



HAL
open science

An improved Immersed Boundary Method for turbulent flow simulations on Cartesian grids

Benjamin Constant, Stéphanie Péron, Heloise Beaugendre, Christophe Benoit

► **To cite this version:**

Benjamin Constant, Stéphanie Péron, Heloise Beaugendre, Christophe Benoit. An improved Immersed Boundary Method for turbulent flow simulations on Cartesian grids. *Journal of Computational Physics*, 2021, 435, pp.110240. 10.1016/j.jcp.2021.110240 . hal-03182402

HAL Id: hal-03182402

<https://hal.science/hal-03182402>

Submitted on 26 Mar 2021

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

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

An improved Immersed Boundary Method for turbulent flow simulations on Cartesian grids

Benjamin Constant¹, Stéphanie Péron¹, Héloïse Beaugendre², Christophe Benoit¹

Abstract

In this paper, we present recent improvements of an Immersed Boundary Method (IBM) for the simulation of turbulent compressible flows on Cartesian grids. The proposed approach enables to remove spurious oscillations at the wall on skin pressure and friction coefficients. Results are compared to a body-fitted approach using the same wall function, showing that the stair-step immersed boundary provides a smooth solution compared to the body-fitted one.

The immersed boundary method has been modified to adapt the location of forced and forcing points involved in the immersed boundary reconstruction to the Reynolds number. This method has been validated either for subsonic and transonic flow regimes, through the simulation of the subsonic turbulent flow around a NACA0012 profile and the transonic flow around a RAE2822 profile and the three-dimensional ONERA M6 wing.

Keywords: Immersed Boundary Method, Wall Model, Cartesian Mesh, Turbulent Flows

1. Introduction

Within the past two decades, Computational Fluid Dynamics (CFD) has dramatically risen in the aerospace industry to simulate more and more complex configurations. Reynolds-averaged Navier-Stokes (RANS) simulations on body-fitted or conformal meshes are commonly performed
5 efficiently onto realistic geometries due to the improvements in terms of robustness and accuracy of CFD solvers within the past two decades. Today, industrial configurations take into account complex geometrical details, such as aircraft track fairings or a helicopter rotor head to evaluate their influence on the overall performances of the aircraft or rotorcraft. As a result, the mesh

¹ONERA, Université Paris Saclay, F-92322, Châtillon, France

²Univ. Bordeaux, INRIA, CNRS, Bordeaux INP, IMB, UMR 5251, F-33400, Talence, France

generation has become the bottleneck of the CFD workflow, as it requires manual interaction and
10 expertise, with several weeks of engineer time in comparison with a day for running a RANS
simulation.

For this reason, immersed boundary methods (IBM) have become popular within the last decade
as the mesh does not need to conform to the obstacles, simplifying significantly the mesh genera-
tion. This concept was introduced by Peskin [1, 2] many decades ago to simulate fluid-structure
15 interactions in the heart, where a Cartesian mesh was used to simulate blood flow. An immersed
boundary condition (IBC) is applied to take into account the obstacles lying in the flow. IBCs
can be classified into two main approaches, as referred by Mittal and Iaccarino [3]. The first class
of methods consists in introducing a forcing term within the equations, that can represent the ex-
change of momentum between the fluid and solid through a law based on the theory of elasticity
20 and is well suited for flows with immersed elastic boundaries [1, 4], but leads to a lack of stability
and accuracy for rigid bodies as the problem becomes stiff. Other methods have been developed,
among which the penalization method [5] or artificial spring [6]. The flow equations are solved on
the whole computational domain, including the solid region, and the forcing is diffused on neigh-
bouring mesh points of the solid interface. IBM can be used on the whole geometry [7, 8] or locally
25 [9] to capture the potential effects of geometrical details.

Another approach, similar to IBMs, is the cut-cell approach, which consists in computing directly
the intersection between the cells and the obstacles, which has proven efficient for inviscid flow
simulations and low Reynolds flows around complex geometries (see Coirier & Powell [10], Berger
& Aftosmis [11] and Harada *et al.* [12]). Contrary to IBM approaches, it has the advantage to be
30 conservative as the flux are directly computed onto the interface. On the other hand, cutting cells
can result in badly shaped cells, requiring some additional geometrical treatments (e.g. merging
some cells).

The use of Cartesian grids with local grid refinement in combination with embedded obstacles
(either with immersed boundary or cut-cell methods) seems to be well-suited for a high-level of
35 automation and computational efficiency [7, 13, 11]. IBMs on Cartesian grids are either used by
Euler or Navier-Stokes solvers or Lattice Boltzmann solvers.

Wall functions are usually required to solve turbulent flows using IBM on adaptive Cartesian
grids around arbitrary obstacles to restrict the number of mesh points, as the cell size is equal both
in the wall normal and tangential directions [14, 15, 16, 17]. Although the use of adaptive Cartesian

40 grids around arbitrary immersed obstacles using wall functions is conceptually attractive, spurious oscillations occur when extracting the skin quantities (pressure and skin friction), which is a strong issue for aerospace applications. Actually, wall-modeled immersed boundary approaches usually use a forcing/reference/image point in the fluid, where the flow field is interpolated [18, 19, 15, 16]. The wall function is applied to compute the friction velocity at this reference point, which is used to
45 reconstruct the tangential velocity at the IBM forced/target point. Consequently, the wall-modeled near-wall resolution is too coarse to capture correctly the nonlinear tangential velocity profile in the wall normal direction. Spurious oscillations appear in the near-wall regions, that can be dissipated far from the wall boundaries provided the numerical scheme is dissipative enough, but are visible on the skin quantities. For that purpose, Capizzano [14] proposed to linearize the velocity profile
50 down to the wall from the reference/image point, while Tamaki *et al.* [20] proposed an extension of this approach by modifying the eddy viscosity profile too in the Spalart-Allmaras turbulence model, to preserve the balance of the shear stress in the boundary layer.

Here, another approach is presented, based on geometrical considerations and an analogy with wall-modeled body-fitted approaches. Near-wall spacing and location of IBM reference/image points
55 (that will be called *IB image points* in the following) must be consistent with the Reynolds number of the CFD simulation (which can be of ten to one hundred millions for a CFD simulation around an aircraft). The proposed approach relies on the previous work of Péron *et al.* [16], adapted to determine the location of IB target points, where the solution is reconstructed, in agreement with the near-wall spacing and the Reynolds number of the simulation.

60 The paper is organized as follows: in section 2, the mathematical models and the flow solver are presented. In section 3, the wall modeling by the immersed boundary method along with the location of image points are detailed. Then the improved immersed boundary treatment is presented in section 4. Simulations of turbulent flows in subsonic and transonic regimes around 2D airfoils and around the ONERA M6 wing are presented to assess the improvements of the proposed method
65 in section 5. Results are compared with results obtained with the former IBM treatments but also with body-fitted wall-modeled solutions and body-fitted solutions where the near-wall discretization enables to capture the boundary layer correctly. Finally, conclusions are drawn in section 6.

2. Description of the flow solver

2.1. Governing equations

The steady-state mean flow is obtained by solving the compressible RANS equations [\(1\)](#):

$$\left\{ \begin{array}{l} \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0 \\ \frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\tau_{ij}) \\ \frac{\partial \rho E}{\partial t} + \frac{\partial}{\partial x_j} ((\rho E + p) u_j) = -\frac{\partial}{\partial x_j} (Q_j) + \frac{\partial}{\partial x_j} (\tau_{ij} u_i) \end{array} \right. \quad i = 1, 2, 3 \quad (1)$$

70 where ρ , u_i , p , E denote the fluid density, the mean velocity components, the pressure and the total energy per unit mass respectively, Q is the heat flux vector and τ_{ij} the total shear stress tensor, decomposed into the viscous stress and the Reynolds stress modeled here by Spalart-Allmaras model [\[21\]](#), which is chosen for its robustness and its low computational cost.

The Spalart-Allmaras model is a one-equation transport model that consists in introducing a pseudo eddy viscosity $\tilde{\nu}$ as follows:

$$\nu_t = \tilde{\nu} f_{v1}, \text{ where } f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}, \chi = \frac{\tilde{\nu}}{\nu} \text{ and } c_{v1} = 7.1, \quad (2)$$

75 where ν is the kinematic viscosity of the fluid. The damping function f_{v1} goes to 0 to the wall and 1 in the log layer, such that this pseudo viscosity corresponds to the eddy viscosity ν_t in the log layer.

2.2. Numerical methods

The compressible RANS equations are solved using FAST CFD solver developed at ONERA [\[22\]](#), [\[23\]](#), relying on a 2^{nd} -order accurate cell-centered Finite-Volume Method on multiblock structured
80 grids. One major feature of this solver is its efficiency in dealing with unsteady simulations (see [\[24\]](#)) as it enables to update 10 million cells per second per core on a single Intel Broadwell core. It also contains a solver dedicated to Cartesian grids which is 2.5 times more efficient in terms of memory and CPU time than the curvilinear block-structured solver with the same numerical ingredients. For this reason, despite the highest cell count resulting from the generation of a block-structured
85 Cartesian mesh in comparison with a body-fitted approach (structured or unstructured), this extra cost is counterbalanced by the reduced memory requirements (neither the mesh nor the metrics need to be stored, numerical fluxes are simplified...).

FastS solver relies on an hybrid MPI/OpenMP framework, where the memory is distributed (by distributing CFD grids) on the processors at high level, i.e. between nodes, whereas multithreading
 90 is managed via OpenMP within a given node. For our purpose, where Cartesian grids are uniform and contain few cells in comparison with grids resolving boundary layers accurately, Cartesian grids are distributed between the cores using OpenMP.

For RANS computations, two second-order accurate spatial schemes are considered, depending on the flow regime: the Roe-MUSCL scheme [25] for transonic flow simulations or an AUSM scheme
 95 [26] for subsonic flow simulations, which is based on a modification of the AUSM+(P) scheme (see Edwards & Liou [27]). Jacobian approximations are those proposed by Jameson & Yoon [28] and Coakley [29], whereas the linear system is solved by the LU-SGS method [28].

In our approach, the octree-based Cartesian mesh is defined by a set of overset grids with a minimum overlapping, as described in [30, 16]. Consequently, two adjacent grids of same refinement
 100 level are extended from two layers of cells on both sides and the solution is directly updated for the two layers of border cells, as depicted in figure 1(a). In the case of two adjacent grids where the left-hand side one is fine and the right-hand side one is coarse, as depicted in figure 1(b), the fine grid is extended from two cells and the coarse one from three cells to ensure that donor cells are always interior points and thus already updated. Interpolation is achieved by a 2^{nd} -order Lagrange interpolation.

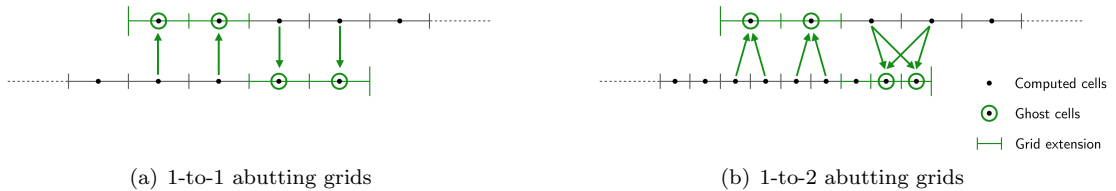


Figure 1: Ghost cell updates on octree-based Cartesian grids.

105

In the following, W will denote the flow solution $W = (\rho, \rho u_i, \rho E, \tilde{v})$.

3. Wall modeling by immersed boundaries

3.1. Principles

The immersed boundary method proposed in this paper relies on a ghost-cell direct forcing formulation, described in [16]. This approach consists in imposing the flow variables W at some points close to the obstacles to mimic a boundary condition. These points will be called *IB target* points in the following.

First, solid and fluid regions are identified geometrically, by a hole-cutting algorithm [31]. If a cell center is found lying inside the solid, the cell is marked as blanked and it will not be updated by the CFD solver during the simulation. If a Cartesian grid is entirely blanked, it is removed during the pre-processing to save memory.

