Monthly Report for November

SUN Yujie

1 Numerical examples

1.1 Turbulent flow benchmark around a square cylinder at $Re = 22,000$

Most of the studies on flow around cylinder have been performed for circular cylinders. Due to the wide range of applications of rectangular section cylinder in the engineering field, the flow around square cylinders has also been extensively studied recently. After full consideration, Many research have chosen the configuration of $Re = 22,000$ [5, 2, 6]. It was designed as a benchmark to assess the accuracy of turbulence models and the quality of numerical algorithms in some turbulence seminars.

The scale of turbulence is much larger than the molecular mean free path range, and still satisfies the continuous medium hypothesis. Most researchers believe that the Navier–Stokes (NS) equations can describe turbulence. The incompressible NS equations of a Newtonian fluid are described as follows:

$$\begin{cases} \nabla \cdot \boldsymbol{u} = 0 \\ \dfrac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} = -\nabla p + \nu \nabla^2 \boldsymbol{u} \end{cases} \tag{1}$$

where p stands for pressure, ν is kinematic viscosity, and $\boldsymbol{u} = (u, v, w)$ presents the velocity field. Distinguishing from the conventional filtered equations, we adopt the variational multiscale residual-based turbulence modeling for large eddy simulation. The basic idea of VMS is to replace the filter with a variational projection. The theory is derived directly from the incompressible NS equations 1, completely avoiding the use of eddy viscosity. It has demonstrated good numerical results in isotropic turbulence and channel turbulence [1]. In addition to being designed as a residual-driven turbulence-based model, the variational multiscale method (VMS) enables equal-order interpolation of velocity and pressure approximations while stabilizing the system with regard to its saddle point characteristics and avoids numerical oscillations if the system is predominantly convective. Besides, we use the generalized α method for full implicit time integration. Specific discretizations can be referred to [3].

1.1.1 Mesh and boundary condition configuration

In this case, the dimensionless Reynolds number $Re = UD/\nu$ is defined based on the inflow velocity U and the width of the square cylinder D. The size of outer boundaries of the fluid domain is $30.5D \times 54D \times 4D$ in the stream-wise, cross-stream and span-wise direction, respectively. The square cylinder is placed at the center of the cross-stream, and the front of the square cylinder is $10D$ away from the inlet and the rear surface is $19.5D$ away from the outlet. The dimensions of the square cylinder in the x,y and z direction are $1D \times 1D \times 4D$. The coordinates origin is located in the center of the square cylinder. The boundary conditions employed for this case are as follows: (i) A constant incoming flow with streamwise velocity $\boldsymbol{u} = (U, 0, 0)$ is applied in the inlet. (ii) In order not to affect the upstream flow, convective boundary conditions are adopted in the outlet: $\partial \boldsymbol{u}/\partial t + U\partial \boldsymbol{u}/\partial x = 0$. (iii) The upper and lower surfaces in the cross-stream direction are set as Neumann boundary condition: $\partial u/\partial y = v = \partial w/\partial y = 0$. (iv) The span-wise direction is subject to a periodic boundary constraint. (v) No-slip wall boundary conditions are imposed for all solid surfaces on the square cylinder. The details of the computational domain and boundary conditions are schematically illustrated in Figure 1. The relevant dimensions and boundary conditions considered in the above geometry is consistent with the reference [5, 2, 6].

