Implicit Large Eddy Simulation for the High-Order Flux Reconstruction Method

Hui Zhu[*], Song Fu
School of Aerospace Engineering
Tsinghua University
Qinghua Yuan, Beijing, 100084 China
zhu-h12@mails.tsinghua.edu.cn

Lei Shi, ZJ Wang
Department of Aerospace Engineering
University of Kansas
Lawrence, Kansas, 66044, USA

ABSTRACT

High-order methods have demonstrated their potential in large eddy simulations (LES) of turbulent flows with relatively low Reynolds numbers. The cost becomes a serious limiting factor for high Reynolds flows. A promising approach to reduce the cost of these simulations is the hybrid Reynolds-Averaged Navier-Stokes (RANS)/LES approach. In this paper, a new hybrid RANS-Implicit LES approach for the high-order FR/CPR method is presented, using a simple algebraic version of the Spalart-Allmaras model in the vicinity of solid walls, and implicit LES approach elsewhere. Despite its simplicity, this approach shows good performance in simulating turbulent flow at relatively high Reynolds numbers.

INTRODUCTION

The past 15 years witnessed the rapid development of high-order CFD methods, including the traditional Finite Volume Method (FVM), Discontinuous Galerkin Method (DG), and new methods such as Spectral Volume Method (SV) and Spectral Difference Method (SD). In 2007, Huynh proposed a compact differential high-order method called Flux Reconstruction (FR), which provides a general framework for many other high-order methods, including DG. In 2009, Wang and Gao extended this method to general mixed grids, and named it as Correction Procedure via Reconstruction (CPR).

High-order methods have shown their potential in simulation of turbulent flow, e.g. Taylor-Green Vortex and Decaying Isotropic Turbulence. Due to their property of high resolution, a relatively coarser mesh can be used when resolving turbulence. However, for wall-bounded turbulent flows at a relatively high Reynolds number, there is still no mature approach suitable for high-order methods. Reynolds Averaged Navier-Stokes (RANS) may introduce stiff turbulence model equations, which is hard to solve by high-order methods. Meanwhile, in the philosophy of RANS, turbulence is all modelled, and this does not take advantage of the resolution capability of high-order methods. On the other hand, wall resolved Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) require too many computational resources for high Reynolds number turbulent flows. Therefore, wall modelled LES or hybrid RANS-LES approaches should be appropriate for high-order methods.

In this paper, a new hybrid RANS-Implicit LES approach for high-order FR/CPR method will be presented. It combines an algebraic eddy viscosity model and Implicit LES, which both have simple formulations with no necessity to solve additional stiff turbulence model equation, and shows good ability in our test cases.

This paper is organized as follows. In the second section, a brief review of the FR/CPR method is given. In section 3, the new hybrid approach is described in detail. In Section 4, several test cases are presented to verify this approach. The conclusions are drawn in Section 5.

BRIEF REVIEW OF THE FR/CPR METHOD

The FR/CPR method was proposed by Huynh originally in 2007 for hyperbolic partial differential equations, and later Wang and Gao to extended it to general unstructured mesh. In 2009, Huynh showed how to use FR/CPR to solve equations with diffusion terms, thus enabling it to solve Navier-Stokes Equations. This method could be categorized into discontinuous finite element methods, like the famous DG method, but also has some advantages. FR/CPR has a differential formulation, involving no explicit numerical quadrature, which means the computation cost is less. Also, FR/CPR offers a general framework for other high-order methods including DG, and makes it possible to implement several kinds of high-order methods without much modification in the code.

In the FR/CPR method, the conservative variables inside one cell are assumed to be polynomials, and expressed by nodal values at certain points called Solution Points (SPs). Based on the solution polynomial we can get a polynomial of flux and its divergence. This step is called Reconstruction. However, this flux polynomial is not continuous in adjacent cells, thus not conservative. Like in DG or FVM, we can use a Riemann solver to get common face flux with discontinuous conservative variables, and then interpolate the difference of common face flux and local face flux to SPs, which is called the Correction step.

In summary, the final form of FR/CPR Method can be written like this:

$$\frac{\partial \mathbf{U}_i}{\partial t} + \mathbf{\Pi}_i\left(\nabla \cdot \mathbf{F}\left(\mathbf{U}_i\right)\right) + \delta_i = 0 \qquad (1)$$

Here, U_i is the solution polynomial in the i-th cell, F is the local flux based on this solution polynomial, and δ_i is the correction term. Generally, when F is not a linear function of U, then the divergence term does not lie in the polynomial space of U_i. In order to make the finite element idea work, we should introduce an operator Π to project this term to the solution polynomial space. In this paper, a Chain Rule (CR) projection operator is used for the inviscid flux term to minimize the aliasing error, and a Lagrange Polynomial (LP) one is used for the viscous flux term to make the formulation as simple as possible.

