VI International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2015
B. Schrefler, E. Oñate and M. Papadrakakis (Eds)

A HYBRID LES/CAA METHOD APPLIED TO A 3D SHEAR FLOW SIMULATION

O. LABBÉ, C. PEYRET

ONERA - The French Aerospace Lab
F-92322 Châtillon, France
email: odile.labbe@onera.fr, www.onera.fr
email: christophe.peyret@onera.fr, www.onera.fr

Key words: Shear flow, Finite Volume method, Discontinuous Galerkin Method, CWIPI, LES/CAA

Abstract. An aero-acoustic coupling method based on the splitting into noise sources generation and acoustic propagation in separate physical domains is presented in this paper. The key idea is to limit, as much as possible, the CFD domain to the noise generation region and to accurately propagate the acoustic waves with a CAA solver. The approach presented here is a domain decomposition method based on the coupling of different equations, methods and meshes, which allows a simulation of both flow and acoustics in one single coupled calculation suitable for far field predictions with reflecting bodies through a coupling boundary.

1 INTRODUCTION

The numerical simulation of both generation and propagation of acoustic waves in one computation is actually a challenge due to the spatio-temporal scale disparities of the problem. The full time-dependent, compressible Navier-Stokes equations describe both aerodynamic and acoustic phenomena, but require a detailed numerical compressible flow simulation, using a grid fine enough to minimize the introduction of sound propagation errors. A large number of grid elements would be necessary to resolve all the scales accurately in an unsteady simulation. Moreover, the handling of long-distance sound propagation remains difficult with usual CFD solvers due to the numerical damping and dissipation. On the other hand, a single approach contains naturally the interaction of the acoustic perturbations with the flow field and with embedded geometries. The dissipation problems generally lead to carry out hybrid methods, in which the computational domain is split into different regions, such that the governing flow field (source region) or acoustic field (acoustic region) can be solved with different equations, numerical techniques and computational grids. Various hybrid methodologies exist, differing from each other in the type of applied propagation equations or in the way the coupling between source and propagation regions is made. Bailly and Bogey [1] proposed a review of the progress in the computational aeroacoustics field and discussed connections between CFD and CAA using hybrid approaches. The coupling methods commonly used for hybrid CFD/CAA applications are divided into two categories: one based on equivalent source formulations and the other based on an acoustic continuation of source region simulation. For the first one, once the sound source is predicted, the approach to describe its propagation is the extension of near-field CFD results to the acoustic far-field with surface or volume integral methods [2]-[6]. The second hybrid CFD/CAA approach solves the Acoustic Perturbation Equations [7]-[9] or the Linearized Euler Equations [10], [11] to extend the CFD solutions to the far-field. These propagation solvers are generally high order accurate but necessitate a mean flow definition. Generally these hybrid methods do not take into account any acoustic feedback, except for the domain decomposition performed by [12], [13], where the coupling approach connects different classes of methods on structured and unstructured grids for the solution of Navier-Stokes, Euler and linearized Euler equations. The approach presented in this paper is also a domain decomposition method based on the coupling of different equations, grids and methods, which allows a simulation of both flow and acoustics in one single calculation suitable for far field predictions with reflecting bodies. The CFD domain solving the Navier-Stokes equations is reduced to the region of viscous effects and initial turbulence development which generally accounts for a small part of the flow. The acoustic propagation is solved with the full non-linear Euler equations with a coupling boundary located in the turbulent flow. The acoustic solver is based on high order Discontinuous Galerkin schemes [14], which offer a high accuracy, low dispersion and low dissipation. This class of solvers is able to accurately propagate waves over large distances and allows using unstructured grids, which present significant advantages such as a highly flexible refinement even in complex geometries.

The coupling method presented in this paper consists in separating the whole computational domain into two complementary parts. The first domain is assumed to contain the region where turbulence develops and its size is reduced as much as possible. The flow in this domain is solved using the Navier-Stokes equations. The second domain is devoted to the propagation of the perturbations generated in the first one, in a flow which is not necessary uniform and may contain reflecting bodies. It may also include noise production, in which viscous effects can be negligible. The problem is solved using the full Euler equations. Such LES/CAA couplings pose, a priori, the problem of the continuity of the solution on both sides of the exchange surface. Indeed, the LES must be performed using sufficiently fine meshes and time steps small enough to correctly simulate the development of turbulence. On the other hand, to be efficient, the CAA must be carried out using meshes and time steps adapted to the acoustic scales which are generally much larger than the turbulence scales. So, in this approach the spatial scales will be very different on both sides of the coupling boundary, but the time step will be identical.