As shown in Table 1, two meshes (LR and HR) are selected to discretize the fluid domain into 4,313,493 and 7,783,060 tetrahedral elements, respectively. Since no wall functions are used, the boundary layer mesh in the near square cylinder wall must be fine enough to ensure that wall units $y^+<1.0$. The wall units y^+ is defined as $y^+ := u_\tau y/\nu$, where $u_\tau := \sqrt{\tau_w/\rho}$ is friction velocity and $\tau_w := \rho\nu\left(\frac{\mathrm{d}\langle u\rangle}{\mathrm{d}y}\right)_{\mathrm{wall}}$ is the wall shear stress. Based on this, Cao et al. [2] and Tamura et al. [4] concludes that the height (Δy) of the nearest mesh closest to the cylinder wall is determined to be less than $0.1D/Re^{0.5} = 6.7\times10^{-4}$. In this work, Δy is set to 5.76×10^{-4} for both the LR and HR meshes, and the growth ratio of the boundary layer mesh near the cylinder in the direction of stream-wise and cross-stream is about 1.05. In the span-wise direction, the LR and HR mesh are uniformly distributed with the intervals of $0.05D$ and $0.04D$, respectively. In the stream-wise direction, the minimum grid distance $(\Delta x)_{\mathrm{min}}$ of LR mesh is 0.01, while that of HR mesh is 0.006. The global mesh, local mesh and boundary layer mesh of fluid domain are displayed in Figure 2.

Trial et al. [5] derives the minimum Kolmogorov time scale $\tau_\eta = (\nu/\langle\epsilon\rangle) \approx 6.8\times10^{-3}$ from DNS analysis, where $\langle\epsilon\rangle = 2\nu\langle s' : s'\rangle$ represents the time-averaged disspation of turbulence kinectic energy and $s' = 1/2(\nabla\boldsymbol{u}' + (\nabla\boldsymbol{u}')^T)$ is for fluctuating rate-of-strain tensor. Therefore, the dimensionless time step ($\Delta t^* = \Delta tU/D$) selected for LR and HR meshes is determined to be 1.0×10^{-3} and 5×10^{-4}, respectively, which both are significantly smaller than τ_η.

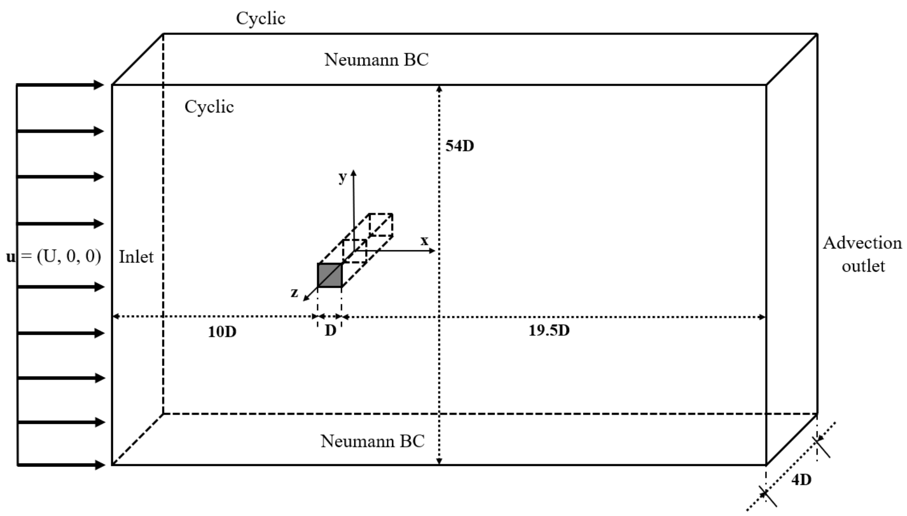

Figure 1: Computational domain and boundary conditions

Case	Δy	y^+	$(\Delta x)_{\mathrm{min}}$	Δz	Type of Element	Grid Points
LR	5.76×10^{-4}	—	1.0×10^{-2}	5.0×10^{-2}	tetrahedra	4,313,493
HR	5.76×10^{-4}	—	6.0×10^{-3}	4.0×10^{-2}	tetrahedra	7,783,060
LES Zeng [6]	6.0×10^{-4}	≤ 1.0	6.0×10^{-3}	4.0×10^{-2}	hexahedra	$270\times280\times101$
LES Cao [2]	5.6×10^{-3}	≤ 1.49	—	5.0×10^{-2}	hexahedra	$300\times300\times81$
DNS Trials [5]	1.44×10^{-3}	≤ 0.15	1.89×10^{-3}	1.454×10^{-2}	hexahedra	$1272\times1174\times216$

