34th European Rotorcraft Forum, September 16th-19th, 2008, Liverpool, UK
Aerodynamic Session

NUMERICAL ANALYSIS OF ADVANCED-INNOVATIVE TILTROTOR CONFIGURATIONS

Alessandro Scandroglio
Aerodynamic Specialist
AgustaWestland, Aerodynamics Dept.
alessandro.scandroglio@agustawestland.com

Andrea D'Andrea
Senior Aerodynamic Specialist
AgustaWestland, Aerodynamics Dept.
andrea.dandrea@agustawestland.com

Stefano Melone
Aerodynamic Specialist
AgustaWestland, Aerodynamics Dept.
stefano.melone@agustawestland.com

ABSTRACT

For the last past years the European Community has been co-founding several research programs aimed at validating and developing a new generation of tiltrotor in Europe based on the ERICA concept proposed by Agusta [1]. ERICA (*Enhanced Rotorcraft Innovative Concept Achievement,* Figure 1) belongs to a *second-generation* tiltrotor architecture, which has as innovative key points the capability to tilt the outboard portion of the wings (located in the slipstream of the rotors), and in its minimum rotors' diameter.
For this kind of innovative configurations, the determination of the right positioning of the external-tiltable wing during the passage from *helicopter* to *airplane mode* surely represents a critical issue. During this phase, indeed, it is mandatory to understand the interactional aerodynamic phenomena such as rotor wake impingement, flow field distortion, and the possible generation of large unsteady separated flow areas on the wings [2], that can dramatically reduce tiltrotor performance and stability. The unsteady aerodynamic interaction between rotating wings, their trailed wakes and tiltrotor fuselage plays an important role even for internal acoustics and aircraft vibratory levels in several ways.

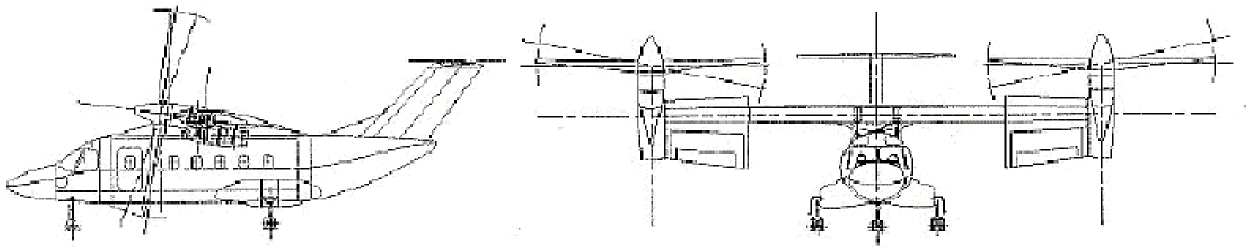

Figure 1 ERICA Tiltrotor General layout

Thus, during the past years several research programs such as TILTAERO [3]-[4] were led with the specific goal to develop a common European experimental database able to cover the main aerodynamic interactional phenomena arising during the different flight conditions of a tiltrotor aircraft based on the ERICA innovative concept. The TILTAERO test campaign, carried out in 2006 in the ¾-open test section of the DNW-LLF Wind Tunnel, covered a wide range of the future tiltrotor ERICA flight-envelope, from hovering to descent flight conditions (in helicopter mode), to forward-flights (in propeller mode). A 40% mach-scaled half-span model was designed, manufactured and tested in order to provide a detailed investigation of the aerodynamic interactional phenomena arising in different tiltrotor flight conditions, with the aim to provide high quality wind tunnel test data to increase the designer's knowledge about aircraft efficiency in this field. A brief summary of the experimental tests carried out during the TILTAERO Project is reported in the next Figure 2.

The present paper is focused on numerical analysis of advanced-innovative tiltrotor configurations and summarizes part of the whole activity performed by AgustaWestland Aerodynamics Dept. in the framework of TILTAERO and NICETRIP [5] research programs. Numerical simulations reported in this paper have been carried out by means of an advanced unsteady panel code, ADPANEL [6]-[7], and the Navier-Stokes code FLUENT [8].
ADPANEL is a Three-Dimensional Full-Unstructured Panel code coupled with a Time-Stepping Full Span Free Wake Vortex model. This tool implements the most advanced aerodynamic features in the field of potential methods, such as the possibility to represent the bodies' surfaces into unstructured-hybrid meshes, for a Constant Vorticity Contour (CVC) modeling of both rotary and fixed wing wakes, and for its Multi-Processor (MPI) implementation. Thanks to the previous features this tool is able to analyze in short computational times and with detailed prediction entire tiltrotor configurations, allowing to obtain accurate predictions of rotor performance and unsteady airloads on tiltrotor fuselage and wings. In this paper we will present the capability of the present tool to simulate innovative tiltrotor configurations.

A detailed numerical analysis of the experimental tests led during the TILTAERO Wind Tunnel test campaign is carried out; numerical results will space from rotor performance, pressure distributions along rotor blades, and unsteady airloads generated on tiltrotor wings and nacelle. In order to enhance as much as possible the capability of the present tool to correlate experimental data, a specific *interactional model* has been developed in ADPANEL in order to properly evaluate the wake impingement on the model surfaces.

