Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 61 (2013) 246 – 250

Procedia Engineering

www.elsevier.com/locate/procedia

ELSEVIER

Parallel Computational Fluid Dynamics Conference (ParCFD2013)

Parallel adaptive mesh refinement simulation of the flow around a square cylinder at Re = 22000

Oscar Antepara[a,*], Oriol Lehmkuhl[a,b], Jorge Chiva[a], Ricard Borrell[b]

[a]*Heat and Mass Transfer Technological Center, Polytechnical University of Catalonia (UPC), c/ Colom 11, 08222, Terrassa, Barcelona, Spain*
[b]*Termo Fluids S.L., Avda. Jaquard, 97 1-E, 08222, Terrassa, Barcelona, Spain*

Abstract

In this paper a parallel adaptive mesh refinement for LES simulation of turbulent flows is presented. The AMR scheme applies a cell-based refinement technique to get enough grid-resolution to solve the small scales structures, adapting the mesh according to physics-based refinement criteria. A flexible tree data structure is used to keeping track of the mesh adaptation and an edge-based data structure to save and search cell connectivity. The AMR framework is combined with parallel algorithms for partitioning and balancing of the computational mesh. Numerical results for turbulent flow around a square cylinder at Re = 22000 are compared to experimental data.

Selection and peer-review under responsibility of the Hunan University and National Supercomputing Center in Changsha (NSCC)

Keywords: Adaptive mesh refinement, Large Eddy Simulations (LES). flow around square cylinder.

Nomenclature

u	velocity in the direction of x(m/s)
v	velocity in the direction of y(m/s)
w	velocity in the direction of z(m/s)
AMR	Adaptive mesh refinement
LES	Large Eddy Simulations
WALE	Wall-adapting eddy viscosity model
VMS	Variational multiscale
MPI	Message Passing Interface
SGS	Subgrid-Scale

1. Introduction

Turbulent flows are characterized by a wide range of scale motions, which limit the impact of numerical studies. Due to its complexity, 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.

In a parallel computing framework, a considerable effect of the adaptation is that the dynamically change of the computational grid leads to a dynamically change of data size, work load, and communication pattern at runtime. This kind

* Corresponding author. Tel.: (+34) 93 739 81 92; fax: (+34) 93 739 89 20.
E-mail address: cttc@cttc.upc.edu

Selection and peer-review under responsibility of the Hunan University and National Supercomputing Center in Changsha (NSCC)
doi:10.1016/j.proeng.2013.08.011

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.

The proposed AMR algorithm borrows from previous work by Berger[1, 2], and Powell [3] that describes an AMR formulation for Cartesian meshes and cell-based AMR methods. Moreover, AMR-LES variants have been presented for turbulent flows around bluff bodies[4, 5], and AMR approach for finite volume methods[6]. The LES model used in the present work is the WALE model [7] within a variational multiscale framework [8] (VMS-WALE).

This article is organized as follows. In Section 2, the system of governing equations using a symmetry preserving discretization is described. In Section 3, a detailed description of the adaptive mesh refinement scheme developed and the parallelization strategy is presented. In Section 4, the solutions for a turbulent flow around a square cylinder are compared to experimental results. Finally, some conclusions are drawn.

2. Mathematical Formulation

The turbulent flow is described by means of LES using symmetry-preserving discretizations. The spatial filtered and discretized Navier-Stokes equations can be written as,

$$Mu = 0 \tag{1}$$

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

where M, C, D and G are the divergence, convective, diffusive and gradient operators, respectively. Ω is a diagonal matrix with the sizes of control volumes, ρ is the fluid density, ν the viscosity, p represents the filtered pressure, u is the filtered velocity, $\boldsymbol{M}$ represents the divergence operator of a tensor, and T_m is the SGS stress tensor.

