems of interest. The error in computed blade deviation just after blade release in cruise conditions (when the angle of attack is small and the drag force is overestimated) is dramatically reduced. Thus one can expect a better prediction of the blade trajectory, in particular during the first instants after blade loss, as well as of the impact locations with the surrounding structure (the engine itself, the front and rear blades, the fuselage).

Finally, another advantage of the EPX code is to allow performing such FSI simulations with a deformable structure, which is generally not possible with purely CFD codes. Future work will focus on the blade debris trajectory taking into account the initial release conditions due for example to bird strikes. In parallel, new investigations of shielding materials (experimental tests and impact simulations) will be performed taking into account the initial impact conditions obtained in the present work.

Acknowledgements The authors would like to thank Clean Sky for trust and support in this project.

References

1. Brunet M., Lafage R., Aubry S., Junior A., Catros J.-Y. The Clean Sky programme: environmental benefits at aircraft level. 15th AIAA Aviation Technology, Integration and Operations Conference, Dallas, TX (2015).
2. Stuermer A., Akkermans R.A.D. Validation of Aerodynamic and Aeroacoustic Simulations of Contra-Rotating Open Rotors at Low-Speed Flight Conditions. 32nd AIAA Applied Aerodynamics Conference, Atlanta, GA (2014).
3. Vlastuin J., Dejeu C., Louet A., Talbotec J., Lepot I., Lonfils T., Leborgne M. Open Rotor Design Strategy: From Wind Tunnel Tests to Full Scale Multi-Disciplinary Design. ASME Turbo Expo, Montreal, Quebec, Canada, June 15-19 (2015).
4. Brandvik T., Hall C., Parry A.B. Angle-of-Attack Effects on Counter-Rotating Propellers at Take-Off. ASME Turbo Expo, Copenhagen, Denmark, June 11-15 (2012).
5. Vion L., Delattre G., Falissard F., Jacquin L. Counter-Rotating Open Rotor (CROR): flow physics and simulation. 20^{ème} Congrs de Mcanique, Besanon, August 29 - September 2 (2011).
6. Sturmer A., Akkermans R.A.D. Multidisciplinary analysis of CROR propulsion systems: DLR activities in the JTI SFWA project. CEAS Aeronautical Journal, Volume 5, issue 3, pp 265-277, September (2014).

7. Negulescu C. Airbus AI-PX7 CROR design features and Aerodynamics. SAE Int. J. Aerosp. 6(2):626-642 (2013).
8. Remacha M., Martino E. Large Damage Residual Strength Analysis: Simulations and Tests. ECCM16 - 16th European Conference on Composite Materials, Seville, Spain, 22-26 June (2014).
9. Seng S., Frankenberger C., Ruggeri C.R., Revilock D.M., Pereira J.M., Carney K.S., Emmerling W.C. Dynamic Open-Rotor Composite Shield Impact Test Report. NASA/TM, DOT/FAA/TC-15/22 (2015).
10. Carney K., Pereira M., Kohlman L., Goldberg R., Envia E., Lawrence C., Roberts G., Emmerling W. Weight Assessment for Fuselage Shielding on Aircraft with Open-Rotor Engines and composite blade loss. NASA/TM, DOT/FAA/TC-13/34 (2013).
11. EUROPLEXUS User's Manual. On-line version: <http://europlexus.jrc.ec.europa.eu>.
12. Casadei F., Larcher M., Leconte N. Strong and weak forms of a fully non-conforming FSI algorithm in fast transient dynamics for blast loading of structures. COMPDYN 2011 Conference, Corfu, Greece, May 25-28 (2011).
13. Casadei F., Leconte N. FLSW: A weak, embedded-type Fluid-Structure interaction model with CCFV in EUROPLEXUS. Technical Note JRC65826 (2011).
14. Beccantini A., Casadei F., Galon P. Improvement of the FLSW model for Cell-Centered Finite Volumes in EUROPLEXUS. Report DEN/DANS/DM2S/STMF/LATF/NT/13-019/A (2013).
15. Casadei F., Díez P., Verdugo F. An algorithm for mesh refinement and un-refinement in fast transient dynamics. Int. J. Comp. Meth. 10 (4), p. 1350018-1-1350018-31 (2013).
16. Casadei F., Valsamos G., Beccantini A. Combination of mesh adaptivity with Fluid-Structure Interaction in EUROPLEXUS. EUR Report 26617 EN (2014).
17. Ortiz R., Delsart D., Sobry J-F. CROR WP1 Blade structure and certification. Technical Report CROR, R131, R132, R133, RT 13/14407 DAPP/DADS (2010).
18. Ortiz R. JTI Clean Sky SFWA-ITD WP 2.2.2.8. PBR-Propeller Blade Kinematics AI-PX4 Blade Trajectory: Study 2012-1. RT 6/19592 DAAP/DADS (2012).
19. Ortiz R. JTI Clean Sky SFWA-ITD WP 2.2.2. Task 2.2.2.8, PBR Propeller Blade Kinematics, AI-PX6 Blade trajectory with ALE GLOB approach. Report RT/7/20894 DAAP/DADS (2013).
20. Mouton S., Rantet E., Gouverneur G., Verbeke C. Combined wind tunnel tests and flow simulations for light aircrafts performance prediction. AAAF 47th International Symposium of Applied Aerodynamics, Paris, 26-28 March (2012).
21. Mouton S. CFD simulations of the flow over open rotor blade fragments. RT 1/20878 DAAP (2012).
22. Mouton S., Bourez J-P., Verbeke C. Wind tunnel testing of open rotor blade fragments. RT 5/20878 DAAP (2013).
23. Ortiz R., Mialon B. Open Rotor Blade Fragment Trajectory Testing and simulation synthesis Report. RT 6/20878 DAAP (2013).