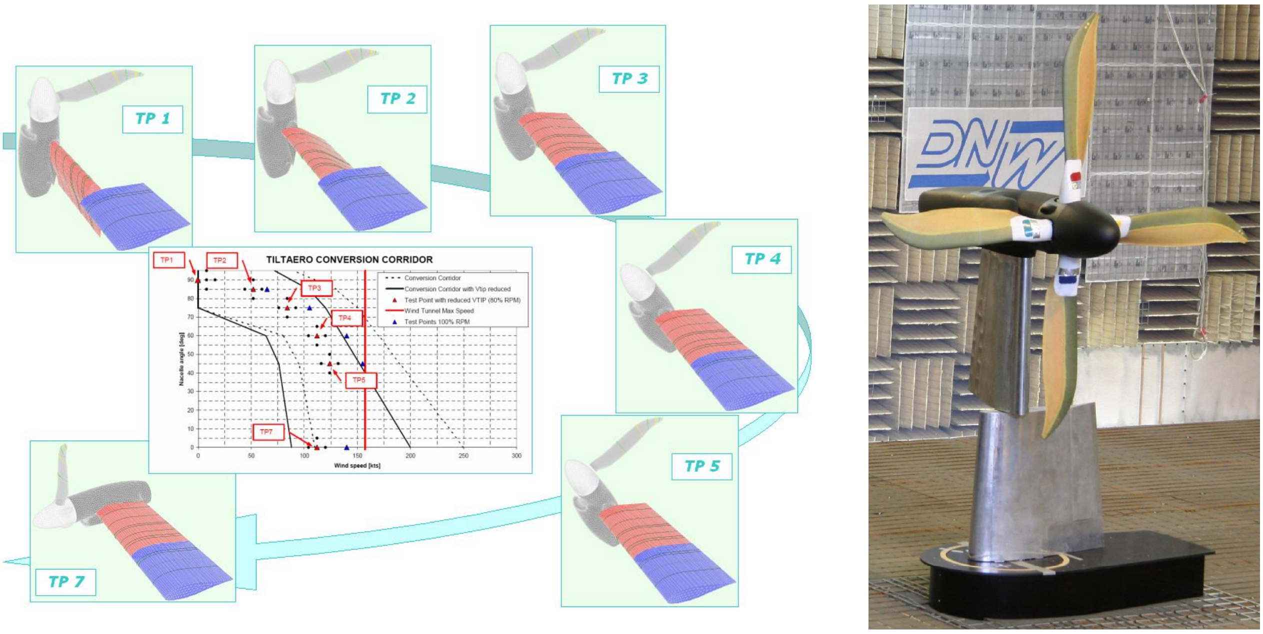

Figure 2 TILTAERO Conversion Corridor and experimental Test Matrix (left) – The TILTAERO half-span mock-up tested in the DNW Wind Tunnel (right)

FLUENT is a world-wide marketed and commercial numerical tool solving the Reynolds Averaged Navier-Stokes equations in three dimensions using a finite-volumes formulation on both structured and unstructured meshes. In this work unstructured triangular surface meshes and tetra volume meshes have been used coupled with a suitable and appropriate set of boundary conditions. Several turbulence models are available; for the present analyses the Wilcox's two-equation k-ω model with SST (shear-stress transport) correction has been chosen as the most accurate.
Navier-Stokes computations of the TILTAERO configuration can be performed, at first, in a *quasi-steady* mode by using an *actuator disk* to model the rotor. Due to this steady state assumption a great reduction in computational time could be achieved allowing in any case to obtain quite accurate predictions of the flowfield around fixed and movable wing. In particular, FLUENT capability to determine the separated flow areas on the outer-movable wing along the entire conversion corridor could be exploited. In the ambit of the actuator disk methodology both the uniform and the non-uniform actuator disk technique could be used.
A further enhancement of Navier-Stokes computations can be achieved by means of the *MRF (Multiple Reference Frame) Technique* that allows to take into account the real three-dimensional geometry of the rotor blade. This methodology can be used to better numerically describe the strong interaction occurring between rotor and wings in different flight conditions and to evaluate rotor performance in hovering.

DESCRIPTION OF THE ADPANEL CODE

Panel methods continue to be widely used for initial design studies on helicopters and tiltrotors due to their ease of use and rapid solution times. However, on the other side, most modern panel methods have several drawbacks in particular caused by the limit on the number of discrete elements allowed for a given simulation. ADPANEL is an in house potential flow solver with a multi-block iterative structure; it has a Multi-Processor implementation (based on simple MPI instructions), that makes this tool able to provide computational times further less than others common panel methods, requiring at the same time a minimum amount of computer RAM memory.

Basic formulation

Consider a body with known boundaries S_b submerged in a potential flow as shown in Figure 3. The flow of interest is in the outer region V where incompressible, irrotational, continuity equation (in the body's frame of reference) in terms of the total potential Φ^*, is given by the Laplace equation

$$\nabla^2\Phi^* = 0 \tag{1}$$

Following the Green's identity, the general solution to the previous equation can be constructed by a sum of *sources* σ and *doublets* μ placed on the boundary S_b, and can be expressed by

$$\Phi^*(x,y,z) = \frac{-1}{4\pi}\int_{S_B}\left[\sigma\left(\frac{1}{r}\right) - \mu\,\bar{n}\cdot\overline{\nabla}\left(\frac{1}{r}\right)\right]dS \quad + \quad \Phi_\infty \qquad (2)$$

Here the vector n points in the direction of the potential jump μ (which is normal to S_b and positive outside of V), and Φ_∞ is the freestream potential given by

$$\Phi_\infty = U_\infty x + V_\infty y + W_\infty z \qquad (3)$$

This formulation does not uniquely describe a solution since a large number of source and doublet distributions will satisfy a given set of boundary conditions. Therefore, an arbitrary choice has to be made in order to select the desirable combination of such singularity elements. It is well known that for simulating the effect of thickness, source elements can be used; whereas, for lifting problems, anti-symmetric terms (such as doublets or vortex rings) shall be utilized. In ADPANEL we simultaneously make use of both sources and doublets.