In a previous study [15], this splitting method was applied to a hot jet simulation, that only dealt with a one way coupling from LES to CAA and thus mainly focused on the acoustic purpose. Here, this splitting method is applied to a 3D shear flow. The computational domain is a 3D cylinder, which includes a smaller one, and can be considered as a model of jet/wing interactions. A shear flow (100m/s-50m/s) is imposed at the inflow of the small cylinder and a rigid wall conditions is imposed at the external boundary of the big cylinder, which induces wave reflections impacting on the shear flow. In this approach, both the LES and CAA solvers are part of the same simulation framework. While the LES solver is based on a finite-volume method for the prediction of the flow field, the CAA approach is based on a high-order discontinuous Galerkin solver for the acoustic field. The exchanges between both solvers are made through the MPI communications.

2 NUMERICAL METHODS

2.1 Finite-volume method for the flow simulation

The aerodynamic solver FUNk [16] developed at ONERA, is based on the unsteady compressible Navier-Stokes equations expressed in conservative form. The LES equations are obtained using Favre filtering and the filtered equations are closed by means of a subgrid scale viscosity and the Prandtl analogy. The model used to compute the subgrid viscosity is the selective mixed scale model introduced for compressible flows by Lenormand et al. [17]. The spatial discretization method is based on the cell-centered Finite Volume methodology (FVM) on structured grid. An upwind biased scheme, with a third-order MUSCL interpolation scheme of AUSM+(P) family without any shock capturing feature is used for the convective terms. A second-order-accurate centered scheme is used for viscous fluxes. The time integration is carried out by means of a third-order compact Runge-Kutta scheme. The solver has been extensively validated and used for various flow problems.

2.2 Discontinuous Galerkin approximation

The CAA computation is carried out with the SPACE solver developed at ONERA [18], [19] which solves the Euler equations using a nodal Discontinuous Galerkin Method. A nodal DGM with an optimized Lagrangian polynomials basis of order N+1 is used to solve the conservative form of Euler equations:

$$\partial_t w + \nabla . F(w) = 0 \tag{1}$$

Considering a domain Ω with an envelope Γ, the derivation of the DG method starts with the weak form of the equation. Therefore, Eq. 1 is multiplied by a test function Ψ and integrated over the domain Ω. A variational formulation of Euler equations reads:

$$\left\{ \forall \Psi \in V(\Omega); \quad \int_{\Omega} (\partial_t w + \nabla . F(w)) \ \Psi d\Omega = 0 \right\} \tag{2}$$

Subdividing the domain $\Omega = \bigcup_i \Omega_i$ and $\Gamma = \partial\Omega = \bigcup_j \Gamma_j$ and using integration by parts on the flux term, we obtain the weak formulations of the differential equations:

$$\left\{ \forall \Psi \in V(\Omega); \sum_i \int_{\Omega_i} \partial_t w \Psi d\Omega - \int_{\Omega_i} F(w) . \nabla \Psi d\Omega + \oint_{\partial\Omega_i|\Gamma} \widetilde{F}(w, w^+) n \Psi d\Gamma + \sum_j \oint_{\Gamma_j} F(w) n \Psi d\Gamma = 0 \right\} \tag{3}$$

where n is the surface normal vector in the reference system and $\partial\Omega_i$ is the envelope of geometric cell Ω_i. On the element boundaries $\partial\Omega_{/\Gamma}$, a numerical flux combines values w and $w+$ from both sides to a single flux. The Local Lax-Friedrichs Numerical Fluxes have been chose:

$$\widetilde{F}(w,w^{+})n = \tfrac{1}{2}(F(w)+F(w^{+}))n - \alpha\lambda_{\max}(w^{+}-w) \qquad \text{where } \alpha \in [0,1]$$

and $\lambda_{\max} = \max(|v.n|, |v.n+c|, |v.n-c|, |v^{+}.n^{+}|, |v^{+}.n^{+}+c^{+}|, |v^{+}.n^{+}-c^{+}|)$

with $c = \sqrt{\gamma p/\rho}$

The solution w is approximated using a polynomial basis N

$$w(\xi,t) \approx \sum_{i,j,k}^{N} \overline{w_{ijk}}(t)\varphi_{ijk}(\xi_{ijk}) \qquad \varphi_{ijk}(\xi) = l_i(\xi_i)l_j(\xi_j)l_k(\xi_k)$$

where the basis functions φ_{ijk} are the product of one-dimensional Lagrange polynomials l_i of degree N in each spatial direction and $\overline{w_{ijk}}(t)$) are the sampling of solutions at (x_i, y_j, z_k). The nodal basis is defined with a set of interpolation points $\{\xi_{i=0}^{N}\}$ on the interval ξ ∈ [−1, 1], which in this work are the Gauss-Lobatto nodes. The fluxes are approximated using the same approach.

