INCREASE OF VORTEX STRUCTURES RESOLUTION BY USE OF HYBRID GRID BASED AND GRID FREE NUMERICAL METHODS

Kornev Nikolai[1], Jakobi Gunnar[1] & Cherunova Irina[2]
[1]*Chair of Modelling and Simulation, University of Rostock, Rostock, Germany*
[2]*Technology Faculty, Don State Technical University, Shakhty, Russia*

Abstract A new numerical hybrid technique based on the coupled grid based and grid free simulations is developed to improve the resolution of vortex structures in CFD. The method is based on decomposition of vortex structures into large scale and small scale parts. The large scale vortices are simulated on grid whereas the small vortices are represented by particles tracked in grid free Lagrangian way. The vorticity transport equation is split into coupled system of transport equation taking into account the interaction between particles and grid based field. Simulation of two simple academic cases shows the advantages of the hybrid method.

INTRODUCTION

Insufficient resolution of vortex structures is one of the key problems in Computational Fluid Dynamics (CFD). First, the turbulent models used in Reynolds Averaged Navier Stokes Equations (RANSE) and Large Eddy Simulations (LES) approaches can be too diffusive. Second, the grid based methods possess rather high numerical diffusion which is proportional to the grid resolution. Both effects result in non- physical flow smoothing making difficult the reproduction of concentrated vortex structures with scales comparable with the cell size Δ. The numerical diffusion can be sufficiently diminished when the grid free Lagrangian methods like the Computational Vortex Method (CVM) [1] are applied. The vorticity domain is represented as a set of vortex particles tracked in the Lagrangian way. The CVM has the following advantages: low numerical diffusion, no restrictions with respect to the CFL stability criteria, convenience in results interpretations in terms of vorticity, etc. Being developed many decades ago, the CVM is still not became a popular tool in the turbulence research because of the following difficulties: formulation of boundary conditions on solid boundaries, artificial noise typical for all particle methods, viscosity effects modeling, stability problems in three dimensional cases, etc. Taking the fact into account that many disadvantages of CVM can be easily solved within grid based methods and vice versa the authors came to idea to combine both methods to improve the resolution of vortex structures in CFD.

HYBRID METHOD

The idea is based on the decomposition of vortex structures and velocity field into large scale (superscript g) and small scale (superscript v) parts : $\vec{u}(\vec{x},t) = \vec{u}^v(\vec{x},t) + \vec{u}^g(\vec{x},t)$. The large scale structures are simulated on the grid, whereas the small scale ones are represented through a set of particles within the CVM method. The idea requires the development of a new formalism which should contain the procedures describing the following effects: generation of new particles from grid based solution, motion of these particles in grid based flow with two way coupling between vortex particles and background field, mapping of particles back to the grid when their size is growing, diffusion of vortex particles and their disappearance. The generation of vortices is based on the idea taken from LES. Let $\vec{u}(\mathbf{x},\mathbf{t})$ be the original velocity field before the generation procedure. Application of a filtering procedure with filter width of $\alpha\Delta$ gives the spatially filtered velocity field $\bar{\vec{u}}(\vec{x},t)$. The vorticity represented by vortex elements should induce the field of pulsations $\vec{u}'(\vec{x},t) = \vec{u}(\vec{x},t) - \bar{\vec{u}}(\vec{x},t)$. Only the strongest vorticity obeying the condition $\lambda_{ci} > \varepsilon$, where ε is a certain threshold and λ_{ci} is the vortex identification criterion [2], is approximated by vortex particles. The approximation procedure is based on the assumptions that the fine vortices are axis- symmetric, their centers correspond to the local maxima of λ_{ci} field and the axes are aligned with the vorticity vector. The vortex structures with radius smaller than $\alpha\Delta$ are represented by particles, the rest remains on the grid. By special choice of vortex particles one can reproduce desirable spectra of velocity fluctuations **??**. The two way coupling motion equations are derived using the splitting according to scales. Substitution of the decomposition into the vorticity transport equation results in the system of two coupled transport equations with terms having a clear physical interpretation

$$\frac{\partial \vec{u}^g}{\partial t} + (\vec{u}^g \nabla)\vec{u}^g = \nabla\big(\pi + \frac{1}{2}\nabla(u^g)^2\big) + \nu\Delta\vec{u}^g + (\vec{u}^v \times \vec{\omega}^g)$$

$$\frac{d\vec{\omega}^v}{dt} = (\vec{\omega}^v \nabla)(\vec{u}^v + \vec{u}^g) + \nu\Delta\vec{\omega}^v$$

The terms $(\vec{u}^v \times \vec{\omega}^g)$, $(\vec{\omega}^v\nabla)\vec{u}^g$ and $(\vec{u}^g\nabla)\vec{\omega}^v$ take the interaction between different scales into account. The boundary conditions are explicitly formulated only for the grid solution. The vortex particles are diffused according to the core spreading method [1] as long as they become larger than $\alpha\Delta$. Then they are mapped back to the grid. They are eliminated if the core is degraded due to stretching. The method allows one to implement any version of grid based and CVM methods. It principally differs from the hybrid vortex- in- cell method.