For viscous flux involving the gradient of conservative variables, directly using the gradient of U_i can give wrong solution. Here, an additional set of variables should be solved together using the FR/CPR method:

$$\mathbf{R} = \nabla \mathbf{U} \qquad (2)$$

In this paper, we use a Bassi-Rebay 2 (BR2) scheme to solve for the gradient variable R, then Eq. (2) can be expressed in an algebraic way, giving both corrected local gradient inside one cell and corrected common gradient on interfaces of cells.

All the test cases in section 4 are solved by an in-house code called MUSIC (MUlitiphysics SImulation Code). MUSIC is a general equation solver on unstructured mesh using the FR/CPR method, which can handle both 2-D and 3-D Navier-Stokes Equations. In this paper, to make results clear, the adaptation part is not used, and time marching method is explicit third-order SSP Runge-Kutta. All the cases are solved with 4th-order FR/CPR method on hexahedral meshes, which we find has good ability to resolve turbulence, and strong robustness to avoid diverging during calculation.

A NEW HYBRID RANS-IMPLICIT LES APPROACH

General Framework

Hybrid RANS-LES approaches can be basically divided into two groups. The first group is DES-type (Detached Eddy Simulation) ones, which calculate most of the flowfield using RANS, and only use LES where there is massively separated flow and enough mesh resolution. The most important characteristic of DES-type hybrid method is that they should always work in a RANS mode in the whole attached boundary layer. However actually this cannot be guaranteed because original DES relies on mesh size to switch from RANS to LES, thus may accidentally trigger LES mode inside the boundary layer when mesh is too fine. This problem has been solved by Delayed DES (DDES).

The other kind of Hybrid RANS-LES method is Wall Modelled LES (WMLES). Since the turbulence structure size is restricted near the wall, the size of the so-called "Large Eddy" here is much smaller than the integral scale of the flowfield. If they are to be resolved, then the computation cost can be close to DNS. Therefore, in practical LES, wall modelling is necessary. This type of hybrid approach treats most of the flowfield with LES, but near the wall introduces a RANS model in some way, such as assigning wall shear stress, embedded RANS mesh, or using RANS eddy-viscosity. When DES was first proposed, Nikitin 2000 tried to use it in a WMLES sense, but the results seems to be not satisfactory, and a well-known phenomenon of log-layer mismatch (LLM) was observed. Later, Travin made a thorough analysis of LLM, and gave a new method called Improved Delayed DES (IDDES). IDDES can work properly as a WMLES method, eliminating LLM, when most of the turbulence can be resolved, and return to DDES in the absence of resolved turbulence. However, IDDES involves too many empirical relations and constants, adding to its complexity.

Recently, Li and Wang found that for the high-order FR/CPR method, Implicit LES can perform better than LES with Sub-Grid Stress (SGS) models, such as static and dynamic Smagorinsky models. Since high-order FR/CPR method has much better resolution ability than second-order FVM, more turbulence information can be captured. Moreover, in the LES concept, SGS models only work well when filter size lies in the inertial subrange, and the numerical error of resolved part does not dominate the SGS modelled stress. Nonetheless, with high-order methods, often a much coarser mesh is used, and the filter size of LES can hardly be smaller than the mesh size. Meanwhile, the SGS models are often dissipative with a positive eddy viscosity coefficient, and typically this coefficient is proportional to the square of filter size, thus in the numerical viewpoint it can be regarded as an additional second-order dissipation term, which may be harmful to the resolution of the high-order FR/CPR method, causing a much larger numerical error than expected.

As mentioned before, algebraic turbulence models are preferred for the high-order FR/CPR method due to the fact that no additional stiff turbulence model equation need to be solved, and they only give an eddy viscosity coefficient to the Navier-Stokes solver, which is very easy to implement. Here, we propose a new hybrid RANS-Implicit LES approach for the high-order FR/CPR method. In the vicinity of wall boundary, a RANS eddy viscosity is calculated, and far away from the wall, this eddy viscosity vanishes and return to Implicit LES.

The hybrid eddy viscosity coefficient formulation is:

$$\mu_{t,hybrid} = \mu_{t,RANS} \cdot \left[0.5 - 0.5\tanh\left(y^{+} - 25\right)\right] \qquad (3)$$

Here, y+ is non-dimensional wall distance based on the inner scale of boundary layer. Close to the wall, y+ is much smaller than 25, and the eddy viscosity is purely RANS, and away from the wall, when y+ exceeds 30, the eddy viscosity returns to zero, making the simulation an Implicit LES. The transitional location is selected to be in the buffer layer between viscous sublayer and log layer, trying to eliminate the Log-Layer-Mismatch phenomenon.

