11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

PARALLEL ADAPTIVE MESH REFINEMENT OF TURBULENT FLOW AROUND SIMPLIFIED CAR MODEL USING AN IMMERSE BOUNDARY METHOD

O. Antepara[*], R. Borrell[†], O. Lehmkuhl[*†], I. Rodríguez[*] and A. Oliva[*]

[*] Heat and Mass Transfer Technological Center(CTTC)
ETSEIAT, Technical University of Catalonia
C/ Colom 11, 08222, Terrassa, Spain
e-mail: cttc@cttc.upc.edu, web page: http://www.cttc.upc.edu/

[†]Termo Fluids, S.L.
c/ Magí Colet 8, 08204 Sabadell (Barcelona), Spain
e-mail: termofluids@termofluids.com - Web page: http://termofluids.com

Key words: Adaptive mesh refinement, LES, turbulent flow, simplified car models, immerse boundary

Abstract. In the present work, a parallel adaptive mesh refinement (AMR) strategy for large-eddy simulations (LES) is proposed and tested for a fully 3D geometry. The underlying discretization of the Navier-Stokes equations is based on a finite-volume symmetry-preserving formulation, with the aim of preserving the symmetry properties of the continuous differential operators to ensure stability and conservation of kinetic-energy balance. The proposed AMR scheme applies a cell-based refinement technique, with a physics-based refinement criteria based on the variational multi-scale(VMS) decomposition theory and an equalized histogram of the vorticity field. This strategy has been tested in other turbulent problems around bluff bodies in 2D and 3D. To carry out the simulation of turbulent flow around complex geometries with AMR, an immerse boundary method is implemented based on a finite volume approach. Finally, the robustness and accuracy of our methodology is shown on the numerical simulation of the turbulent flow over an Ahmed car at $Re_h = 7.68 \times 10^5$, which reproduces the basic fluid dynamics features of real cars, i.e. vortex shedding, flow reattachment and recirculation bubbles.

1 INTRODUCTION

The use of numerical simulations to predict the fluid-dynamics features of real cars has increased in the last years. The principal improvements developed on this topic are related to the physical phenomena prediction capabilities, get accurate numerical results and the efficient utilization of computational resources. One of the main difficulties on the numerical simulation of flow around vehicles is the the ability to handle complex geometries. In order to solve such problem, Immerse Boundary (IB) method allows the computation of the flow around complex objects which is not aligned with the grid lines [1]. In the IB approach a mass source/sink as well as a momentum forcing, are introduced in the governing equations to model the effect of the presence of an object on the flow [2]. Moreover, the use of adaptive mesh refinement algorithms give the spatial resolution needed to identify, in an accurate way, the boundary between the fluid and the body surface.

The latter approach is an attractive platform to evaluate the IB-AMR capabilities in eddy resolving techniques such as the large-eddy simulation (LES) approach, which is playing an important role in the modelization and understanding of complex turbulent flows. The use of AMR schemes and LES models are used to get accurate results for complex turbulent flows with an efficient utilization of computational resources. With the use of adaptive mesh refinement algorithms, the computational grid is changed which is effective in treating problems with a wide range of length scales. This kind of approach permit local mesh refinement, minimizing the number of computational cells and providing the spatial resolution required for the study of turbulent flows past bluff bodies.

In the present work, the turbulent flow is described by means of Large Eddy Simulation (LES) using symmetry-preserving discretizations [3]. The LES model used is the WALE model [4] within a variational multiscale framework [5] (VMS-WALE). The proposed AMR algorithm borrows from previous work by Berger [6] and Powell [7] that describes an AMR formulation for Cartesian meshes and cell-based AMR methods. To carry out the simulation of turbulent flow around complex geometries with AMR, an immerse boundary method is implemented based on a finite volume approach [2, 8, 9, 10]. This article is organized as follows. In Section 2, the system of governing equations using a symmetry preserving discretization, the IB method and the adaptive mesh refinement scheme are described. In Section 3, the solutions for a turbulent flow over an Ahmed car are compared to experimental results. Finally, some conclusions are drawn.