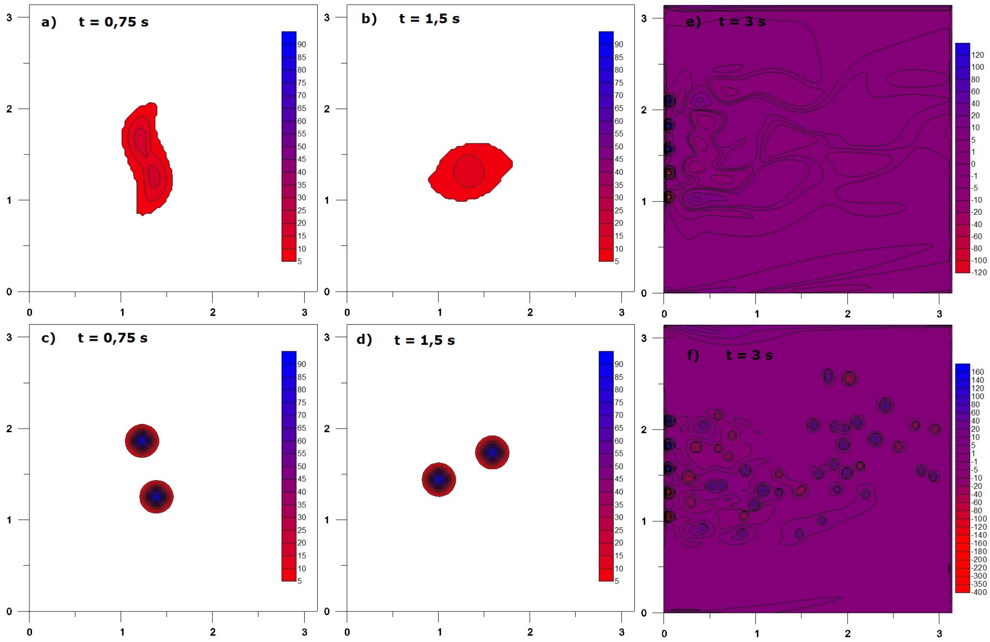

Figure 1. Simulation of vortex dynamics using Finite difference Method (a,b,e) and Hybrid Method (c,d,f). a)-d) simulation of two rotating vortices, e)-f) 2D turbulence.

TESTS

We present numerical simulations of a few academic cases illustrating advantages of the hybrid method. The simulations have been performed in the rectangular domain $\pi \times \pi$ with the uniform grid of 100×100 cells and time step of 10^{-3}s. The kinematic viscosity is zero ($\nu = 0$, $Re = \infty$), i.e. only artificial viscosity is present. The upwind scheme was applied for the grid based simulations using Finite difference Method (FDM). Two Lamb- Oseen vortices with vorticity distribution $\omega = 2\gamma Exp[-r^2/\sigma^2]/\sigma^2$ are first placed onto the grid. The vortices with $\sigma = 0.1$m are well separated and have opposite signs $\gamma = \pm 0.5 m/s$. Within the hybrid method they are identified using the two dimensional version of the algorithm described above. As seen from Fig. 1 a) and b) the vortices are merged and diffuse in the pure grid based simulations whereas they rotate and keep their identity in hybrid simulations (Fig. 1 c) and d)). Moreover, the maximum vorticity is kept almost constant in hybrid simulations.

In the second test case the two dimensional turbulence was generated by placing five Lamb Oseen vortices with random sign of vorticity uniformly on the line ($x = 0.05, \pi/3 < y < 2\pi/3$) every 0.3 s. Then they move driven by the mean flow with the velocity $U_{mean} = 1m/s$ and mutual induction. The strength of vortices was equal to $\gamma = 0.1 m^2/s$ and radius $\sigma = 0.02m$. Such a vortex occupies about three cells. Results presented in Fig. 1 show that the vortices disappear quickly in FDM (Fig. 1 e)) whereas they live in the hybrid FDM+CVM simulations up to the outlet (Fig. 1 f)).

The method implemented into the code OpenFoam is still being under development. More complicated test cases will be presented during the conference.

References

[1] P. Koumotsakos and J. Cottet. Vortex methods: theory and practice. Cambridge university press, 2000.

[2] P. Chakraborty, S. Balachandar and R.J. Adrian. On the relationships between local vortex identification schemes. *J. Fluid Mech.* **535**, No. 5: 189 - 214, 2005.

[3] N. Kornev and E. Hassel. Synthesis of homogeneous anisotropic divergence free turbulent fields with prescribed second-order statistics by vortex dipoles. *Physics of Fluids* **19**, Issue 5, 2007.