The integrals in Eq. 3 are approximated by Gauss-Lobatto quadrature. Generally, Gauss quadrature of an arbitrary function f (x) on the interval [−1, 1], with N+1 nodes can be written as:

$$\int_{-1}^{1} f(x)dx = \sum_{i=0}^{N} \omega_i f(x_i) = \sum_{i=0}^{N} \omega_i \sum_{i=0}^{N} l_j(x_i) f_j = \sum_{i=0}^{N} \omega_i \; f_i \,,$$

because $l_j(x_i) = \delta_{ij}$, where the weights ω_i and the integration nodes x_i are specific to the chosen quadrature. These weights are pre-calculated and stored to make the algorithm efficient. With the interpolation points $\{\xi_i\}$ collocated at the Gauss nodes, all sums collapse into single values.

In the next step, the variational formulations are integrated in time to obtain the solution at the next time step, for which a low-storage three-order Runge-Kutta scheme is used, the same as the one used in the CFD solver.

The coupling from LES to CAA is achieved by a numerical integration along the coupling surface Γ_c :

$$\int_{\Gamma_c} F(w)n\Psi d\Gamma = 0$$

For each CAA cell along the coupling boundary, the integral is computed using the values of LES solution interpolated on Gauss points.

2.3 Coupling procedure

The CWIPI [20], [21] library aims at providing a fully parallel communication layer for mesh based coupling between several parallel codes with MPI communications. CWIPI is a static tool in the sense that all the components of the simulations are started at the beginning, exchange data during the run phase and finish together at the end. The coupling is made through 1D, 2D or 3D exchange zones that can be discretized in different ways in the coupled codes. The library takes into account all types of geometrical elements (polygon, polyhedral) with an unstructured description. CWIPI functionalities involve the construction of the communication graph between distributed geometric interfaces through geometrical localization, interpolation on non coincident meshes, exchanges of coupling fields for massively parallel applications as well as visualization files.

At the coupling boundary, the DGM solver is supplied at each stage of the Runge-Kutta method with the conservative variables from the CFD solver at the Gauss points belonging to the coupling boundary. These variables are computed with an interpolation of same order as the DGM scheme starting from the nodal values of the LES cells containing the Gauss points under consideration. And vice-versa, the CAA solver fills the ghost cells of Finite Volume method in order to apply the scheme at the coupling boundary. This coupling procedure works with structured/unstructured and non-conform meshes on both sides of the exchange surface.

3 LES/CAA COUPLING APPLIED TO 3D SHEAR FLOW SIMULATION

The simulation configuration consists of two cylinders fitted together as shown in Figure 1. The length and diameter of the external cylinder are 1m and 0.0794m, whereas the dimensions of the internal cylinder are respectively 0.06m and 0.04m. The meshes are built around a square of 14 cells × 14 cells with radial layers. There are 281 points in the axial direction and 57 points in the azimuthal direction. The internal cylinder has 225 120 elements, while the external one 1 061 760. The cylinder mesh is uniform in the axial direction with Δx =0.0005 m up to x=0 .06 and then a geometric progression is used in such a way that the cell size at the exit boundary Δx=0.02261 m.

In x=0, a velocity of 100m/s is imposed in the disc centered in 0 with a radius of 0.01m, on the remaining part of the entrance boundary the velocity is 50m/s as indicated in Figure 1, which shows the plane z=0, giving the different boundary conditions. In x=1m, a non-reflective condition is imposed at this boundary. The external envelope of the cylinder is a wall, on which is applied a slip condition. The coupling boundary is located at the interface of both cylinders. An explicit three-stage Runge-Kutta scheme is used in both solvers with a time step $\Delta t= 10^{-7}$ s. The exchanges between the solvers FUNk and SPACE occur at each stage of the RK3 scheme.

