Fluid-Structure interaction framework based on structured RANS solver.

Davide Cinquegrana* and Pier Luigi Vitagliano

1CIRA, Italian Aerospace Research Center. 81043 Capua (CE), Italy.

E-mail: d.cinquegrana@cira.it

Abstract. A Fluid Structure Interaction framework developed at CIRA to deal with multi-physics problems in a partitioned approach is presented. The CIRA multi-block structured flow solver for unsteady Navier-Stokes equations UZEN was updated and tightly coupled with an open-source solver for non-linear structural dynamics in a modular approach. The solvers are glued in space and time through an open source library, able to control the executing processes and to deliver exchanging data by specific adapters. The validity of the framework is tested on vortex-induced vibration effects of a cantilevered beam in low Reynolds flow and on as three-dimensional wing flutter in transonic turbulent flow.

1. Introduction
The interaction of flexible structures in relative motion with fluids can generate static and dynamic instabilities, due to exchanging from fluid to structural elastic and inertial energy. The numerical capabilities in simulating those phenomena with high fidelity tools become fundamental in order to understand physical phenomena and to improve the final product. Fields of application range from rotorcraft dynamics to the project of new aircrafts, characterized by thinner, longer and lighter wings, designed to achieve higher efficiency and to save fuel. All configurations can be prone to structural instabilities like wing flutter, flow induced deformations and vibrations, that can affect aerodynamic stability, flight quality and durability [1],[2].

Partitioned approach for Fluid-Structure interaction (FSI) simulations exploits the high fidelity of solvers able to model non-linear features of structures and aerodynamics, respectively. Efficient communication among solvers, in terms of loads and deformations transfer and in terms of time synchronization, must address a wide range of topics, in order to guarantee stability and to avoid lagging phenomena that could generate spurious solutions [3]. During the last decade effort is grown in the development of tools offering coupling techniques among solvers, in forms like libraries or software interfaces [4],[5]-[9]. In this work, the validation of a new framework developed for FSI simulations is proposed. A multi-block structured flow solver[10] for unsteady Reynolds Averaged Navier-Stokes (URANS) equations has been upgraded with dynamic mesh capabilities and then coupled with an open source structural solver, capable of non-linear dynamics analyses[11], through the open source library PreCICE [4], that manages displacements and loads transfer together with time coupling. In this paper are proposed the analyses of vortex-induced vibration effects of a cantilevered beam in low Reynolds flow and three-dimensional wing flutter in transonic turbulent flow.
2. Aerodynamic Solver and Dynamic Mesh Implementation

UZEN is a multi-block structured flow solver developed at CIRA for the URANS equations with classical Arbitrary Lagrangian Eulerian (ALE) formulation\[^3\],[12]. The spatial discretization is based upon second order finite volumes central schemes cell-centred, with artificial dissipation. The unsteady computations are carried out by using a second order backward implicit time derivatives discretization; convergence in each time step is achieved with dual time stepping through explicit Runge-Kutta multi-stages and acceleration techniques like local time stepping, residual averaging and multigrid\[^13\]. CIRA has recently developed a technique \[^14\] which allows the non-conformal block to block coupling (i.e. sliding mesh boundary condition).

Within the frame of FSI studies the flow solver has been upgraded with the possibility to adopt dynamic meshes on block base (i.e. some specific blocks in the flow field can be deformed and updated at each time step). The concurrent use of dynamic meshes and non-conformal mesh coupling makes it possible the flow simulation of complex configurations. To deal with large displacements and keep grid quality, the mesh generation procedure can modify the blocks shape in a parametric way, following specific directives starting from a set of updated geometric entities like surfaces, curves and vertices. In order to allow for a strong-coupling in FSI problems, the current time step calculations can be repeated (and the updated mesh reloaded), when required, under control of external routines that check for suitable convergence criteria.

Dynamic meshes are implemented by following the 3 step backward implicit time scheme in such a way to satisfy the Discrete Geometrical Conservation Law (DGCL\[^2\]). The basic concept is that the scheme should not perturb a uniform flow when cells volume and shape change in time.