A Near-Wall algebraic version of SA model

Traditional algebraic turbulence models, such as Baldwin-Lomax model and Cebeci-Smith model, often involve too many non-local variables, making it hard to implement in unstructured solvers. Since the new hybrid method only need turbulence model to provide eddy viscosity in the vicinity of wall, we can use a near-wall algebraic version of Spalart-Allmaras model, following Durbin 2004.

Calculation of non-dimensional wall distance

By sampling flow variables including density, tangential velocity and viscosity coefficient at a solution point in the first wall cell, the non-dimensional wall distance y+ at this solution point can be calculated iteratively with a relation between u+ and y+. Then, the y+ value can be interpolated to all the solution points and flux points inside the first wall cell. Check the maximum y+ value, if it is larger than 30, than use the eddy viscosity model above, otherwise interpolate the y+ into next cell in the wall normal direction, and so forth.

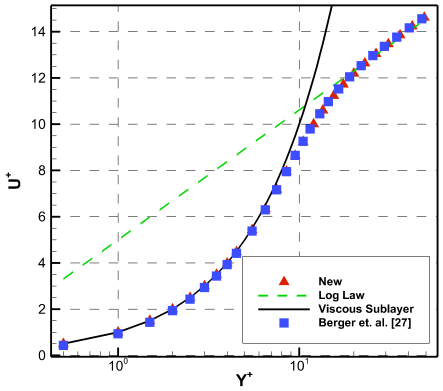


Figure 1. Comparison of velocity profile derived by the new approach and Berger's equation.

TEST CASES

Cylinder Flow at Re=3900

This test case is at a relatively low Reynolds number. No hybrid approach is used since the attached boundary layer is laminar, and a pure Implicit LES is performed to show the resolution ability of high-order Implicit LES method.

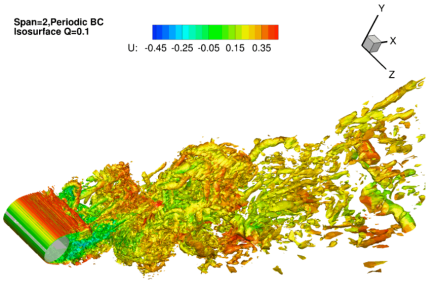


Figure 2. Overview of p3 calculation of cylinder flow at Re=3900. Isosurface of Q=0.1 coloured by x-velocity.

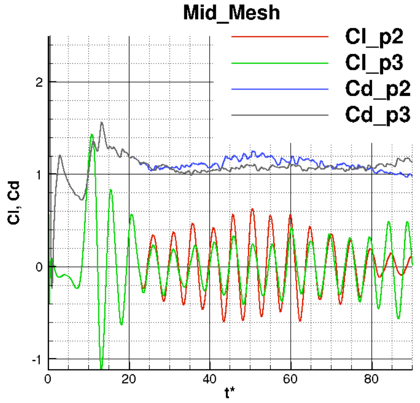


Figure 3. Force history comparison between p2 and p3 calculation at initial stage.

Turbulent Channel Flows

Turbulent Channel Flows at three different Reynolds numbers are calculated, ranging from $Re_\tau=395$ to $Re_\tau=1113$. The results show that our new hybrid method can calculate the velocity profile correctly.

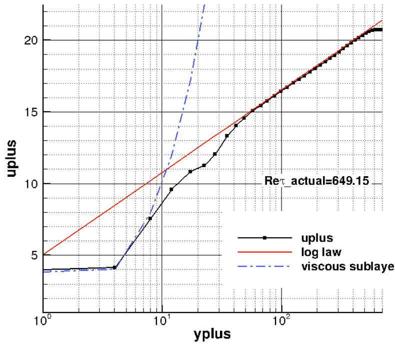


Figure 4. Averaged velocity profile calculated by new hybrid method at Reynolds number of 649

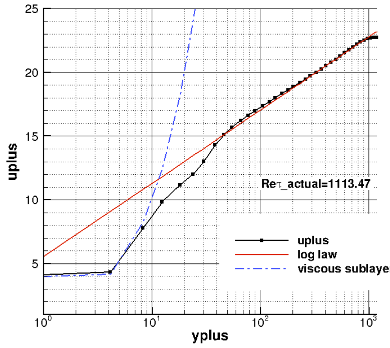


Figure 5. Averaged velocity profile calculated by new hybrid method at Reynolds number of 1113

Periodic Hill

This case is calculated to extend the hybrid method to new separated flows. Third order curved mesh is generated with Gmsh. Cases at two different Reynolds numbers, $Re_b = 2800$ and $Re_b = 10595$ are calculated. The hybrid approach shows great potential in the calculation of separation points and reattachment points, comparing to the pure implicit LES method.

