

Proposal of Numerical Methods for Computationally Affordable and High-Fidelity Large Eddy Simulation of Automotive Engines

Hiroki Yao¹⁾ Takayuki Ito²⁾ Toru Takabayashi³⁾

1) Ryoyu Systems Co., Ltd.

6-19 Oe-cho, Minato-ku, Nagoya, Aichi, 455-0024, Japan

2) Japan Automobile Research Institute

2530 Karima, Tsukuba, Ibaraki, 305-0822, Japan

3) Honda R&D Co., Ltd.

4630 Shimotakanezawa, Haga-machi, Haga-gun, Tochigi, 321-3393, Japan

KEY WORDS: heat engine, spark ignition engine, numerical calculation, Large Eddy Simulation [A1]

As the circumstances surrounding the development of automotive internal combustion engines have become increasingly complex, there is a growing need to reduce the cost of Computational Fluid Dynamics (CFD) simulations, one of the key analysis tools, and to improve its prediction accuracy. Reynolds-Averaged Navier-Stokes (RANS) simulations have been conventionally used in the development of engines. However, the predictive capability of RANS is limited by its turbulence modeling concept. Hence, the expectations for predictions using Large Eddy Simulation (LES) have been rising instead.

To conduct high-fidelity LES simulations, the most important thing in terms of numerical methods is to employ a low-dissipative scheme for the convection term in the Navier-Stokes equation to eliminate extra and unphysical viscosity (called numerical viscosity) as much as possible. Also, higher-order schemes are preferable because they can capture high-wavenumber turbulence. Furthermore, in practice, those schemes are required to be applicable to difficult flows in engines: flows including shocks, reacting flows and spray flows, as well as to turbulent flows.

In the CFD history, Finite Volume Method (FVM) has been used for practical and engineering simulations because arbitrary meshes are available. On the other hand, Finite Difference Method (FDM) has been used for more research purposes rather than engineering purposes. Here, it is notable, improving the order of accuracy of an FDM scheme is more easily and computationally low-costly achieved compared to FVM-based schemes.

In this study, according to the above, authors propose a hybrid method that high-order FDM schemes are used for far-wall region and FVM-based schemes are used for near-wall region. In addition, Cut-Cell Method is employed for the boundary treatment to ensure the conservation of fluid. Figure 1 shows a conceptual image of them.

In this paper, TENO-family schemes are used for far-wall region as high-order FDM-based schemes. For an example, TENO6-A scheme, which is one of them, requires 6-point stencil and has 4th order accuracy. Note that those schemes were designed to capture turbulence and shock simultaneously.

To reduce computational cost, cell-based Adaptive Mesh Refinement (AMR) is used in this study. Thus, there could be AMR-level interfaces in meshes. Ghost cells (or hanging nodes) are placed along the level interfaces so that wide stencils can be placed across the level interfaces and then high-order FDM-based schemes can be applied. Here, a physical value of each ghost cell is defined by interpolations using values of lower-level cells.

From point of view of the cost reduction, improving order of accuracy has a huge impact. When high-order schemes are used, meshes could be coarsened. If a mesh is coarsened as twice, the computational cost is reduced to 1/16 in theory. That is remarkable and is another reason to use high-order FDM-based schemes even though their cost is higher than low-order schemes.

In this paper, the feasibility of above proposed methods is checked through several problems including basic and practical. As a basic problem, numerical results of the enstrophy evolution of the Taylor-Green Vortex ($Re=1,600$) are shown in Figure 2. One can find that TENO6 (128^3 cells) is roughly comparable to CENTRAL4 (128^3) and MUSCL3 (256^3). Where DNS, CENTRAL4 and MUSCL3 in Fig.2 denotes Direct Numerical Simulation, 4th order skew-symmetric central difference and 3rd order MUSCL scheme respectively. Note that 2nd order MUSCL schemes are widely used in common CFD codes.

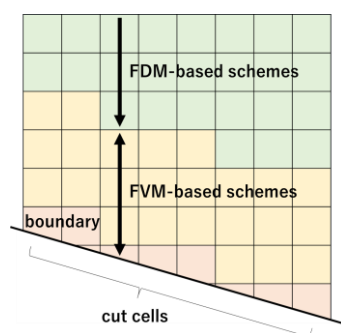


Fig.1 Conceptual image of proposed methods

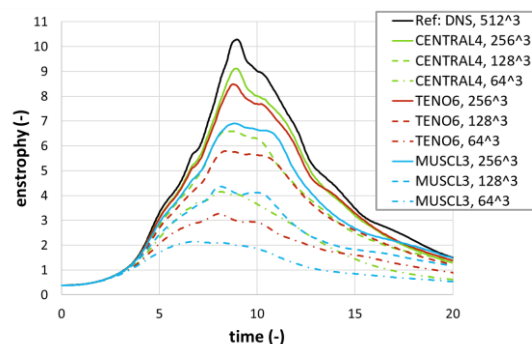


Fig.2 Enstrophy evolution with different schemes and meshes