30th International Conference
ENGINEERING MECHANICS 2024
Milovy, Czech Republic, May 14 – 16, 2024

REYNOLDS-AVERAGED SIMULATION OF TURBULENT FLOWS WITH IMMERSED BOUNDARIES

Kubíčková L.*, Isoz M.**

Abstract: *Simulating turbulent flows in complex real-life geometries faces two major problems. First, direct simulation of turbulent flow is extremely costly. Second, a complex geometry-conforming mesh is required, and such mesh presumably suffers from several mesh-quality related problems lowering the solution accuracy and prolonging the simulation time. To solve the first problem, phenomenological turbulence models based on, e.g. Reynolds-averaging, are commonly utilized. To address the second one, a variant of an immersed boundary (IB) method can be used where the complex geometry is projected onto a simple mesh by an indicator field and adjustment of governing equations. Consequently, a connection of Reynolds-averaging and an immersed boundary method shall resolve both the problems and provide a simulation approach favorable for e.g. optimizations. However, such a connection is not common. In this contribution, we utilize our custom IB variant, the hybrid fictitious domain-immersed boundary method (HFDIB) and aim on extending the HFDIB by tools of the Reynolds-averaged simulation (RAS). In comparison with standard simulation approaches, the new HFDIB-RAS approach shows acceptable results in wide range of flow Reynolds numbers and in several testing geometries.*

Keywords: Immersed boundary method, computational fluid dynamics, Reynolds-averaged simulation, wall functions.

1. Introduction

In the modern world, engineers are increasingly using simulation tools to design, test and optimize devices and components. Moreover, with more affordable and always growing computational power, the tested geometries can be more complex, and simulations more accurate and faster. However, simulations of complex geometries affected by turbulent flows are still problematic and require specific approaches to be computationally feasible.

One problem is with the turbulent flow itself. Direct numerical simulation of such a flow is extremely costly, since the number of required mesh nodes grows steeply with the Reynolds number of the flow (Wilcox, 2006). To alleviate simulation costs, several turbulence modeling approaches have been developed. In this work, we focus on the Reynolds-averaged simulation (RAS) approach extended by the Boussinesq hypothesis and wall functions for boundary layer modeling. Despite the assumptions adopted and the consequent inaccuracies of the approach, its low computational cost makes it a prominent part of the engineering toolbox. The value of RAS is the most apparent in optimizations or parametric studies, where trends in the solution are of the most importance.

Another problem related to simulations of industrial flows is the complexity of the geometry and mesh construction. A standard in computational fluid dynamics (CFD) is to create the mesh to be geometry-conforming. Consequently, the complexity and irregularity of the real-life geometry is reflected in the mesh and may lead to lower mesh quality. In particular, high nonorthogonality and skewness of the mesh lead to lower solution robustness and accuracy. An answer to this problem may be a kind of an immersed boundary

* Ing. Lucie Kubíčková: Institute of Thermomechanics, Czech Academy of Sciences, Dolejškova 1402/5; 182 00, Prague; CZ (ITCAS), and Department of Mathematics, Informatics, and Cybernetics, University of Chemistry and Technology, Technická 5; 166 28, Prague; CZ (DMICUCT), luciekub@it.cas.cz

** Ing. Martin Isoz, PhD.: ITCAS, and DMICUCT

doi: 10.21495/em2024-174

(IB) method. In the IB method, the geometry-conforming mesh is replaced by a simple one and the complex geometry is projected onto it using a scalar indicator field and adjustment of governing equations. Moreover, in optimizations and parametric studies, the IB method has a clear advantage, since for different geometries, only the scalar field has to be redefined and no remeshing is required, saving large amounts of computational time (Kubíčková and Isoz, 2022).