The governing equations have been discretized on a collocated unstructured grid arrangement by means of second-order spectro-consistent schemes [9]. Our schemes are conservative, i.e. they preserve the symmetry properties of the continuous differential operators and ensure both, stability and conservation of the kinetic-energy balance even at high Reynolds numbers and with coarse meshes. 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 exactly equal to the divergence operator ($-(\Omega G)^* = M$) and if the diffusive operator D 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 for the pressure gradient term an implicit first-order scheme has been used. This methodology has been previously used with accurate results for solving the flow over bluff bodies with massive separation [10, 11].

2.1. Adaptive mesh refinement algorithm

In this section the basic algorithm for mesh refinement is described. 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 into four (two dimensions) or eight (three dimensions) children (Fig. 1). However, in areas that are over resolved, the refinement process can be reversed by coarsening four or 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 and the maximum resolution is determined by the Kolmogorov scales derived for this problem.

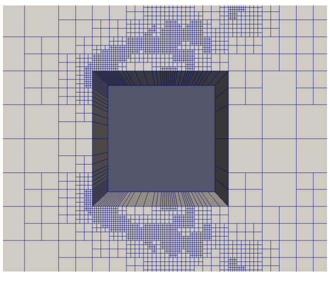

Fig. 1. Illustration of AMR technique applied to a 2D Mesh.

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 AMR scheme applies the edge based data structure in searching and storing cell connectivity related data. With this edge-based data structure, the cells containing a common refined edge can be identified, wherewith common nodes are added to the cells that shares the common refined edge. The coarsening algorithm follows a similar process, where the selected child cells become a parent cells. In this case, the edge-based data structure is used to eliminate common nodes if the neighbor cell has the same or a lower level of refinement compared with the new coarse cell level. After the addition-elimination of nodes, the new refined/coarsened cells and faces are created resulting in a new connectivity data.

2.2. Parallel strategy and domain decomposition.

Writing a parallel algorithm of mesh refinement for scientific computations is a critical work. These kinds of algorithms are developed on application basis, highly optimized to solve a certain problem. The parallelization is based on a fully distributed computational grid approach, where every process required during the solution procedure is performed in parallel, without representing any process step on one core.

The proposed overall procedure for parallel adaptive mesh refinement is implemented on a memory distributed parallel computer using MPI as the data communication protocol and is carried out every time unit fraction derived for the studied case. The parallel adaptive mesh refinement consists on divide the computational grid on subdomains where each one belongs to a specific processor that reads the corresponding grid data. Cells, in which refinement or coarsening must be done, are flagged and those processes are performed on the subdomain that contains each processor according to the criteria mentioned in the last section. Thereby, the communication process is held with nodes and faces data corresponding to the boundaries of the neighboring processors. The next step consist in unify the global node, face and cell indexes, due to the new mesh data, among all processors and build up a new connectivity data for all cells. Finally, all processors are synchronized and the data is gathered to create a HDF5 data file that contains the information of the new mesh that will be parallel partitioned and extruded to be used in the next step of the solution procedure.

Parallel partitioning generally refers to a redistribution of the cells among processes without changing the global mesh topology. Thus, a weighted dual graph is used as input for a graph partitioner (for this purpose is used ParMETIS [12]) that produces new partitions which is an optimal way to achieve load-balance. This will lead to a new partitioned mesh, with a good load-balance (fig. 2), that will be used for the next simulation step.