2 MATHEMATICAL FORMULATION

In large-eddy simulations (LES) the spatial filtered and discretized Navier-Stokes equations are defined as

$$\mathcal{M}\overline{\boldsymbol{u}} = 0 \tag{1}$$

$$\Omega\frac{\partial\overline{\boldsymbol{u}}}{\partial t} + \mathsf{C}\left(\overline{\boldsymbol{u}}\right)\overline{\boldsymbol{u}} + \nu\mathsf{D}\overline{\boldsymbol{u}} + \rho^{-1}\Omega\mathsf{G}\overline{\boldsymbol{p}} = \mathsf{C}\left(\overline{\boldsymbol{u}}\right)\overline{\boldsymbol{u}} - \overline{\mathsf{C}\left(\boldsymbol{u}\right)\boldsymbol{u}} \approx -\mathcal{M}\mathcal{T} \tag{2}$$

where $\boldsymbol{u}$ and $\boldsymbol{p}$ represent the filtered velocity vector and pressure, respectively, ρ is the fluid density and ν is the kinematic viscosity, Ω is a diagonal matrix with the sizes of control volumes. Convective and diffusive operators in the momentum equation for the velocity field are given by $\mathsf{C}\left(\boldsymbol{u}\right) = (\mathbf{u}\cdot\nabla)$ and $\mathsf{D} = -\nabla^2$, respectively. Gradient and divergence operators are given by $\mathsf{G} = \nabla$ and $\mathcal{M} = \nabla\cdot$, respectively. The term that requires modelling is the filtered non-linear convective term. $\mathcal{T}$ is the SGS stress tensor, which is defined as [11],

$$\mathcal{T} = -2\nu_{sgs}\overline{\mathcal{S}_{\mathsf{ij}}} + (\mathcal{T} : \mathsf{I})\mathsf{I}/3 \tag{3}$$

$$\overline{\mathcal{S}_{\mathsf{ij}}} = \frac{1}{2}[\mathsf{G}(\overline{\boldsymbol{u}}) + \mathsf{G}^{*}(\overline{\boldsymbol{u}})] \tag{4}$$

where $\overline{\mathcal{S}_{\mathsf{ij}}}$ is the rate-of-strain tensor and G^* is the transpose of the gradient operator. To close the formulation, a suitable expression for the subgridscale (SGS) viscosity, must be introduced. LES studies have been performed using a SGS model suitable for unstructured formulations: the wall-adapting local-eddy viscosity model within a variational multi-scale framework (VMS-WALE) [4, 5].

Second-order spectro-consistent schemes on a collocated unstructured grid arrangement were adopted for the discretization of the governing equations. Discrete conservation properties are related to the symmetries of the continuous differential operators as studied in detail by Verstappen and Veldman [3]. These conservation properties are held if and only if the discrete convective operator is skew-symmetric $C(u) = -C(u)^*$, if the negative conjugate transpose of the discrete gradient operator is equal to the divergence operator $M = -G^*$ and if the diffusive operator is symmetric and positive-definite. For the temporal discretization of the momentum equation a two-step linear explicit scheme on a fractional-step method has been used for the convective and diffusive terms, while the pressure is solved using an implicit first-order scheme. This methodology has been extensively tested and verified with accurate results for solving the flow over bluff bodies with massive separation [12, 13, 14, 15].

The immerse boundary method used in this work consist on the addition of a mass source/sink as well as a momentum forcing on the momentum equation, and if this equation is discretized in time, we obtain

$$\frac{\boldsymbol{u}^p - \boldsymbol{u}^n}{dt} = RHS + f \tag{5}$$

where RHS contains the convective and viscous terms and the pressure gradient. The forcing term will be a non-zero value on the inner body nodes (interior points) and on the fluid nodes that has a neighbour node inside the body or in the interface body-fluid (forcing points), but if $\boldsymbol{u}^p = V$ on the immersed boundary this will yield to a equation that gives the forcing value,

$$f = \frac{V - \boldsymbol{u}^n}{dt} - RHS \tag{6}$$

where V in the interior points is the body velocity in that position, and for the forcing points a second order linear interpolation is used [8], to keep the global accuracy. To do this interpolation, three external neighbours nodes (no forcing points) and the nearest object node are used. Because the interpolation is held on the predictor velocities, an error appears due to the linear relation between the velocities is not kept after the projection step of the fractional-step method. To reduce this error, the idea introduced by [2] and improved on [9, 10] is used.

2.1 Adaptive mesh refinement algorithm

Mesh adaptation is accomplished by coarsening and dividing a group of cells following refinement criteria based on our physical understanding of the problem. In regions where spatial resolution needs to be increased, a parent cell is refined by dividing itself eight (three dimensions) children (Fig. 1). However, in areas that are over resolved, the refinement process can be reversed by coarsening eight children into a single parent cell. In any case, the grid adaptation is constrained such as the cell resolution changes by only a factor of two between adjacent cells.