Ultimately, for simulations of turbulent flows in or around complex geometries, the connection of the RAS turbulence modeling approach and the IB method would be favorable. However, such a connection is not common (Verzicco, 2023). Several attempts have been reported in the literature; see e.g. (Capizzano, 2011; Troldborg et al., 2022). The methods showed acceptable solution accuracy, but to stabilize the computation, they mostly used mesh refinement near the immersed boundary. Such refinements are costly and may locally reduce the quality of the mesh, lowering the IB-inherent advantage to be used in geometry optimization.

In this contribution, we present recent advances in the development of a connection of the RAS approach and our custom IB method variant, the hybrid fictitious domain-immersed boundary method (HFDIB), see (Isoz et al., 2022). The main motivation is application of the resulting HFDIB-RAS approach in an automated geometry optimization. Therefore, we focus on robustness and general applicability of the approach. The approach combines the HFDIB method with two-equation RAS turbulence models and wall functions. Compared with standard CFD approaches, the HFDIB-RAS shows good accuracy in a variety of verification and validation tests.

2. Description of HFDIB-RAS

The presence of the solid body in the computational domain is indicated by a scalar field λ, see Fig. 1a. This field is constructed given the distance from the body surface and surface normal. In HFDIB-RAS, the λ field is used to divide mesh cells into three groups, in-solid cells, boundary cells and free-stream cells, see Fig. 1b. Each group is then treated differently in the construction of IB-induced sources.

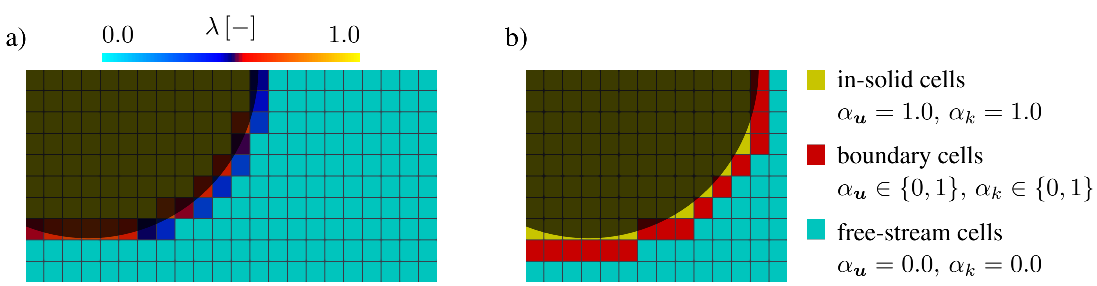

Fig. 1: a) Scalar field λ indicating presence of a solid body (cylinder). b) Division of mesh cells based on the λ field. Values of α fields are given.

The used governing equations stem from the Reynolds-averaged Navier-Stokes equations with the Boussinesq hypothesis. To close the problem, we focus on two-equation turbulence models to compute the turbulent viscosity. In short form, the equations can be written as:

$$\begin{gathered}\mathcal{M}(\boldsymbol{u}) = -\nabla\hat{p} + \boldsymbol{f}_{\text{ib}}, \quad \boldsymbol{f}_{\text{ib}} = \alpha_{\boldsymbol{u}}(\lambda)\,\left[\mathcal{M}(\boldsymbol{u}_{\text{ib}}) + \nabla\hat{p}\right], \quad \nabla\cdot\boldsymbol{u} = 0 \\ \mathcal{N}(k) = S_{\text{ib}}, \quad S_{\text{ib}} = \alpha_k(\lambda)\,\mathcal{N}(k_{\text{ib}}), \quad \mathcal{P}(\omega|\varepsilon) = 0\end{gathered} \tag{1}$$

where $\hat{p}$ is the turbulence pressure, $\boldsymbol{u}$ the velocity, k the turbulence kinetic energy, ω the specific rate of dissipation of k and ε the rate of dissipation of k. The operator $\mathcal{M}$ contains terms of the momentum equation where $\boldsymbol{u}$ is present, operator $\mathcal{N}$ sums up the k conservation equation and $\mathcal{P}$ the ω or ε $(\omega|\varepsilon)$ conservation equation (based on the chosen turbulence model). Exact definitions of the operators may be found in Kubíčková and Isoz (2023).