The transport equations of a generic flow variable \(\phi\) at any mesh cell is discretized as follows:

\[
\frac{3}{2} (\phi V)_{t+1} - 2(\phi V)_t + \frac{1}{2} (\phi V)_{t-1} + \Delta t \left[ \sum_i \phi (u + w) \cdot n S_i \right]_{t+1} - D_{t+1} = 0
\]

The apex \(t\) represents the time step where the flow variable \(\phi\) is computed in a cell which volume is \(V\), \(\Delta t\) is the time step. The first three terms on the left represent the time variation, summation in square brackets extends to all cell faces (six for a hexahedral cell) and represents the convective flux of \(\phi\) from the cell. It is made of two contributions: one depending upon flow velocity \(u\), the other is produced by the mesh apparent velocity \(w\) at the face \(i\). \(n\) is the face \(i\) unit normal outward vector, \(S_i\) is the \(i\)-th face surface area, \(D\) represents diffusive and source terms. When the flow is uniform (\(\phi\) and \(u\) are constant) the equation simplifies:

\[
\frac{3}{2} V_{t+1} - 2V_t + \frac{1}{2} V_{t-1} + \Delta t \left[ \sum_i w \cdot n S_i \right]_{t+1} = 0
\]
By setting the cell volume variation from time step \( t-1 \) to time step \( t \) (see Figure 1):

\[
\Delta V^t = V^t - V^{t-1} / \Delta t
\]

(3)

and the cell face \( i \) apparent volume fluxes at time \( t+1 \):

\[
W_i = w \cdot n S_i
\]

(4)

we get:

\[
\frac{3}{2} \Delta V^{t+1} - \frac{1}{2} \Delta V^t + \sum_i W_i = 0
\]

(5)

That is a relation useful to compute the mesh apparent fluxes at the cell face \( W_i \) from the cell volume variation in time. Boundary conditions at mesh external surfaces are applied by considering the meshes in all three-time steps, in order to keep the second order accuracy.

3. Structural Solver

The Finite Element solver CalculiX [11] is employed to carry out non-linear analysis of structural dynamic problems. The Finite Element Method solves the weak formulation of the balance momentum equation in material form, generally known as the principle of virtual work:

\[
\rho_s \frac{D^2 U}{Dt^2} + V \cdot \mathbf{P} + \rho_s f = 0
\]

(6)

where \( P \) is the first Piola–Kirchhoff stress tensor, \( f \) the force per unit mass, \( U \) the displacement vector and \( \rho_s \) is the solid density. Equation (6) is discretized as a set of linear algebraic equations over space:

\[
[K][U] + [M] \frac{D^2}{Dt^2} [U] = [F]
\]

(7)

where \([K]\) is the global stiffness matrix, \([M]\) is the global mass matrix and \([F]\) is the global force vector. For non-linear analysis they depend upon displacements and the equation is solved by subsequent iterations. The equation of motion is integrated in time using the \( \alpha \)-method [15]. This implicit scheme is unconditionally stable and second-order accurate.

4. Fluid - Structure coupling

In a partitioned approach for FSI, loads and deformations have to be transferred between fluid and structural domains. Unsteady simulations also require synchronisation of the two solutions. Particular care is taken to prevent sources of spurious momentum and energy by the interface scheme affecting the stability of the solution procedure and the properties of the aeroelastic system [16]. Main properties are consistency (meaning that constant values are interpolated exactly) and conservation (meaning that energy is globally conserved over the interface) [2].

While the structural solver CalculiX is already interfaced with preCICE [17], a specific adapter was developed for the aerodynamic solver UZEN. The adapter translates aerodynamic output in order to send forces on the relevant fluid-solid boundary surfaces and receives deformations data for remeshing the fluid domain. In our test cases loads and deformations are transferred using a radial basis function (RBF) method [18]. Basis functions adopted are thin plate spline (TPS) [19] and Gaussian spline.

It is possible to select between weakly (or explicit) and strongly (or implicit) coupled schemes. In weakly coupled schemes solvers are executed once for each time step; strong coupling rather requires both solvers to be executed multiple times with convergence checks. Typical motivations for using strong coupling are to obtain more accurate solutions and to overcome stability issues [20]. In order to stabilize and to accelerate strong coupling iterations between CFD and FEM solvers, postprocessing techniques are required to be applied at data exchanged. Among different techniques, we found the most
convenient to be the interface quasi-Newton inverse least square method (IQN-ILS) [21]. Convergence is evaluated by setting threshold on displacements and forces.