The proposed mesh refinement scheme is based on linear interpolation; this scheme is performed by averaging the adjacent vertex coordinates of the parent cell. In addition, a tree data structure is used to keeping track of the computational cell connectivity to transmit the information between the old and new mesh, wherein the information on the tree data structure is corresponding to the level of refinement and the indexes representing each cell.

The refinement criteria is based on the variational multi-scale(VMS) decomposition theory that allows the refinement of the cells where spatial resolution is needed to solve the small structures, this criteria has been tested with good results for turbulent problems around bluff bodies [16, 17]. Moreover, an equalized histogram of the vorticity field is used, modifying the actual vorticity field into an equalized one with a linear cumulative distribution, thus the maximum and minimum values of the equalized vorticity are established between 0 and 1, and this wont requires continual tuning depending on the flow problem.

3 NUMERICAL RESULTS OF TURBULENT FLOW OVER AN AHMED CAR AT $Re_h = 7.68 \times 10^5$

Numerical simulations of the flow over an Ahmed car were performed at $Re_h = 7.68 \times 10^5$, where the Reynolds number is based on the inlet velocity, U_{ref}, and the car height, h. Solutions are obtained on a computational domain of dimensions 9.1944 x 1.87 x 1.4, where the front of the body is located at a distance from the inlet of 2.1024m. The boundary condition at the inflow consist of a uniform axial velocity. At the lateral and top walls, slip boundary conditions are prescribed. A pressure-based boundary condition is applied at the outlet for the downstream. No-slip conditions at the bottom surface are considered. The use of an adaptive mesh, with four mesh levels for the interface fluid/object at the slant back and up to three mesh levels for the fluid, has allowed to cluster more control volumes around the body surface and in the near wake. Some illustrative results obtained

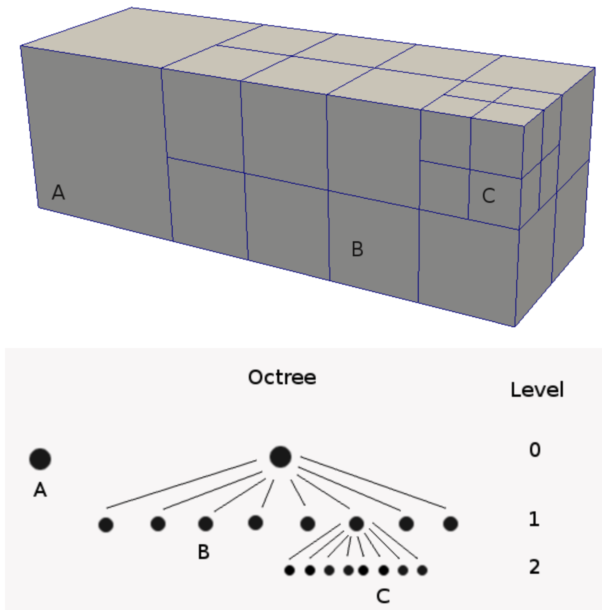


Figure 1: Illustration of AMR technique applied to a 3D Mesh and its corresponding octree

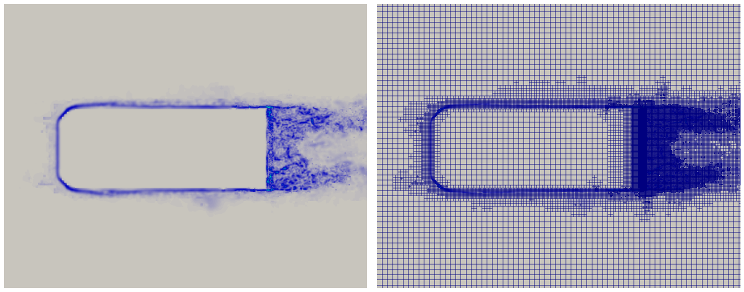
are depicted in Figure 2. Vorticity structures in the near wake obtained with the adaptive grid are plotted in Fig. 2(left) and the computational grid is plotted in Fig. 2(right).

Results have been obtained based on the integration of instantaneous data over a sufficiently time period (from 10 to 30 time units), and a mesh with approximately 5M cells. Furthermore, preliminary results for the averaged streamwise velocity for the symmetry plane (y=0) at the rear end of the body are compared with the experimental data by Leinhart et al. [18] in figure 3 and 4. As can be seen, results obtained with a parallel adaptive mesh refinement are in good agreement with the experimental data, but some minor differences appear on the prediction of the mean flow in the final part of the slant back. In the results, the flow separates in the slant corner and forms a recirculation zone. So, minor discrepancies are obtained in the prediction of the Reynolds stresses as can be seen in Fig 4.