The effect of the solid body on flow is accounted for by two source terms, $\boldsymbol{f}_{\text{ib}}$ and S_{ib}. These source terms are switched on and off based on the fields $\alpha_{\boldsymbol{u}}$ and α_k. The α fields are non-zero in cells where the body has the greatest effect on the flow behavior; see Fig. 1b. The calculation of the source terms is based on the prescribed values of the respective fields $\boldsymbol{u}_{\text{ib}}$ and k_{ib}. These immersed values are then enforced in the most affected cells by iterative solution of the equations. Lastly, the effect of the solid body on ω or ε cannot be accounted for by source terms, because the fields go to infinity near the solid surface. Still, the values of $\omega_{\text{ib}}|\varepsilon_{\text{ib}}$ are computed, but to enforce them, direct matrix manipulation is utilized.

In each group of cells, the immersed values are computed differently. In the in-solid cells, it is considered that $\boldsymbol{u}_{\text{ib}} = \mathbf{0}$, $k_{\text{ib}} = 0$, $\omega_{\text{ib}}|\varepsilon_{\text{ib}} = \max(\omega^{\text{old}}|\varepsilon^{\text{old}})$. In the boundary cells, wall functions in the forms given by Kalitzin et al. (2005) are used to calculate the values of $\omega_{\text{ib}}|\varepsilon_{\text{ib}}$. For k_{ib}, the wall functions cannot be used directly, since it severely deteriorates the stability and accuracy of the solution. Instead, there is a switch for cases with a resolved and unresolved boundary layer. In resolved cases, the wall functions are used to give the value at the solid surface and a simple polynomial interpolation is used to compute the value in the boundary cell center. In the unresolved case, α_k is set to zero and the k behavior is corrected by $\omega|\varepsilon$.

Lastly, the values of $\boldsymbol{u}_{\text{ib}}$ are treated similarly to those of k_{ib}. Only in cases with resolved boundary layer, the value at the surface is given by the no-slip boundary condition. In unresolved cases, the turbulent viscosity, computed from k and $\omega|\varepsilon$ corrects the velocity behavior near the body surface.

3. Verification & validation tests

Several verification and validation tests were conducted. In each test, two similar meshes were created: (i) one simple and structured mesh for the HFDIB-RAS method and (ii) one geometry-conforming mesh for simpleFoam solver available in OpenFOAM (Weller et al., 2021). The simpleFoam solver is used to provide referential solutions and geometry-conforming meshes were made to fit a surface-representing contour of the λ fields. This ensures that simpleFoam works with the same geometries as HFDIB-RAS even with non-ideal λ field generation.