To be consistent with the CFD solver, based on a five-point stencil and thus requiring two layers of ghost cells, two layers of IB target cells must be defined. The previous method described in [16] consists in marking as IB target cells the first two layers of cells at the fringe of blanked cells in the fluid part of the mesh, represented by red dots in figure 2. Flow solution W is obtained at *IB target point A* by using an *IB image point* (point B marked by a blue dot in figure 2) located within the fluid region along the wall normal direction at point A. As this point B, also known as a reference point [17], is not a discretization point, the flow field at this point is obtained by a 2^{nd} -order interpolation from its surrounding points (also called *donor points*, represented by green dots in figure 2).

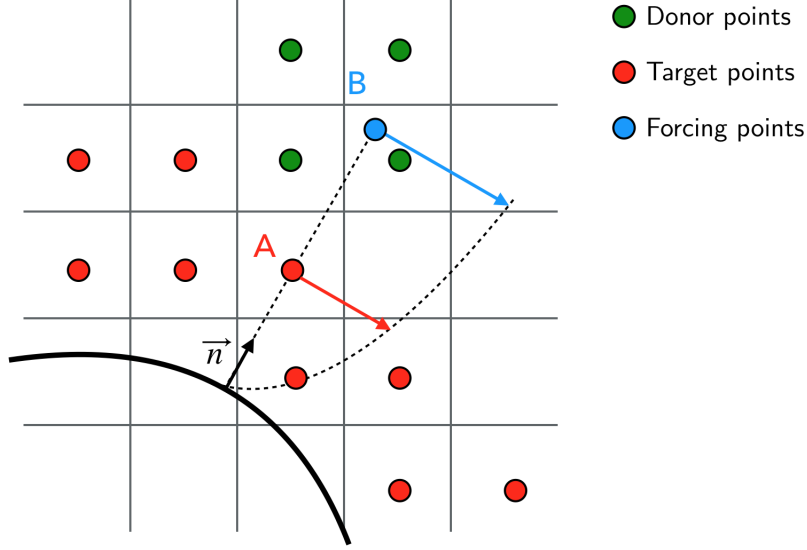


Figure 2: Direct forcing on an IB target point (A) using its corresponding image point (B).

As described in [16], solution W at image point B is used to reconstruct the flow field W at target point A. Since the pressure p is assumed to be constant in the wall normal direction in the turbulent boundary layer, we then get $p_A = p_B$. In order to take into account the compressibility effects within the boundary layer, the Crocco-Busemann relationship with the adiabatic wall condition provides the temperature at point A T_A as follows:

$$T_A = T_B + \frac{\sqrt[3]{Pr}}{2C_P} (u_{t,B}^2 - u_{t,A}^2), \quad (3)$$

where Pr is the Prandtl number and C_P is the specific heat coefficient at constant pressure. This relationship is used to recover the density at point A, that is

$$\rho_A = \frac{p_A}{RT_A}, \text{ where } R \text{ is the perfect gas constant.} \quad (4)$$

For high Reynolds number flows, solving directly the RANS equations on Cartesian grids, where the near-wall spacing is the same in the wall normal and the wall tangential directions, would lead to a mesh that is too prohibitive to capture the velocity gradient accurately. For that purpose, IBM methods on Cartesian grids usually apply wall functions, either based on algebraic models [17, 16, 32] or TBLE-based (for thin boundary layer equation) models [33, 15].

As detailed in [16], an algebraic wall function proposed by Musker [34], that fits either the log, buffer and viscous sublayers, is used to reconstruct the velocity at IB target point A. First, the friction velocity u_τ is evaluated at IB image point B and is used to reconstruct the tangential velocity u_t at target point A as

$$u_{t,A} = u_\tau f(y_A^+), \text{ with } y_A^+ = \frac{y_A u_\tau}{\nu}. \quad (5)$$

The normal velocity $u_{n,A}$ at point A is estimated by a linear reconstruction as

$$u_{n,A} = \frac{y_A}{y_B} u_{n,B}, \quad (6)$$

where y_A and y_B are the distances of points A and B to the wall boundary.

As the expression $\nu_t = \kappa u_\tau y D$ is valid in the log layer and down to the wall (where u_τ is the friction velocity, y is the wall distance, $\kappa = 0.41$ is the Von Kármán constant, D is the Van Driest damping term, with $D = (1 - e^{-\frac{y^+}{19}})$, with $y^+ = \frac{u_\tau y}{\nu}$), the pseudo-viscosity $\tilde{\nu}$ can be evaluated at target point A solving the nonlinear equation

$$\tilde{\nu}^4 - \kappa u_\tau y D \tilde{\nu}^3 - \kappa u_\tau y D \nu^3 C_{v1}^3 = 0, \quad (7)$$

derived from equation (2), as described in details in [16].

135 3.2. Location of image points

In order to reconstruct the solution $W_A^{(n)}$ at iteration n , $W_B^{(n)}$ at image point B must be interpolated from donor points already updated at iteration n , thus from computed points only. This means that the image point B has to be chosen far enough to prevent target points from being donor points. On the other hand, these points must not be too far from the wall to prevent them
 140 from being out of the boundary layer. In the literature, most approaches define image points at a given iso-distance to the wall [20, 17].

A good compromise is to set that distance to $d_B = \sqrt{2} N_{diag} \times h_{local}$ in 2D and $d_B = \sqrt{3} N_{diag} \times h_{local}$ in 3D, where h_{local} the local spacing of the mesh in the vicinity of target point A and N_{diag} depends on the spatial stencil for the flow solver. Indeed, the worst case is obtained when the
 145 normal direction from the wall to target point A is oriented along the diagonal of the cell. For a spatial scheme in the solver that requires one cell per diagonal direction, then N_{diag} is equal to 1.

However, the lack of robustness of this approach can occur for complex geometries, especially at concave corners, where the condition that target or blanked points must not be donor points for the

image point is not guaranteed. Moreover, this distance depends strongly on the local spacing near
 150 the wall, which may not be constant in the tangential direction. Typically, different components of
 a complex configuration, such as a fuselage with a wing, may not be defined by the same near-wall
 spatial resolution to save mesh points and memory.

For that purpose, a method that computes automatically the location of image points that
 ensures both previous conditions has been developed [16] and is applied here. It consists of de-
 155 termining the fringe of first computed cells surrounding the target points. This fringe is a set of
 surface grids, represented by stair steps, that define a watertight surface, even if that front inter-
 sects Cartesian grids of different levels of refinement. First, target points (displayed in figure 3 by
 red dots) are projected onto that front, resulting in blue dots in figure 3, following the wall normal
 direction \mathbf{n} , obtained by a normalization of the gradient of the distance to the wall at target points.
 160 Then, corresponding IB wall points are obtained by the same projection in the opposite direction
 (represented in black dots in figure 3).

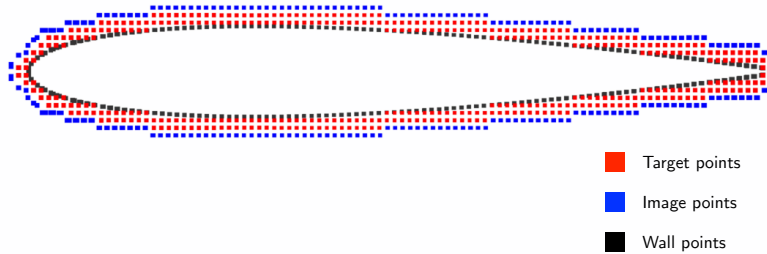


Figure 3: Location of IB points for the classical method around the NACA0012 airfoil.

3.3. Post-processing in an immersed boundary context

165 Unlike body-fitted approaches, it is not possible to extract the flow fields directly at wall bound-
 aries. A reconstruction must be performed to obtain some quantities such as skin friction or loads
 and to visualize them on the obstacles. For that purpose, the Moving Least Squares (or MLS)
 method is performed. It has been initially built up for the generation of surfaces [35] and has been
 derived to provide spatial approximations for meshless methods [36, 37].

170 In our approach, the flow quantities (pressure, density, friction velocity) are extracted at IB wall
 points at a given iteration of the flow solver. As detailed in section 3.1, and similarly to the flow
 field reconstruction implemented for target points, the wall pressure is extrapolated at each wall

point w from its corresponding image point B , which leads to $p_w = p_B$. The wall density is then recovered using the Crocco-Busemann relationship between the image point and the wall, where
 175 the tangential velocity is assumed to be zero:

$$T_w = T_B + \frac{\sqrt[3]{Pr}}{2C_P}(u_{t,B}^2), \quad (8)$$

$$\rho_w = \frac{p_w}{RT_w}. \quad (9)$$

These variables are interpolated using a 3^{rd} -order accurate MLS interpolation onto the vertices of a triangular mesh describing the obstacles on which the skin quantities are required. For each vertex VT of the tessellation, a point cloud made by at least 10 IB wall points surrounding the vertex VT is used to project the solution on that vertex using the MLS algorithm. Consequently, in
 180 order **not** to introduce interpolation errors due to strong discrepancies between the distribution of IB wall points and the tessellation (e.g. a MLS stencil 10 times larger than the characteristics length of the target triangle), the tessellation should be consistent with the Cartesian mesh discretization in the vicinity of the obstacles.

After extraction, the skin pressure and friction coefficients can be computed on the surface:

$$C_p = \frac{p_w - p_\infty}{\frac{1}{2} \rho_\infty U_\infty^2} \quad (10)$$

$$C_f = \frac{\tau_w}{\frac{1}{2} \rho_\infty U_\infty^2} = \frac{\rho_w u_\tau^2}{\frac{1}{2} \rho_\infty U_\infty^2} \quad (11)$$

185 4. Improved immersed boundary treatment

4.1. Motivation

The aforementioned IBM treatment leads to spurious oscillations on the skin pressure and friction coefficients. This has also been observed by Capizzano [14], Tamaki *et al.* [20] and Wilhelm *et al.* [17]. Actually, the numerical scheme in combination with the near-wall uniform spacing is not
 190 capable of capturing the strong variations of the tangential velocity in the wall normal direction. In addition, the stair-step distribution of IB target and image points can lead to neighbouring points that do not lie within the same sublayer of the turbulent boundary layer, e.g. the first one being

in the viscous sublayer and the second one in the log layer, leading to oscillations in the near-wall region of the solution, as their velocity gradients are different although they represent two boundary
195 points in the stencil of the first computed point above.

Capizzano [14] and Tamaki *et al.* [20] proposed a linearization of the velocity profile between the wall and the image point location and a modification of the eddy-viscosity profile to maintain the balance of the shear stress.

In this paper, another approach is proposed, based on geometrical considerations about the
200 location of IB target and image points. The novel treatment consists in applying the immersed boundary condition at a fixed y^+ , such that IB target and image points are located in the log layer, for cell count concerns. The objective is to mimic the wall-modeled body-fitted approach using the wall-modeled IBM treatment on Cartesian grids. It can be noted that this method has the advantage of being independent from the chosen wall model and the turbulence model.

205 4.2. Estimation of IB target and image point location

Contrary to the improvements of the IBM treatment found in the literature [14, 20], for which the turbulence model and the numerical scheme are modified, the present approach is based upon geometrical aspects that concern the location of target and image points with respect to the wall. It relies on the flat-plate boundary layer theory that enables to compute the height of the first cell
210 off the wall for a desired value of y^+ .

Let us consider the case where the boundary layer is fully resolved by the mesh (figure 4(left)). Then consider a Cartesian mesh of uniform near-wall spacing y_n , as depicted in figure 4(right). In order to be consistent with a mesh where cells stretch off the wall, the wall-modeled IBM treatment on the Cartesian mesh of uniform cell spacing y_n implies to locate the IB image points at distance
215 H_{mod} from the wall, hereafter called *modeling height*. This implies that the near-wall spacing alone is not sufficient to model the boundary layer correctly, since it must also be consistent with the wall distance at which the wall model is applied (at IB image points). This is why no mesh convergence was possible using the former treatment [38], as the first computed points still fall in a region where the nonlinearity of the velocity cannot be well represented by the uniform spacing.