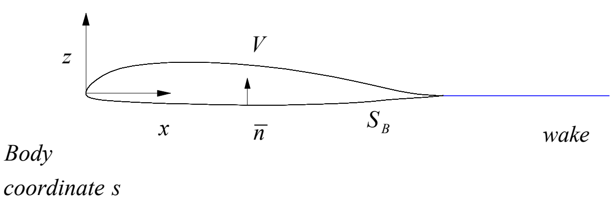

Figure 3 Potential flow over a lifting body

To uniquely define the solution of this problem, firstly the boundary conditions of zero flow normal to the surface S_b must be applied. In the general case of three-dimensional flows, specifying the boundary conditions will not immediately yield a unique solution because of the need to introduce some physical considerations in order to fix the amount of circulation around the surface S_b. These considerations mainly deal with the wake modeling and with fixing the wake-shedding lines and their initial orientation and geometry.
By modeling the wake with thin doublets or vortex sheets, therefore the Eq. (2) can be rewritten as:

$$\Phi^*(x,y,z) = \frac{1}{4\pi}\int_{body+wake}\left[\mu\,\bar{n}\cdot\overline{\nabla}\left(\frac{1}{r}\right)\right]dS \quad - \quad \frac{1}{4\pi}\int_{body}\left[\sigma\left(\frac{1}{r}\right)\right]dS \quad + \quad \Phi_\infty \qquad (4)$$

Boundary conditions

The boundary conditions needed for solving the previous equation, could directly specify a zero normal velocity component on the surface S_b (in which case this formulation is called *Neumann problem*). It is possible however to specify Φ^* on the boundary as well, so that indirectly the zero normal flow condition will be met; this formulation is known as *Dirichlet problem*.
It is important to underline that ADPANEL is based on the latter approach that was found to be more robust and computationally efficient [7].
Lamb demonstrated that the internal potential in the case of no-singularity inside the body has to be constant. In this case we can write the previous equation as:

$$\Phi_i^*(x_i,y_i,z_i) = \frac{1}{4\pi}\int_{body+wake}\left[\mu\,\bar{n}\cdot\overline{\nabla}\left(\frac{1}{r}\right)\right]dS \quad - \quad \frac{1}{4\pi}\int_{body}\left[\sigma\left(\frac{1}{r}\right)\right]dS \quad + \quad \Phi_\infty = const \qquad (5)$$

where $P_i = (x_i,y_i,z_i)$ is a generic point lying inside S_b.

Eq. (5) represents the *starting point* for solving the Laplace equation by means of a panel method based on Dirichlet boundary conditions. It can be demonstrated, that by setting the strength of the sources σ locally proportional to the freestream velocity component normal to the surface S_b, the much simpler form can be obtained:

$$\underbrace{\frac{1}{4\pi}\int_{body+wake}\left[\mu\,\bar{n}\cdot\nabla\left(\frac{1}{r}\right)\right]dS}_{Doublet} \quad - \quad \underbrace{\frac{1}{4\pi}\int_{body}\left[\sigma\left(\frac{1}{r}\right)\right]dS}_{Sources} \quad = 0 \qquad \textit{(6)}$$

where sources strengths are known and set by initial conditions, i.e.

$$\sigma = \overline{V}_\infty \cdot \bar{n} \qquad \textit{(7)}$$

Wake representation (cvc)

Since the flow is assumed to be inviscid, a Kutta condition is generally applied in boundary element methods at all wing/blade trailing edges.
This condition prescribes a jump discontinuity in the surface potential across the geometric cusp representing the trailing edges. In order to prescribe the streamwise vorticity released from wings and rotor blades, it is common to make use of the following relationship:

$$\left\{\Gamma_{TE} = 0 \quad \rightarrow \quad hence: \quad \mu_u + \mu_l + \mu_w = 0\right\} \qquad (8)$$

Here, the subscripts upper *(u)* and lower *(l)* refer to cells on the upper and lower surfaces of the trailing edge of the wing.

The wake modeling implemented in ADPANEL is composed of two parts: a *dipole buffer wake sheet*, and a set of *Constant Vorticity Contour (CVC) vortex filaments*. It is worth noting that even a more simple model, based on a buffer wake and a sequence of quadrilateral vortex rings, is in any case included in this tool (Vortex-Lattice wake model; see [7] for more details).

Buffer wake and CVC vortex filaments are used to represents the vorticity released from the wing for both its components, *trailed* and *shed* (respectively generated by spanwise and temporal variations of the bound circulation).
The buffer wake sheet is a *short dipole* (consisting in a sequence of doublets) trailing all the lifting surface and starting from its trailing edge. In the spanwise direction, the buffer wake is comprised of the same amount of panels as the wing trailing edge. This dipole is generated every time step and is converted (after the resolution of the Laplace equation) in CVC vortex filaments. Before the conversion, starting from the second time iteration, an equivalent vortex is generated along the confinement of the buffer region (Figure 4) in order to erase the not-balanced amount of circulation.
Other panel codes (e.g. FASTAERO) commonly use this approach to transform the vorticity trailed from wings in *vortex particles.*

The Constant Vorticity Contour wake modeling developed in the present tool, allows to generate refined roll-up and high spanwise resolution along the blade without enforcing an unnecessary large number of wake elements.

Recent and well-validated "vortex dissipation laws" have been implemented in ADPANEL in order to represent the increasing of the vortex core with the time passing. Further details with regard to this point can be found in [7].

