

UPCommons

Portal del coneixement obert de la UPC

<http://upcommons.upc.edu/e-prints>

Aquesta és la versió revisada per parells del següent article:

Ortega, E. [et al] (2013) A meshless finite point method for three-dimensional analysis of compressible flow problems involving moving boundaries and adaptivity. " International journal for numerical methods in fluids". Vol. 73, issue. 4. pp.323-343.
Doi:10.1002/fld.3799,

la qual ha estat publicada en la versió definitiva a
<http://dx.doi.org/10.1002/fld.3799>.

Aquest article pot ser utilitzat per a fins no comercials, d'acord amb els [termes i condicions d'auto-arxiu de Wiley](#).

This is the peer reviewed version of the following article:

Ortega, E. [et al] (2013) A meshless finite point method for three-dimensional analysis of compressible flow problems involving moving boundaries and adaptivity. " International journal for numerical methods in fluids". Vol. 73, issue. 4. pp.323-343.
Doi:10.1002/fld.3799,

which has been published in final form at
<http://dx.doi.org/10.1002/fld.3799>.

This article may be used for non-commercial purposes in accordance with [Wiley Terms and Conditions for Self-Archiving](#).

A MESHLESS FINITE POINT METHOD FOR THREE-DIMENSIONAL ANALYSIS OF COMPRESSIBLE FLOW PROBLEMS INVOLVING MOVING BOUNDARIES AND ADAPTIVITY

Enrique Ortega, Eugenio Oñate, Sergio Idelsohn¹ and Roberto Flores

International Center for Numerical Methods in Engineering (CIMNE)
Universidad Politécnica de Cataluña
Edificio C1, Campus Norte, UPC
Gran Capitán, s/n, 08034 Barcelona, España

Abstract. A Finite Point Method (FPM) for solving compressible flow problems involving moving boundaries and adaptivity is presented. The numerical methodology is based on an upwind-biased discretization of the Euler equations, written in arbitrary Lagrangian-Eulerian form, and integrated in time by means of a dual-time stepping technique. In order to exploit the meshless potential of the method, a domain deformation approach based on the spring network analogy is implemented and h -adaptivity is also employed in the computations. Typical movable boundary problems in transonic flow regime are solved to assess the performance of the proposed technique. In addition, an application to a fluid-structure interaction problem involving static aeroelasticity illustrates the capability of the method to deal with practical engineering analyses. The computational cost and multi-core performance of the proposed technique is also discussed through the examples provided.

1. INTRODUCTION

The simulation of problems involving moving or deforming boundaries is of paramount importance in many fields of aerospace, mechanical and civil engineering. Since the beginning of the last century, the study of such problems has been extensively addressed through experimental, analytical and simplified numerical approaches, and numerous techniques involving different levels of complexity have been proposed. Most of the typical analyses of aeroelastic instabilities such as lift divergence, flutter and gust response are an example of this. However, present design trends pursuing reduced structural weight and enhanced operational performances usually demand improved analyses which, in many cases, go beyond the scope of classical methodologies (an interesting discussion on challenges and emerging trends in the field of airplane aeroelasticity is presented in [1]). Hence, more

¹ ICREA Research Professor at CIMNE

complex fluid-structure coupled solution approaches based Euler or Navier-Stokes equations must be considered.

The solution of Euler or Navier-Stokes equations on moving or deformable domains involves difficulties regarding mesh motion and deformation. Moreover, adaptive refinement techniques are also frequently needed to achieve enough mesh resolution to capture local behaviour and/or keep acceptable mesh topology in highly deformed zones of the analysis domain. These facts make the problem solution more complex and increase the computational cost, causing sometimes failure of the numerical approach to suit the requirements of a practical design tool. In this context, meshless methodologies (see [2] for an overview) represent an alternative to conventional discretization techniques. The fact that meshless methods do not exhibit any of the topological restrictions associated with a mesh makes them potentially well suited for tackling moving or deformable boundary problems.

In the last two decades, numerous meshless techniques aimed at dealing with steady and unsteady compressible flow problems [3-10] have been developed (cf. [11] for a comparative analysis of popular discretization approaches). Moreover, applications to unsteady problems accounting for body motion have also been explored with success in a meshless context through small perturbation boundary conditions or domain deformation techniques, see for instance [12-14]. All in all, meshless features have been increasingly exploited and numerous advantages over conventional discretization techniques have been brought to light. However, some implementation aspects affecting robustness and efficiency of meshless applications and a lack of numerical results evaluating their performance in practical engineering analyses still arise doubts about their real benefits. With the objective to contribute to this discussion, we present an application of the meshless Finite Point Method (FPM) [4, 6, 9] to the adaptive solution of compressible flow problems in moving/deformable domains, for considering this a field in which the exploitation of meshless features could render important benefits.

This work is organized as follows. Section 2 outlines the main aspects of the finite point method and gives implementation details. The equations to be solved are presented in Section 3 and the scheme proposed for its solution is outlined in Section 4. Sections 5 and 6 describe the domain deformation and h -adaptive techniques respectively. Aimed at assessing the performance of the proposed methodology, several numerical applications are presented in Section 7. Finally, the most relevant conclusions of this work are drawn in Section 8.

2. THE FINITE POINT METHOD

Suppose $u(\mathbf{x})$ is a field function defined in a closed analysis domain $\Omega \in \mathfrak{R}^d$ discretized by a set of points \mathbf{x}_i ($i=1, n$) and Ω_i are local clouds of points covering Ω . Let each local cloud consist of np points resulting from a point \mathbf{x}_i called *star point* and a set of points \mathbf{x}_j surrounding it. Then, an approximation of $u(\mathbf{x})$ in Ω_i can be stated by

$$\hat{u}(\mathbf{x}) = \sum_{l=1}^m p_l(\mathbf{x}) \alpha_l = \mathbf{p}^T(\mathbf{x}) \boldsymbol{\alpha} \quad (1)$$

where p_l are monomial basis functions and α_l are unknown coefficients which must be determined. In this work complete quadratic polynomial bases are employed ($\mathbf{p}^T(\mathbf{x}) = [1, x, y, z, xy, xz, yz, x^2, y^2, z^2]$ in \mathfrak{R}^3 , see [15]). Next, assuming $np > m$ in the sampling region Ω_i , the following Weighted Least-Squares (WLSQ) functional is defined to compute the vector $\boldsymbol{\alpha}$

$$J_i = \sum_{j=1}^{np} \varphi_i(\mathbf{x}_j) [\hat{u}_j - u_j]^2 = \sum_{j=1}^{np} \varphi_i(\mathbf{x}_j) [\mathbf{p}^T(\mathbf{x}_j) \boldsymbol{\alpha} - u_j]^2 \quad (2)$$

in which $\varphi_i(\mathbf{x}_j) = \varphi(\mathbf{x}_j - \mathbf{x}_i)$ is a compact support exponential weighting function centred on the star point of the cloud [15] (Fixed Least-Squares (FLS)). The minimization of Eq. (2) with respect to $\boldsymbol{\alpha}$ leads to the following system of equations

$$\mathbf{A} \boldsymbol{\alpha} = \mathbf{B} \mathbf{u} \quad (3)$$

where $\mathbf{A} = \mathbf{P}^T \boldsymbol{\Phi}(\mathbf{x}) \mathbf{P}$, $\mathbf{B} = \mathbf{P}^T \boldsymbol{\Phi}(\mathbf{x})$, $\boldsymbol{\Phi}(\mathbf{x}) = \text{diag}(\varphi_i(\mathbf{x}_j))$ and $\mathbf{P} = [\mathbf{p}(\mathbf{x}_1), \mathbf{p}(\mathbf{x}_2), \dots, \mathbf{p}(\mathbf{x}_{np})]^T$. Then, the vector of unknown coefficients can be obtained by

$$\boldsymbol{\alpha} = \mathbf{A}^{-1} \mathbf{B} \mathbf{u} \quad (4)$$

and the approximate value of $u(\mathbf{x})$ at the star point of the cloud (i.e. the point where the weighting function is located) is computed by Eq. (1) as

$$\hat{u}(\mathbf{x}_i) = \mathbf{p}^T(\mathbf{x}_i) \mathbf{A}^{-1} \mathbf{B} \mathbf{u} = a_{ij} u_j \quad (5)$$

where summation over repeated indices is assumed. Note that Eq. (5) does not interpolate the nodal values $\mathbf{u}(\mathbf{x})$. These are simply internal values used to construct $\hat{\mathbf{u}}(\mathbf{x})$, the true approximation for which the governing equations and boundary conditions are enforced.

Having adopted a fixed weighting function in Eq. (2), matrices \mathbf{A} and \mathbf{B} become constant in Ω_i , thus simplifying the procedure to obtain the derivatives of Eq. (5) and reducing its computational cost. First-order derivatives of the unknown function at \mathbf{x}_i are approximated by

$$\frac{\partial \hat{u}(\mathbf{x}_i)}{\partial x_k} = \frac{\partial \mathbf{p}^T(\mathbf{x}_i)}{\partial x_k} \mathbf{A}^{-1} \mathbf{B} \mathbf{u} = b_{ij}^k u_j \quad (6)$$

and higher-order derivatives can be obtained by successive differentiation of the approximation bases vector.

It is possible to demonstrate that the approximation (metric) coefficients satisfy the *partition of unity* (sum in j $a_{ij} = 1$) and the *partition of nullities* (sum in j $b_{ij}^k = 0$) properties. Moreover, if the approximation bases are complete of order p , polynomials functions up to order p can be exactly reproduced by the numerical approximation at the star point of the cloud [16].

The metric coefficients in the FPM lack of symmetry properties due to the fact that the FLS procedure is discontinuous, e.g. $|b_{ij}^k| \neq |b_{ji}^k|$. This entails significant consequences regarding implementation aspects (related to data structures, storage requirements and the number of operations needed to compute nodal gradients) and the properties of the resulting discrete schemes, such as positivity and conservation. The discontinuous character of the approximation also makes collocation techniques to be a natural choice to discretize the problem governing equations.

2.1 Discretization of the analysis domain: the point generator

Even though not any point discretization is valid for meshless computations (especially when the problem exhibits complex geometrical/flow features), the fact that partition topology is not a matter of concern allows suitable point distributions filling the analysis domain can be generated quickly and efficiently by using dedicated algorithms. Moreover, simpler CAD representations of the computational models can be often employed, and this fact reduces developing times considerably. Therefore, it is expected that any point generation technique employed makes the most of these important advantages. In this work we adopt a very effective technique that is developed by Calvo [17]. The point generation starts from an user-supplied closed external boundary grid (internal isolates surfaces are also allowed) and inserts new points in the centre of empty spheres filling Ω through an optimization driven point insertion procedure. This incremental quality technique based on unconstrained Delaunay tetrahedralization allows achieving a quality point discretization with approximated cost $O(n)$ [18, 19] which is highly competitive with respect to typical mesh generation methods. In addition to the low computational cost, this technique has other advantages that reduce model preparation times significantly, e.g. the point spacing inside the domain does not need to be specified, as it is automatically assigned by computing linear variations between grid

boundary sizes (internal and external). The methodology described above is coded in a software package called MeshSuite (see <http://www.cimec.org.ar>) and has recently been incorporated in the GiD pre and post-processing software developed by CIMNE [20].