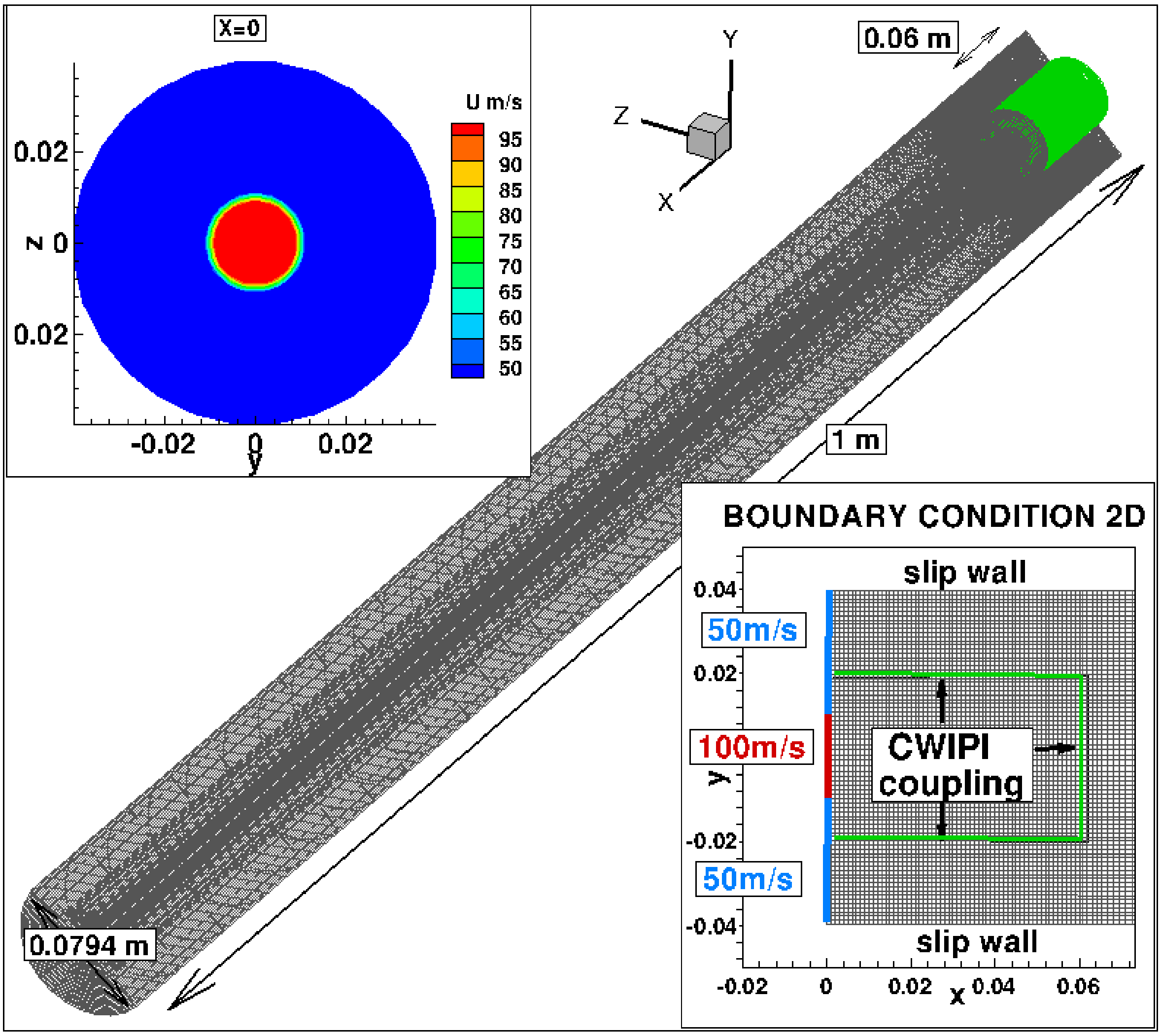

Figure 1 : Two cylinders fitted together, 2D view (z=0) of the boundary conditions and velocity at the entrance (x=0).

A first validation has been carried out with the aerodynamic solver FUNk, with and without CWIPI coupling in order to control if exchanges are correctly made. The results are identical. All the following results are plotted at t=0.001s, when the shear flow is crossing the coupling boundary at x=0.06m. In Figure 2 the pressure iso-contours are plotted in the plane z=0, the black bold line marks the coupling CWIPI boundary. An enlargement around this boundary is displayed with the iso-contours of velocity u. Moreover, the plane x=0.061m downstream to the coupling boundary shows the iso-velocity v. This case acts as reference for other simulations.

The aim of the hybrid method is first to use less points and high order in the CAA domain than in the same part of the previous CFD/CFD simulation. For that, each cell of the CAA part is divided by two in each direction. Then, the DGM method can access to high order schemes.