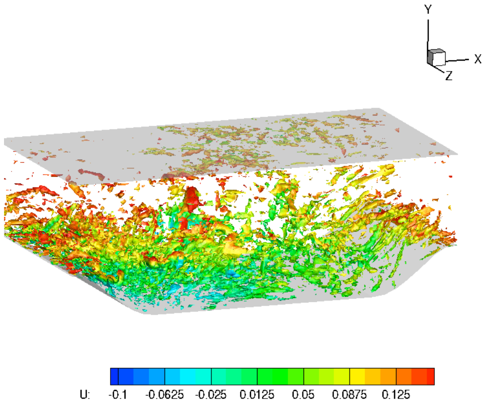


Figure 6. Overview of the Periodic Hill test case. Isosurface of Q=0.1 Coloured by x-velocity

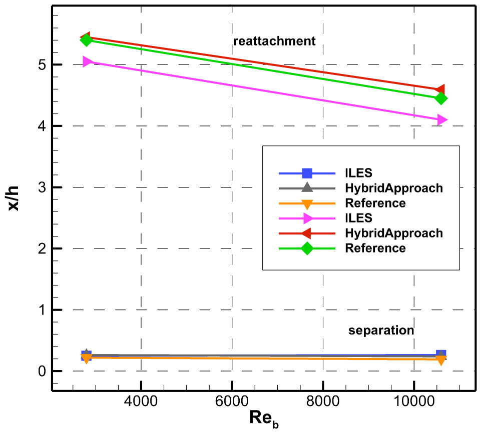
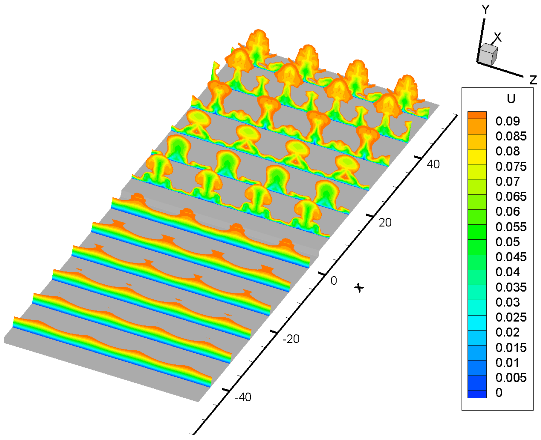


Figure 7. Separation points and reattachment points of the Periodic Hill test case

Forward-Facing Steps

The implicit Large Eddy Simulation approach based on the high-order FR/ CPR method is applied to explore the mechanism flow transition of forward facing steps in a subsonic boundary layer. The step height is a third of the local boundary-layer thickness. The Reynolds number based on the step height is 720. Inlet disturbances are introduced giving rise to streamwise vortices upstream of the step. It is observed that these small-scale streamwise structures interact with the step and hairpin vortices are quickly developed after the step leading to flow transition in the boundary layer.



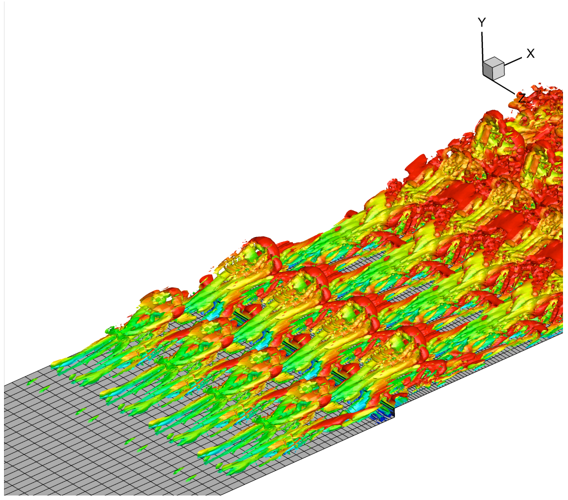


Figure 9. Isosurface of Q = 0.005 of the Forward-Facing step flows. Coloured by x-velocity

REFERENCES

Huynh, H. T. “A flux reconstruction approach to high-order schemes including discontinuous Galerkin methods.” AIAA paper 4079 (2007): 2007.

Wang, Z. J., and Gao, H. “A unifying lifting collocation penalty formulation including the discontinuous Galerkin, spectral volume/difference methods for conservation laws on mixed grids.” Journal of Computational Physics 228, No. 21 (2009): 8161-8186.

Kalitzin, G., Medic, G., Iaccarino, G., and Durbin, P. "Near-wall behavior of RANS turbulence models and implications for wall functions." Journal of Computational Physics 204, No. 1 (2005): 265-291.

Nikitin, N. V., Nicoud, F., Wasistho, B., Squires, K. D., and Spalart, P. R. "An approach to wall modeling in large-eddy simulations." Physics of Fluids (1994-present) 12, No. 7 (2000): 1629-1632.

Lodato, G., Castonguay, P. and Jameson, A. "Structural Wall-modelled LES Using a High-order Spectral Difference Scheme for Unstructured Meshes." Flow, Turbulence and Combustion 92, No. 1-2 (2014): 579-606.