2.2 Construction of local clouds

Cloud construction techniques in WLSQ-based meshless approximations must be aimed at achieving point distributions which facilitate (from a mathematical point of view) the solution of the minimization problem. Moreover, these procedures should be able to deal with all the geometrical features found in practical application problems with robustness and a low computational cost. A technique combining these requisites was proposed by Löhner *et al.* [6] and we follow its main ideas here. Given a point discretization bounded by a triangulation with associated geometrical data, for each star point \mathbf{x}_i in Ω :

- a₁. A set of neighbouring points within a given radius is sought by using a spatial search algorithm based on bins [21]. These points are employed to generate an initial local cloud.
- a₂. The points in the initial cloud are filtered in order to match boundary restrictions (needed to ensure physical compatibility in the local cloud). Basically, if a ray from \mathbf{x}_i to another point \mathbf{x}_j in the local cloud pierces a boundary, the point \mathbf{x}_j is discarded. If the resultant number of admissible points is lower than a given threshold (about 120 in 3D clouds) the search radius is increased and the procedure returns to a₁.
- a₃. A Delaunay grid is generated with the admissible points and the first layer of nearest neighbours of \mathbf{x}_i is retained and stored. This guarantees the necessary overlapping of the clouds in order to cover the complete analysis domain.

After generating the layers of neighbouring points, the numerical approximation is computed (complete quadratic polynomial bases are used in this work). For each star point \mathbf{x}_i in Ω :

- b₁. A local cloud is initialized with its layer of nearest neighbours. If the number of points is lower than np_{\min} (about 30 in 3D), further points are added from an auxiliary neighbours list. This auxiliary list is constructed by adding points according to layers and ordering them by increasing distances from \mathbf{x}_i .
- b₂. The minimization problem is solved and a quality check of the local approximation is performed, see [6]. If the test is not successful, additional points are added from the auxiliary list while $np \leq np_{\max}$. In our experience $np_{\max} = 50$ is enough to achieve a proper approximation in highly distorted 3D clouds of points.

The procedure described above has proven reliable in general problems in which the data available are the coordinates of the points and a boundary grid delimiting the analysis domain. However, this procedure can be simplified (and its cost reduced) in some problems, for instance when the domain is fixed or only moderate movement/deformation is involved. In such cases, the graph of nearest neighbours can be obtained from the global cloud connectivity after a filtering stage to enforce boundary restrictions (step a₂). In this way, a considerable time saving can be achieved by avoiding the local Delaunay gridding at each cloud. This is the procedure followed in this work, with the exception of the construction or reconstruction of clouds affected by h-adaptation, where local triangulation is required.

3. FLUID GOVERNING EQUATIONS

The problems considered in this work are governed by the compressible Euler equations and an arbitrary Lagrangian-Eulerian (ALE) frame of reference is adopted to facilitate the treatment of moving/deforming bodies. These equations can be written as

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}^k}{\partial x_k} + \mathbf{S} = \mathbf{0} \quad (7)$$

where

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u_i \\ \rho e_t \end{bmatrix}, \quad \mathbf{F}^k = \begin{bmatrix} \rho(u_k - w_k) \\ \rho u_i(u_k - w_k) + \delta_{ik} p \\ \rho e_t(u_k - w_k) + u_k p \end{bmatrix}, \quad \mathbf{S} = \mathbf{U} \frac{\partial w_k}{\partial x_k} \quad (8)$$

being ρ , p and e_t the fluid density, pressure and total energy respectively. The Cartesian components of the fluid velocity are u_i and w_i denotes the components of the discrete points velocity. The following state relation for a perfect gas is assumed

$$p = \rho(\gamma - 1) \left[e_t - \frac{1}{2} u_k u_k \right] \quad (9)$$

where γ is the specific heat ratio. The boundary conditions adopted in this work are far-field conditions on outer boundaries Γ_∞ and slip conditions on solid body boundaries Γ_w . In the case of far-field boundaries, the flux in the outward normal direction at each $\mathbf{x}_i \in \Gamma_\infty$ is prescribed by solving an approximate Riemann problem between \mathbf{U}_i and the far-field state \mathbf{U}_∞ . Over body boundaries, slip wall conditions are enforced by

$$(\mathbf{u} - \mathbf{w}) \cdot \hat{\mathbf{n}} = 0 \quad \forall \mathbf{x}_i \in \Gamma_w \quad (10)$$

being $\hat{\mathbf{n}}$ the unit boundary normal vector at point \mathbf{x}_i . The spatial position of the discrete points \mathbf{x}_i and their velocities \mathbf{w}_i as well as the boundary normal vectors $\hat{\mathbf{n}}_i$ are obtained from prescribed or computed body movements.

4. THE FLOW SOLVER

The semi-discrete problem is obtained by replacing the continuous spatial variables in Eqs. (7) by their discrete approximations (Eqs. (5) and (6)). The resulting equations can be recast in a form resembling LED schemes by exploiting the partition of nullities property of the FP approximation. This leads to [9],

$$\frac{\partial \hat{\mathbf{U}}_i}{\partial t} = -2 \sum_{j \neq i} b_{ij}^k [\tilde{\mathbf{F}}_{ij}^k - \tilde{\mathbf{F}}_i^k] - \hat{\mathbf{U}}_i \sum_j b_{ij}^k \tilde{\mathbf{w}}_j^k \quad \forall j \in \Omega_i \quad (11)$$

where $\hat{\mathbf{U}}_i$ is a discrete approximation at \mathbf{x}_i of the conservative variables vector, $\tilde{\mathbf{F}}_{ij}$ is an unknown numerical flux computed at the midpoint of the line segment connecting \mathbf{x}_i to another point $\mathbf{x}_j \in \Omega_i$, $\tilde{\mathbf{F}}_i^k = \mathbf{F}^k(\mathbf{U}_i)$ and $\tilde{\mathbf{w}}_i$ is the discrete point (grid) velocity vector. Among the different possibilities that exist for defining the numerical flux, following [6], the Roe's approximate Riemann solver [22] is adopted in this work. Thus,

$$\tilde{\mathbf{F}}_{ij}^k = \frac{1}{2} (\tilde{\mathbf{F}}_j^k + \tilde{\mathbf{F}}_i^k) - \frac{1}{2} |\mathbf{A}_{\hat{\mathbf{n}}}(\mathbf{U}_i, \mathbf{U}_j)| (\mathbf{U}_j - \mathbf{U}_i) \hat{\mathbf{n}}_{ij}^k \quad (12)$$

where $\hat{\mathbf{n}}_{ij}$ is a unit vector in the direction of $\mathbf{l}_{ij} = \mathbf{x}_j - \mathbf{x}_i$ and $|\mathbf{A}_{\hat{\mathbf{n}}}(\mathbf{U}_i, \mathbf{U}_j)|$ is the positive Roe matrix calculated in the same direction (see [23] for details). Aimed at increasing the spatial accuracy of the Roe solver, the variables $(\mathbf{U}_i, \mathbf{U}_j)$ in Eq. (12) are replaced by leftward and rightward higher-order reconstructions $(\mathbf{U}_i^+, \mathbf{U}_j^-)$ obtained by limited MUSCL extrapolation along the vector \mathbf{l}_{ij} [6]. The Van Albada limiter is adopted.

4.1 Discretization in time

The Jameson's dual-time stepping approach [24] is adopted to integrate in time the semi-discrete equations (11). This procedure allows solving implicitly each physical time increment by means of inner explicit iterations in a fictitious time. To this end, a 2nd-order backward difference operator is applied to the time derivative in Eq. (11) leading to

$$\frac{3\mathbf{U}_i^{n+1} - 4\mathbf{U}_i^n + \mathbf{U}_i^{n-1}}{2\Delta t} = -\mathbf{R}(\mathbf{U}_i^{n+1}) \quad (13)$$

where $\mathbf{R}(\bullet)$ denotes the right-hand side of Eq. (11). Next, for a given increment in physical time Δt , a modified (unsteady) residual is defined from Eq. (13) as

$$\mathbf{R}_i^*(\mathbf{U}_i^*, \mathbf{U}_i^n, \mathbf{U}_i^{n-1}, \Delta t) = \frac{1}{2\Delta t} (3\mathbf{U}_i^* - 4\mathbf{U}_i^n + \mathbf{U}_i^{n-1}) + \mathbf{R}_i(\mathbf{U}_i^*) = \mathbf{0} \quad (14)$$

verifying that \mathbf{U}_i^* approaches \mathbf{U}_i^{n+1} as $\mathbf{R}_i^* \rightarrow \mathbf{0}$. In order to solve Eq. (14) in an explicit manner, a temporal derivative with respect to a fictitious time t^* is introduced. This leads to

$$\frac{\partial \mathbf{U}_i^*}{\partial t^*} + \mathbf{R}_i^*(\mathbf{U}_i^*, \mathbf{U}_i^n, \mathbf{U}_i^{n-1}, \Delta t) = \mathbf{0} \quad (15)$$

and this system is driven to the steady state by means of a multi-stage scheme. Therefore, the intermediate solution \mathbf{U}_i^* is advanced from a fictitious time level m to a level $m+1$ by

$$\hat{\mathbf{U}}_i^{*(m+j)} = \hat{\mathbf{U}}_i^{*m} - \alpha_j \Delta t_i^* \mathbf{R}^*(\mathbf{U}_i^{*(m+j-1)}, \mathbf{U}_i^n, \mathbf{U}_i^{n-1}, \Delta t) \quad (16)$$

being α_j integration coefficients depending on the number of stages employed (see for instance [25]) and Δt_i^* a fictitious local time step which must be bounded due to stability requirements. This is computed by

$$\Delta t_i^* = \mathcal{C} \min \left(\frac{\|\mathbf{l}_{ij}\|}{|\mathbf{u}_i \cdot \hat{\mathbf{n}}_{ij}| + c_i} \right) \quad \forall j \in \Omega_i \quad (17)$$

where \mathcal{C} denotes the Courant number, c_i is the speed of sound and the rest of the variables have been previously defined. In addition to local time stepping, implicit residual smoothing [26] is applied to accelerate the convergence of the system (15) to the steady state. Notice that the explicit treatment of the term $3\mathbf{U}_i^*/2\Delta t$ could lead to numerical instabilities if the physical time step is small [27]. Fortunately, this problem can be easily overcome by treating implicitly that term in Eq. (16), see [28].

Due to the fact that the FPM approximation does not interpolate nodal data and taking into account that $\mathbf{R}^*(\bullet)$ in Eq. (16) is a function of $\mathbf{U}_j \forall \mathbf{x}_j \in \Omega_i$, a linear system has to be solved to recover the internal nodal values from the approximate solution at each integration stage. According to Eq. (5) this system is

$$\sum_{\forall j \in \Omega_i} a_{ij} \mathbf{U}_j = \hat{\mathbf{U}}_i \quad i = 1, n \quad (18)$$

and can be solved by a few Gauss-Seidel iterations with little computational cost.