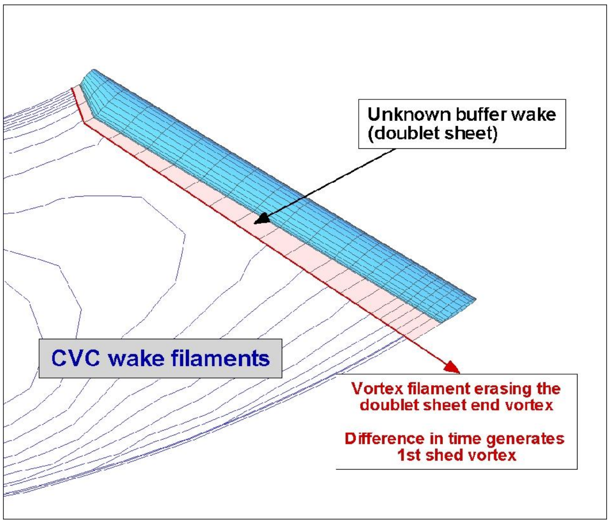

Figure 4 ADPANEL wake model – Buffer wake sheet + Constant Vorticity Contour vortex filaments

DESCRIPTION OF THE FLUENT MODEL

FLUENT SETTINGS

The steady 3D node-based segregated model has been chosen in FLUENT 6.3.26 for the simulation. An unstructured tetrahedral grid has been used coupled with a prism cell region for the boundary layer.
To take into account the turbulence effects, the k-ω SST model has been chosen. A test has also been done using the k-ε realizable model.

In order to achieve coherence and compatibility, the problem has been represented in FLUENT using the following boundary condition types:

- Pressure inlet used for the external volume inlet: it requires the total temperature and total pressure of the flow.
- Pressure outlet used for the external volume outlet: it requires the static pressure.
- Wall for all the solid surfaces (wing, blades).
- Symmetry for the remaining external volume surfaces (upper, lower, and side surfaces).

The following solution strategy (Figure 5) has been used:

1) first order solution until convergence;
2) adapt respect to the y+ in order to reduce its mean value; iterate until convergence;
3) adapt respect to the total pressure gradients; iterate until convergence;
4) second order solution until convergence.

The y+ and total pressure gradient adaptions were necessary in order to increase the stability and the accuracy of the solution.

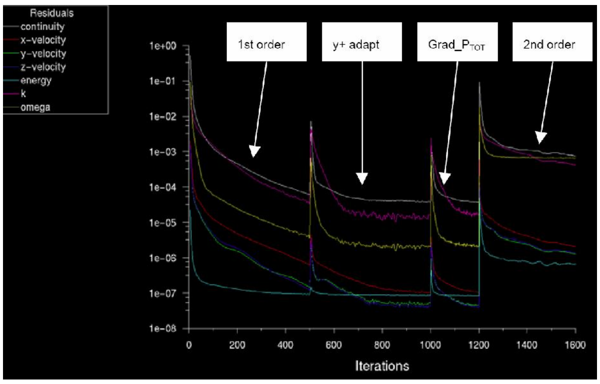

Figure 5 Residuals convergence history

The final y+ mean value was about 80 with a total number of cells of 7500000.

The boundary layer has been solved using the wall function approach whose validity depends on y+ value

$$y^+ = \frac{u_\tau y}{\nu}$$

The y+ range in which the wall function approach is valid is 50 ÷ 300.

The boundary layer region has been modeled by means of 5 prism cell layers (Figure 6).

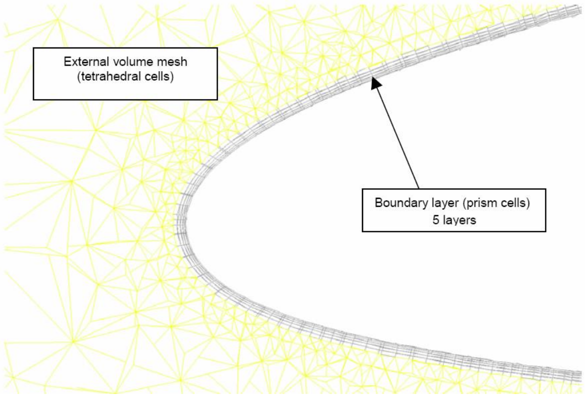

Figure 6 Boundary layer grid

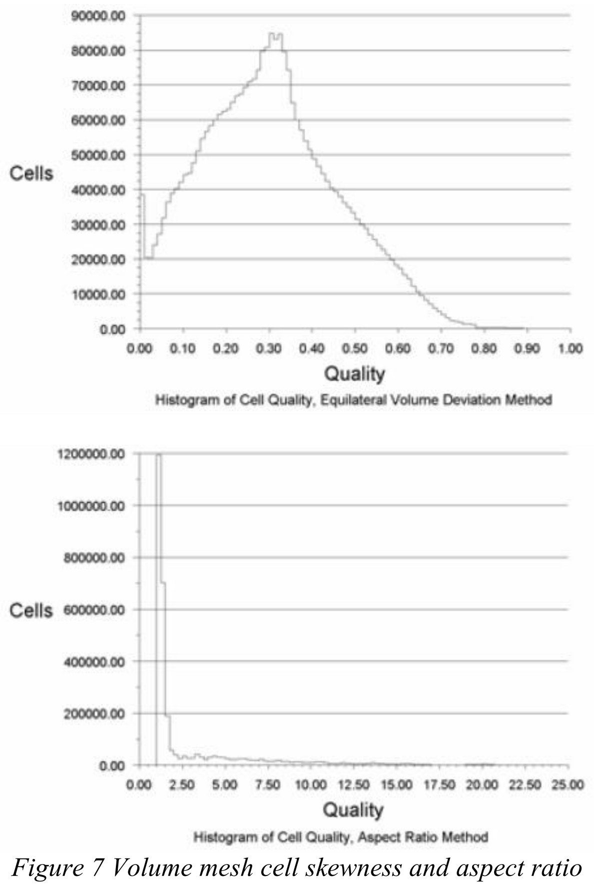

Figure 7 Volume mesh cell skewness and aspect ratio histogram

In order to simulate the mutual interference between the non-rotating and the rotating parts of an aircraft three different techniques could be used with different level of accuracy and effort in the model set-up:

1) Uniform actuator disk model;
2) Non-Uniform actuator disk model;
3) MRF technique.