Table 1: Spatial mesh details

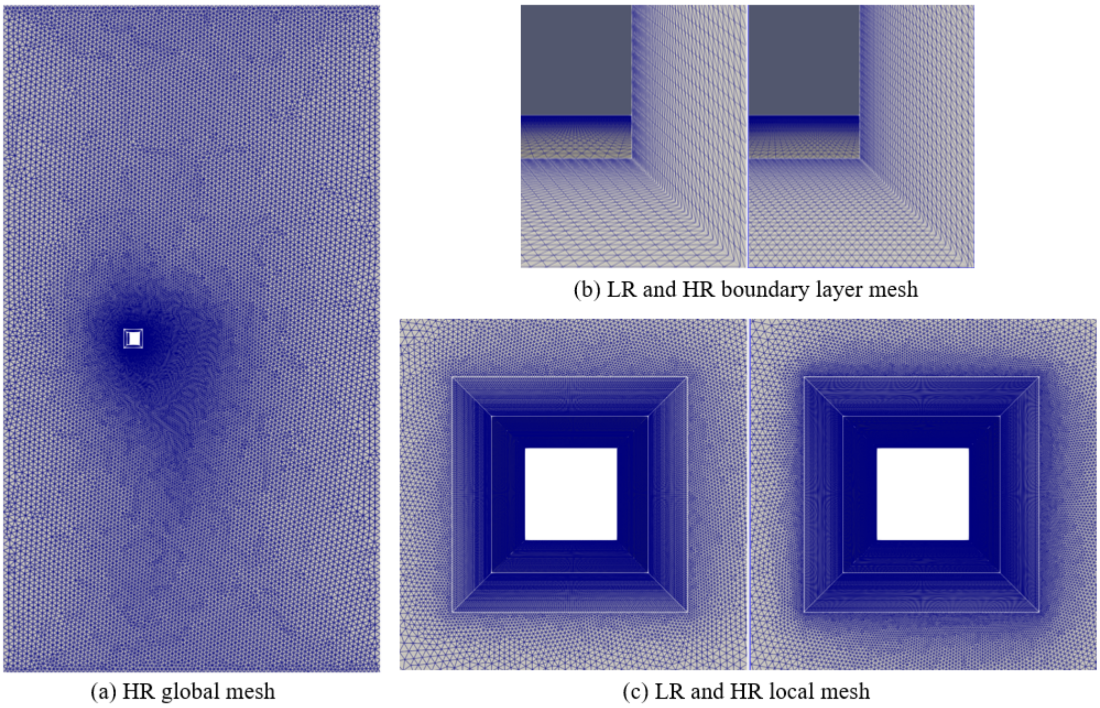

Figure 2: Meshes of the fluid domain

References

[1] Y Bazilevs et al. “Variational multiscale residual-based turbulence modeling for large eddy simulation of incompressible flows”. In: *Computer methods in applied mechanics and engineering* 197.1-4 (2007), pp. 173–201.

[2] Yong Cao and Tetsuro Tamura. “Large-eddy simulations of flow past a square cylinder using structured and unstructured grids”. In: *Computers & Fluids* 137 (2016), pp. 36–54.

[3] Ju Liu et al. “The nested block preconditioning technique for the incompressible Navier–Stokes equations with emphasis on hemodynamic simulations”. In: *Computer methods in applied mechanics and engineering* 367 (2020), p. 113122.

[4] Tetsuro Tamura and Yoshiyuki Ono. “LES analysis on aeroelastic instability of prisms in turbulent flow”. In: *Journal of wind engineering and industrial aerodynamics* 91.12-15 (2003), pp. 1827–1846.

[5] FX Trias, Andrei Gorobets, and A Oliva. “Turbulent flow around a square cylinder at Reynolds number 22,000: A DNS study”. In: *Computers & Fluids* 123 (2015), pp. 87–98.

[6] Kai Zeng et al. “Implicit large eddy simulations of turbulent flow around a square cylinder at Re= 22,000”. In: *Computers & Fluids* 226 (2021), p. 105000.