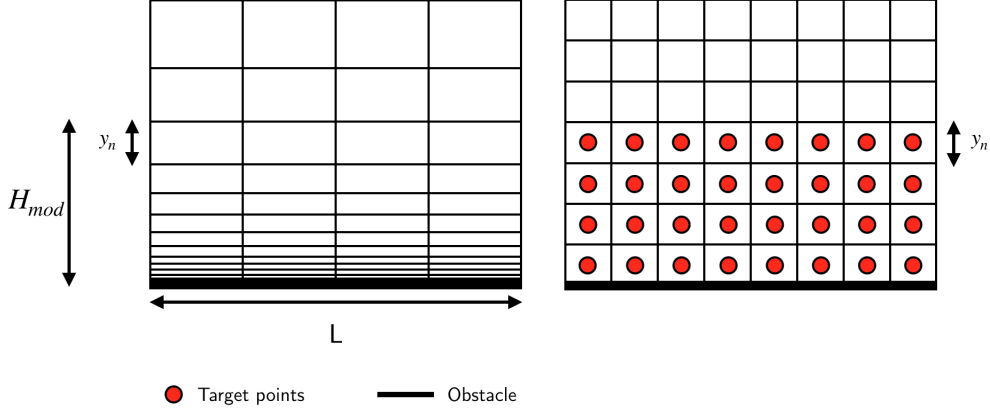


Figure 4: Comparison between Cartesian and body-fitted meshes for a flat plate of length L .

In practice, it is proposed to provide a value of y^+ for the first computed points just above the last layer of IB target cells, called y_{target}^+ . As the IB image points are located on the front of first computed cells in our approach, this implies that the value of y^+ at image points is nearly y_{target}^+ . In our framework, the near-wall spacing y_n is imposed first, as it can depend on the geometrical features that must be represented by the mesh, such as a variation of the curvature or gaps between very close bodies. Then, a flat plate approximation is performed to evaluate the distance to the wall of the first computed cells H_{mod} , given the values of the Reynolds number, the characteristic length L of the geometry and the flow conditions. For a fixed y_{target}^+ , this resulting height H_{mod} provides a first approximation of the threshold between computed points and modeled IB target points where the solution is reconstructed. In order to apply the wall model within the log sublayer, the y_{target}^+ should be set between 50 and 300. Moreover, the choice of a specific y_{target}^+ value in this region should be made so as to ensure that the cell spacing of the first calculated points inside the Cartesian mesh is consistent with the cell spacing required for a body-conformal mesh at the same distance to the wall. The algorithm is described as follows: Let denote $y^+ = \frac{u_\tau y}{\nu}$, where $u_\tau = \sqrt{\frac{\tau_\omega}{\rho}}$, with τ_ω the wall shear stress and $C_f = \frac{\tau_\omega}{\frac{1}{2} \rho_\infty U_\infty^2} = f(Re)$, the modeling height H_{mod} is then defined by:

$$H_{mod} = \sqrt{2} \frac{y_{target}^+ L}{Re \sqrt{f(Re)}}, \quad (12)$$

where the function $f(Re) = 0.058 Re^{-0.2}$ is a flat plate approximation for the turbulent boundary layer.

Conversely, the optimal y_{target}^+ for a given near-wall spacing y_n can be obtained by using the same flat plate approximation. Considering that the spacing of the body-fitted mesh displayed in figure 4(left) is following a geometric progression with common ratio $q = 1.2$, and given that y_n also corresponds to its n-th cell, we can deduce that $y_n = y_0 q^n$, where y_0 is the height of the first cell. Since this first cell is generally chosen to impose $y^+ = 1$ at the wall and considering previous calculations, we may write:

$$y_0 = \sqrt{2} \frac{L}{Re \sqrt{f(Re)}}. \quad (13)$$

$H_{mod} = \sum_{i=1}^n y_i$ can also be rewritten:

$$H_{mod} = y_0 \frac{1 - q^{n+1}}{1 - q} = \frac{y_0 - q y_n}{1 - q}. \quad (14)$$

Finally, using equations 12, 13 and 14, the optimal y_{target}^+ is given by:

$$y_{target}^+ = H_{mod}/y_0 \quad (15)$$

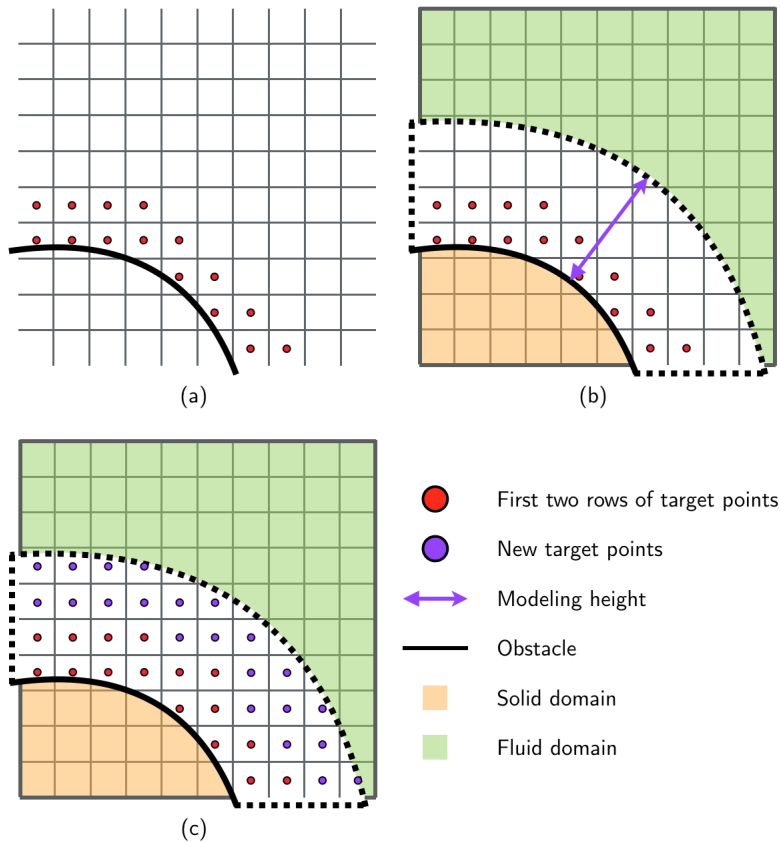


Figure 5: IB target point: former location (a) and new positioning (c) according to the threshold distance between the modeled and computed regions (b).

Figure 5 describes the modified IBM preprocessing.

- The former method consists in marking the two first layers of cells at the fringe of solid cells as IB target points (figure 5(a)).
- The threshold distance to wall H_{mod} is estimated using the flat plate approximation (figure 5(b)).
- IB target points are the two layers of cells just below the surface of distance H_{mod} (figure 5(c)).

For three-dimensional test-cases, a change in the near-wall spacing is often required in the

tangential direction to the wall to prevent a huge amount of mesh points. As the Cartesian mesh is
 octree-based, one refinement level can be crossed between neighbouring grids. Thus, in the vicinity
 of the border between a fine and a twice as coarse grid near the wall, the IBM reconstruction is
 achieved at IB target points using image points that might be located in different regions of the
 boundary layer, as depicted in figure 6-(left). This leads to a strong discrepancy in the estimation
 of the velocity at target points that are neighbours in the tangential direction but whose refinement
 level is not the same. This can be observed on the skin friction coefficient, where a spurious
 oscillation occurs in the vicinity of the change of the mesh refinement level near the wall. This will
 be discussed in section 5.3 on the ONERA M6 wing configuration, where the leading edge is refined
 to take into account the variation of the curvature while the other near-wall regions are twice as
 coarse to avoid the cell count to be too high.

A consequence of the novel treatment is the location of IB image points at roughly the same
 distance to the wall, as depicted in figure 6-(right). The location of the new target points is also
 modified, resulting in nearly the same threshold height between the modeled and computed regions,
 independently from the refinement level of the near-wall grid. In section 5.3 the removal of the
 spurious oscillations on the skin friction in the vicinity of the change of refinement level will be
 demonstrated.

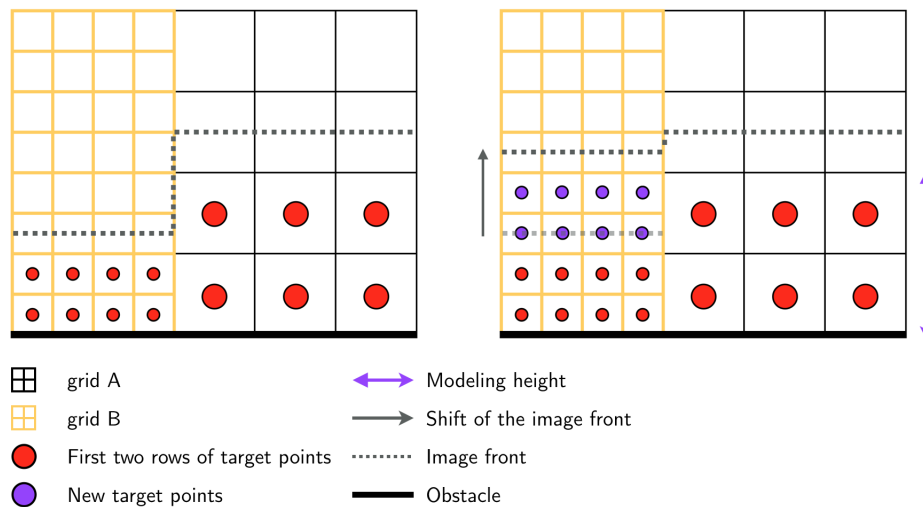


Figure 6: IBM location: impact of the change of refinement level in the tangential direction to the wall.

As several IBM treatments will be compared in the following, table 1 summarizes their characteristics regarding the location of IB target and image points:

-	Layers of target points	Front type of image points
$F0$	2	fixed distance to the wall (smooth front)
$F1$	2	stair-step front
$F2$	3	stair-step front
$F42$	adaptive	stair-step front

Table 1: Classification of algorithms for IB target and image point location.

265 An illustration of the location of IB target and image points is provided in figure 7 for the four IBM treatments in the case where a change in the refinement level occurs tangentially to the wall.

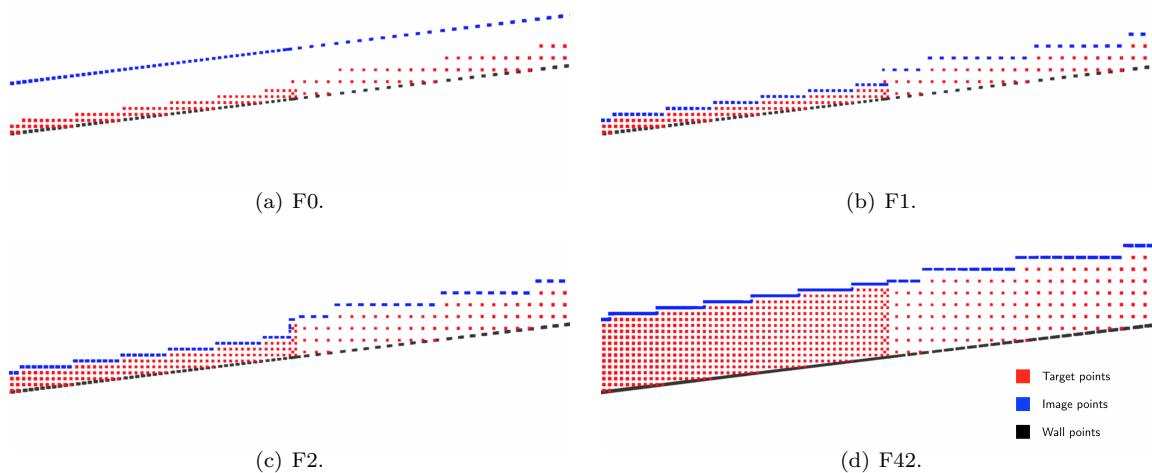


Figure 7: Comparison of the location of IB points for the four treatments near a wall boundary (in black), with a change of refinement level along the wall.

5. Numerical results

In order to demonstrate that the modified near-wall geometrical treatment of the proposed immersed boundary method enables to improve significantly the distribution of the skin pressure and friction, bidimensional and three-dimensional test-cases are considered in subsonic and transonic

regimes. The influence of the location of IB target points and image points is highlighted by a comparison between the different approaches for determining their positions. Results are compared with a body-fitted approach where the boundary layer is resolved by the mesh. In addition, a comparison between the IBM results and a body-fitted approach with a wall function applied at same y^+ assesses the influence of the wall function onto the accuracy of the solution. This study demonstrates that the improved IBM approach provides similar results to wall-law body-fitted simulation, despite the stair-step fashion of IB target and image points.