The uniform actuator disk model assumes that the rotor will be replaced by an actuator disk (of the same area of the rotor disk) across which a constant (both in radial and azimuthal directions) pressure jump is obtained. The constant pressure jump across the disk could be evaluated starting from the experimental thrust value. The pressure jump, then, can be applied as boundary condition to the actuator disk using the *FAN* module available in FLUENT. This model allows to apply a constant or non-uniform pressure jump across a thin layer disk in order to simulate the fan effects.

The non-uniform actuator disk model assumes, instead, that the rotor will be replaced by an actuator disk across which a non-uniform pressure jump (both in radial and azimuthal directions) is obtained.
The non-uniform pressure jump could be evaluated starting from the thrust distribution along the blade span. The calculated pressure jump distribution can be assigned as boundary condition to the *FAN* module making use of an external "*.profile"* file.

The MRF *(Multiple Reference Frame)* technique, available in FLUENT, allows to completely model the rotating blades without the necessity to put them physically in rotation (dynamic mesh technique). In fact, using the MRF the blades are still while the rotational/translational speed is imposed.
The MRF technique is a steady-state approximation in which individual cell zones "move" at different rotational/translational speeds. Although the multiple reference frame approach is clearly an approximation, it has proven excellent numerical solutions of time-averaged flows for rotor applications.

No assumptions have to be made regarding the pressure jump across the disk but only the correct blade pitch angle has to be set.
Using this technique a more physical representation of the flow field around the rotating/non-rotating bodies can be obtained but it requires more CPU time during calculations than the actuator disk techniques and a lot of efforts in the model set-up and mesh generation.

All these techniques have been widely used and validated by the Aerodynamics Dept. of AgustaWestland for different types of rotorcraft.
The preliminary results hereafter reported are MRF technique results; ongoing and future activities are focused on further MRF analyses and actuator disk simulations to extend the validation of these techniques to ERICA configuration. This activity constitutes the preliminary phase for an accurate evaluation of the tiltrotor performances including viscous effects (wing download in helicopter mode, wing efficiency in forward flight). The use of a Navier-Stokes solver for these types of problems is the natural completion of the studies carried out with the panel method.

ANALYSIS OF FULL-AIRCRAFT INNOVATIVE TILTROTOR CONFIGURATION (ERICA)

For the last past years the European Community has been co-founding several research programs aimed at validating and developing a new generation of tiltrotor in Europe based on the ERICA concept proposed by Agusta [1].

ERICA (Enhanced Rotorcraft Innovative Concept Achievement) belongs to a second-generation tiltrotor architecture, which has as innovative key points the capability to tilt the outboard portion of the wings (located in the slipstream of the rotors), and in its minimum rotors' diameter. For this kind of innovative configurations, the determination of the right positioning of the external-tiltable wing during the passage from helicopter to airplane mode surely represents a critical issue.

During this phase, indeed, it is mandatory to understand the interactional aerodynamic phenomena such as rotor wake impingement, flow field distortion, and the possible generation of large unsteady separated flow areas on the wings [2] that can dramatically reduce tiltrotor performance and stability. The unsteady aerodynamic interaction between blades, their trailed wakes and tiltrotor fuselage plays an important role even for internal acoustics and aircraft vibratory levels in several ways.

Thus, during the past years several research programs such as TILTAERO [3]-[4] were led with the specific goal to develop a common European experimental database able to cover the main aerodynamic interactional phenomena arising during the different flight conditions of a tiltrotor aircraft based on the ERICA innovative concept.

The TILTAERO test campaign, carried out in 2006 in the ¾-open test section of the DNW-LLF Wind Tunnel, covered a wide range of the future tiltrotor ERICA flight-envelope, from hovering to descent flight conditions (in helicopter mode), to forward-flights (in propeller mode). A 40% mach-scaled half-span model was designed, manufactured and tested (Figure 8) in order to provide a detailed investigation of the aerodynamic interactional phenomena arising in different flight conditions, with the aim to provide high quality wind tunnel test data to increase the designer's knowledge about aircraft efficiency in this field.

Several numerical correlations have been performed by AW during the TILTAERO Research Program, in order to validate ADPANEL in case of aerodynamic analysis of demanding tiltrotor configurations.

Other numerical studies are at now going on by means of this tool within the European Research Program NICETRIP [5]. A full description of this work can be found in [4], whereas aim of this section is only to briefly show part of it.

All the numerical simulations described in [7] have been carried out in ADPANEL by using an unstructured-hybrid surface mesh of the entire TILTAERO model. More in detail (Figure 8) we made use of: *a)* a wall resembling the wind tunnel floor discretized by using 1,000 triangular panels; *b)* the model support, discretized with 1,825 triangular panels; *c)* the wing system, meshed with a structured-quadrilateral grid (60 chordwise × 100 spanwise); *d)* the nacelle, meshed with 5,814 triangular panels.

Each of the four Tiltaero blades have been represented by using 60 chordwise × 23 spanwise panels. Numerical simulations were carried out for both hovering, vertical cruise, descent flight conditions (in helicopter mode), and for medium-speed forward flight (in airplane mode).

In the next Figure 10, we represented the CVC wake development obtained with ADPANEL for TP7 airplane mode cruise condition. The strong restriction of the wake is quite evident, as well as the strong acceleration of the flowfield in the external part of the TILTAERO model movable wing.

Through Figure 11 to Figure 13 the comparisons between analytical results and experimental data show a quite good level of agreement even though looking at the unsteady pressure time histories the amplitudes and the frequencies are well captured while the phase suffers of some delay.