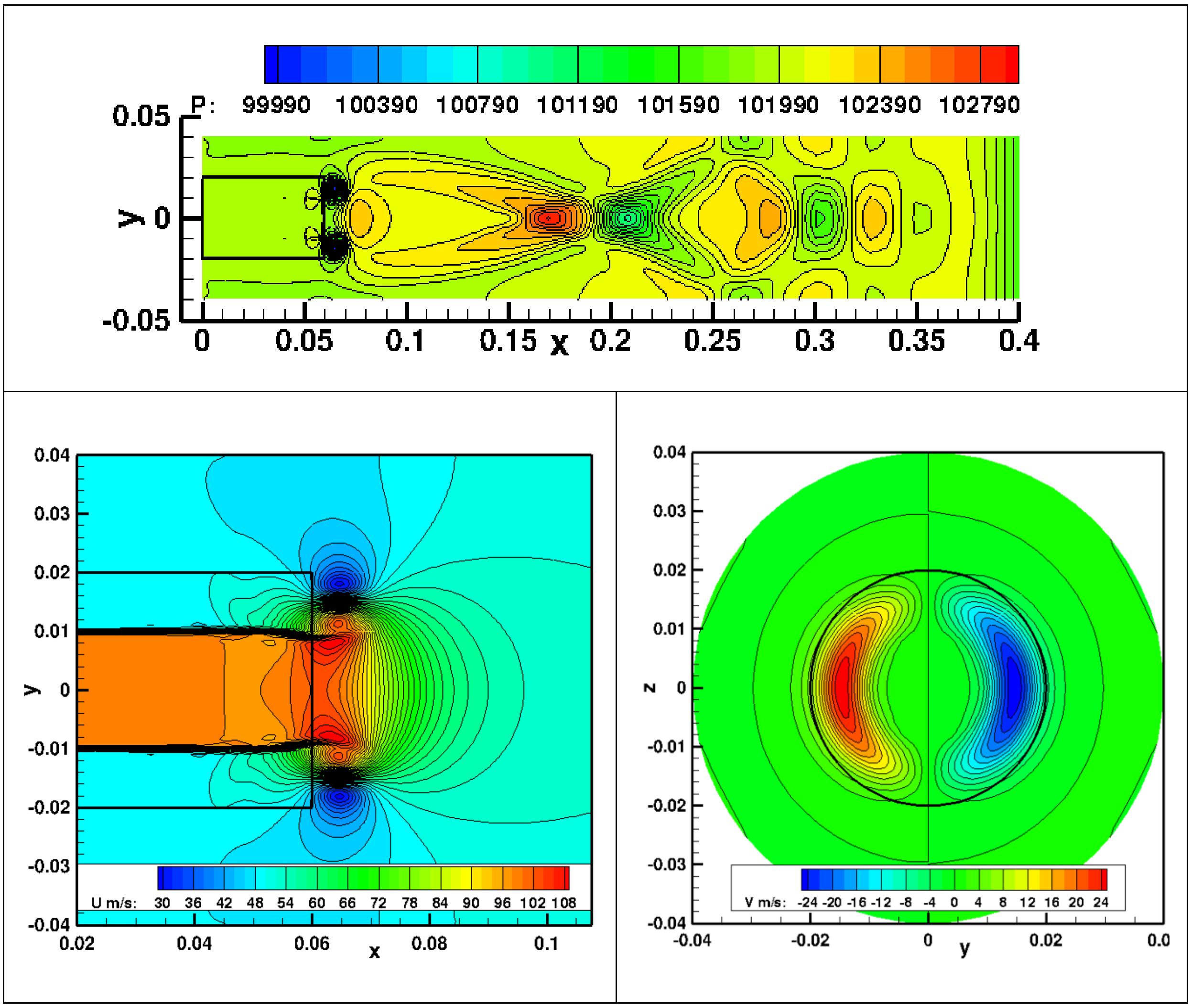

Figure 2 : FUNk/FUNk iso-contours: pressure and u-velocity (z=0), v-velocity (x=0.061).

The first FUNk/SPACE simulation uses the same mesh in the LES domain, but contains only 132 720 elements in the external cylinder, instead of 1 061 760 hexahedra in FUNk. A Q_1 Lagrangian basis is used for each SPACE hexahedra, which corresponds to approximately the same order of the LES scheme. The same iso-contours as previously are displayed in Figure 3. The results are similar to those obtained in the FUNk/FUNk simulation with much fewer points.

The second FUNk/SPACE simulation was to use a Q_2 hexahedra in the SPACE solver with the same coarse mesh, without changing anything for the FUNk solver. The results are plotted in Figure 4. The iso-pressure contours show that the pressure field is richer, especially between x=0.25 and 0.4m. That means the acoustic propagation is less numerically dissipative and dispersive, which is one of the main aims of this coupling method. The iso velocity-contours display spurious oscillations close to the coupling boundary, maybe due to the coarse mesh in this region. To solve this problem, either the use of refined hexahedra or the use of tetrahedra in the CAA domain only in the vicinity of the coupling boundary and in the remaining part of the domain a stretched mesh is sufficient.