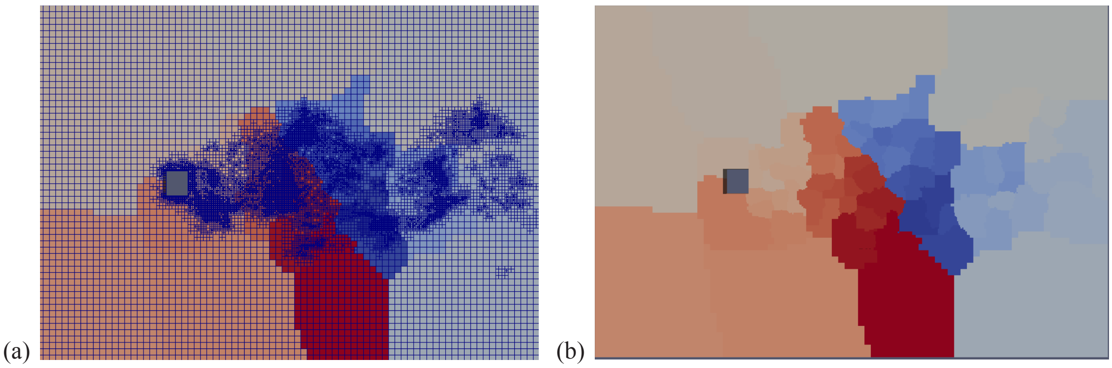

Fig. 2. Illustration of the computational domain for a turbulent problem (a) computational grid resulting from the application of the AMR scheme (b) decomposition domain for 64 processors.

3. AMR-LES of turbulent flow around a square cylinder at Re =22000

Numerical simulations of the flow around a square cylinder are performed at Re = 22000, where Reynolds number is defined in terms of the free-stream velocity U and the square length L. Solutions are obtained on a computational domain of dimensions [-5.5L,14.5L]; [-7L,7L]; [0,4L], where the square cylinder is located at $x = 0$, $y = 0$. The governing equations are solved on an adaptive mesh generated from the extrusion around the axis of a two-dimensional grid in a (x,y) plane. The boundary conditions at the inflow consist of a uniform velocity (u,v,w)=(1,0,0). A pressure based condition is used at the outlet boundary for the downstream. No-slip conditions on the square cylinder are imposed.

Some illustrative results obtained are depicted in Figure 3 and 4. Vorticity structures in the near wake obtained with the adaptive grid are plotted in Fig. 3(a) and the computational grid for that time step is plotted in Fig. 3(b). Furthermore, preliminary results for the averaged streamwise velocity at different locations are compared with experiment available results[13,14] in figure 4. As can be seen, results obtained with a parallel adaptive mesh refinement are very promising as they are able to predict the first and second order statistics successfully.

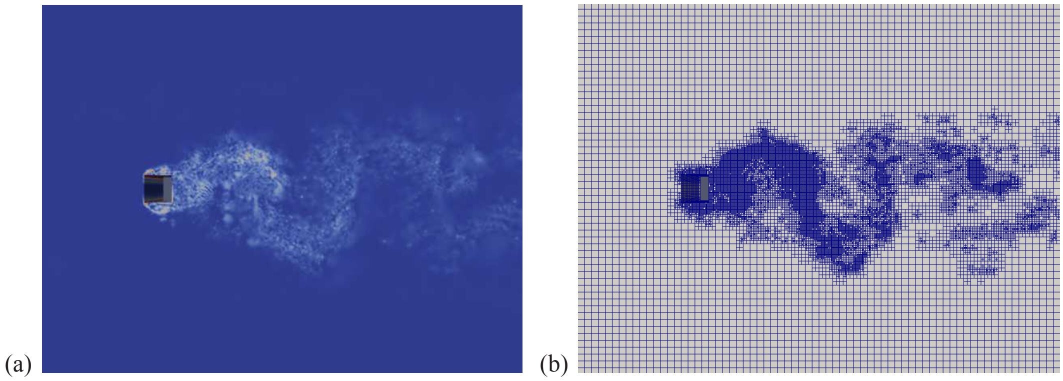

Fig. 3. Illustration LES of turbulent flow around a square cylinder (a)Vorticity structures (b) computational grid.

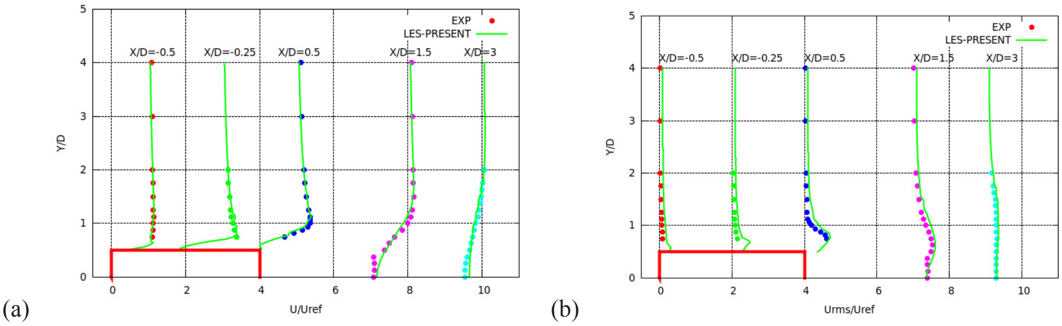

Fig. 4. Preliminary results of statistics at different locations (a)Averaged streamwise velocity (b) and Reynolds stresses.