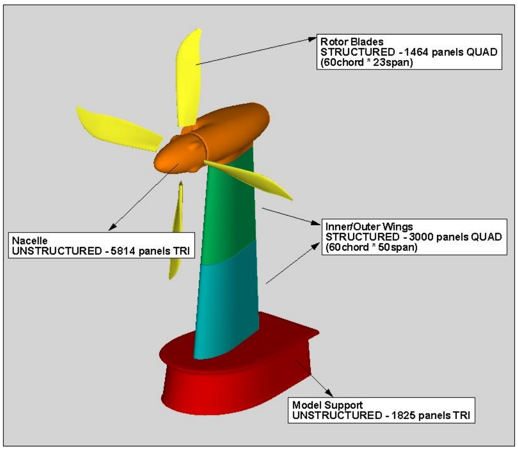

Figure 8 ADPANEL model of TILTAERO mock-up

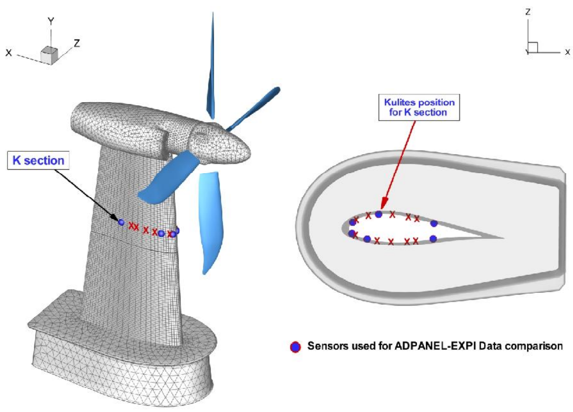

Figure 9 TILTAERO model unsteady pressure sensors (KULITES) location

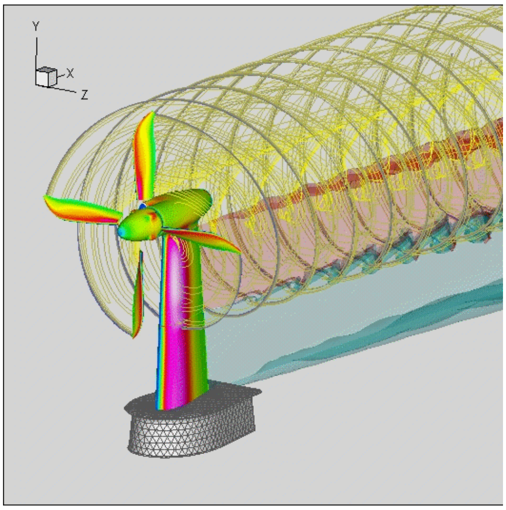

Figure 10 ADPANEL simulation of the TILTAERO TP7 test case - Airplane Propeller mode forward-flight

In the next figure (Figure 11) the wing spanwise distribution of lift (as normal force coefficient multiplied by the square of Mach number CnM²) has been reported; the comparison between the analytical lift distribution with (blue curve) and without (green curve) the rotor effect clearly shows the beneficial effect of the rotor downwash on the outer wing increasing the lift capabilities.

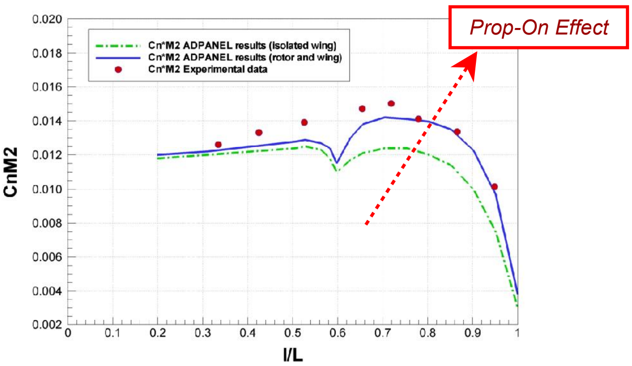

Figure 11 TILTAERO wing spanwise lift coefficient distribution - Test-case TP7

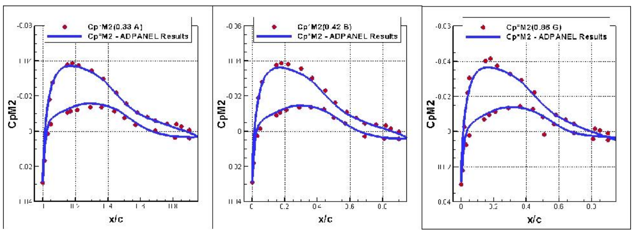

Figure 12 Comparison between numerical and experimental steady CpM2 along the outer-movable wing of the TILTAERO model - Test-case TP7

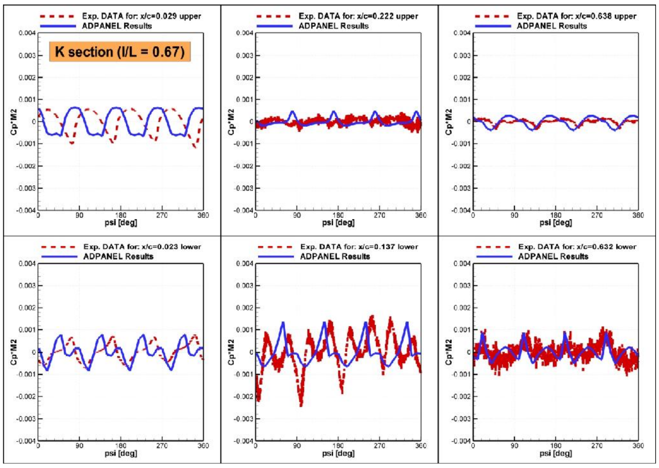

Figure 13 Comparison between numerical and experimental unsteady CpM2along the outer-movable wing of the TILTAERO model - Test-case TP7

In Figure 14, we depicted instead the CVC wake development obtained for the TILTAERO TP4 test case in helicopter mode (Figure 2).