Steady RANS simulations are performed for IBM on Cartesian grids and body-fitted mesh using FAST solve, with the same numerical ingredients, in particular the same spatial schemes and turbulence model. Musker’s wall function is applied either within the body-fitted approach for the wall-law simulations and within the IBM approach on Cartesian grids to reconstruct the velocity and friction at IBM points.

In the following, most of the IBM simulations using the improved algorithm are performed with $y^+ = 100$ for the image points along the geometry.

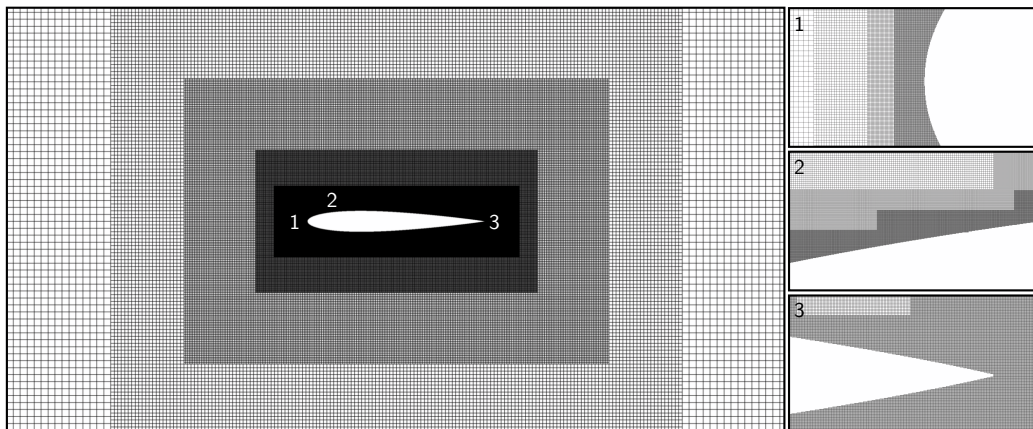
Imposing $y^+ = 100$ enables to apply the wall model in the log sub-layer and at the same time to guarantee that the cell spacing of the first calculated points above target points are in agreement to the cell spacing required for a body-conformal mesh at the same distance to the wall.

Two bidimensional turbulent flow simulations around airfoils in the subsonic and transonic regimes are considered to assess the improvements in terms of skin pressure and friction of the proposed method. Results are compared to a solution obtained on a wall-resolved conformal mesh and to experiments. The modified immersed boundary treatment is compared with previous methods. The influence of the near-wall spacing and the angle of attack is also considered. A comparison between a body-fitted wall-modeled solution and an IBM solution is also achieved to make a distinction between the influence of the wall model and of the stair-step IBM. Finally, a three-dimensional turbulent flow simulation around the ONERA M6 wing in the transonic regime is performed. Results are compared with a body-fitted solution and experimental data.

5.1. Simulation of the subsonic turbulent flow around a NACA0012 profile

The first test-case is the subsonic turbulent flow around a NACA0012 airfoil under the following flow conditions: freestream Mach number $M_\infty = 0.15$, a zero angle of attack and Reynolds number based on the chord is $Re = 6$ millions. The Cartesian mesh is automatically generated according

to the mesh resolution imposed around the airfoil. The extent of the computational domain corresponds to $40c$, where c is the chord length and is equal to 1. Farfield boundary conditions are defined at the external boundaries and an immersed boundary condition is applied on the airfoil.



305

Figure 8: Views of the IBM Cartesian mesh around the NACA0012 airfoil.

The steady RANS equations are discretized with the 2^{nd} -order accurate AUSM spatial scheme described in paragraph 2.2. An implicit backward Euler scheme is performed to solve the pseudo-time integration.

-	IBM5	IBM6	IBM7	IBM8	BF
Minimum spacing h_{min}/c	0.0002	0.0001	0.00005	0.00002	10^{-6}
Number of pts (millions)	4.1	7.8	16.4	34.9	16.1

310

Table 2: Cartesian mesh characteristics for the subsonic NACA0012 airfoil test-case

5.1.1. Influence of the location of IBM target and image points

This paragraph aims at evaluating how the location of target and image points is crucial to capture accurately skin quantities. The mesh settings for this test-case are gathered in table 2.

Figure 9 displays the isocontours of the Mach number for a body-fitted simulation where the boundary layer is solved by the mesh and an IBM wall-modeled simulation on Cartesian mesh IBM7, showing a good agreement between both solutions.

Figure 10 compares the different IBM treatments for the same near wall spacing (10^{-4}), which corresponds to the IBM6 Cartesian mesh defined in table 2. FAST IBM6 F0 and FAST IBM6 F1 results present strong oscillations along the chord for both skin coefficients, especially near the leading edge. Despite the oscillations, the trend of the skin pressure distribution compares fairly to the body-fitted solution FAST BF. But the skin friction coefficient is over-estimated and deviates significantly from the reference solution near the leading edge.

The F2 algorithm appears as a better option as it suppresses all the oscillations on FAST IBM6 F2 for both coefficients for the chosen near-wall resolution. The skin pressure distribution matches the reference solution, whereas it still results in an over-estimation of the skin friction near the leading edge, at a lower level than F0 and F1 algorithms. The improvement is probably due to a better agreement with the location of IB image points and the chosen near-wall resolution. We observed that the spurious oscillations eventually appeared when the near-wall resolution was increased.

Finally, the improved method is the only treatment among the four treatments considered here that is capable of both removing spurious oscillations and matching the body-fitted reference, as the skin friction coefficient is no longer over-estimated near the leading edge for FAST IBM6 F42.

Furthermore, table 3 confirms that ensuring that both target and image points lay in the same sub-layer is indeed important to remove spurious oscillations, especially in regions where the wall presents a high curvature. The average y^+ obtained with F0 might appear sufficient to ensure such a requirement, but one should keep in mind that the IB image points are projected at a fixed distance to the wall during preprocessing, as explained in table 1, whereas the IB target points can lie in the log layer but also in the viscous sublayer due to their stair-step distribution near curved boundaries.

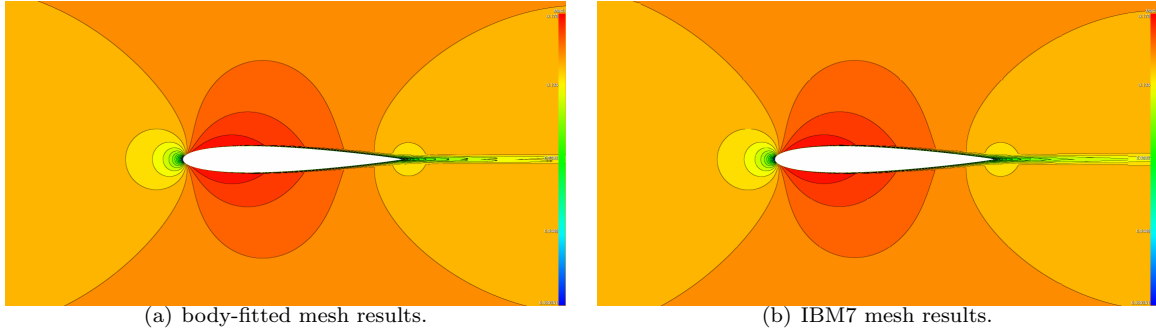


Figure 9: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: Mach number flowfield.

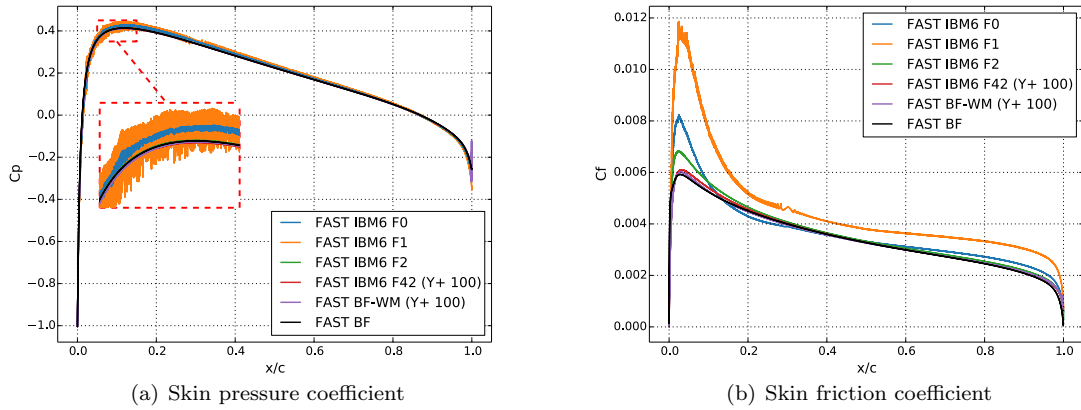


Figure 10: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: influence of the location of target and image points.

-	IBM6 F0	IBM6 F1	IBM6 F2	IBM6 F42	BF-WM
y^+	82	54	66	123	124

Table 3: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: mean values of y^+ for image points with respect to the location of image and target points.

5.1.2. Influence of the near-wall resolution

In this study, the improved treatment (*F42*) is chosen. Several uniform near-wall resolutions are evaluated, from coarse (IBM5) to ultra-fine (IBM8), summarized in table 2. In [38], Renaud *et al.* observed no mesh convergence for the same test-case using the F0 IBM treatment. They observed

a good agreement of the solution in terms of pressure coefficient, but the friction coefficient was under-predicted for coarser meshes and became over-predicted for finer meshes. Spurious oscillations appeared, similar to what was observed previously for F0 treatment. The same general behaviour has also been reproduced ever since with the F1 and F2 treatments.

350 Here, whatever the mesh refinement, FAST IBM F42 correctly reproduces the pressure coefficient and the friction coefficient distributions, as displayed in figure 11. The obtained solutions are in agreement with the reference FAST BF solution. The wall model slightly influences the friction coefficient, in particular near the leading edge.

355 Table 4 also proves that the new algorithm enables to locate image points according to the desired $y_{target}^+ = 100$.

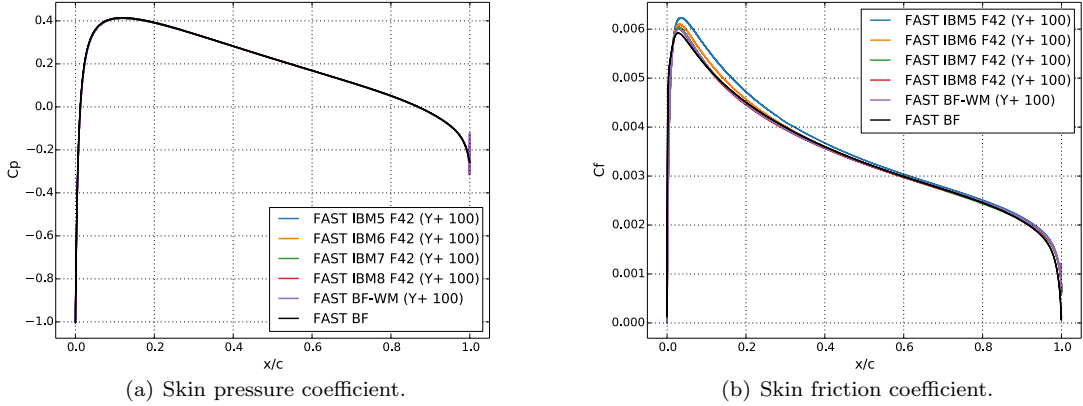


Figure 11: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: influence of the near-wall resolution.

-	IBM5 F42	IBM6 F42	IBM7 F42	IBM8 F42	BF-WM
y^+	134	123	117	115	124

Table 4: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: mean values of y^+ for image points with respect to the near-wall resolution.

360 Figure 12 displays longitudinal velocity profiles at several positions along the chord for FAST IBM7 F42 simulation, showing that these velocity profiles obtained with the new IBM treatment are in good agreement with the FAST BF-WM solution, where the body-fitted mesh is used in combination with the same wall model as for the IBM approach. Yet, variations between these two wall-modeled

solutions and the FAST BF reference appear at the beginning of the airfoil, which could explain the small discrepancies observed on the skin friction coefficient previously.