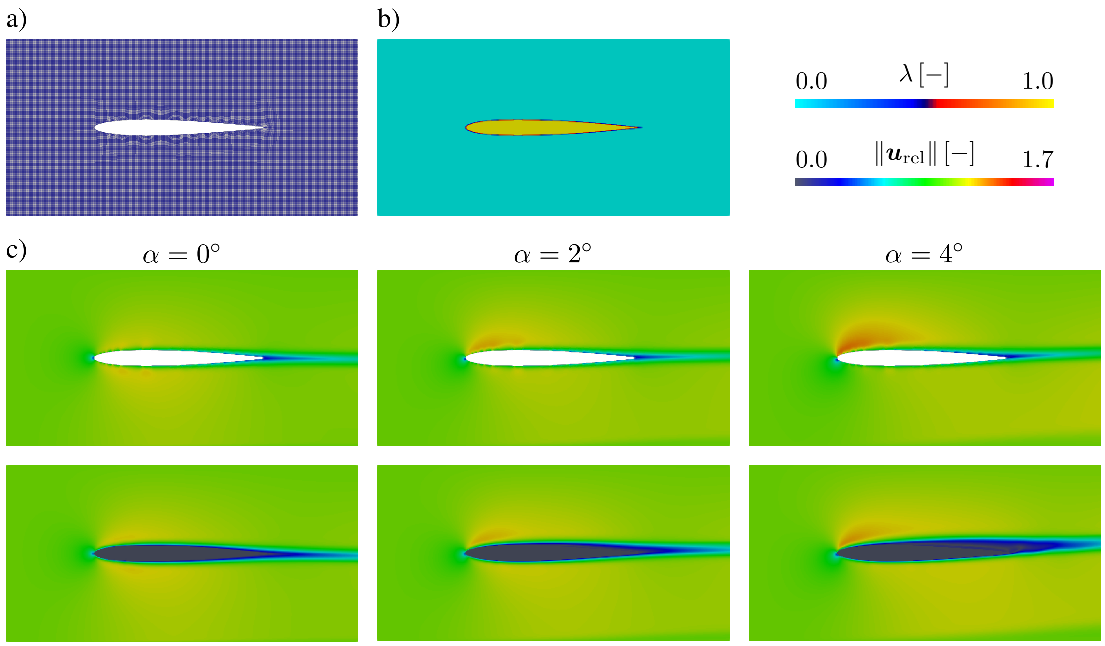

Fig. 2: Verification test on NACA-0009 airfoil with $\mathrm{Re} = 200\,000$*. a) Mesh used for simpleFoam. b)* λ *field used for HFDIB-RAS. c) Comparison of velocity fields for different angles of attack* α*. Colored by magnitude of relative velocity* $\boldsymbol{u}_{\text{rel}} = \boldsymbol{u}/\|\boldsymbol{u}_{\text{inlet}}\|$*.*

One chosen verification test and one validation test is presented. In the verification test, simulations of flow around a NACA-0009 airfoil with $\mathrm{Re} = 200\,000$ were performed for several angles of attack. The results are depicted in Fig. 2. As a validation test, a backward facing step benchmark was selected with experimental data provided by Driver and Seegmiller (1985). The results achieved are presented in Fig. 3. In both tests, the agreement is not perfect; the HFDIB-RAS overestimates the boundary layer thickness. However, for use in optimization, the HFDIB-RAS behavior is considered acceptable, since it captures the trends sufficiently well.

4. Conclusions

In this work, we introduce the HFDIB-RAS approach, which is a connection of our custom immersed boundary method variant and the Reynolds-averaged simulation approach. In particular, two-equation turbulence models with wall functions for boundary layer modeling. The code with illustrative tutorials is available from https://github.com/techMathGroup/openHFDIBRANS. Moreover, we present recently