As in the case of airplane mode, for the tested helicopter mode configuration we found the same good agreement between analytical results and experimental data (Figure 15 and Figure 16) with the same considerations on the unsteady pressure time histories.

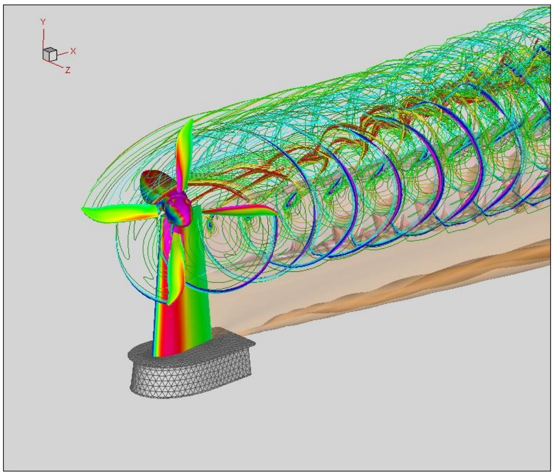

Figure 14 ADPANEL simulation of the TILTAERO TP4 test case - Helicopter mode forward-flight

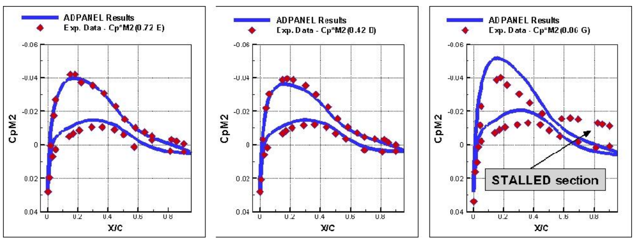

Figure 15 Comparison between numerical and experimental steady CpM2 along the outer-movable wing of the TILTAERO model - Test-case TP4

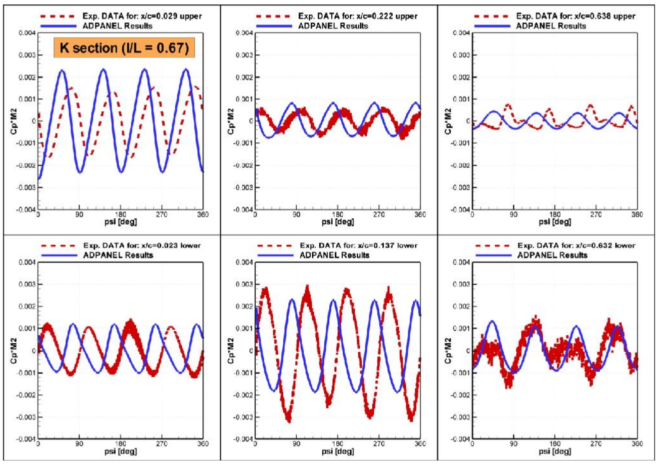

Figure 16 Comparison between numerical and experimental unsteady CpM2along the outer-movable wing of the TILTAERO model - Test-case TP4

In the next two figures () the FLUENT MRF results are shown for the initial phase of the TILTAERO convertion corridor (low speed forward flight in helicopter mode configuration).

The analytical simulation clearly shows the attached flow on the tiltable outer wing avoiding strong flow separations and reducing the download effects due to the rotor downwash. For these reasons the choice of a tiltable outer wing for tiltrotor applications is preferable in order to improve the performances in helicopter mode and the safety in the convertion corridor.

From the aerodynamics and performance point of view the Navier-Stokes simulations have been fundamental to verify the correct outer wing attitude.

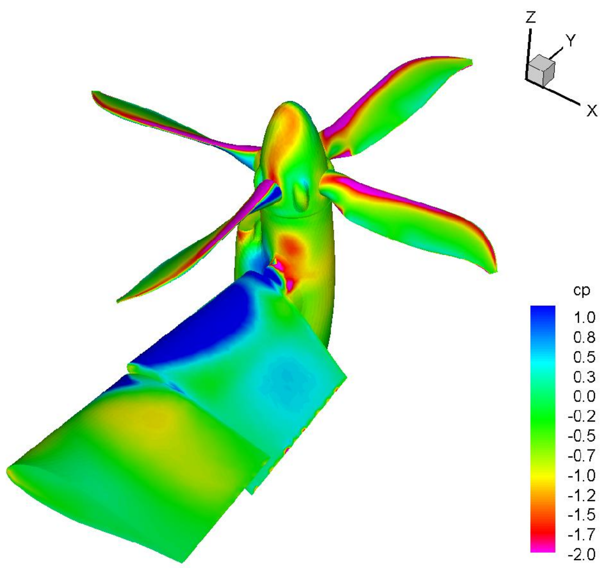

Figure 17 FLUENT MRF pressure coefficient distribution in the initial phase of the TILTAERO convertion corridor