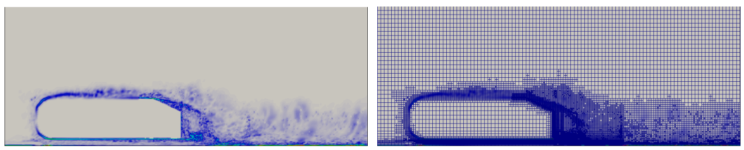
4 CONCLUSIONS

- A parallel adaptive mesh refinement has been used with an immersed boundary method to carry out the simulation of turbulent flow over complex geometries. The numerical simulation of a turbulent flow over an Ahmed car at $Re_h = 7.68 \times 10^5$ has been carried out by means of Large Eddy Simulation using symmetry-preserving discretizations.

- The main fluid dynamics features of real cars, i.e. vortex shedding, flow reattach-



top view



side view

Figure 2: Illustration LES of turbulent flow over an Ahmed car at $Re_h = 7.68 \times 10^5$ (left)Vorticity structures (right) computational grid.

ment and recirculation bubbles were successfully captured with the use of the refinement criteria mentioned before.

- Finally, numerical results of the velocity average and fluctuations demonstrated the potential of our approach, where our numerical results and the experimental references has been analyzed in different zones. It is important to highlight the benefit of the use immersed boundary with AMR to accomplish the grid requirements on the interface fluid/object in order to get accurate results in the slant back of the Ahmed car.

5 ACKNOWLEDGEMENTS

This work has been financially supported by the Ministerio de Ciencias e Innovación, Spain (ENE2010-17801). Calculations have been performed on the JFF cluster at the CTTC. The authors thankfully acknowledge these institutions.

REFERENCES

[1] B. Khalighi, S. Jindal, G. Iaccarino. Aerodynamic flow around a sport utility vehicle-Computational and experimental investigation. *Journal of Wind Engineering and Industrial Aerodynamics* (2012) **107-108**:140–148.

[2] J. Kim, D. Kim, H. Choi. An Immersed-Boundary Finite-Volume Method for Simulations of Flow in Complex Geometries. *Journal of Computational Physics* (2001) **171**:132–150.

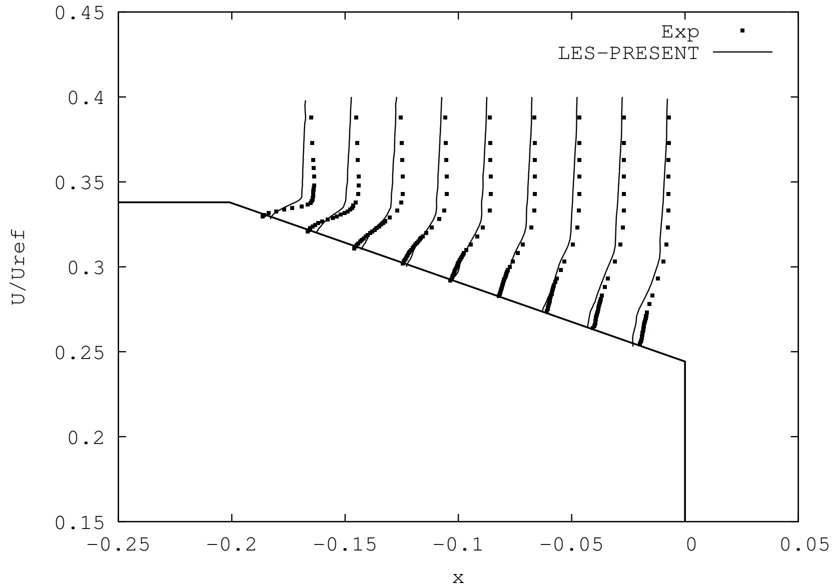


Figure 3: Mean streamwise velocity profiles in the symmetry plane: solid line VMS-WALE model, dots experiments [18]

[3] R.W.C.P. Verstappen, A.E.P. Veldman. Symmetry-preserving discretization of turbulent flow. *Journal of Computational Physics* (2003) **187**:343–368.

[4] F. Nicoud and F. Ducros. Subgrid-scale stress modeling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion* (1999) **62**:183–200.

[5] T.J.R. Hughes, L. Mazzei, and K.E. Jansen. Large eddy simulation and the variational multiscale method. *Computing and Visualization in Science* (2000) **3**:47–59.