4.2 Spatial accuracy of the FPM solver

Considering one-dimensional, equally spaced point distributions and symmetric local clouds, Fischer and Oñate [29] have proven that quadratic (and even linear) FLS yield an approximation to the flux derivatives which is second-order accuracy ($O(2)$). However, they have also found that for non-symmetric clouds of points the accuracy of the approximation degrades tending to $O(1)$, showing a notable dependence on the weighting function settings and the spatial distribution of points. It should be noticed that the deterioration of spatial convergence rates in irregular stencils of points is not a particular issue in meshless, as it can be also observed in mesh-based approximation methods.

4.3 Conservation of the discrete scheme

In most meshless approximations the properties of the metric coefficients \mathbf{b}_{ij} and their dependence on the distribution of points in the local clouds and weighting function settings mean conservation can not be demonstrated in general application cases (e.g. through typical telescopic flux properties). However, in spite of the fact that to date there is a lack of proofs demonstrating that meshless schemes can be conservative at the discrete level, the empirical evidence does not reveal the typical problems which are typically related to a lack of conservation. Overall, numerical evidence by itself is not sufficient and conservation in meshless methods remains an important theoretical weakness which requires further research. In this regard, it is worth mentioning some recent works in which conservation and other desirable properties in meshless schemes are enforced by the construction of the approximation metric coefficients; see for instance [30].

5. POINTS MOVEMENT STRATEGY

Problems involving deforming or moving bodies require the domain discretization to conform continuously to the instantaneous body shape. An overview of typical solution strategies to deal with domain deformation is presented in [31] while more recent promising techniques based on Delaunay mapping can be found in [32] (see a 2D meshless application in [14]). In this work a classical spring network approach is adopted [33, 34] to maintain the meshless character of the solution the methodology. Therefore, the displacement of any interior point \mathbf{x}_i in response to instantaneous displacements of body points are obtained by enforcing the static equilibrium of the forces exerted by all the points \mathbf{x}_j connected through the \mathbf{x}_i 's layer of nearest neighbours in the local cloud (outer boundary points are considered to be fixed). This leads to a system of equations in terms of displacements which is solved by Jacobi iterations,

$$\delta_i^{k+1} = \frac{\sum_j k_{ij} \delta_j^k}{\sum_j k_{ij}} \quad (19)$$

being k the iteration counter, δ_i and δ_j displacement vectors of points \mathbf{x}_i and \mathbf{x}_j respectively and $k_{ij} = \|\mathbf{x}_j - \mathbf{x}_i\|^{-1}$ a link stiffness which prevents the clouds near the moving boundary from excessive distortion. After some iterations (10-50 iterations showed enough to propagate satisfactorily the body displacements) the position of the discrete points is updated according to $\mathbf{x}_i^{new} = \mathbf{x}_i^{old} + \delta_i$ and its velocities are simply estimated by $\mathbf{w}_i = \delta_i / \Delta t$ being Δt the physical time increment employed in the simulation.

6. ADAPTIVITY

The solution of compressible flow problems involving moving or deforming boundaries typically exhibits evolving discontinuities which must be properly resolved. An efficient way to deal with these problems with accuracy while keeping the computational cost acceptable is by using h -adaptivity. This technique also constitutes a good complement to domain deformation because highly distorted zones in the analysis domain can be locally regenerated, thus improving the quality of the discretization. The reduced topological restrictions found in meshless methods make them an ideal context to implement h -adaptive strategies. In this work the main lines of the methodology developed by the authors in [9, 35] are followed, although some modifications are introduced to increase the effectiveness and robustness of the technique. Next, the main aspects of the adaptive strategy are described.

6.1 Refinement/coarsening indicator

Local clouds in the analysis domain where either refinement or coarsening is required are identified as follows. First, a sensor measuring in an approximate manner the curvature of the density solution field is computed for all points in the domain by

$$r_i = \sum_{j=1}^{n_{nei}} \left| \mathbf{l}_{ij} \cdot (\nabla \rho_j - \nabla \rho_i) \right| \quad (20)$$

being n_{nei} the number of points in the layer of nearest neighbours of \mathbf{x}_i and $\mathbf{l}_{ij} = \mathbf{x}_j - \mathbf{x}_i$. Next, a smoothed non-dimensional refinement indicator is computed at each star point as

$$\varphi_i = \frac{1}{\rho_i} \frac{1}{n_{nei}} \sum_{j=1}^{n_{nei}} r_j \quad (21)$$

Finally, the logarithm of the smoothed indicator (21) is taken in order to compress the scale of the distribution and its mean value φ_m and standard deviation s_φ are computed. These values are used to determine local clouds in which refinement or coarsening is required: the star point \mathbf{x}_i is tagged for refinement when $\varphi_i > \varphi_m + N_{ref}s_\varphi$ and, conversely, the point \mathbf{x}_i is marked to be removed if $\varphi_i < \varphi_m - N_{rem}s_\varphi$. The threshold parameters N_{ref} and N_{rem} must be set according to the problem under study, typical values employed in the examples shown in this work are $N_{ref} = 1$ and $N_{rem} = 0$. It was observed that this indicator performs better than that employed previously in [9, 35] in capturing strong and smooth features of the flow; in addition, it presents a reduced dependence of the user defined threshold parameters.

6.2 Removal and insertion of points

The adaptive stage begins with the removal of points tagged for deletion. The coarsening of surface grids is performed by collapsing edges with marked nodes and interior volume points are simply deleted from the vertices list. After the removal of points the data structure is updated. It is important to note that the removal of points is restricted only to the existing points that have been inserted in prior refinement levels.

The surface grids are refined after the coarsening stage. To this end, surface elements having tagged all their nodes are selected and a new point is inserted at its centroid (and the element is subdivided) if the distance from the latter to any other point in the discretization is greater than a minimum cut-off distance (h_{min}) which determines the desired level of refinement. The latter is computed for the original points distribution as an user-defined percentage of the distance between each point and the nearest neighbour in its cloud. For new points added during refinement stages, h_{min} is inherited from the original points distribution. Once new boundary points are inserted, their positions (originally coincident with the centroid of the underlying element) are slightly improved by interpolation [36] to avoid excessive faceting of the surfaces in successive element subdivisions. Finally, edge swapping is applied on the affected boundary elements to improve its connectivity and facilitate the insertion and removal of points in future adaptive passes.

In the interior volume, when a cloud of points is selected to be refined, the Voronoi vertices surrounding the star point \mathbf{x}_i are computed by means of its Delaunay grid of nearest neighbours. Next, a new point is added at each Voronoi vertex location if the distance from the latter to any existing point is greater than h_{min} for the cloud and if, in addition, boundary

constraints are satisfied (basically, the ray from the star point to any new point added to the cloud cannot pierce any boundary surface). In order to perform the many spatial search operations required during the adaptive stage efficiently, a dynamically updated bins structure is employed.

6.3 Update

Once coarsening and refinement steps are finished, a few steps of a Laplacian smoothing are carried out on the affected area. After that, the clouds of points and the local approximation concerning the new points are constructed. In addition, the data concerning existing clouds of points affected by deletion, insertion of new points or smoothing are re-constructed. Finally, the flow and other problem variables at the new points are obtained as an average of the variables at their previously existing first nearest neighbours.

7. NUMERICAL APPLICATIONS

Four numerical applications are presented in this section to illustrate the performance of the proposed technique. The first two examples, which involve a pitching NACA 0012 wing and a wing with oscillating flap, are aimed at assessing the overall performance of the methodology. Then, an ONERA M6 wing subject to twist deformation is presented and the adaptive solution approach is analyzed from the point of view of accuracy and computational cost. In the last example, a coupled fluid-structure interaction problem is solved to determine the static aeroelastic deformation of the HiReTT N44 wind tunnel model. Comparisons with available experimental/numerical data and details of the computational cost are given.

7.1 NACA wing subject to pitching oscillations

This example is a typical AGARD benchmark for unsteady flow which involves a NACA 0012 airfoil subject to prescribed pitching oscillations [37]. The problem is solved in 3D using a wing with unit chord ($c = 1$) and span $b = c/2$. The analysis domain considers two symmetry lateral planes at the wing tips to force two-dimensional flow and the outer boundary, where Riemann freestream boundary conditions are prescribed, is located 10 chords away from the wing. The discrete model consists of an unstructured distribution of 33409 points. The freestream Mach number is set to $M_\infty = 0.755$ and the instantaneous angle of attack of the wing (in degrees) is varied according to $\alpha(t) = 0.016 + 2.51\sin(\omega_\alpha t)$. The reduced frequency is $\kappa = \omega_\alpha c / 2U_\infty = 0.0814$ and the pitch axis is located at $c/4$. The converged steady solution of the problem is employed to initialize the unsteady simulations.

Simulation runs with 4, 8, 12, 16 and 32 physical time steps per oscillation period (T) are performed firstly to assess the performance of the time integration scheme. At each time increment, the unsteady residual is reduced at least 4 orders of magnitude (500 iterations were enough). The time evolution of the lift coefficients presented in Figure 1 shows the convergence of the computed solutions as the physical time step is reduced. The time accuracy is evaluated next by defining a temporal error as the magnitude of the difference between the lift coefficient computed at time $t = 115$ for a given solution and that obtained at the same instant time with the smaller time step computation ($\Delta t = T/32$). A convergence rate close to the design order of accuracy of the scheme is observed in Figure 2.

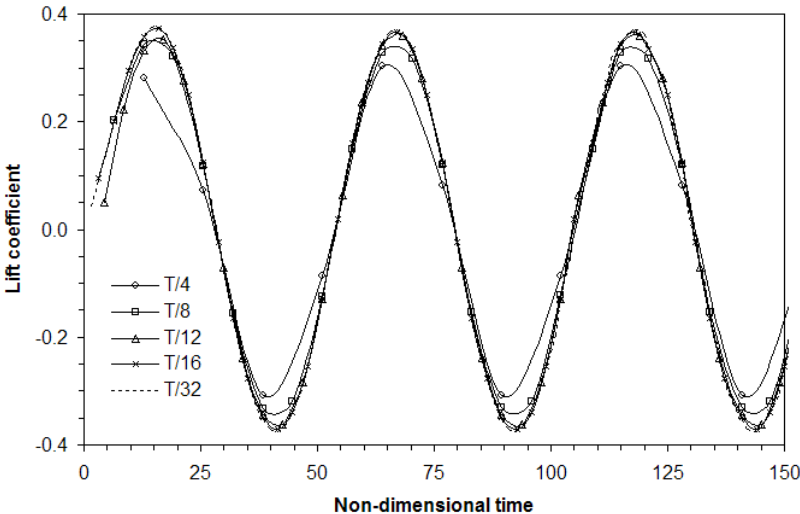


Figure 1. Time evolution of the lift coefficient for the NACA 0012 wing computed with different time step sizes.