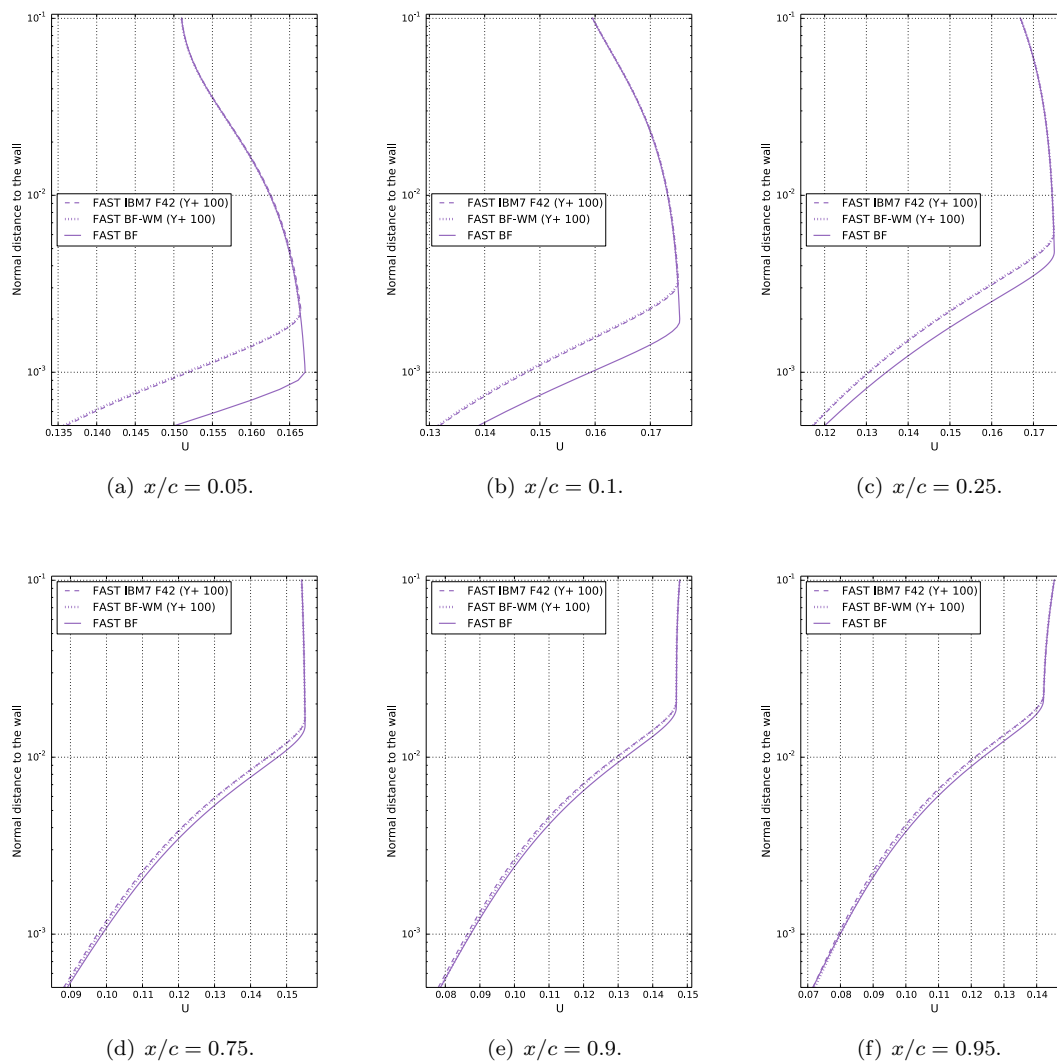


Figure 12: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: x-velocity profiles.

5.1.3. Influence of the wall model

365 Following the previous observations, figure 13 compiles the information on the impact of the wall model on the skin friction prediction. To do so, various y_{target}^+ are studied with the body-fitted

mesh, first for different boundary layer subregions (figure 13(a)), then inside the log sub-layer (figure 13(b)).

It can be observed that the skin friction distribution depends on the sublayer where the IB image points fall. This confirms that the IB target and image points should both lay in the same given sublayer and can explain the spurious oscillations previously encountered.

First, in the inner subregion ($y^+ = 1$), that is unattainable for a Cartesian mesh, the wall model is capable of accurately predict the close wall physics. Eventually, bringing the IB target points closer to the wall increases the quality of the solution that correctly converges toward the FAST BF reference.

Then, in the buffer zone ($y^+ = 10$), the forcing points fall in an area that is not well defined by the wall model and the solution eventually differs from the reference.

Finally, in the log sub-layer ($y^+ \in [50, 300]$), the solution is again well predicted by the wall model. However, as noticed before, solutions slightly vary near the leading edge from one another depending on the y_{target}^+ . According to the figure 12, a reasonable hypothesis would be that the boundary layer has yet to be fully turbulent and remains laminar near the leading edge. Hence, a new wall model might be investigated in the future to obtain the correct modeling in this area.

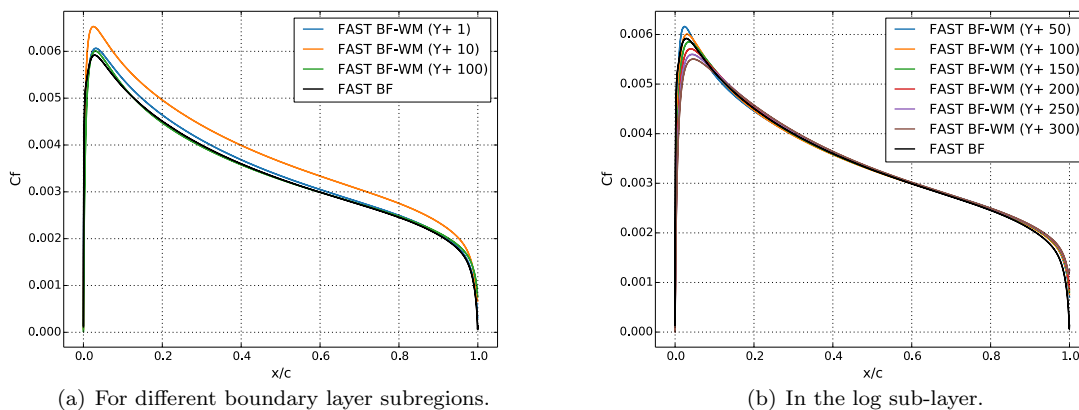


Figure 13: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: effect of the wall model on the skin friction distribution on a body-fitted mesh for different modeling heights.

5.1.4. Influence of the angle of attack

Figures [14](#) and [15](#) summarize the behaviour of the improved IBM treatment when increasing
385 the angle of attack, the other flow and solver parameters being unchanged. Without any additional
treatment, solution FAST IBM7 F42 demonstrates the robustness of the method as the solution
is still accurate as the angle of attack increases. Yet, as already highlighted, a slight difference is
observed near the leading edge for the friction coefficient. However, since FAST IBM7 F42 and
FAST BF-WM match for all the angles of attacks that are considered here, it can be assessed
390 that this is due to the wall model, as mentioned earlier, rather than to the Cartesian mesh or the
stair-step IBM.

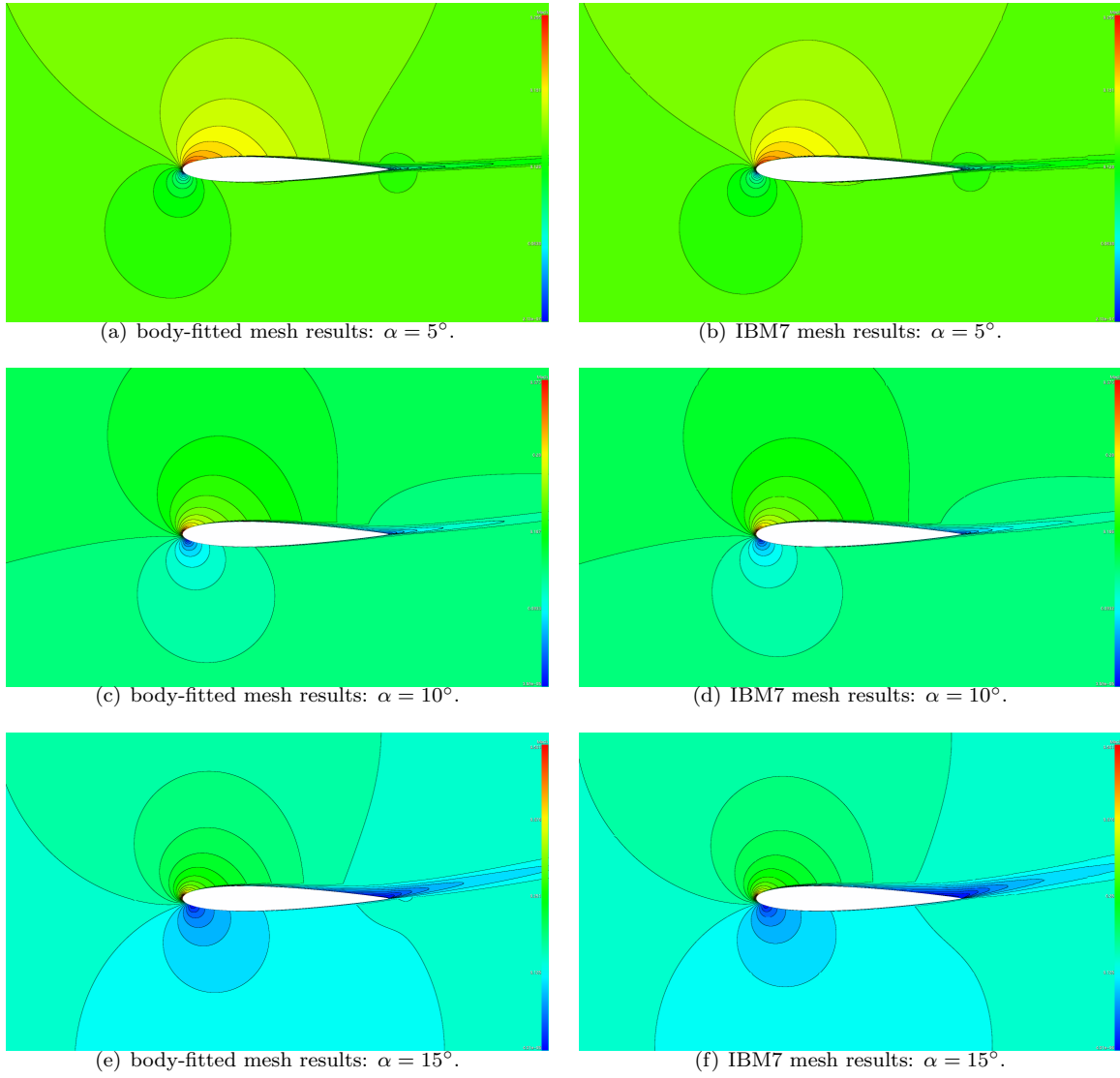
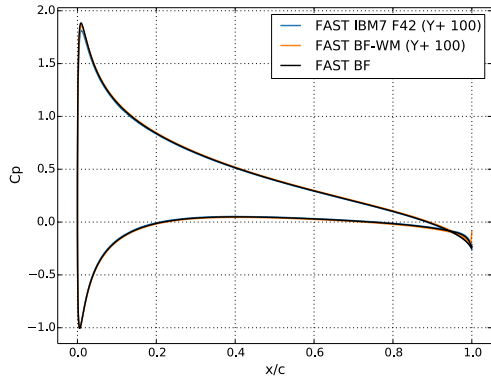
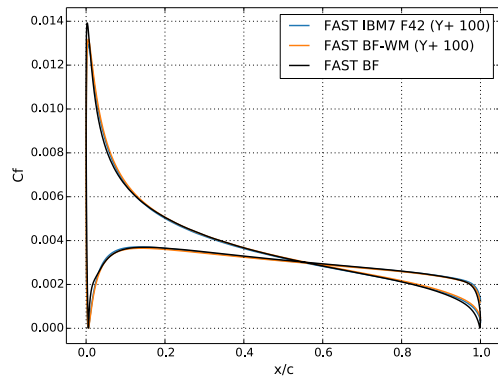


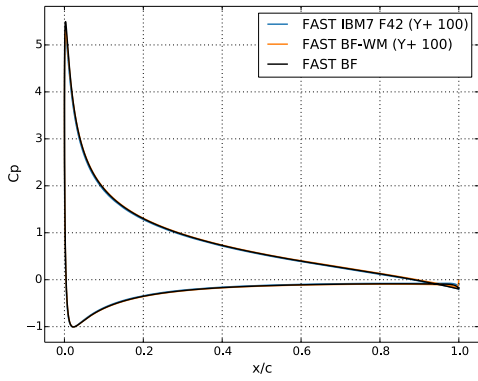
Figure 14: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: influence of the angle of attack α on the Mach flowfield.