5. Results

5.1. Vortex Induced Vibrations
This test case was proposed in [22] to provide a numerical validation of coupled simulations in which large deformations play a key role. It presents a rigid cylinder square in a channel at Reynolds number 330, with attached an elastic cantilever. The channel walls are inviscid, no slip conditions are imposed to cylinder and cantilever walls. The material parameters are taken from [22]: elastic modulus $E = 2.5 \times 10^6 N/m^2$, Poisson's ratio $\nu = 0.35$ and density $\rho_s = 0.1 kg/m^3$. The fluid density is $\rho_f = 1.18 kg/m^3$, and the free stream velocity $u_\infty = 0.5 m/s$. The square side length $L = 0.01 m$.

The Von Karman vortex sheet, that starts at square corners and develops downstream, excites the flexible beam tip, that starts oscillating and tip displacement magnitude becomes comparable to the cylinder size.

Figure 2. Grid and dynamic blocks.
Many authors replicated the numerical test case [9], [22]-[31], [32]. Parameters considered to compare the results are the maximum tip displacement and its vibrating frequency. In reviewed papers average value of the maximum tip oscillation is 0.011 m. However, in results obtained with finer CFD grids and finer time resolutions, the maximum displacement decreases.

In the present work the fluid field is solved with a structured grid of 10 blocks. The effects of grid size are investigated with three levels of refinement: L1 grid contains 28160 cells, L2 contains 41344 cells and the finest L3 contains 112640 cells. FEM model of the cantilever is made with 650 nodes of quadratic elements.

A first simulation was carried out with rigid beam: a vortex frequency $f = 5.825 Hz$ was obtained. From modal analysis we computed the first and second cantilever mode frequencies: 3.15 Hz and 19.75 Hz, respectively. The coupled simulations started with uniform flow, no perturbations of structural loads and no initial displacements. It was observed that the beam starts to vibrate at frequency of about 6 Hz. This frequency disappears after a certain time, as the motion is dominated by the first frequency of the structure. A similar phenomenon was reported in [26].

By comparing solutions obtained with different grids and time steps, it can be observed that the main frequency is not affected, while the second frequency, relative to the vortex shedding, is better solved with the finer grids and the smaller time steps; furthermore, the second mode frequency of about 20 Hz is present only for the finer meshes with the smallest time step tested [Table 1]. When such frequency is resolved, the maximum oscillations amplitude decreases significantly. Those effects are reported also in [24] and [26]. Considering the solutions with time step of 0.01 s, it is possible to notice that the transient time to reach maximum level of oscillations is shorter with the finer grids.
**Figure 3.** Velocity magnitude contour plot from -0.7 to 1.6 m/s with 0.01 m/s steps. Left: maximum downward deflection. Right: maximum upward deflection. Grid L2.

**Table 1** Beam tip maximum displacement and main frequencies of tip vibration obtained with different grids and time steps.

| dt [s]   | Grid Level | avg $y_{\text{max}}$ [cm] | st.dev. $y_{\text{max}}$ [cm] | $f_1$ [Hz] | $f_2$, $f_3$ [Hz] |
|---------|------------|----------------------------|--------------------------------|------------|-----------------|
| 1.0E-03 | L1         | 0.690                      | 0.0525                         | 3.19       | 5.8             |
|         | L1         | 0.678                      | 0.0752                         | 3.19       | 5.8             |
| 2.0E-03 | L2         | 0.053                      | 0.0086                         | 3.21       | 6.2, 19.4       |
|         | L3         | 0.050                      | 0.0006                         | 3.27       | 6.18, 19.6      |
| 5.0E-03 | L1         | 1.015                      | 0.0525                         | 3.20       | -               |
|         | L2         | 0.818                      | 0.0549                         | 3.21       | 6.1             |
|         | L3         | 0.942                      | 0.0323                         | 3.19       | -               |
| 1.0E-02 | L2         | 1.043                      | 0.0065                         | 3.13       | 5.9             |
|         | L3         | 1.028                      | 0.0028                         | 3.18       | 5.94            |

**Figure 4.** Time history and FFT of cantilever tip oscillations. Gray line: 5.0e-3 s (dtf=0.25), black line: 1.0e-3 s (dtf=0.05).

5.2. **AGARD 445.6.**

Several authors considered the results of AGARD 445.6 wing test campaign [33],[34] to validate FSI solvers [35]-[46],[1],[9]. In this work the so-called weakened Model 3 is considered. It is a 45 degrees swept-wing with NACA 65A004 airfoil section, surface 0.353 m$^2$, root chord 0.559 m, taper ratio 0.66. The wing is modeled as an orthotropic material with the properties of mahogany, whose fibers are oriented in the wing sweep direction: elastic characteristics can be found in literature.
\[ V^* = U_\infty \frac{c_r}{0.5c_r \omega_2 \sqrt{\mu}} \] (8)

where \( c_r \) is the wing root chord, \( \omega_2 \) is the natural torsion frequency equal to 239 [rad/s], and \( \mu \) is the wing mass ratio equal to 225.82. The speed index was modified by keeping constant freestream flow variables like density and Mach number, changing static pressure \( p_\infty \) and temperature \( T_\infty \). Reynolds number was kept constant in the simulations performed.