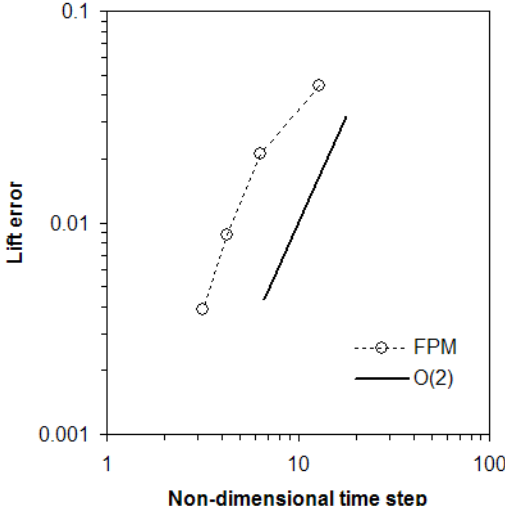


Figure 2. Convergence of the temporal error as a function of the time step size for the NACA 0012 wing.

The simulation run having 16 physical time steps per oscillation cycle is repeated, but adding 2 h -adaptivity passes per physical time step. On average, no more than 7000 points are added

during the adaptive steps while the number of removed points does not exceed 3000, achieving this maximum when the shock wave moves from the upper to the lower side of the wing and vice versa. The computed coefficient of pressure (C_p) along the mid-span of the wing at two instant times (corresponding to phase angles $\varphi = 67.8$ and 253.8) are compared in Figure 3 with those obtained for the original discretization and experimental results [37]. Moreover, the variation of lift and pitching moment coefficient with the instantaneous angle of attack for the adapted solution is compared with experimental results in Figure 4. As the difference found between adapted and non-adapted solutions was very small, the latter is not included in this figure (possibly the original coarse discretization was already fine enough to be in the range of converged grid solutions). A good agreement between numerical and experimental results is observed in Figures 3 and 4. Figure 5 shows refined points distributions in a cut plane along the mid-span of the wing at two different instant positions.

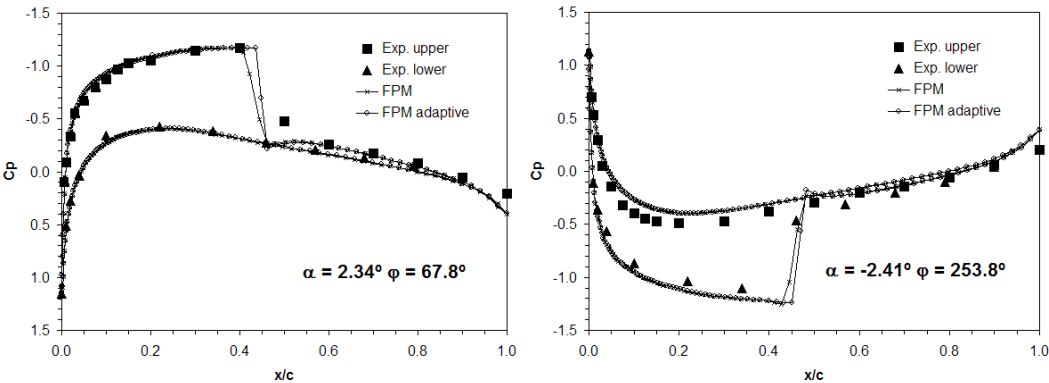


Figure 3. Surface pressure distributions along the NACA 0012 wing for two different time instants during the oscillation cycle.

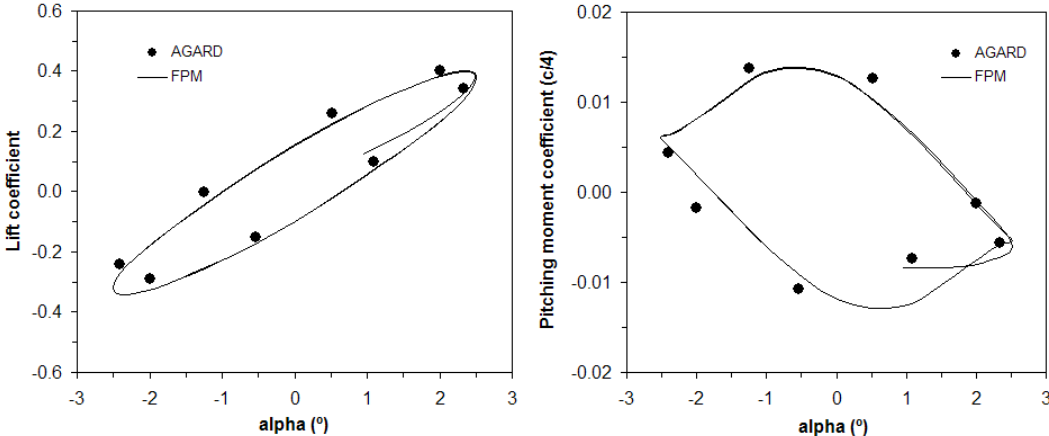


Figure 4. Comparison between computed and experimental force and moment variation with the instant angle of attack for the NACA 0012 wing (adapted solution, $\Delta t=T/16$).

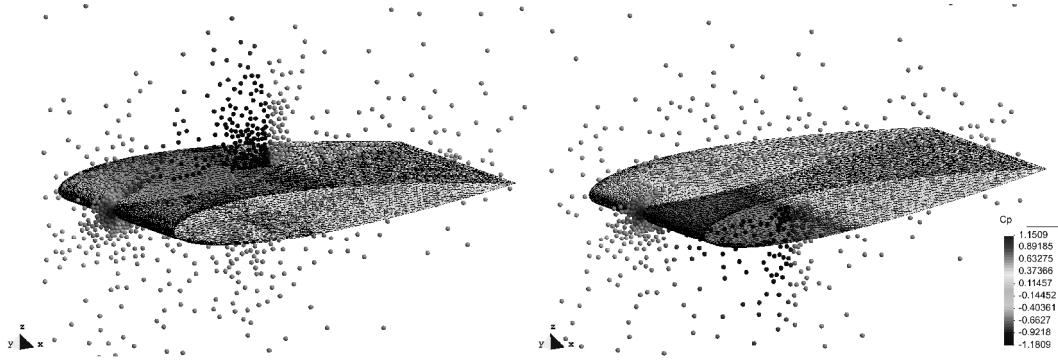


Figure 5. Refined point distributions along a plane passing through the wing mid-span. Left $\alpha=2.34^\circ$ and $\varphi=67.8^\circ$; Right: $\alpha=-2.41^\circ$ and $\varphi = 253.8^\circ$.

7.2 Wing with oscillating flap

An adaptive calculation of a wing having an oscillating flap is solved in this example to assess the capability of the method to deal with problems involving relative body motion. The section of the wing corresponds to the William airfoil (configuration B) [38] and the flow conditions are defined as in [39]: the freestream Mach number is set to $M_\infty = 0.58$, the instantaneous flap angle (in degrees) is $\delta_f(t) = 7.0 + 7.0 \sin(\omega_\delta t)$, the reduced frequency $\kappa = 0.0814$ and the flap hinge position is located at coordinates $(x, z) = (0.98, -0.07)$. The coordinates are non-dimensional with the chord of the main wing section and the rotation of the flap is counted with respect to its original position in the Williams airfoil. The analysis domain employed is defined similarly to the previous example and discretized by an unstructured distribution of 20713 points. The numerical simulation employs 12 physical time steps per flap oscillation cycle and 2 h -adaptive stages in each are performed. The computed variation of the normal force and pitching moment coefficients as a function of the instantaneous flap angle is compared in Figure 6 with results presented by Dubuc *et al.* [39] showing a satisfactory agreement.

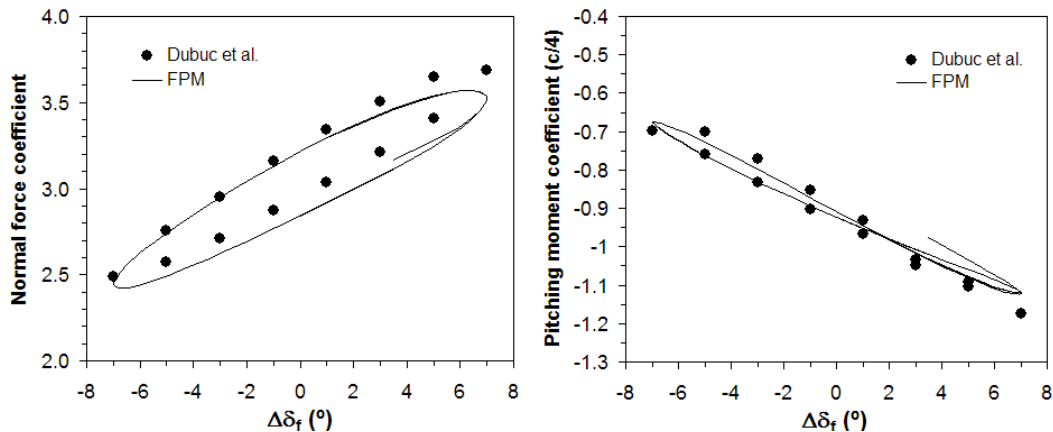


Figure 6. Normal force and pitching moment coefficient (wing+flap) of the Williams wing as a function of the flap instant rotation ($\Delta t=T/12$).

Figure 7 shows two plane cuts of the adapted domain taken at different instant times during the simulation. It is possible to observe the displacement of the shock wake along the main wing and the appearance of a shock on the flap for its maximum incidence angle.

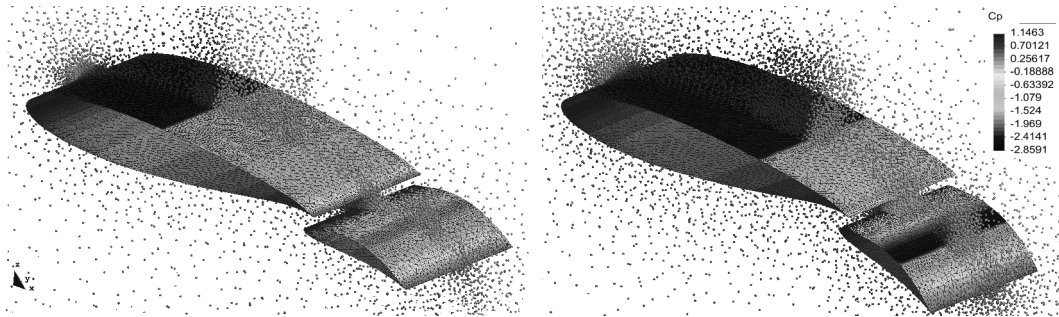


Figure 7. Refined point distributions along a plane passing through the mid-span of the Williams wing. Left: $\delta_f=0^\circ$ and $\varphi=270^\circ$; Right: $\delta_f=14.0^\circ$ and $\varphi=90^\circ$.

7.3 Twist deformation of an ONERA M6 wing

This example concerns the calculation of an ONERA M6 wing subject to twisting deformation about a line passing through the tip quarter chord point [40]. The freestream Mach number is $M_\infty=0.84$ and the wing incidence is $\alpha=3.06^\circ$. The twist deformation is achieved by forcing the tip section to pitch with $\alpha_{ip}(t) = 2.51\sin(\omega_\alpha t)$ (in degrees) while keeping the root section fixed. The instant angle of attack of the intermediate sections is varied linearly along the wing semi-span and the reduced frequency is $\kappa=0.1628$. The analysis domain includes a symmetry plane and a hemispherical outer freestream boundary.