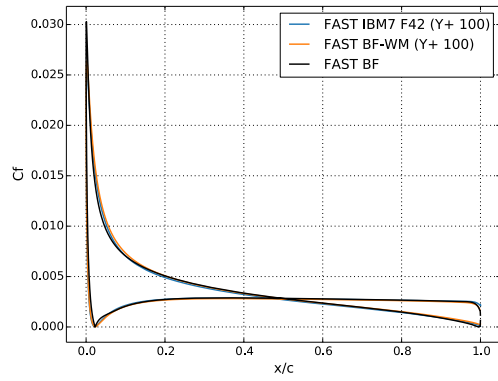
(a) Skin pressure coefficient: $\alpha = 5^\circ$.



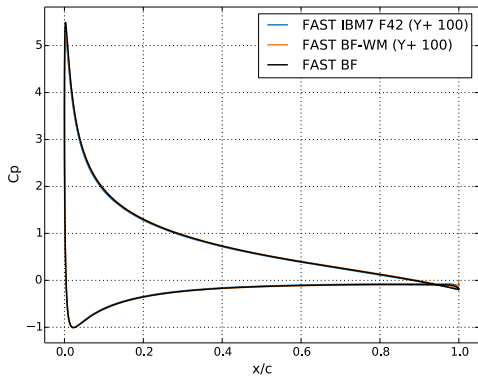
(b) Skin friction coefficient: $\alpha = 5^\circ$.



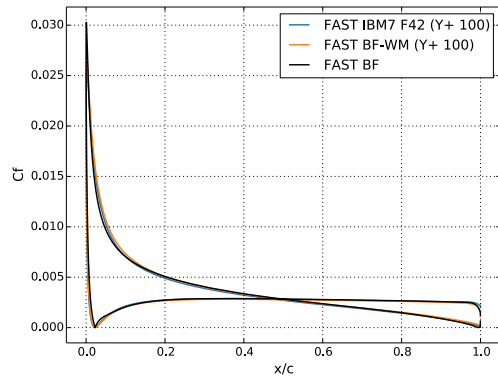
(c) Skin pressure coefficient: $\alpha = 10^\circ$.



(d) Skin friction coefficient: $\alpha = 10^\circ$.



(e) Skin pressure coefficient: $\alpha = 15^\circ$.



(f) Skin friction coefficient: $\alpha = 15^\circ$.

Figure 15: NACA0012 airfoil, $M_\infty = 0.15$, $Re = 6$ millions: influence of the angle of attack α .

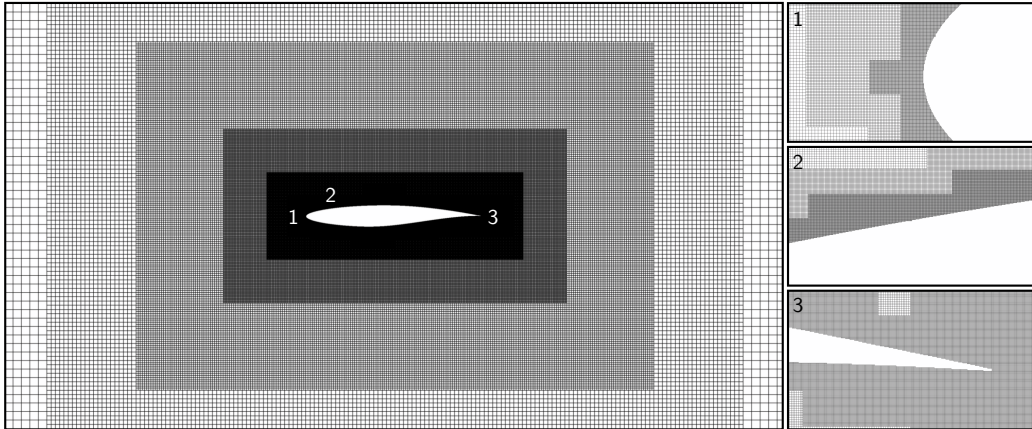


Figure 16: Views of the IBM Cartesian mesh around the RAE2822 airfoil.

5.2. Simulation of the transonic turbulent flow around a RAE2822 airfoil

The second test-case is the simulation of the transonic turbulent flow around a RAE2822 airfoil at freestream Mach number $M_\infty = 0.729$, an angle of attack α of 2.79° and a Reynolds number
 395 based on the chord c equal to $Re_c = 6.5$ millions. This test-case aims at evaluating the accuracy of the method in the case of an interaction between the boundary layer and a shock, that is located at mid-span of the airfoil on the suction side.

The Cartesian mesh is automatically generated with an extent of $20c$ and a uniform near-wall spacing. Figure 16 presents a global view around the airfoil and close-up views in the vicinity of
 400 the leading edge, the suction side and the trailing edge, showing more clearly the distribution of the refinement levels in the near-wall region. The method is evaluated for different near-wall spacings, for which the mesh characteristics are summarized in table 5.

Similarly to the previous validation case, the steady-state RANS equations are solved using FASTS solver. A backward Euler implicit scheme is applied in combination with the Roe-MUSCL
 405 scheme for the spatial discretization. The standard Spalart-Allmaras turbulence model is used and Musker's wall model is applied for the reconstruction of the velocity at IB target points.

-	IBM6	IBM7	IBM8	BF (body-fitted)
Minimum spacing h_{min}/c	0.0001	0.00005	0.00002	10^{-6}
Number of points (millions)	7.2	15.9	36.9	0.137

Table 5: Cartesian mesh characteristics for the RAE2822 airfoil test-case.

Figure 17 compares the iso-contours of the Mach number for the IBM wall-modeled simulation and the wall-resolved body-fitted solution, showing a good agreement between both results, even if the shock appears thinner for the IBM simulation, as in its vicinity, the mesh is finer than the body-fitted mesh.

Figure 18 demonstrates the mesh convergence of the method to the reference body-fitted solution. The shock location is better predicted as the near-wall spacing increases. At convergence, the skin pressure distribution is identical to that of a body-fitted solution where the boundary layer is modeled by a wall function. The skin friction distribution, depicted in figure 18-(b), is overall underestimated compared to the body-fitted solution where the near-wall region is resolved by the mesh. As the skin friction is identical to that of a body-fitted wall-modeled solution, these discrepancies are probably linked to the wall model, since it does not take into account the pressure gradient and convection in the vicinity of the shock, as discussed by Capizzano 14.

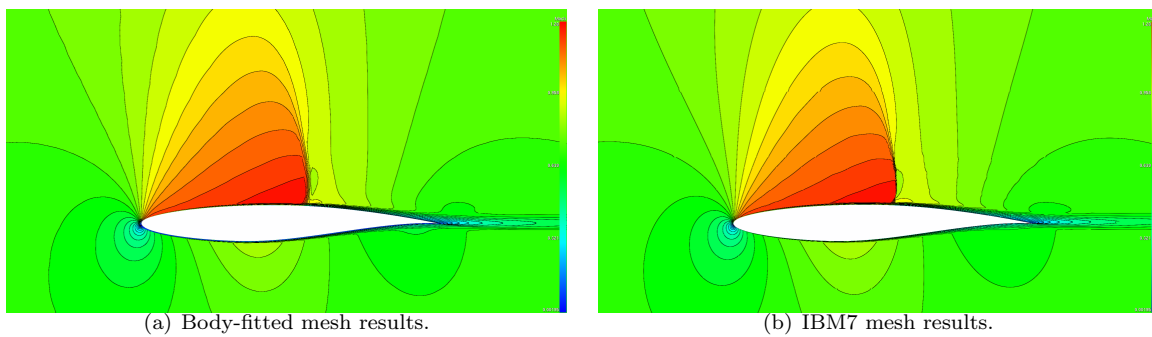


Figure 17: RAE2822 airfoil: comparison of iso-contours of the Mach number between wall-modeled IBM and wall-resolved body-fitted solutions.

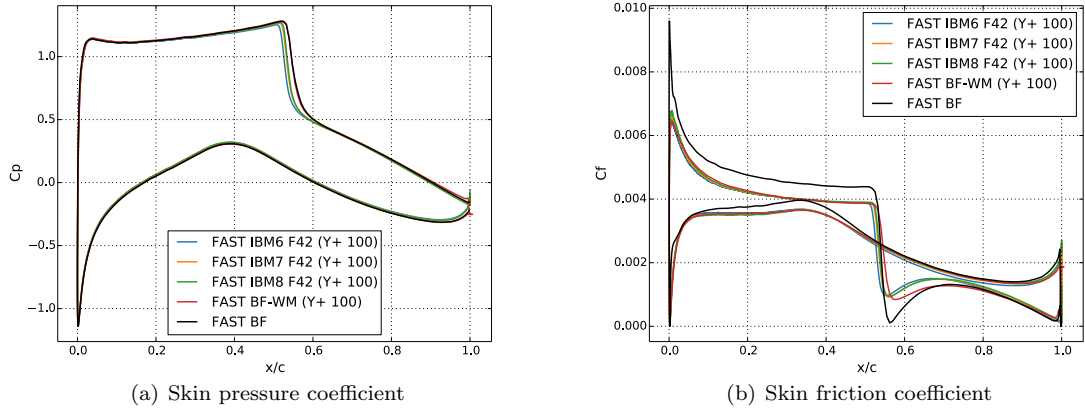


Figure 18: RAE2822 airfoil: influence of the near-wall resolution.

5.3. Simulation of the transonic turbulent flow around the ONERA M6 wing

The final test is the attached transonic turbulent flow around a ONERA M6 wing at freestream Mach number $M_\infty = 0.839$, angle of attack $\alpha = 3.06^\circ$ and Reynolds number $Re_c = 11.72$ millions. This test case has been originally described in [39] by Schmitt and Charpin in 1979 and is still widely popular in the CFD community to evaluate turbulent models in CFD solvers [40].

The Cartesian mesh is automatically generated with an extent of $15c$. It is composed of 1.7 billion points, with a near-wall spacing of $h_w = 4 \times 10^{-4}c$ around the wing and a local refinement of $h_{LE} = 2 \times 10^{-4}c$ in the vicinity of the leading edge. A slice at spanwise section $y/b = 0.55$ of the final mesh can be found in figure 19, where the change of refinement level along the chord can be seen.

The regular surface mesh of the ONERA M6 wing is also modified with INRIA surface remeshing tool MMGS [41] such that the discretization of the surface is compliant with the near-wall spacing of the Cartesian mesh, which is usually necessary to prevent from solution faceting due to the projection of IB target points onto the triangular mesh which can be observed if the triangles are roughly ten times larger and more than the near-wall cells.

Here, IBM simulations using the improved method are performed with a target $y_{target}^+ = 1000$ for the image points. Actually, the near-wall cell spacing h_{min} must be in agreement with y_{target}^+ , such that the first calculated points above IB target points is compliant with the cell spacing required for a body-conformal mesh at the same distance y_{target}^+ . In order to prevent a huge number of

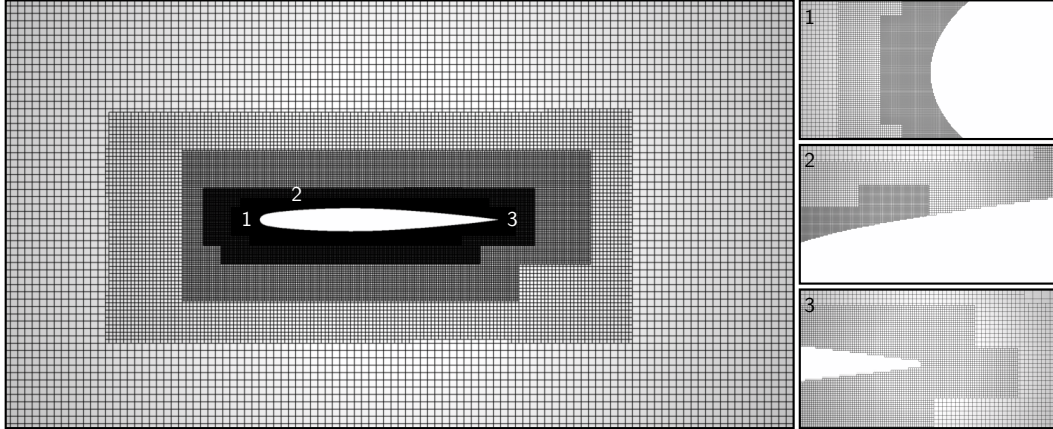
440 mesh points, the near-wall spacing around the wing is set to $h_w = 4 \times 10^{-4}c$, corresponding to $y_{target}^+ = 1000$. Note that the ideal y_{target}^+ is imposed by the coarsest near-wall spacing h_w and not h_{LE} in the vicinity of the trailing edge. As already observed on bidimensional test-cases, this value of y_{target}^+ should have been decreased for a better prediction of the skin friction in this region. But imposing a value $y_{target}^+ = 500$ would eventually lead to spurious oscillations in the near-field
445 around the rest of the wing as the discretization is not sufficient to compute accurately the gradients given that distance y_{target}^+ .