To identify the flutter velocity and frequency, the damping coefficient of the maximum vertical displacement envelope, relative to the wing-tip leading edge, was evaluated by a non-linear fitting, assuming that the time-history takes an exponential form \([40],[41]\): \( w(t) = Ae^{-\xi t} + K \). Flutter onset velocity \( V_f \) is assumed when zero damping \( \xi \) is found. The simulations were performed with different time-steps, in order to verify the convergence in terms of flutter velocity: by reducing the time step, a decreasing flutter velocity is found.
6. Conclusions
The present study demonstrated the capabilities of a new framework developed at CIRA to simulate FSI problems characterized by non-linear instabilities, with a partitioned approach and implicit coupling.
Partitioned approach was adopted to take advantage of already existing codes: the in house structured multi-block CFD solver, UZEN, upgraded with the possibility to use dynamic meshes, and open source FEM structural solver Calculix, coupled through the PreCICE software library.

The computational framework was able to reproduce experimental results of AGARD 445.6 wing flutter at $M_\infty = 0.96$, adopting fully turbulent flow simulations, in agreement with other numerical results found in literature.

In the frame of vortex induced vibrations, high frequencies effects are well captured with regard to a viscous low Reynolds flow characterized by large deformations.

Further investigations of FSI phenomena and applications to other aeronautic configurations are foreseen in the next future.

References
[1] Farhat, C., and Geuzaine, P., “Design and analysis of robust ALE time-integrators for the solution of unsteady flow problems on moving grids,” Computer Methods in Applied Mechanics and Engineering, Vol. 193, No. 39, 2004, pp. 4073 – 4095.
[2] Farhat, C., Lesoinne, M., and LeTallec, P., “A conservative algorithm for exchanging aerodynamic and elastodynamic data in aeroelastic systems,” Aerospace Sciences Meetings, American Institute of Aeronautics and Astronautics, 1998, pp. –.
[3] Felippa, C. A., Park, K., and Farhat, C., “Partitioned analysis of coupled mechanical systems,” Computer Methods in Applied Mechanics and Engineering, Vol. 190, No. 24, 2001, pp. 3247 – 3270
[4] Bungartz, H.-J., Lindner, F., Gatzhammer, B., Mehl, M., Schuëfle, K., Shukaev, A., and Uekermann, B., “PreCICE – A Fully Parallel Library for Multi-Physics Surface Coupling,” Computers and Fluids, Vol. 141, 2016, pp. 250 - 258
[5] Kingsley, G., Jr., J. S., Harrand, V., Lawrence, C., and Luker, J., “Development of a multidisciplinary computing environment (MDICE),” 7th AIAA/USAF/NASA/ISSMO Symposium on Multidisciplinary Analysis and Optimization
[6] Joppich, W., and Kurschner, M., “MpCCI - A tool for the simulation of coupled applications,” Concurrency and Computation: Practice and Experience, Vol. 18, No. 2, 2006, pp. 183–192.
[7] Kataoka, S., Minami, S., Kawai, H., Yamada, T., and Yoshimura, S., “A parallel iterative partitioned coupling analysis system for large-scale acoustic fluid–structure interactions,”
Computational Mechanics, Vol. 53, No. 6, 2014, pp. 1299–1310.

[8] Duchaine, F., Jauré, S., Poitou, D., Quémerais, E., Staffelbach, G., Morel, T., and Gicquel, L., “Analysis of high performance conjugate heat transfer with the OpenPALM coupler,” *Computational Science and Discovery*, Vol. 8, No. 1, 2015, 015003.

[9] Thomas, D., Cerquaglia, M., Boman, R., Economon, T., Alonso, J., Dimitriadis, G., and Terrapon, V., “CUPyDO - An integrated Python environment for coupled fluid-structure simulations,” *Advances in Engineering Software*, Vol. 128, 2019, pp. 69 – 85.