In order to investigate the impact of h -adaptivity from the point of view of accuracy and computational cost two different unstructured domain discretizations are constructed: a fine discretization having 258919 points and a coarse one with 107847 points. In addition, an adapted discretization is obtained from the latter by performing a single h -adaptive pass per physical time step (see Figure 8).

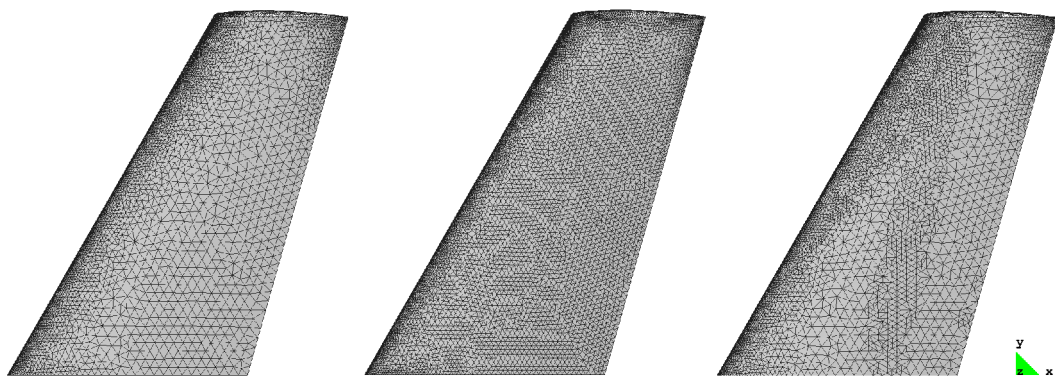


Figure 8. Top view of the ONERA M6 wing surface grids. Left: coarse; Center: fine and Right: adapted-coarse at $\varphi=0^\circ$ (the adapted discretization does not change substantially during the twist cycle).

Several twist cycles are simulated with the discretizations defined above running 8 physical time steps per cycle. The number of fictitious time iterations is fixed to 500 and the resultant unsteady residual is reduced approximately 4 orders of magnitude. Figure 9 displays the time evolution of the normal force coefficient and some instantaneous C_p distributions along the wing are compared in Figure 10. Even though the number of new points added during the h -adaptivity stages reaches as much the 12% of the number of original points, it can be seen that the adapted solution improves noticeably over the non-adapted coarse simulation.

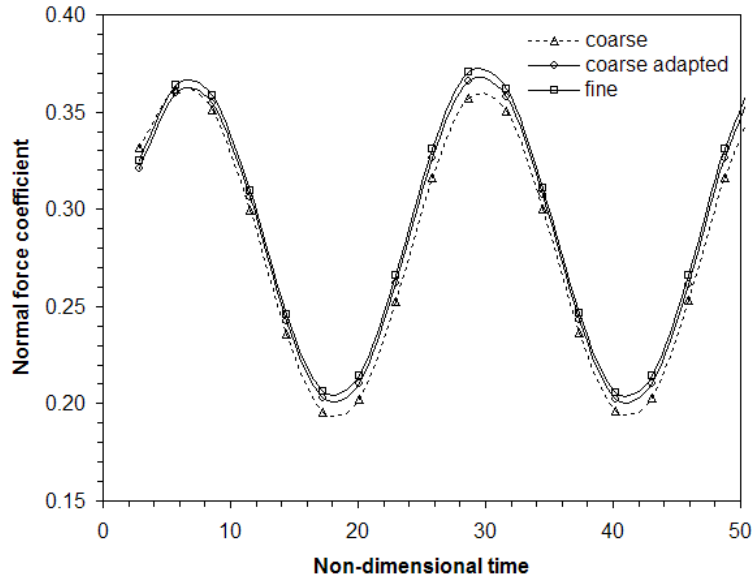


Figure 9. Time evolution of the normal force computed for the twisting ONERA M6 wing with the coarse, fine and adapted-coarse discretizations ($\Delta t=T/8$).

The CPU-time required to advance the simulation one physical time step ($\Delta t=T/8$, 500 fictitious time iterations) is compared in Table 1 for the coarse, fine and adapted-coarse discretizations. The computational code is written in Fortran 77/90 and parallelized through OpenMP directives. The simulations are run in a desktop computer with Intel Core2 Quad Processor Q9550 @ 2.83 GHz. Table 1 shows that the cost in the adapted simulation is slightly higher than that corresponding to the coarse discretization and low if compared with the fine discretization run. At the same time, the adapted solution improves considerably over that obtained with the coarse discretization and matches closely the solution obtained with the fine discretization. Although there could exist some problem (and user setting) dependence in the adaptive results, meshless adaptivity shows effective.

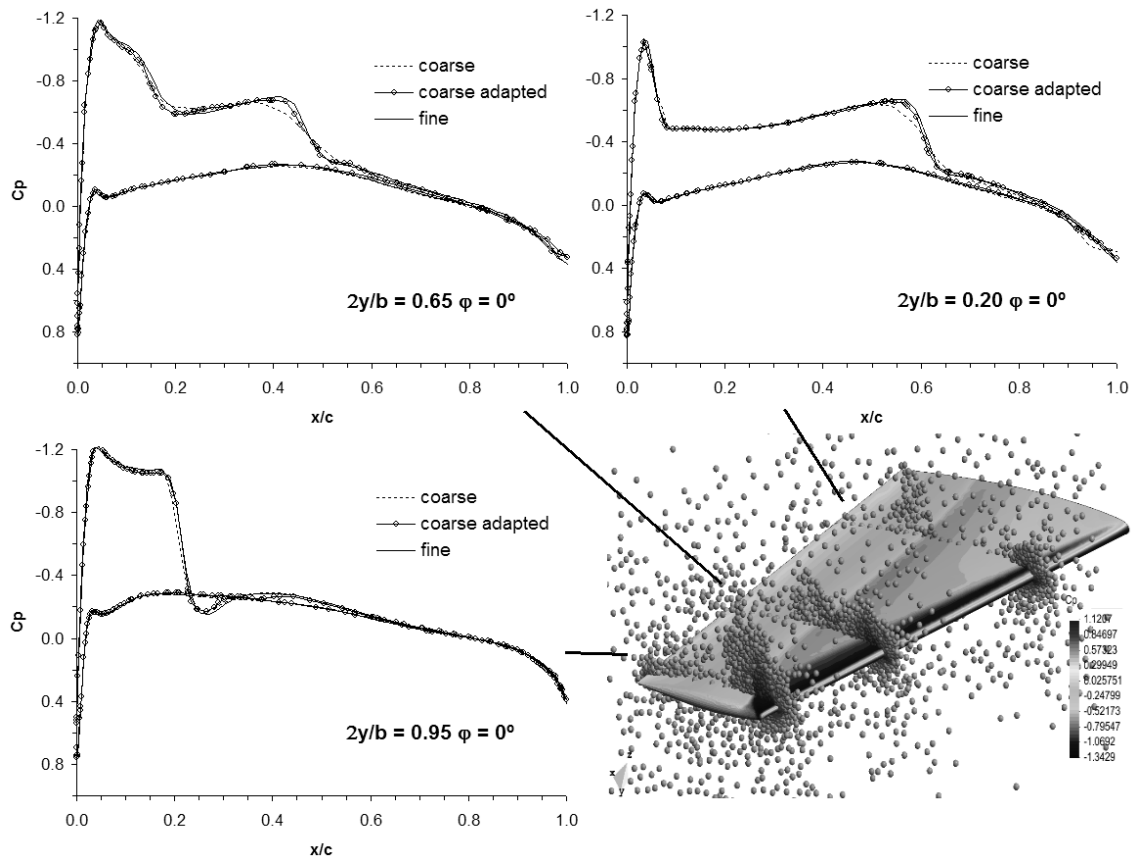


Figure 10. Comparison of instantaneous C_p distributions along 3 sections of the ONERA M6 wing computed for the coarse, fine and adapted discretizations (a view of the latter is displayed at the right bottom).

	CPU-time (secs)	relative cost
coarse	170.8	0.54
adapted coarse	192.6	0.60
fine	318.6	1.00

Table 1. Comparison of CPU-time needed to advance the solution of the twisting ONERA M6 wing a single physical time step ($\Delta t = T/8$ and 500 fictitious time iterations) using 4 running cores.

The parallel speedup observed for the adaptive simulation is depicted in Figure 11. FPM low data dependence allows most of the typical operations in the flow computation (including adaptive stages) to be carried out in an independent manner, thus achieving good parallel performance. However, as it will be seen in the next example, attainable speedups in multi-core (single processor) hardware platforms can degrade when working with larger data sets due to cache misses and memory bandwidth limitations.

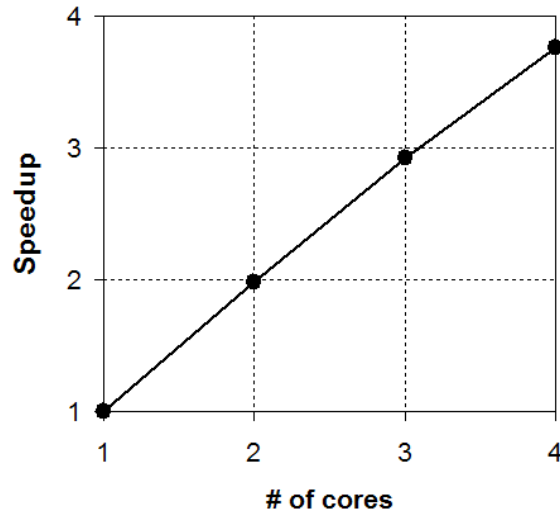


Figure 11. Observed speedup in the ONERA M6 wing adapted simulation ($\Delta t=T/8$ and 500 iterations in fictitious time loop).

7.4 Coupled aeroelastic analysis of the HiReTT wind tunnel model

The static aeroelastic deformation of a wing-body configuration is solved in this example to assess the proposed meshless technique in a practical application. The geometry of analysis corresponds to a cryogenic wind tunnel model employed in the European research project HiReTT [41] (High Reynolds Number Tools and Techniques for Civil Aircraft Design) to investigate scale effects on aircraft performance. A half-span symmetric version of the N44 test model [42] with a clean F7-1 wing configuration is adopted ($b/2=790\text{mm}$).

7.4.1 Computational models

In order to accomplish the coupled aeroelastic solution of the analysis configuration, aerodynamic and structural models are constructed. The analysis domain for the aerodynamic problem (CFD-model) includes a symmetry vertical plane along the fuselage centerline and a hemispherical outer freestream boundary located 15 half-spans away from the body. The resultant discrete model has 338886 boundary triangles and a total of 3100159 points generated with the technique described in Section 2.1. A view of the model boundary discretization is shown in Figure 12. As large flow separation areas are not expected in the range of simulation conditions to be adopted here, the flow is considered to be inviscid and the boundary conditions applied are similar to those employed in the previous examples.