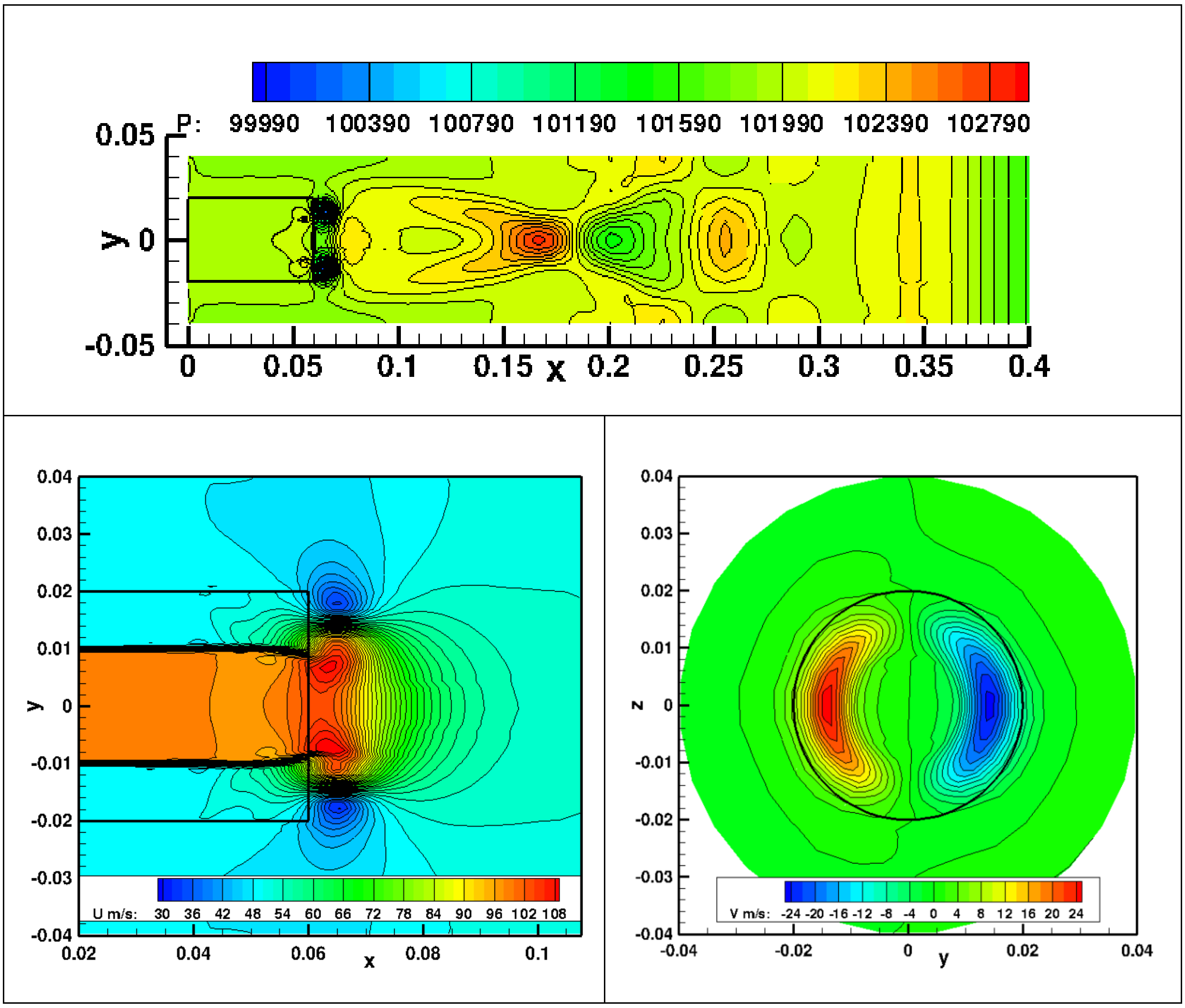

Figure 3 : FUNk/SPACE(Q_1) iso-contours: pressure and u-velocity (z=0), v-velocity (x=0.061).