[10] Marongiu, C., Catalano, P., and Amato, M., “U-ZEN: a computational tool solving U-RANS equations for industrial unsteady applications,” 34th AIAA fluid dynamics conference, Portland OR, 2004

[11] Dhondt, G., CalculiX CrunchiX USER’S MANUAL version 2.13, 08 2017

[12] Catalano, P., and Amato, M., “An Evaluation of RANS Turbulence Modeling for Aerodynamic Applications,” *Aerospace Science and Technology*, Vol. 7, No. 7, 2003, pp. 493–590.

[13] Jameson, A., “Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings,” 10th Computational Fluid Dynamics Conference, 1991, p. 1596.

[14] Vitagliano, P. L., Minervino, M., Quagliarella, D., and Catalano, P., “A conservative sliding mesh coupling procedure for U-RANS flow simulations,” *Aircraft Engineering and Aerospace Technology*, Vol. 88, No. 1, 2016, pp. 151–158.

[15] Dhondt, G., The Finite Element Method for Three-Dimensional Thermomechanical Applications, Wiley, 2004.

[16] Romanelli, G., Castellani, M., Mantegazza, P., and Ricci, S., “Coupled CSD/CFD non-linear aeroelastic trim of free-flying flexible aircraft,” Structures, Structural Dynamics, and Materials and Co-located Conferences, American Institute of Aeronautics and Astronautics, 2012, pp. –. https://doi.org/10.2514/6.2012-1562.

[17] Rusch, A., “Extending SU2 to fluid-structure interaction via preCICE.” Master’s thesis, Munich School of Engineering, Technical University of Munich, 2016.

[18] Rendall, T. C. S., and Allen, C. B., “Improved radial basis function fluids-structure coupling via efficient localized implementation,” *International Journal for Numerical Methods in Engineering*, Vol. 78, No. 10, 2009, pp. 1188–1208.

[19] Beckert, A., and Wendland, H., “Multivariate interpolation for fluid-structure-interaction problems using radial basis functions,” *Aerospace Science and Technology*, Vol. 5, No. 2, 2001, pp. 125–134.

[20] Kassiotis, C., “Nonlinear Fluid-Structure Interaction: a Partitioned Approach and its Application Through Component Technology,” Theses, Université Paris-Est, Nov. 2009.

[21] Degroote, J., Bruggeman, P., Haelterman, R., and Vierendeels, J., “Stability of a coupling technique for partitioned solvers in FSI applications,” *Computers & Structures*, Vol. 86, No. 23, 2008, pp. 2224 – 2234. https://doi.org/10.1016/j.compstruc.2008.05.005.

[22] Wall, W., and Ramm, E., “Fluid-structure interaction based upon a stabilized (ALE) finite element method,” *Computational Mechanics*. New Trends and Applications, 1998.

[23] Mehl, M., Ueckermann, B., Bijl, H., Blom, D., Gatzhammer, B., and van Zuijlen, A., “Parallel coupling numerics for partitioned fluid-structure interaction simulations,” *Computers & Mathematics with Applications*, Vol. 71, No. 4, 2016, pp. 869 – 891.

[24] Dettmer, W., and Peric, D., “A computational framework for fluid structure interaction: Finite element formulation and applications,” *Computer Methods in Applied Mechanics and Engineering*, Vol. 195, No. 41, 2006, pp. 5754 – 5779.

[25] Habchi, C., Russeil, S., Bougeard, D., Harion, J.-L., Lemenand, T., Ghanem, A., Valle, D. D., and Peerhossaini, H., “Partitioned solver for strongly coupled fluid-structure interaction,” *Computers & Fluids*, Vol. 71, 2013, pp. 306 – 319.

[26] Sanchez, R., Palacios, R., Economon, T. D., Kline, H. L., Alonso, J. J., and Palacios, F., “Towards a Fluid-Structure Interaction Solver for Problems with Large Deformations Within the Open-
Source SU2 Suite,” *AIAA SciTech Forum*, 2016, pp. –.http://dx.doi.org/10.2514/6.2016-0205.

[27] Matthies, H., and Steinendorf, J., “Partitioned strong coupling algorithms for fluid-structure interaction,” *Computers & Structures*, Vol. 81, 2003, pp. 805–812.

[28] Wood, C., Gil, A., Hassan, O., and Bonet, J., “Partitioned block-Gauss Seidel coupling for dynamic fluid-structure interaction,” *Computers & Structures*, Vol. 88, No. 23, 2010, pp. 1367–1382.