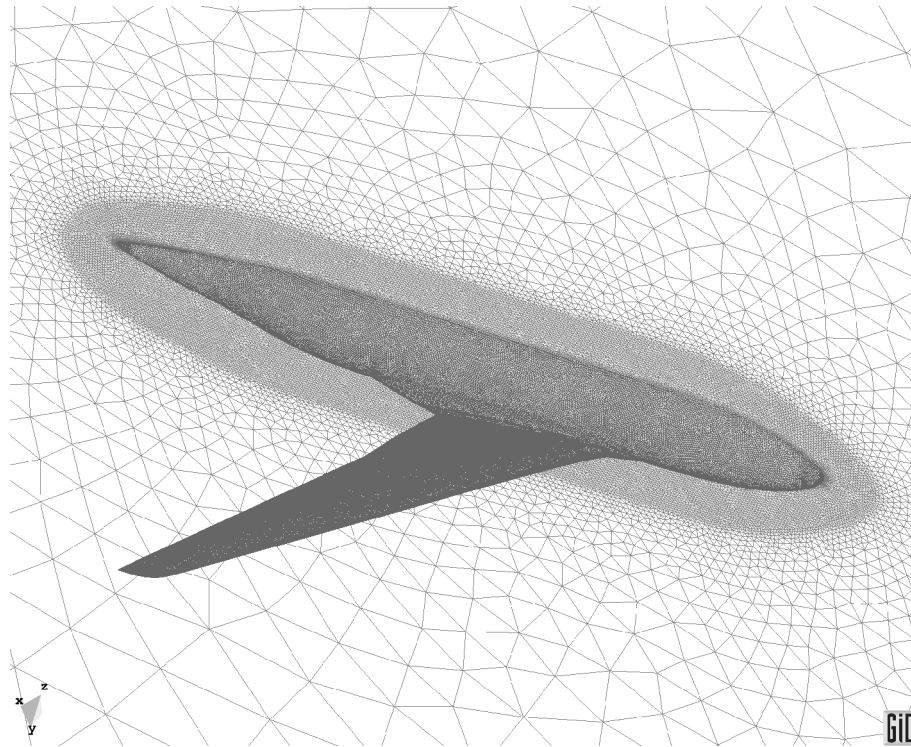


Figure 12. Boundary grid of the half-span model employed for flow computations (approx. 340K triangles).

The deformation of the model under aerodynamic loading is studied by means of a finite-element structural model (FE-model) which has been developed assuming that the fuselage and the support system (single rear-sting) flexibility can be neglected. This assumption, intended to reduce the model complexity, can be explained by the fact that the expected body deformations are negligible compared to those of the wing, and the experimental results in [42] already account for the deflection of the supporting mechanism. As a further simplification, the internal wing cavities needed to install and route measurement instrumentation are also neglected in the FE-model and the wing is considered to be solid (internal cavities are designed to have a minimum influence on the model stiffness). Regarding the modelling of the wing-fuselage junction, the wing is considered to be clamped at the interface. A detailed representation of this junction is considered to be not necessary in this study because wing deformation is only relevant for outboard wing stations. The structural model of the wing developed in this way is discretized by means of 19456 20-node hexahedral elements and 94749 nodes. The material adopted is a high-strength maraging steel with Young's modulus $E = 196 \text{ GPa}$ and Poisson's ratio $\nu = 0.30$. A view of the FE-model is presented in Figure 13. Note that in both, CFD and FE models, the wing geometry corresponds to the manufacturing or jig-shape experimental test model.

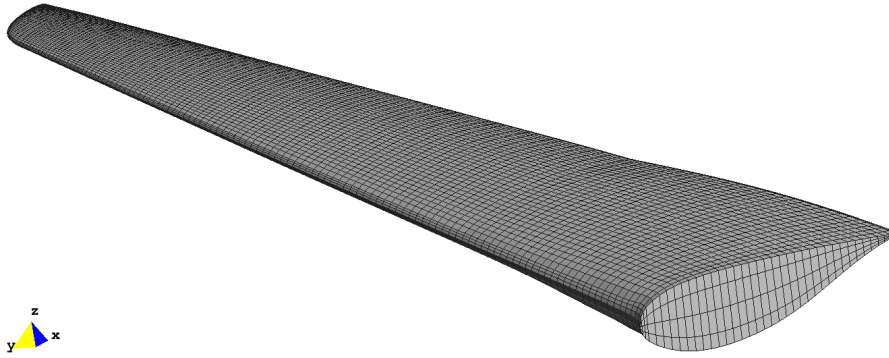


Figure 13. Structural model of the HiReTT N44 F7-1 wing (approx. 19.5K 20-node hexahedra).

7.4.2 Coupled solution strategy

The coupled solution methodology is based on the time-marching procedure presented in Sections 4 and 5. At each physical time step of the CFD-model, aerodynamic pressure loads are transferred to the FE-model, which is solved in order to obtain the displacements (the computations are performed using an in-house linear elastic solid solver). Next, the computed deformation is mapped back onto the CFD-model boundaries, the volume discretization is deformed accordingly and the flow solution is advanced in fictitious time until convergence is achieved. Then, a new interaction cycle is started by transferring the converged aerodynamic pressures to the FE-model. The iterations continue until a given displacement tolerance is achieved. Due to the fact that only the steady solution is of interest in this example, the problem is solved using large physical time steps and loads under-relaxation is also applied to accelerate the convergence of the coupled solution.

The mapping of variables between the CFD and FE grids is performed as follows. For each FE node on the wing surface the CFD boundary triangle where it lays on is identified and the aerodynamic pressure acting on that node is interpolated by using the shape functions of the underlying CFD triangle. Then, averaged pressures at the FE facets are computed and converted into surface tractions which are passed to the structural solver. Once the structure is solved, averaged (barycentre) displacements are obtained at each CFD triangle by using the displacements of the structural nodes falling on it and a smoothing procedure is accomplished to obtain the nodal values. This procedure guarantees a smooth deformation of the CFD boundary grid which matches very closely that computed in the FE-model.

The correspondence between surface FE nodes and the CFD elements needed to transfer variables between the models is obtained by using a spatial search algorithm based on bins [21]. As both structural and fluid boundary grids deform in a similar way, the spatial search

and the construction of the data structure required to transfer variables are performed only once at the beginning of the computations and stored to be used later.

7.4.3 Numerical results

The aeroelastic computations presented in the next adopt the flow conditions defined for the wind tunnel run No 204 in [42], which involves a freestream Mach number $M = 0.85$ and a Reynolds number $Re = 32.5M$. The analysis is performed for angles of attack ranging from -2.62 to 3.69 degrees and the fluid is considered to be inviscid. The stationary solution obtained for the rigid model is employed to initialize the coupled simulations.

Pressure distributions computed for the rigid and elastic models at three wing sections and angles of attack $\alpha = -2.625^\circ$, 0.0613° and 2.234° are presented in Figure 14. It is possible to observe that in the case of $\alpha = -2.625^\circ$ a poorly loaded wing (model lift is near zero) results only in a small model deformation which makes the results obtained for the rigid and elastic models very similar for the inboard wing sections. However, more marked differences are observed near the wing tip. There, the rotation of the sections (stiffness is lower) has an evident effect on the shock position which is well captured by the aeroelastic solution. For the runs at positive angles of attack the aeroelastic effects become more relevant and the negative twist modifies notoriously the wing suction area and the shock wave position, particularly at outboard spanwise sections (the impact of bending is usually small). As observed in Figure 15, this reduces the resultant model lift in the elastic computation. The results presented in Figures 14 and 15 show that the aeroelastic solution improves considerably over the conventional rigid model analysis. It should be noticed that the inviscid flow assumption adopted can introduce some discrepancies with measured data for runs at higher angles of attack where flow separation effects become more relevant (the lift curve measured in the wind tunnel test loses its linearity about $\alpha = 3^\circ$). This can be seen in the behaviour of the difference between computed and experimental lift coefficients shown in Figure 15.

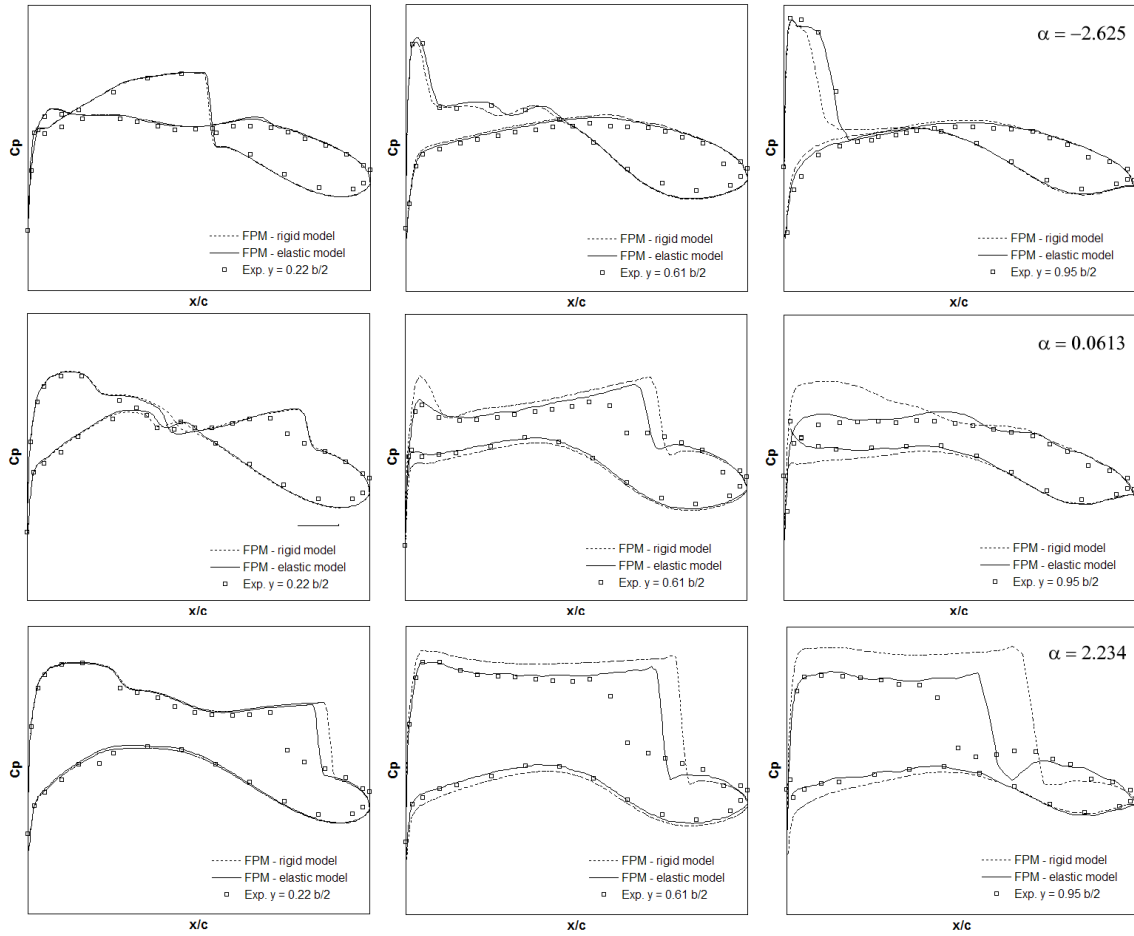


Figure 14. Effects of wing flexibility on pressure distributions computed at different spanwise stations. HiReTT N44 model, $M = 0.85$ $\alpha = -2.625^\circ$ (top), $\alpha = 0.0613^\circ$ (middle) and $\alpha = 2.234^\circ$ (bottom).