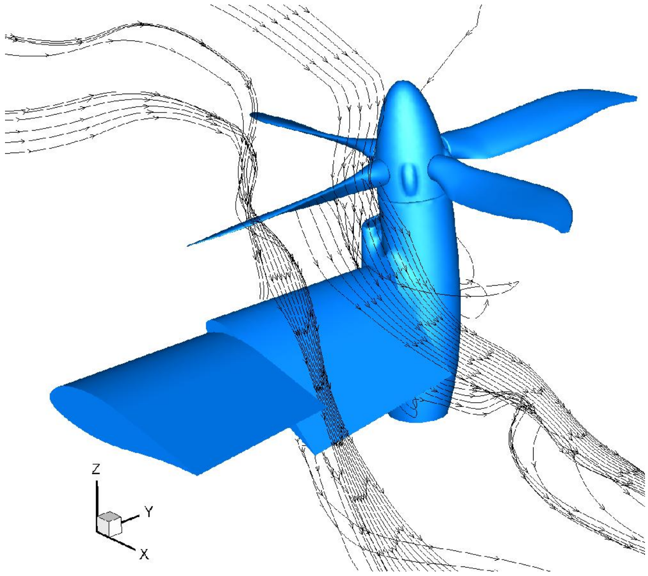

Figure 18 FLUENT MRF simulation of the TILTAERO model in low speed forward flight (initial phase of the TILTAERO convertion corridor) - streamlines

CONCLUSIONS AND FUTURE ACTIVITIES

Both an in house panel method code (ADPANEL) and a commercial Navier-Stokes solver (FLUENT) have been widely used in order to evaluate the performances of the TILTAERO configuration and compare the numerical results with the experimental ones. Some validation activities are still ongoing in order to cover all the conversion corridor but the preliminary results are very encouraging and satisfactory.

ERICA full aircraft configuration has been already developed (Figure 19) and some preliminary results are shown in Figure 20.

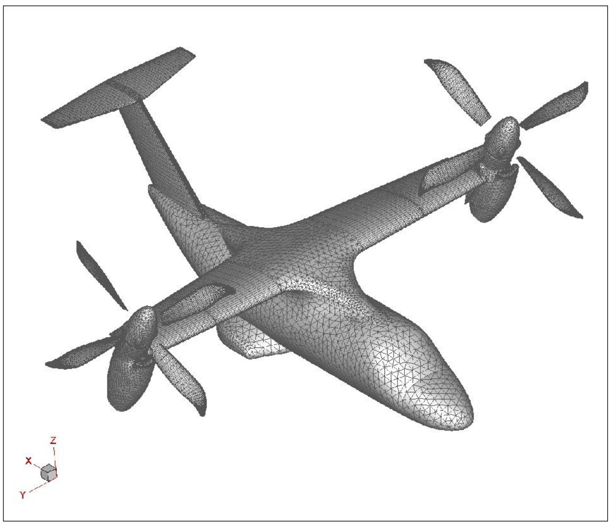

Figure 19 ERICA full aircraft configuration: surface mesh

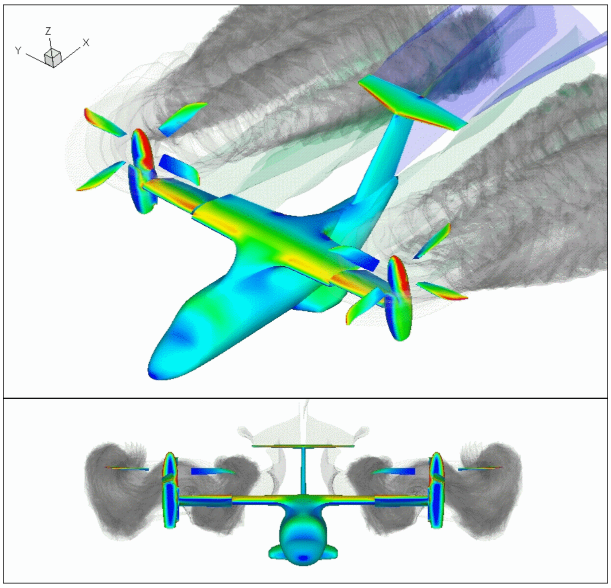

Figure 20 ERICA full aircraft configuration in forward flight

References

[1] Nannoni, F., Giancamilli, M., Cicalè, M., "ERICA: The European Advanced Tiltrotor", 27th European Rotorcraft Forum, Moscow, Russia, 11-14 September, 2001.

[2] Decours, J., Lefebvre, T., "Navier-Stokes Computations applied to Tilt-rotors", 33rd European Rotorcraft Forum, Kazan, 11-13 September 2007.

[3] Agusta S.p.A., "Tilt Rotor Interactional Aerodynamics – TILTAERO. Project Nr. GRD1 2000-25610: Annex I, Description of work", December 2000.

[4] Bianchi, E., Preatoni, G. and Dalla Rovere, N., "Mach scaled wind tunnel tests on a 4-bladed half-span advanced Tilt-Rotor", American Helicopter society (AHS) 63rd Annual Forum, Virginia Beach, VA, USA; May 1-3, 2007.

[5] VERTAIR, "Nicetrip – Integrated Project Reference Nr. 30944: Annex I – Description of work", October 2006.

[6] D'Andrea, A., "Development of a Multi-Processor Unstructured Panel Code coupled with a CVC Free Wake Model for Advanced Analyses of Rotorcraft and Tiltrotors", Accepted for publication at the American Helicopter Society 64th Annual Forum, Montréal, Canada, April 29 – May 1, 2008.

[7] D'Andrea, A., "ADPANEL – 3D Unsteady Full Unstructured Panel Code coupled with a Time Stepping Non-linear Free Wake Vortex Method", theory manual, AgustaWestland Report, N. 200-90-032/02.

[8] Fluent 6.3 User's Guide

[9] F. R. Menter, M. Kuntz, and R. Langtry. Ten Years of Experience with the SST Turbulence Model. In K. Hanjalic, Y. Nagano, and M. Tummers, editors, Turbulence, Heat and Mass Transfer 4, pages 625-632. Begell House Inc., 2003.

[10] D. C. Wilcox. Turbulence Modeling for CFD. DCW Industries, Inc., La Canada, California, 1998.