[29] Froehle, B., and Persson, P.-O., “A high-order discontinuous Galerkin method for fluid-structure interaction with efficient implicit-explicit time stepping,” *Journal of Computational Physics*, Vol. 272, 2014, pp. 455 – 470.

[30] Gilmanov, A., Le, T. B., and Sotiropoulos, F., “A numerical approach for simulating fluid structure interaction of flexible thin shells undergoing arbitrarily large deformations in complex domains,” *Journal of Computational Physics*, Vol. 300, 2015, pp. 814 – 843.

[31] Olivier, M., Morissette, J.-F., and Dumas, G., “A Fluid-Structure Interaction Solver for Nano-Air-Vehicle Flapping Wings,” 19th AIAA Computational Fluid Dynamics, 2009.

[32] Liu, F., Cai, J., Zhu, Y., Tsai, H. M., and F. Wong, A. S., “Calculation of Wing Flutter by a Coupled Fluid-Structure Method,” *Journal of Aircraft*, Vol. 38, No. 2, 2001, pp. 334–342.

[33] Yates, J. E., “Subsonic and Supersonic Flutter Analysis of a Highly Tapered Swept-Wing Planform, Including Effects of Density Variation and Finite Wing Thickness, and Comparison with Experiments,” Tech. Rep. NASA-TN-D-4230, NASA

[34] Yates, J. E., “AGARD standard aeroelastic configurations for dynamic response. 1: Wing 445.6,” 1988.

[35] Lee-Rausch, E., and Batina, J. T., “Calculation of AGARD wing 445.6 flutter using Navier-Stokes aerodynamics,” NASA Technical Memorandum 107732, NASA, March 1993.

[36] Melville, R., Morton, S., and D.P., R., “Implementation of a fully-implicit, aeroelastic Navier-Stokes solver,” 1997.

[37] Goura, G. S. L., “Time marching analysis of flutter using computational fluid dynamics,” Ph.D. thesis, University of Glasgow, 2001.

[38] Xiao, J., and Gu, C., “Wing Flutter Simulations Using an Aeroelastic Solver Based on the Predictor Corrector Scheme,” *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, Vol. 224, No. 11, 2010, pp. 1193–1210.

[39] Zhang, B., Ding, W., Ji, S., and Zhang, J., “Transonic flutter analysis of an AGARD 445.6 wing in the frequency domain using the Euler method,” *Engineering Applications of Computational Fluid Mechanics*, Vol. 10, No. 1, 2016, pp. 243–254.

[40] Pahlavanloo, P., and forskningsinstitut, T., Dynamic Aeroelastic Simulation of the AGARD 445.6 Wing Using Edge, FOI-R, Defence and Security, Systems and Technology, Swedish Defence Research Agency (FOI), 2007.

[41] Zhang, W., Lv, Z., Diwu, Q., and Zhong, H., “A flutter prediction method with low cost and low risk from test data,” *Aerospace Science and Technology*, Vol. 86, 2019, pp. 542–557.

[42] Gordnier, R. E., and Melville, R. B., “Transonic Flutter Simulations Using an Implicit Aeroelastic Solver,” *Journal of Aircraft*, Vol. 37, No. 5, 2000, pp. 872–879. https://doi.org/10.2514/2.2683

[43] Xiangying, C., Ge-Cheng, Z., and Ming-Ta, Y., “Numerical simulation of 3-D wing flutter with fully coupled fluid–structural interaction,” *Computers & fluids*, Vol. 36, No. 5, 2007, p. 856–867.

[44] Beaubien, R. J., Nitzsche, F., and Feszyt, D., “Time and frequency domain flutter solutions for the AGARD 445.6 wing,” Paper No. IF-102, IFASD, 2005.

[45] Silva, W. A., Chwalowski, P., and III, B. P., “Evaluation of linear, inviscid, viscous, and reduced-order modelling aeroelastic solutions of the AGARD 445.6 wing using root locus analysis,” *International Journal of Computational Fluid Dynamics*, Vol. 28, No. 3-4, 2014, pp. 122–139.

[46] Zhao, X., Zhu, Y., and Zhang, S., “Transonic wing flutter predictions by a loosely-coupled method,” *Computers & Fluids*, Vol. 58, 2012, pp. 45 – 62.
[47] Kok, J. C., “Resolving the Dependence on Freestream Values for the k- Turbulence Model,” *AIAA Journal*, Vol. 38, No. 7, 2000, pp. 1292–1295.