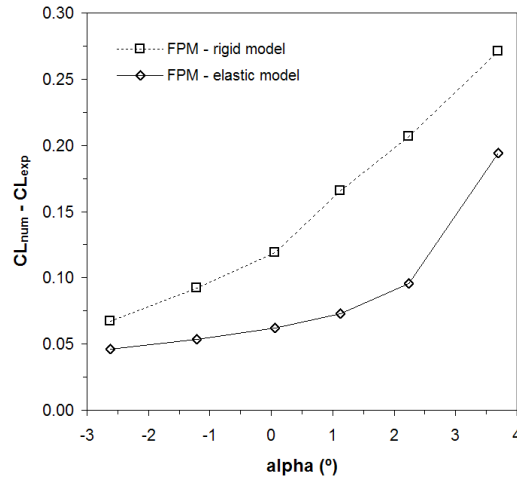


Figure 15. Differences between numerical and experimental lift coefficients computed for the rigid and elastic HiReTT N44 model at $M = 0.85$.

Twist and bend deformation of a section near the wing tip are depicted in Figure 18 for the complete alpha sweep; the trailing edge deflection (Δz_{TE}) is measured from the original (wind-off) position and the twist angle is evaluated as $\tan^{-1}[(z_{LE} - z_{TE}) / (x_{LE} - x_{TE})]$. Figure 19

shows bending and twisting effects along the wing span for $\alpha = 2.234^\circ$. A view of the original and deformed model configuration is displayed in Figure 20.

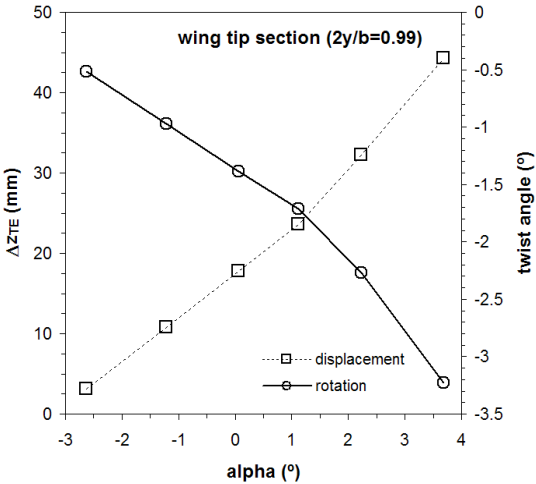


Figure 18. Rotation and vertical displacement of a wing tip section computed for the HiReTT N44 model at different angles of attack at $M = 0.85$ ($b/2=790\text{mm}$).

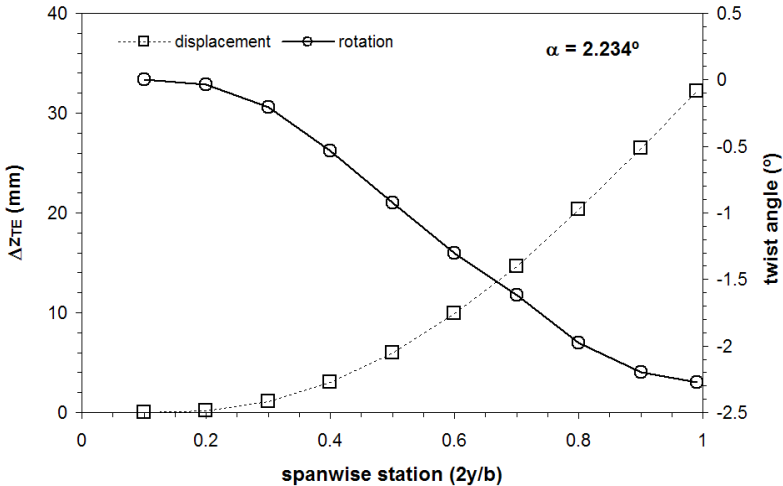


Figure 19. Spanwise wing deformation of the HiReTT N44 model ($b/2=790\text{mm}$), $M = 0.85$ and $\alpha = 2.234^\circ$.

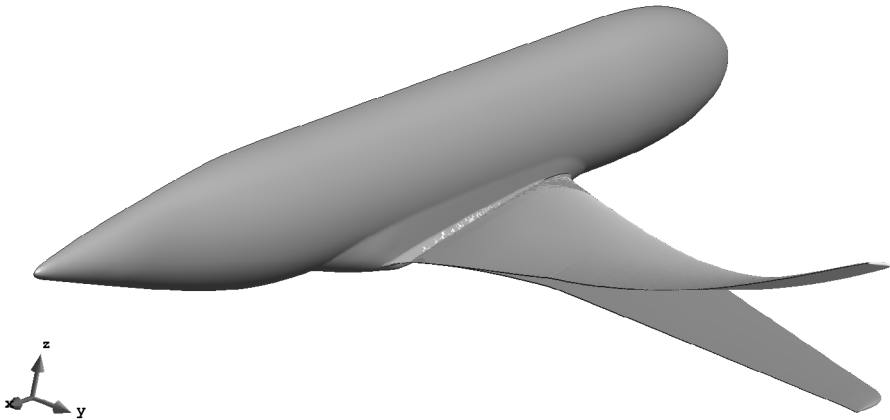


Figure 20. Original (wind-off) and deformed model configurations (5x magnification factor). HiReTT N44, $M = 0.85$ and $\alpha = 2.234^\circ$.

The convergence history of the coupled simulation for $\alpha = 2.234^\circ$ is presented in Figure 21. At each physical time step a maximum number of 3000 iterations in fictitious time or a density residual $< 1.0e-6$ is set as exit condition. A load relaxation factor of 0.5 is adopted. Starting from the rigid model solution, a few coupled iterations are required to achieve the static deformed shape of the model.

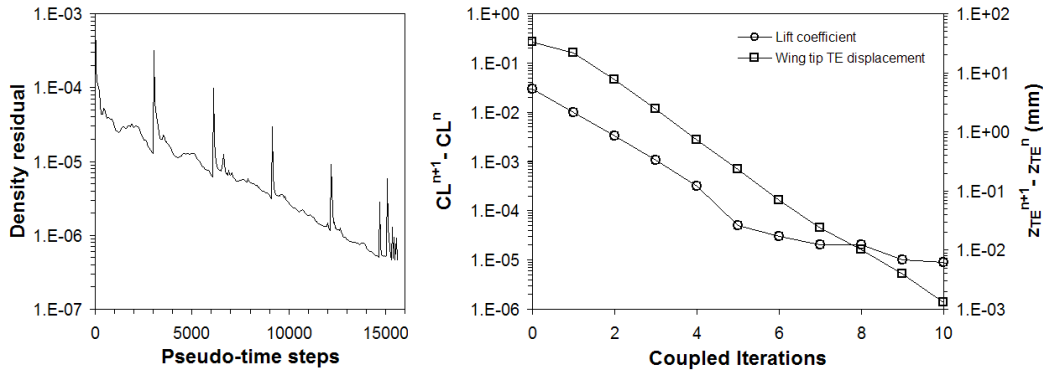


Figure 21. Convergence history for the HiReTT N44 model at $M = 0.85$ and $\alpha = 2.234^\circ$.

CPU-times and multi-core performance measurements are presented next to give an idea of the computational cost involved in the coupled simulation. With this aim, the meshless flow solution is divided into two parts, a pre-process stage which includes the construction of the numerical approximation and its related data structures (clouds of points, shape functions and derivatives) and a flow computation stage which refers to the inner Euler steps in the pseudo-time integration procedure. The structural solution stage is also evaluated. The tasks performed at each structural step involve the transfer of loads and displacements between the fluid and structural models, the solution for the displacements (FE), the deformation of the interior (CFD) volume points according to the computed model position and the update of the CFD problem data structures, mainly cloud shape functions and derivatives (cloud connectivity is kept fixed along the simulation). The runs are performed on a cluster node having 2 Intel Xeon E5645 processors @ 2.4 GHz with 12 Mb L2 cache and 48 Gb ram.

CPU-times for the pre-process, flow solution and structural computation stages as well as multi-core performances are displayed in Figure 22. As far as pre-process tasks are concerned, the inherent complexity of the operations involved in the meshless approximation probably makes it costlier than in conventional mesh-based methods. However, as these tasks are performed only once at the beginning of the computations, the cost does not affect greatly the entire simulation time. Moreover, in cases in which the data structures must be updated, for instance due to excessive domain deformation or adaptive refinement/coarsening, this is performed, where necessary, in a local manner with a low computational cost (see problem in

Section 7.3). On the other hand, the cost involved in the flow solution, which takes most of the total simulation time, does not differ from that which could be expected in a typical mesh-based technique, making the FPM technique quite competitive.

Regarding multi-core performance, it is possible to see in Figure 22 that the speedup degrades noticeably when more than 4 cores are employed. This effect is more accentuated in the pre-process stage (presumably due to the complexity of the operations involved) than in the flow computation where a higher data locality and totally independent computations make it possible for the scaling to continue, albeit at a reduced rate. The loss of performance observed when the number of running cores is augmented can be explained by the fact that the resources of the physical processors (two in this case) must be forcibly shared (specifically, the L2-cache) and this increases the likelihood of cache misses. This effect, coupled with the constant main-memory bandwidth (which depends only on the number of physical processor sockets and therefore does not scale with core count), causes the loss of performance observed. Note that a larger data set, as that employed in the present example, exacerbates the problems of competition for resources and memory. A better scalability should be expected in higher-performance multi-processors hardware platforms. Concerning the structural computation stage, the CPU-time presented in Figure 22 corresponds to a serial run because this part of the code is not fully parallelized. It should be noticed that most of the computational efforts at this stage are devoted to the solution of the linear system involved in the structural problem.

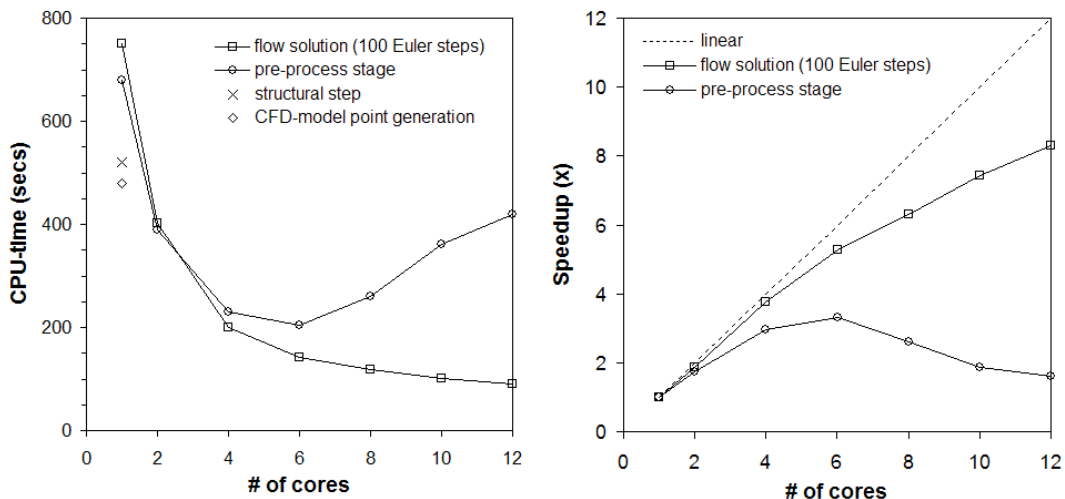


Figure 22. CPU-times and parallel performance of the coupled simulation (approx 3.1M fluid points and 95K structural nodes). HiReTT N44 model at $M = 0.85$ and $\alpha = 2.234^\circ$.