The simulations have been performed with the same solver parameters as for the RAE2822 test case, on 20×28 Intel cores of the ONERA SATOR cluster.

Final IBM results are compared with experimental data from [39] and a solution obtained on
450 a fine wall-resolved conformal mesh proposed on the NASA Turbulence Model Resources website using the CFL3D solver and the standard Spalart-Allmaras turbulence model.

Similarly to the previous bidimensional cases, no extra cost was observed during the pre-processing when using the improved method. However, for the present tridimensional case, a slight increase of 2% of the CPU time was observed with the F42 algorithm in comparison with the
455 F1 algorithm.

Finally, it has to be noted that the equation [12] used to assess the modeling height with the F42 algorithm, still applies in a satisfactory manner for tridimensional cases. Hence, no further developments were undertaken to extend the validity of our method from 2D to 3D simulations.



460

Figure 19: Views of the IBM Cartesian mesh around the ONERA M6 wing at $y/b = 0.55$.

Figure 20 displays the isocontours of the Mach number at a constant spanwise section $y/b = 0.55$, showing no oscillations near the obstacles.

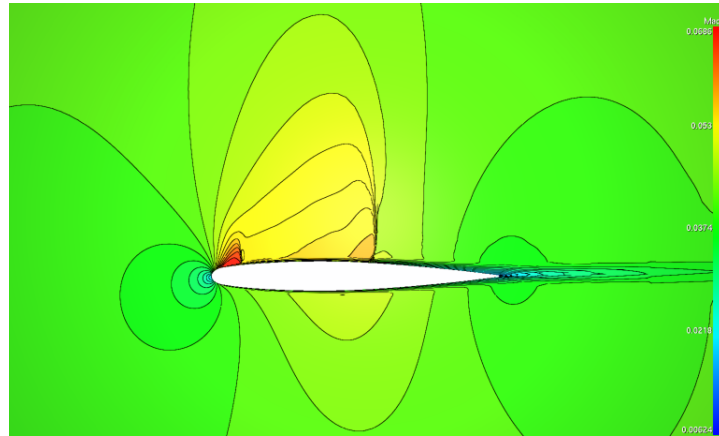


Figure 20: ONERA M6 wing, $M_\infty = 0.839$, $Re = 11.72$ millions: Mach number flowfield at $y/b = 0.55$.

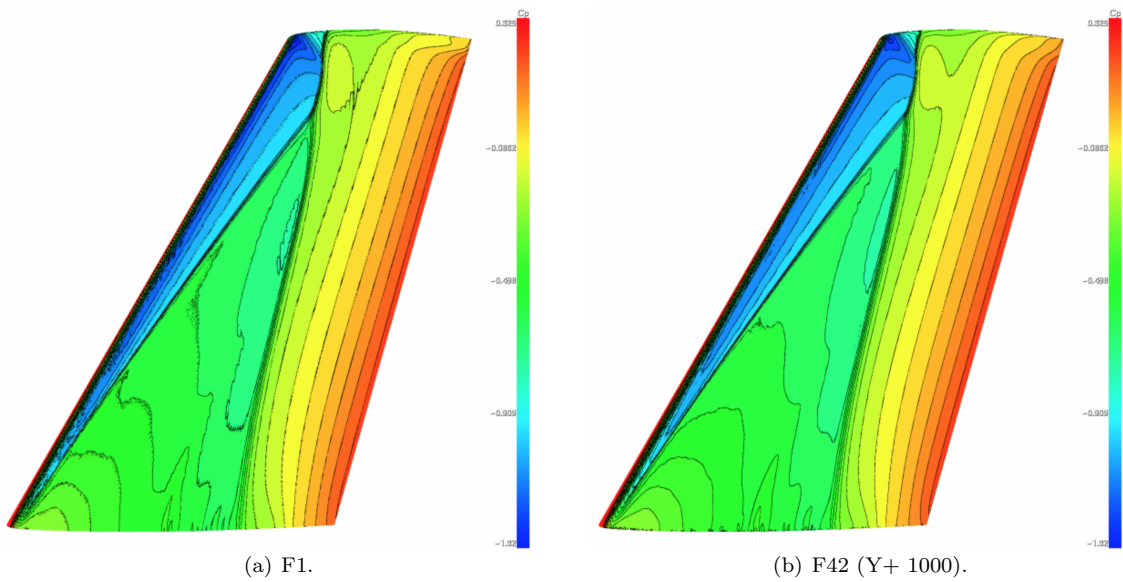
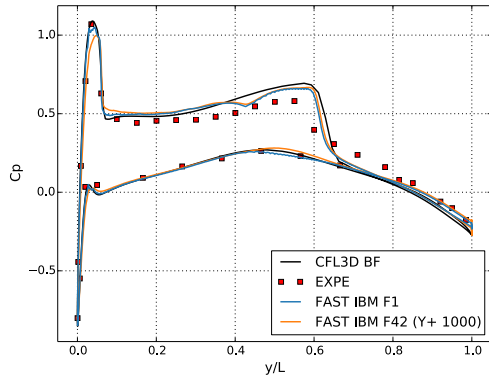
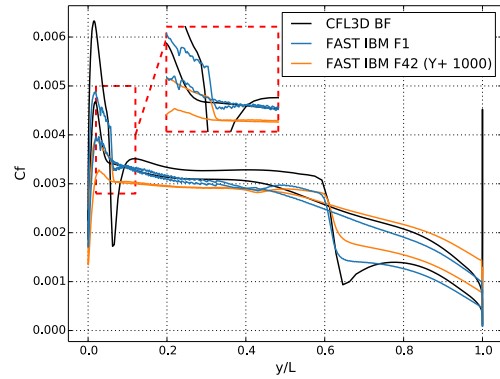


Figure 21: ONERA M6 wing, $M_\infty = 0.839$, $Re = 11.72$ millions: comparison of C_p distributions between two wall-modeled IBM.

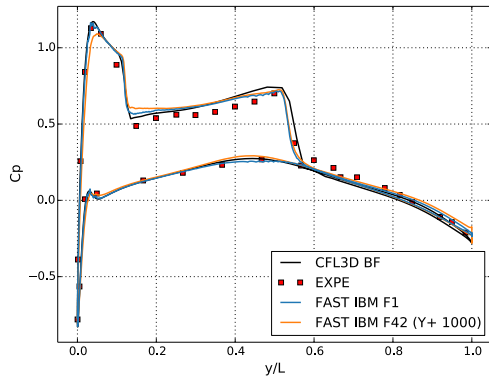
465 Figure 21 compares the skin pressure coefficient for the F1 algorithm (figure 21-(a)) and the F42 algorithm (figure 21-(b)), showing that the improved IBM treatment captures well the λ -shaped shock on the suction side of the wing without the spurious oscillations observed with former F1 algorithm.



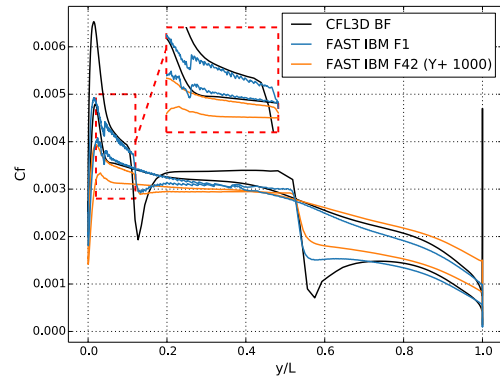
(a) Skin pressure coefficient: $y/b = 0.2$.



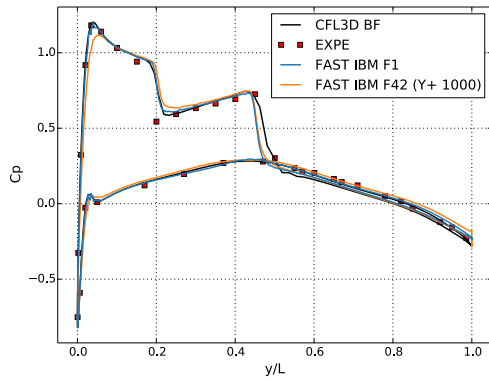
(b) Skin friction coefficient: $y/b = 0.2$.



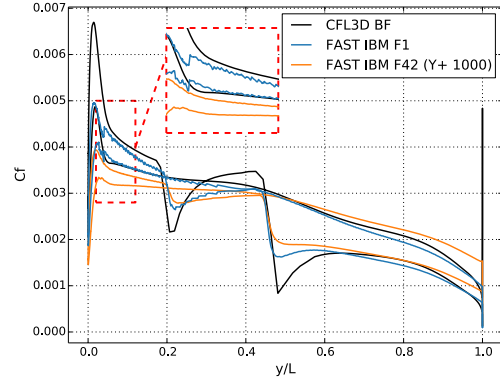
(c) Skin pressure coefficient: $y/b = 0.44$.



(d) Skin friction coefficient: $y/b = 0.44$.

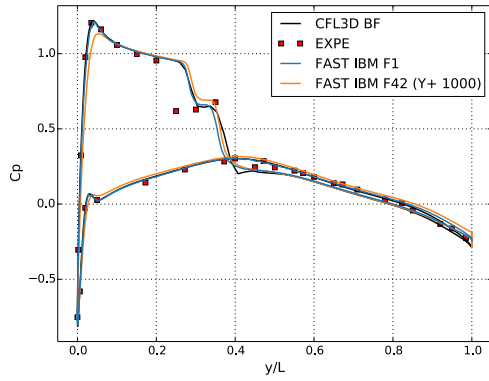


(e) Skin pressure coefficient: $y/b = 0.65$.

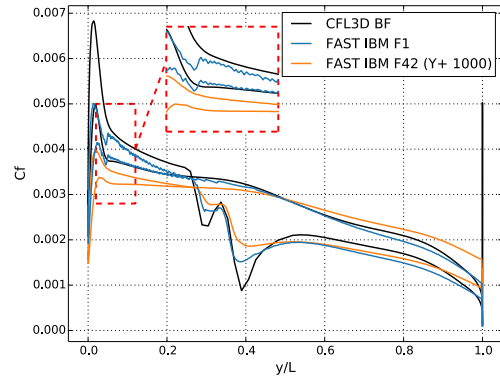


(f) Skin friction coefficient: $y/b = 0.65$.

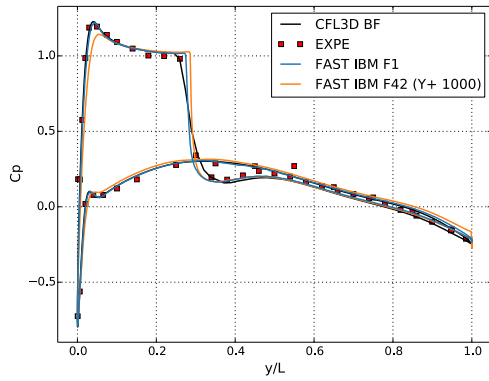
Figure 22: ONERA M6 Wing, $M_\infty = 0.84$, $Re = 11.72$ millions: comparison of skin pressure and skin friction distributions at different spans on the wing (improved treatment versus former treatment and body-fitted solution).



