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)
July 20–25, 2014, Barcelona, Spain

BOUNDARY LAYER ADAPTIVITY FOR INCOMPRESSIBLE TURBULENT FLOWS

Kedar C. Chitale[1], Michel Rasquin[2], Onkar Sahni[1], Mark S. Shephard[1] and Kenneth E. Jansen[3]

[1] Rensselaer Polytechnic Institute, Troy, NY
[2] Argonne National Laboratory, IL
[3] University of Colorado Boulder, CO

Key words: *Adaptive Mesh Generation, Boundary Layers, Incompressible Flows, Turbulence*

The accuracy of numerical simulations strongly depends on the mesh resolution and quality. In complex flow problems, it is difficult to determine the adequate mesh resolution *a priori*. In such cases, an initial mesh is used to get an approximate flow solution; this mesh is then adapted using *a posteriori* error or correction indicators, i.e., based on the approximate numerical solution. This process is carried out iteratively to attain a given level of accuracy. In order for the overall adaptive process to be efficient, the resolution needs to be changed or adapted in a local fashion. This can be done by locally modifying the mesh elements based on a size field. One option is to use scalar error indicators to determine the desired mesh size field, leading to isotropic elements. However, most flow problems of interest exhibit highly anisotropic solution features such as boundary layers, shear layers, shock waves etc. These features are most efficiently resolved with anisotropic elements, i.e., where elements are oriented and stretched in a certain manner.

For viscous flows, boundary layers need to be resolved efficiently and accurately as they are a prominent flow feature. Additionally when the boundary layers are turbulent, which is often the case for high Reynolds number flows, mesh spacing needs to be properly controlled. Meshing boundary layer region with isotropic elements will put an excessive demand on the computational resources due to an extremely large mesh. Furthermore, a fully unstructured anisotropic mesh results in poorly shaped elements (e.g., elements with aspect ratio above 1000) and in-turn leads to a numerical solution of poor quality [4]. To remedy these problems, layered, orthogonal and graded elements are used near the walls whereas rest of the domain is filled with unstructured elements; this is referred to as a *boundary layer mesh*. Such hybrid meshes have been extensively used for flow simulations [2]. During adaptivity, it is desirable to maintain this layered structure of

[1]Email: chitak2@rpi.edu

elements. Adaptation procedures based on local mesh modifications for boundary layer meshes have been presented in previous work [4]. These procedures have recently been extended to work in parallel for distributed boundary layer meshes [3]. One of the main limitations of the adaptive strategies described in the above references is that the wall normal mesh spacing in the boundary layer is kept constant. The next step in the area of adaptive research for boundary layers is the ability to adapt in the wall normal direction and finding suitable indicators to set the mesh sizes in that direction.

Turbulent boundary layers have been studied extensively, both experimentally and computationally, and the mesh spacing requirements near the walls are well understood in terms of in-plane/lateral and thickness/normal resolution needs for different turbulence modeling approaches. These near-wall resolution requirements are usually defined in a dimensionless form of wall or plus units (e.g., Δx^+, Δy^+ and Δz^+) and vary according to the turbulence mode (e.g., RANS, LES or DNS) and the type of wall treatment (i.e., resolved or modeled). Since the mesh spacing requirements in the boundary layer region largely depend on the flow physics, it is advantageous to use this insight to drive the local adaptivity. Mesh resolution and structure in these regions need to be carefully chosen, with an emphasis on setting the wall normal spacing in a correct manner. Some strategies for selecting parameters of meshes inside the boundary layer region are explained in [1].

In this paper we explore ways to set parameters for boundary layer meshes in the wall normal direction using the flow physics. Other regions of the mesh are adapted using a mesh size field derived from the Hessian of the flow solution quantities. We apply this technique to incompressible turbulent flows to study its effectiveness and efficiency.

REFERENCES

[1] K. Chitale, O. Sahni, S. Tendulkar, R. Nastasi, K. E. Jansen, and M. Shephard. Boundary layer adaptivity for transonic turbulent flows. In *Proceedings of the 21st AIAA Computational Fluid Dynamics conference*, San Diego, CA, June 2013. AIAA 2013–2445.

[2] R. V. Garimella and M. S. Shephard. Boundary layer mesh generation for viscous flow simulations. *International Journal for Numerical Methods in Engineering*, 49:193–218, 2000.

[3] A. Ovcharenko, K. Chitale, O. Sahni, K. E. Jansen, and M. Shephard. Parallel adaptive boundary layer meshing for CFD analysis. In *Proceedings of the 21st International Meshing Roundtable*, 82, pages 437–455. IMR, 2012.

[4] O. Sahni, K. E. Jansen, M. S. Shephard, C. A. Taylor, and M. W. Beall. Adaptive boundary layer meshing for viscous flow simulations. *Engineering with Computers*, 24(3):267–285, 2008.