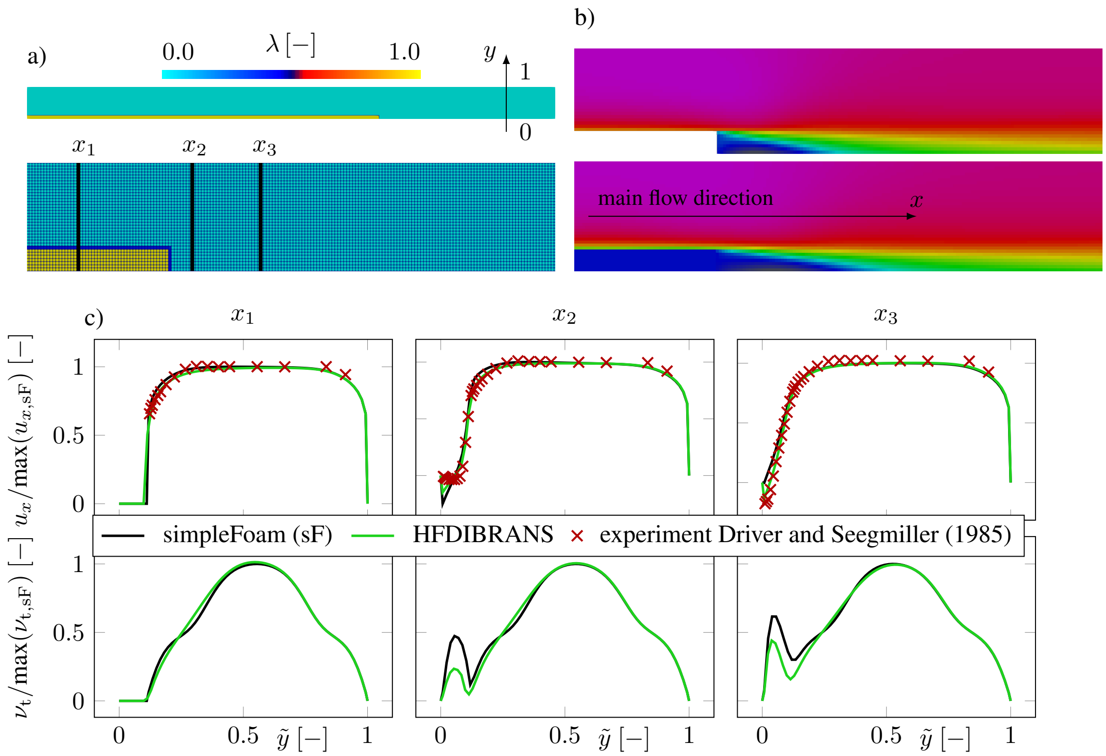

Fig. 3: Validation test on the backward facing step benchmark. a) Used λ field with highlighted sampling lines. b) Velocity field from the simpleFoam (top) and HFDIB-RAS (bottom) simulations. c) Comparison of $\mathbf{u}$ and ν_t profiles along sampling lines.

achieved HFDIB-RAS results that show good agreement with standard computational fluid dynamics approaches. In future development, the HFDIB-RAS will be used in geometry optimization.

Acknowledgments

The work was financially supported by the institutional support RVO:61388998 and by the grant project with No. TN02000069/001N of the Technology Agency of the Czech Republic. The authors acknowledge the financial support provided by the Ministry of Education, Youth, and Sports of the Czech Republic via the project No. CZ.02.01.01/00/22_008/0004591 (Ferroic Multifunctionalities), co-funded by the European Union. This work was supported from the grant of Specific university research – grant No. A2_FCHI_2024_022.

References

Capizzano, F. (2011) A turbulent wall model for immersed boundary methods. *AIAA J.*, 49, 11, pp. 2367–2381.

Driver, D. and Seegmiller, H. (1985) Features of a reattaching turbulent shear layer in divergent channel flow. *AIAA Journal*, 23, 2, pp. 163–171.

Isoz, M., Kotouč Šourek, M., Studeník, O., and Kočí, P. (2022) Hybrid fictitious domain-immersed boundary solver coupled with discrete element method for simulations of flows laden with arbitrarily-shaped particles. *Computers & Fluids*, 244, pp. 105538–1–105538–22.

Kalitzin, G., Medic, G., Iaccarino, G., and Durbin, P. (2005) Near-wall behavior of rans turbulence models and implications for wall functions. *Journal of Computational Physics*, 204, pp. 265–291.

Kubíčková, L. and Isoz, M. (2022) Hybrid fictitious domain-immersed boundary method in CFD-based topology optimization. In *Proceedings of Topical Problems of Fluid Mechanics 2022*. IT CAS, pp. 119–126.

Kubíčková, L. and Isoz, M. (2023) On Reynolds-averaged turbulence modeling with immersed boundary method. *Topical Problems of Fluid Mechanics 2023*, pp. 104–111.

Troldborg, N., Sørensen, N. N., and Zahle, F. (2022) Immersed boundary method for the incompressible reynolds averaged navier–stokes equations. *Computers and Fluids*, 237, pp. 105340.

Verzicco, R. (2023) Immersed boundary methods: Historical perspective and future outlook. *Annual Review of Fluid Mechanics*, 55, pp. 129–155.

Weller, H., Greenshields, C., and Bainbridge, W. (2021) The OpenFOAM foundation.

Wilcox, D. (2006) *Turbulence modeling for CFD*. DCW Industries, USA, 3rd edition.