The time involved in the discretization of the aerodynamic model (also displayed in Figure 22) shows that the efficiency of the point generator could make it possible to regenerate the

complete CFD model discretization with a cost which is low if compared with the total simulation time (comparable to that in a single structural step). This could be an attractive solution approach when very large body movement or deformations must be faced. The point discretization in this example was generated using a desktop computer with AMD Opteron 246 processor @ 1.99 GHz running on a Windows 64-bit system.

8. CONCLUSIONS

A meshless adaptive Finite Point Method (FPM) for solving compressible flow problems involving moving/deforming boundaries has been presented. The overall solution methodology was designed following well-established numerical techniques but aiming at exploiting meshless features in full. Overall, the results obtained are satisfactory and show various capabilities of the FPM to solve problems involving moving/deformable boundaries; among them it is worth mentioning the easiness of combining domain deformation with adaptive strategies in a straightforward and cost-effective manner. The versatile and faster point generation technique adopted in this work not only reduces turn-around times during model setup but has also revealed that a complete regeneration of the model discretization could be an affordable solution strategy, even in large simulation problems. The application of the method to the aeroelastic solution of the HiReTT wind tunnel model has provided insight about the possibilities of the present technique to be applied in engineering problems.

The results obtained so far are encouraging but further investigation is needed to determine the real capabilities of the present technique for practical analyses. An efficient exploitation of meshless features in problems of industrial relevance would require further optimizations of the method, the use of higher-performance hardware platforms and the solution of real viscous flows. In particular, problems at high Reynolds numbers should be a focus of attention. Potential improvements to the current solution scheme should be also explored. As recent meshless implementations have shown [43-45], the application of some convergence acceleration concepts, such as implicit approaches and meshless multigrid, could bring about important benefits to the present FPM technique.

ACKNOWLEDGEMENTS

This work was partially carried out during the ALEF (Aerodynamic Loads Estimation at Extremes of the Flight Envelope) project under the European Commission's 7th Framework Programme (contract number ACP7-GA-2009-211785). The authors gratefully acknowledge the financial support and the problem data provided.

REFERENCES

- [1] Livne, E. Future of airplane aeroelasticity. *Journal of Aircraft*, 40: 1066-1092, 2003.
- [2] Fries, T. and Matthies, H. Classification and overview of meshfree methods. *Department of Mathematics and Computer Science, Technical University of Braunschweig. Inf. 2003-3*, 2004.
- [3] Batina, J. T. A gridless Euler/Navier-Stokes solution algorithm for complex two-dimensional applications. *NASA-TM-107631*, 1992.
- [4] Oñate, E., Idelsohn, S., Zienkiewicz, O. C., Taylor, R. L., and Sacco, C. A Finite Point Method for analysis of fluid mechanics problems. Applications to convective transport and fluid flow. *International Journal for Numerical Methods in Engineering*, 39: 3839-3866, 1996.
- [5] Ghosh, A. K. and Deshpande, S. M. Least squares kinetic upwind method for inviscid compressible flow. *AIAA Paper 1995-1735*, 1995.
- [6] Löhner, R., Sacco, C., Oñate, E., and Idelsohn, S. A Finite Point Method for compressible flow. *International Journal for Numerical Methods in Engineering*, 53: 1765-1779, 2002.
- [7] Sridar, D. and Balakrishnan, N. An upwind finite difference scheme for meshless solvers. *Journal of Computational Physics*, 189: 1-29, 2003.
- [8] Praveen, C. A. A positive meshless method for hyperbolic equations. *Report 2004-FM-16 ARDB Centre of excellence for aerospace CFD, Indian Institute of Science*, 2004.
- [9] Ortega, E., Oñate, E., and Idelsohn, S. A finite point method for adaptive three-dimensional compressible flow calculations. *International Journal for Numerical Methods in Fluids*, 60: 937-971, 2009.
- [10] Chen, H. Q. and Shu, C. An efficient implicit mesh-free method to solve two-dimensional compressible Euler equations. *International Journal of Modern Physics*, 16: 439-454, 2005.
- [11] Katz, A. and Jameson, A. A comparison of various meshless schemes within a unified algorithm. *AIAA paper 2009-596*, 2009.
- [12] Anandhanarayanan, K. Development of three-dimensional grid-free solver and its applications to multi-body aerospace vehicles. *Defence Science Journal*, 60: 653-662, 2010.
- [13] Kirshman, D. J. and Liu, F. Flutter prediction by a cartesian mesh Euler method with small perturbation gridless boundary conditions. *AIAA Paper 2004-0584*, 2004.

- [14] Wang, J., Chen, H. Q., and Periaux, J. A study of gridless method with dynamic clouds of points for solving unsteady CFD problems in aerodynamics. *International Journal for Numerical Methods in Fluids*, 64: 98-118, 2009.
- [15] Ortega, E., Oñate, E., and Idelsohn, S. An improved finite point method for three-dimensional potential flows. *Computational Mechanics*, 40: 949-963, 2007.
- [16] Oñate, E., Idelsohn, S., Zienkiewicz, O. C., Taylor, R. L., and Sacco, C. A stabilized Finite Point Method for analysis of fluid mechanics problems. *Computer Methods in Applied Mechanics and Engineering*, 139: 315-346, 1996.
- [17] Calvo, N. Generación de mallas tridimensionales por métodos duales. *PhD thesis, Universidad Nacional del Litoral*, 2005.
- [18] Idelsohn, S., Calvo N., and Oñate, E. Polyhedrization of an arbitrary 3D point set. *Computer Methods in Applied Mechanics and Engineering*, 192: 2649-2667, 2003.
- [19] Calvo, N. and Idelsohn, S. Generación de mallas tridimensionales en tiempo lineal. *Mecánica Computacional (AMCA)*, 23: 3048-3059, 2002.
- [20] GiD. The personal pre and post processor. Web page: <http://www.gidhome.com>, CIMNE 2012.
- [21] Löhner, R. Applied Computational Fluid Dynamics Techniques. An introduction based on Finite Element Methods. John Wiley & Sons Ltd., 2001.
- [22] Roe, P. L. Approximate Riemann solvers, parameter vectors and difference schemes. *Journal of Computational Physics*, 43: 357-372, 1981.
- [23] Turkel, E. Improving the accuracy of central difference schemes. *ICASE Report 88-53*, 1988.
- [24] Jameson, A. Time-dependent calculations using multigrid with applications to unsteady flows past airfoils and wings. *AIAA Paper 91-1596*, 1991.
- [25] Jameson, A. Artificial diffusion, upwind biasing, limiters and their effect on accuracy and multigrid convergence in transonic and hypersonic flows. *AIAA-93-3359*, 1993.
- [26] Jameson, A. and Baker, T. J. Improvements to the aircraft Euler method. *AIAA Paper 87-0452*, 1987.
- [27] Melson, N. D., Sanetrik, M. D., and Atkins, H. L. Time-accurate Navier-Stokes calculations with multigrid acceleration. *6th Copper Mountain conference on multigrid methods*, 423-439, 1993.
- [28] Venkatakrisnan, V. and Mavriplis, D. J. Implicit method for the computation of unsteady flows on unstructured grids. *ICASE Report 95-60*, 1995.

- [29] Fischer, T. A contribution to adaptive numerical solution of compressible flow problems. *PhD thesis, Universitat Politècnica de Catalunya*, 1996.
- [30] Chiu, E., Wang, Q., and Jameson, A. A conservative meshless scheme: general order formulation and application to Euler equations. *AIAA 2011-651 49th Aerospace Sciences Meeting, Orlando, Florida*, 2011.
- [31] Löhner R. *Applied Computational Fluid Dynamics Techniques. An introduction based on Finite Element Methods*. John Wiley & Sons Ltd., 2001.
- [32] Liu, X., Qin, N., Xia, H. Fast dynamic grid deformation based on Delaunay graph mapping. *Journal of Computational Physics*, 211: 405-423, 2006.
- [33] Batina, J. T. Unsteady Euler algorithm with unstructured dynamic mesh for complex-aircraft aeroelastic analysis. *AIAA Paper 89-1189*, 1989.
- [34] Blom, F. J. Considerations on the spring analogy. *International Journal for Numerical Methods in Fluids*, 32: 647-668, 2000.
- [35] Ortega, E., Oñate, E., Idelsohn, S., and Buachart, C. An adaptive Finite Point Method for the shallow water equations. *International Journal for Numerical Methods in Engineering*, 87: doi: 10.1002/nme.3171, 2011.
- [36] Karbacher, S., Seeger, S., and Häusler, G. Refining triangle meshes by non-linear subdivision. *Chair of Optics, University of Erlangen, Germany*, 2001.
- [37] Landon, R. H. NACA 0012. Oscillatory and transient pitching. *AGARD Report N° 702 Compendium of unsteady aerodynamic measurements*, 1982.
- [38] Williams, B. R. An exact test case for the plane potential flow about two adjacent lifting airfoils. *Aeronautics Research Council, Reports and Memoranda No. 3717*, 1971.
- [39] Dubuc, L., Cantariti, F., Woodgate, B., Gribben, K., Badcock, K. J., and Richards, B. E. A grid deformation technique for unsteady flow computations. *International Journal for Numerical Methods in Fluids*, 32: 285-311, 2000.
- [40] Yang, Z. and Mavriplis, D. J. Unstructured dynamic meshes with high-order time integration schemes for the unsteady Navier-Stokes equations. *AIAA Paper 2005-1222*, 2005.
- [41] Rolston, S. High Reynolds number tools and techniques for civil aircraft design - an overview of the HiReTT programme. *Air & Space Europe*, 3: 2001.
- [42] Wright, M. HiReTT Task 1.1.2: High speed wind tunnel tests on the full span model N44. *Test Report E 9011 TR019, European Transonic Windtunnel*, 2000.
- [43] Singh, M. K., Ramesh, V., and Balakrishnan, N. Convergence acceleration of mesh-less Euler solver. *12th Annual CFD Symposium, Bangalore*, 2010.

- [44] Katz, A. and Jameson, A. Multicloud: multigrid convergence with a meshless operator. *Journal of Computational Physics*, 228: 5237-5250, 2009.
- [45] Kennet, D. J., Timme, S., Angulo, J., and Badcock, K. J. An implicit meshless method for application in computational fluid dynamics. *International Journal for Numerical Methods in Fluids* (published online, DOI 10.1002/fluid.3698), 2012.