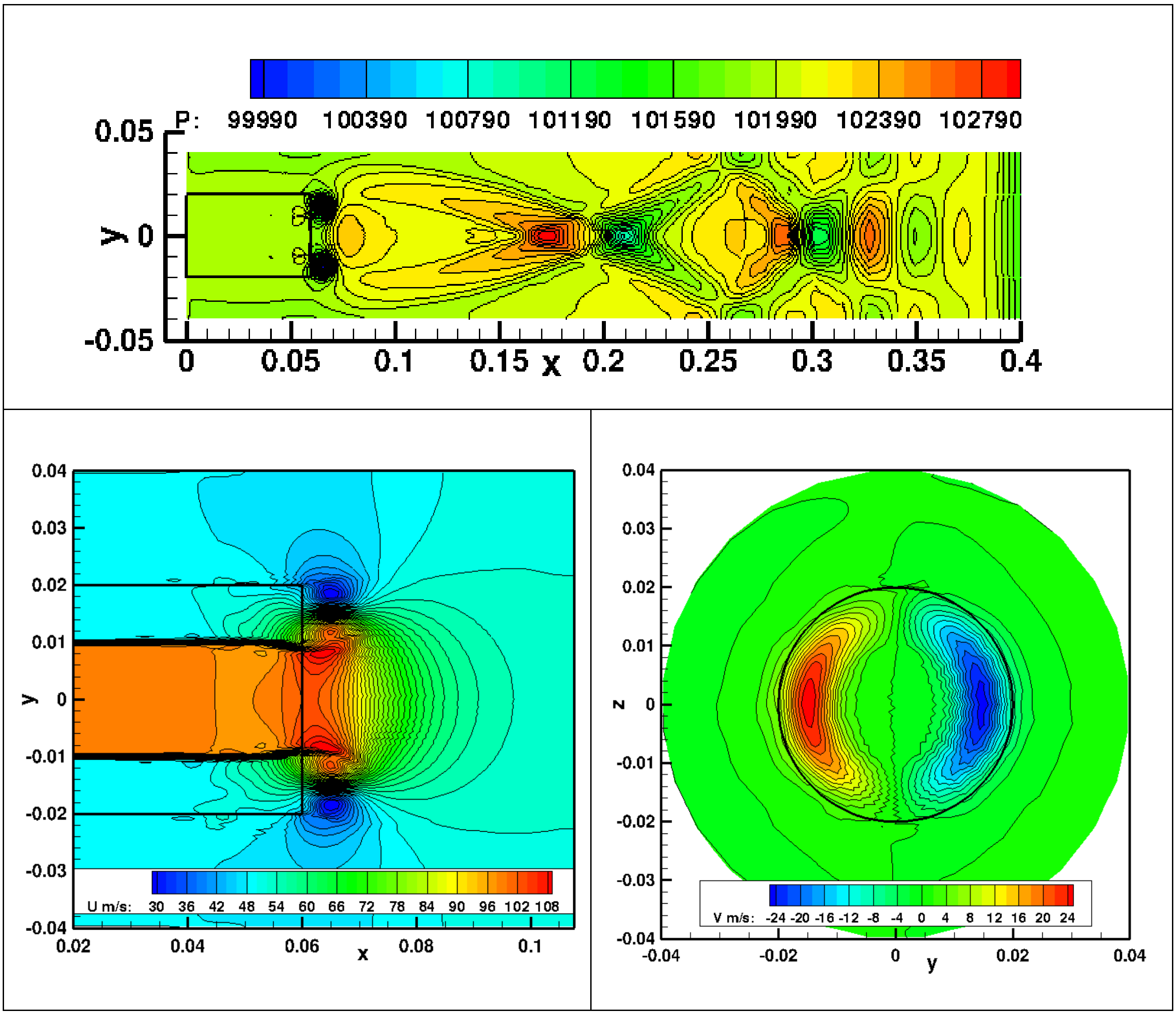

Figure 4: FUNk/SPACE (Q_2): iso-contours: pressure and u-velocity (z=0), v-velocity (x=0.061).

4 CONCLUSION

A domain decomposition approach for the coupling of aerodynamic and acoustic calculations is presented and applied to a 3D shear flow simulation. The aerodynamic domain is computed by a LES based on a Finite Volume method with a structured grid, while the acoustic one is performed using a Discontinuous Galerkin method solving the full non-linear Euler equations on an unstructured mesh. These results constitute the first validation of the hybrid method. First of all, the iso-contours obtained in the FUNk/SPACE (Q_1) are similar to those obtained in the reference case FUNk/FUNk. So, with less (8 times) elements in the external cylinder, the same results are recovered. The first aim of the hybrid method is obtained. Secondly, with the order increase (Q_2) of the DGM, the results show that the acoustic contain is improved and the ability of the CAA method to better propagate acoustic waves compared to the LES.

The next step in this study is to exploit the DGM ability of highly flexible refinement in order to improve the exchanges at the vicinity of the coupling boundary and to use a coarse mesh elsewhere. Moreover, this hybrid method will be very useful applied to a jet simulation, in which reflecting bodies may occur in the flow field.

REFERENCES

[1] Bailly, C., and Bogey, C., Contributions of Computational Aeroacoustics to Jet Noise Research and Prediction, *International Journal of Computational Fluid Dynamics*, **18**, No. 6, (2004), 481-491.