[6] M. J. Berger and J. Oliger. Adaptive mesh refinement for hyperbolic partial differential equations. *Journal of Computational Physics* (1984) **53**:484–512.

[7] K. G. Powell, P. L. Roe, and J. Quirk. Adaptive-mesh algorithms for computational fluid dynamics. *In M. Y. Hussaini, A. Kumar, and M. D. Salas, editors, Algorithmic Trends in Computational Fluid Dynamics* (1993) **53**:303–337.

[8] Fadlun EA, Verzicco R, Orlandi P, Mohd-Yusof J. Combined Immersed-Boundary finite-difference methods for three-dimensional complex flow simulations. *J. Computational Physics* (2000) **161(1)**:35–60.

[9] S. Kang, G. Iaccariono and P. Moin. Accurate Immersed-Boundary Reconstructions for Viscous Flow Simulations. *AIAA Journal* (2009) **47(7)**.

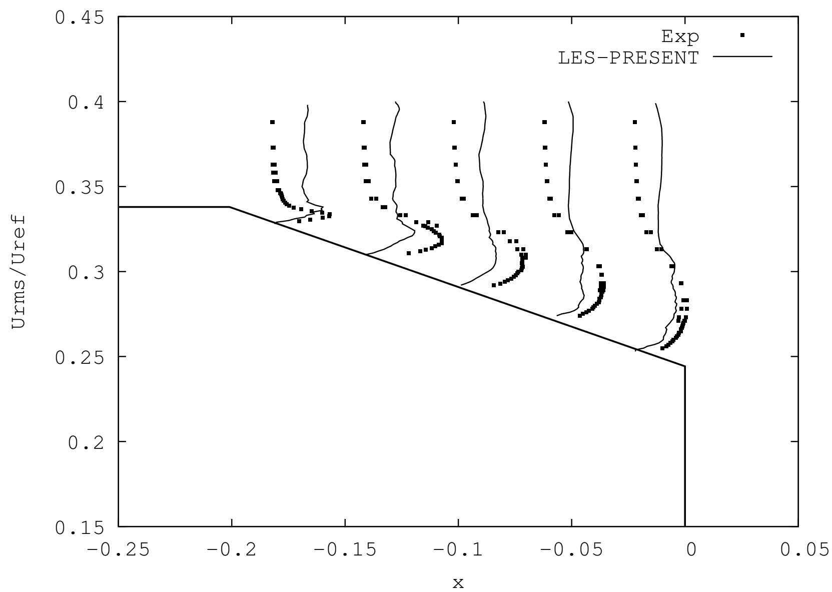


Figure 4: Root mean square of the streamwise velocity fluctuations profiles in the symmetry plane: solid line VMS-WALE model, dots experiments [18]

[10] Wei-Xi Huang and Hyung Jin Sung. Improvement of mass source/sink for an immersed boundary method. *International Journal for Numerical Methods in Fluids,* (2007) **53**:1659–1671.

[11] Sagaut P, Germano M. Large eddy simulation for incompressible flows. *Springer-Verlag,* (2001).

[12] D. Aljure, O. Lehmkuhl, R. Borrell, A. Oliva. Flow and turbulent structures around simplified car models. *Proc. Conf. Modelling Fluid Flow CMFF12* (2012).

[13] O. Lehmkuhl, I. Rodríguez, R. Borrell and A. Oliva. Low-frequency unsteadiness in the vortex formation region of a circular cylinder. *Phys. Fluids* (2013) **25(8)**.

[14] I. Rodríguez, R. Borrell, O. Lehmkuhl, C.D. Perez-Segarra, and A. Oliva. Direct Numerical Simulation of the Flow Over a Sphere at Re = 3700. *Journal of Fluid Mechanics* (2011) **679**:263–287.

[15] I. Rodríguez, O. Lehmkuhl, R. Borrell, A. Oliva. Flow dynamics in the wake of a sphere at sub-critical Reynolds numbers. *Comput Fluids* (2013) **80**:233–243.

[16] O. Antepara, O. Lehmkuhl, J. Chiva, R. Borrell. Parallel Adaptive Mesh Refinement Simulation of the Flow Around a Square Cylinder at Re = 22000. *Procedia Engineering* (2013) **61**:246–250.

[17] O. Antepara, O. Lehmkuhl, A. Oliva, F. Favre. Large-Eddy Simulations of turbulent flow around a wall-mounted cube using an adaptive mesh refinement approach. *14th European Turbulence Conference* (2013).

[18] H. Lienhart and S. Becker. Flow and turbulent structure in the wake of a simplified car model. *SAE Paper* (2003).