4. Concluding remarks and future work

A parallel adaptive mesh refinement has been developed and implemented on a parallel CFD environment. AMR-LES of a turbulent flow around a square cylinder at Re = 22000 has been carried out using a symmetry-preserving formulation. Numerical results of the velocity average and fluctuations demonstrated the potential of the approach for performing AMR-LES for turbulent flow past bluff bodies. A comparison between the AMR-LES results and the experimental references has been analyzed in different zones, where the grid adaptation has been applied. The computational cost benefits using LES-AMR approach will be studied. More simulations for different kind of configurations, schemes and meshes will be presented in order to assess the potential of this kind of numerical simulations in the final paper.

Acknowledgements

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

References

[1] M. J. Berger., 1984. Adaptive mesh refinement for hyperbolic partial differential equations. Journal of Computational Physics, 53:484–512.
[2] M. J. Berger and R. J. LeVeque., 1989. An adaptive cartesian mesh algorithm for the euler equations in arbitrary geometries. AIAA, Paper 89-1930.
[3] K. G. Powell, P. L. Roe, and J. Quirk., 1993. Adaptive-mesh algorithms for computational fluid dynamics. In M. Y. Hussaini, A. Kumar, and M. D. Salas, editors, Algorithmic Trends in Computational Fluid Dynamics, pages 303–337, Springer-Verlag, New York.
[4] J.Hoffman, , 2005. Computation of mean drag for bluff body problems using Adaptive DNS/LES, SIAM J. Sci. Comput. Vol.27(1):184-207.
[5] J.Hoffman and C.Johnson, , 2006. A new approach to Computational Turbulence Modeling, Comput. Methods Appl. Mech. Engrg.
[6] J.P.P. Magalhães, D.M.S. Albuquerque, J.M.C. Pereira, J.C.F. Pereira, 2013. Adaptive Mesh Finite-Volume Calculation of 2D Lid-Cavity Corner Vortices, Journal of Computational Physics.
[7] F. Nicoud and F. Ducros., 1999. Subgrid-scale stress modeling based on the square of the velocity gradient tensor. Flow, Turbulence and Combustion, 62:183–200.
[8] T.J.R. Hughes, L. Mazzei, and K.E. Jansen., 2000. Large eddy simulation and the variational multiscale method. Computing and Visualization in Science, 3:47–59.
[9] R. W. C. P. Verstappen and A. E. P. Veldman., 2013. Symmetry-Preserving Discretization of Turbulent Flow. Journal of Computational Physics, 187:343–368.
[10] O. Lehmkuhl, R. Borrell, J. Chiva and C.D. Perez-Segarra., 2009. Direct numerical simulations and symmetry-preserving regularization simulations of the flow aver a circular cylinder at Reynolds number 3900. In Turbulence, Heat and Mass Transfer.
[11] I. Rodriguez, R. Borrell, O. Lehmkuhl, C.D. Perez-Segarra, and A. Oliva.,2011. Direct Numerical Simulation of the Flow Over a Sphere at Re = 3700. Journal of Fluid Mechanics, 679:263–287 .
[12] K. Schloegel, G. Karypis, and V. Kumar. ParMETIS, Parallel graph partitioning and sparse matrix ordering library, 2011.
[13] Lyn, D.A., Einav, S., Rodi, W. and Park, J.H., 1994. A laser-Doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder. Rept. SFB 210 /E/100.
[14] Lyn, D.A. and Rodi, W, 1994. The flapping shear layer formed by flow separation from the forward corner of a square cylinder. J. Fluid Mech., 267-353.