[2] Gamet, L., and Estivalezes, J.L., Application of Large-eddy Simulations and Kirchhoff Method to Jet Noise Prediction, *AIAA Journal*, **36**, No. 12, (1998), 2170-2178.

[3] Lyrintzis, A., Surface integral methods in computational aeroacoustics -From the (CFD) near-field to the (Acoustic) far-field, *International Journal of Aeroacoustics*, **2**, No 2, (2003), 95-128.

[4] Rahier, G., Prieur, J., Vuillot, F., Lupoglazoff, N. and Biancherin, A., Investigation of integral surface formulations for acoustic post-processing of unsteady aerodynamic jet simulations, Aerospace Science and Technology, 2004, Vol. 8, pp. 453-467.

[5] Uzun, A., Lyrintzis, A., and Blaisdell, G., Coupling of Integral Acoustics Methods with LES for Jet Noise Prediction, 42nd AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, 2004-517.

[6] Perez, G., Prieur, J., Rahier, G., and Vuillot, F., Theoretical and Numerical Discussion on Volume Integral Methods for Jet Noise Prediction, 13th AIAA/CEAS Aeroacoustics Conference, 2007-3593, Rome, Italy.

[7] Ewert, R., and Schröder, W., Acoustic perturbation equation based on flow decomposition via source filtering, *Journal of Computational Physics*, **188**, (2003), 365-398.

[8] De Roeck, W., Baelmans, and M., Desmet, W., Aerodynamic/Acoustic Splitting Technique for Computational Aeroacoustics Applications at Low Mach Numbers, *AIAA Journal*, **46**, No. 2, (2008), 463-475.

[9] Cunha, G. and Redonnet, S., Towards a robust and accurate CFD/CAA Coupling Procedure for Hybrid Methods in Aeroacoustics, Part 1: On the Optimization of CFD/CAA Coupled Calculations, 18th AIAA/CEAS Aeroacoustics Conference, Colorado Springs, Co, AIAA 2012-2063.

[10] Viswanathan, K., and Sankar, L. N., Toward the direct calculation of noise: fluid/acoustic coupled simulation, *AIAA Journal*, **33**, No 12, (1995), 2271–2279.

[11] Bogey, C., Bailly, C., and Juvé, D., Noise investigation of a High Subsonic, Moderate Reynolds Number Jet Using a Compressible Large Eddy Simulation, *Theoretical and Computational Fluid Dynamics*, **16**, (2003), 273-297.

[12] Utzmann, J., Munz, C.-D., Dumbser, M., Sonnendrücker, E., Salmon, S., Jund, S., and

Frinod, E., Fluid-Acoustic Coupling and Wave Propagation, Num. Sim. of Turbulent Flows & Noise Generation, NNFM 104, Springer-Verlag (Ed.) (2009), 47-74.

[13] Birkefeld, A., Beck, A., Dumbser, M., Munz, C.-D., König, D., and Schröder,W., Advances in the Computational Aeroacoustics with the Discontinuous Galerkin Solver NoisSol, 16th AIAA/CEAS Aeroacoustics Conference Stockholm, Sweden, (2010), AIAA 3823.

[14] Lockard, D. P., and Atkins, H. L., Efficient Implementations of the Quadrature-Free Discontinuous Galerkin Method, 14th AIAA Conference, Norfolk, VA, (1999), AIAA 3309.

[15] Labbé, O. Peyret, C. Rahier, G. and Huet, M., A CFD/CAA coupling method applied to jet noise prediction. *Computers & Fluids*, **86** (2013) 1–13.

[16] Larchevêque, L., Sagaut, P., Mary, I., Labbé, O., and Comte, P., Large-eddy simulation of a compressible flow past a deep cavity, *Physics of Fluids*, **15**, No 1, (2003), 193-210.

[17] Lenormand, E., Sagaut, P., Phuoc, T. and Comte, P., Subgrid-scale models for large-eddy simulation of compressible wall bounded flows, *AIAA Journal,* **38**, 8, 1340-1350, (2000).

[18] Delorme, P., Mazet, P.-A., Peyret, C., and Ventribout, Y., Computational Aeroacoustics Applications based on a Discontinuous Galerkin Method, *Comptes rendus de Mécanique*, **333**, (2005), 676-682.

[19] Léger, R., Peyret, C., and Piperno, S., Coupled Discontinuous Galerkin/Finite Difference Solver on Hybrid Meshes for Computational Aeroacoustics, *AIAA Journal*, **50**, No. 2, 338- 349, (2012).

[20] http://sites.onera.fr/cwipi/

[21] http://www.aerospacelab-journal.org/sites/www.aerospacelab-journal.org/files/AL2-11_0.pdf page 7