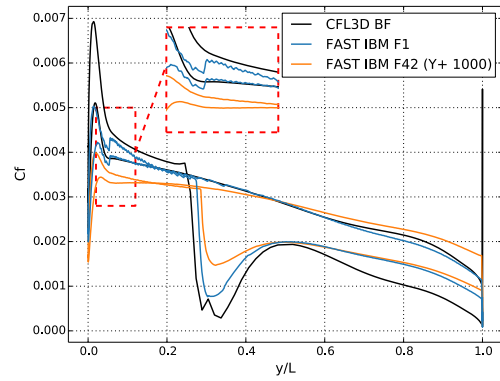
(a) Skin pressure coefficient: $y/b = 0.8$.



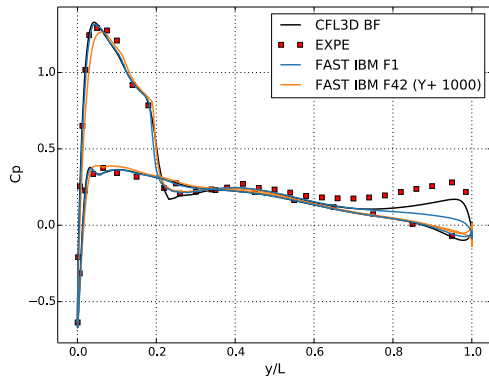
(b) Skin friction coefficient: $y/b = 0.8$.



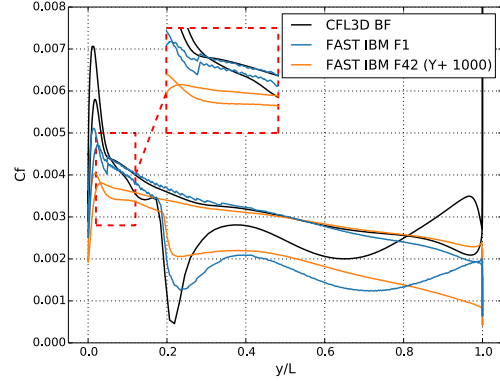
(c) Skin pressure coefficient: $y/b = 0.9$.



(d) Skin friction coefficient: $y/b = 0.9$.



(e) Skin pressure coefficient: $y/b = 0.99$.



(f) Skin friction coefficient: $y/b = 0.99$.

Figure 23: ONERA M6 Wing, $M_\infty = 0.84$, $Re = 11.72$ millions: comparison of skin pressure and skin friction distributions at different spans on the wing (improved treatment versus former treatment and body-fitted solution).

Figures 22 and 23 display the skin pressure and skin friction distributions for different sections
470 of the ONERA M6 wing.

The FAST IBM F42 and FAST IBM F1 solutions show similar trends for the skin pressure
distribution and compare well with both numerical and experimental reference solutions. The
shock location is also well predicted for all the sections. Besides, the improved method enables to
remove the oscillations appearing on the sections close to the wing root for the former method, as
475 displayed in figures 22-(a,c,e).

As previously observed on the RAE2822 test case, the skin friction distribution is still un-
derpredicted for the IBM solutions compared to the CFL3D BF reference solution, especially on
the suction side and near the leading edge. Once again, this underestimation is likely due to the
limitations of our wall modeling for transonic flows, as stated in section 5.2

480 A slightly better prediction of the peaks near the leading edge and the shock is observed in favor
of the F1 algorithm, as its actual y_{target}^+ is lower than 1000, but the improved method provides
smoother results, as it suppresses all spurious oscillations as well as the deviations and perturbations
observed on the FAST IBM F1 solutions in the vicinity of the change of refinement level.

6. Conclusions

485 An improved immersed boundary method on Cartesian grids for RANS simulations of turbu-
lent compressible flows has been proposed. Smooth pressure and skin friction coefficients have
been obtained and compared with body-fitted approaches. In the subsonic regime, results on the
NACA0012 airfoil demonstrate that the grid convergence to the body-fitted solution is reached for
this flow regime and that the method is not sensitive to the angle of attack. In the transonic flow
490 regime, skin pressure distribution is well captured and compares well to the body-fitted solution.
The skin friction distribution is under-estimated in comparison with the wall-resolved body-fitted
solution but compares to the wall-modeled body-fitted solution, showing that the stair-step distri-
bution is no longer responsible for the discrepancies but the wall model. The ONERA M6 wing
test-case highlight the extension to three-dimensional cases of the present approach. It can be
495 noticed that the present approach is neither dependent from the wall model nor from the turbu-
lence model. Future work will consist in improving the wall model to take into account for laminar
regions and pressure gradients within the IBM treatment.

7. Acknowledgment

The authors would like to acknowledge Marc Terracol for our fruitful discussions, Ivan Mary
500 and Thomas Renaud from ONERA for their helpful remarks.

References

- [1] C. Peskin, Flow patterns around heart valves: a numerical method, *Journal of Computational Physics* 10 (2) (1972) 252–271.
- [2] C. Peskin, The immersed boundary method, *Acta numerica* 11 (2002) 479–517.
- 505 [3] R. Mittal, G. Iaccarino, Immersed Boundary Methods, *Annu. Rev. Fluid Mech.* 37 (2005) 239–261.
- [4] R. Beyer, R. LeVeque, Analysis of a One-Dimensional Model for the Immersed Boundary Method, *SIAM Journal on Numerical Analysis* 29 (2) (1992) 332–364.
- [5] P. Angot, C.-H. Bruneau, P. Fabrie, A penalization method to take into account obstacles in
510 incompressible viscous flows, *Numerische Mathematik* 81 (1999) 497–520.
- [6] M.-C. Lai, C. S. Peskin, An immersed boundary method with formal second-order accuracy and reduced numerical viscosity, *Journal of Computational Physics* 160 (2) (2000) 705–719.
- [7] K. Nakahashi, Immersed boundary method for compressible Euler equations in the Building-Cube Method, *AIAA paper 2011-3386*, 2011.
- 515 [8] Y.-H. Tseng, J. Ferziger, A ghost-cell immersed boundary method for flow in complex geometry, *Journal of Computational Physics* 192 (2) (2003) 593–623.
- [9] W. Zhu, T. Behrens, W. Shen, J. Sørensen, Hybrid immersed boundary method for airfoils with a trailing-edge flap, *AIAA Journal* 51 (1) (2012) 30–41.
- [10] W. Coirier, K. Powell, Solution-adaptive Cartesian cell approach for viscous and inviscid flows,
520 *AIAA Journal* 34 (5) (1996) 938–945.
- [11] M. Berger, M. Aftosmis, Progress towards a Cartesian cut-cell method for viscous compressible flow, *AIAA paper 2012-1301*, 2012.

- [12] M. Harada, Y. Tamaki, Y. Takahashi, T. Imamura, Simple and robust cut-cell method for high-Reynolds-number-flow simulation on Cartesian grids, *AIAA Journal* 55 (8) (2017) 2833–2841.
- [13] C. Brehm, M. Barad, C. Kiris, Open rotor computational aeroacoustic analysis with an immersed boundary method, in: *54th AIAA Aerospace Sciences Meeting*, 2016, p. 0815.
- [14] F. Capizzano, Turbulent wall model for immersed boundary methods, *AIAA Journal* 49 (11) (2011) 2367–2381.
- [15] M. Berger, M. Aftosmis, An ODE-based wall model for turbulent flow simulations, *AIAA Journal* (2017) 1–15.
- [16] S. Péron, C. Benoit, T. Renaud, I. Mary, An immersed boundary method on Cartesian adaptive grids for the simulation of compressible flows around arbitrary geometries, *Engineering with Computers* (2020) 1–19.
- [17] S. Wilhelm, J. Jacob, P. Sagaut, An explicit power-law-based wall model for lattice Boltzmann method–Reynolds-averaged numerical simulations of the flow around airfoils, *Physics of Fluids* 30 (6) (2018) 065111.
- [18] G. Kalitzin, G. Iaccarino, Turbulence modeling in an immersed-boundary RANS method, *CTR Annual Briefs* (2002) 415–426.
- [19] T. Pu, C. Zhou, An immersed boundary/wall modeling method for rans simulation of compressible turbulent flows, *International Journal for Numerical Methods in Fluids* 87 (5) (2018) 217–238.
- [20] Y. Tamaki, M. Harada, T. Imamura, Near-wall modification of Spalart–Allmaras turbulence model for immersed boundary method, *AIAA Journal* (2017) 3027–3039.
- [21] P. R. Spalart, S. R. Allmaras, A one-equation turbulence model for aerodynamic flows, *AIAA Journal* 94 (1992).
- [22] I. Mary, Flexible Aerodynamic Solver Technology in an HPC environment, *Maison de la Simulation Seminars*, 2016, [http://www.maisondelasimulation.fr/seminar/data/\\$201611\\$.slides_\\$1\\$.ppt](http://www.maisondelasimulation.fr/seminar/data/201611.slides_1.ppt).

- [23] <https://w3.onera.fr/FAST>
- 550 [24] J. Dandois, I. Mary, V. Brion, Large-eddy simulation of laminar transonic buffet, *Journal of Fluid Mechanics* 850 (2018) 156–178.
- [25] P. L. Roe, Approximate riemann solvers, parameter vectors, and difference schemes, *Journal of Computational Physics* 43 (2) (1981) 357–372.
- [26] I. Mary, P. Sagaut, Large Eddy Simulation of Flow Around an Airfoil Near Stall, *AIAA Journal*
555 40 (6) (2002) 1139–1145.
- [27] J. Edwards, M.-S. Liou, Low-diffusion flux-splitting methods for flows at all speeds, *AIAA journal* 36 (9) (1998) 1610–1617.
- [28] A. Jameson, S. Yoon, Lower-upper implicit schemes with multiple grids for the euler equations, *AIAA Journal* 25 (7) (1987) 929–935.
- 560 [29] T. Coakley, Implicit upwind methods for the compressible Navier-Stokes equations, *AIAA Journal* 23 (3) (1985) 374–380.
- [30] S. Péron, C. Benoit, Automatic off-body overset adaptive Cartesian mesh method based on an octree approach, *Journal of Computational Physics* 232 (1) (2013) 153 – 173.
- [31] C. Benoit, S. Péron, S. Landier, Cassiopee: A CFD pre- and post-processing tool, *Aerospace Science and Technology* 45 (2015) 272 – 283.
565
- [32] S. Péron, T. Renaud, M. Terracol, C. Benoit, I. Mary, An Immersed Boundary Method for preliminary design aerodynamic studies of complex configurations, in: *23rd AIAA Computational Fluid Dynamics Conference*, AIAA paper 2017-3623, 2017.
- [33] F. Capizzano, Coupling a wall diffusion model with an immersed boundary technique, *AIAA Journal*
570 54 (2) (2016) 728–734.
- [34] A. Musker, Explicit expression for the smooth wall velocity distribution in a turbulent boundary layer, *AIAA Journal* 17 (6) (1979) 655–657.
- [35] P. Lancaster, K. Salkauskas, Surfaces generated by Moving Least Squares methods, *Mathematics of computation* 37 (155) (1981) 141–158.

- 575 [36] A. Nealen, An as-short-as-possible introduction to the least squares, weighted least squares and moving least squares methods for scattered data approximation and interpolation, 2004, <http://www.nealen.com/projects>.
- [37] Z.-Q. Cheng, Y.-Z. Wang, B. Li, K. Xu, G. Dang, S.-Y. Jin, A survey of methods for moving least squares surfaces, in: Proceedings of the Fifth Eurographics/IEEE VGTC conference on Point-Based Graphics, 2008, pp. 9–23.
- 580 [38] T. Renaud, C. Benoit, S. Péron, I. Mary, N. Alferez, Validation of an immersed boundary method for compressible flows, in: AIAA Scitech 2019 Forum, AIAA paper 2019-2179.
- [39] V. Schmitt, F. Charpin, Pressure Distribution on the ONERA-M6 Wing at Transonic Mach Number, in: AGARD Report AR 138, Experimental Data Base for Computer Program Assessment, 1979.
- 585 [40] J. Mayeur, A. Dumont, D. Destarac, V. Gleize, Reynolds-Averaged Navier-Stokes Simulations on NACA0012 and ONERA-M6 Wing with the ONERA elsA Solver, Vol. 54:9, 2016, pp. 2671–2687.
- [41] C. Dapogny, C. Dobrzynski, P. Frey, Three-dimensional adaptive domain remeshing, implicit domain meshing, and applications to free and moving boundary problems, Journal of Computational Physics 262 (2014) 358 – 378.
- 590