A Hybrid RANS-Implicit LES Approach for the High-Order FR/CPR Method

Hui Zhu and Song Fu
Tsinghua University, Beijing, 100084, China

Lei Shi and ZJ Wang
University of Kansas, Lawrence, KS, 66045

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 number problems. 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 (ILES) 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 an ILES approach elsewhere. Despite its simplicity, this approach has demonstrated good performance in simulating turbulent flow at relatively high Reynolds numbers.

I. Introduction

The recent past has witnessed a rapid development of high-order CFD methods, including the finite volume method (FVM)[1-3], discontinuous Galerkin method (DG)[4-6], spectral volume method (SV)[7] and spectral difference Method (SD)[8]. In 2007, Huynh proposed a compact differential high-order method called flux reconstruction (FR)[9] or correction procedure via reconstruction (CPR), which provides a general framework for many other high-order methods, including DG, and later in 2009, Wang and Gao extended this method to hybrid mixed grids[10]. Since then, the FR/CPR method has been further analyzed and developed for a wide variety of problems[11-14].

High-order methods have shown their potential for simulating turbulent flows using large eddy simulation (LES), e.g. the Taylor-Green vortex and decaying isotropic turbulence at low to moderate Reynolds numbers. For many high order methods, a relatively coarse mesh can be used to achieve the accuracy required for an engineering design. However, for wall-bounded turbulent flows at a relatively high Reynolds number, the mesh resolution required for LES near the wall using high-order methods is still too demanding for present super-computers since the mesh requirement in the wall layer increases dramatically with the Reynolds number. Therefore, hybrid RANS-LES approaches are proper for turbulent flows at high Reynolds number.

Hybrid RANS-LES approaches can be basically divided into two categories. The first one 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 only on the 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 size of turbulence structures are 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, 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 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 scale (SGS) stress 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.

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.

II. Brief Review of the FR/CPR Method

The FR/CPR method was originally proposed by Huynh in 2007 for hyperbolic partial differential equations, and later Wang and Gao extended it to hybrid unstructured meshes. Further developments on the FR/CPR method are contained in [11-14, 15]. This method belongs to discontinuous finite element methods, like the DG method, but also has some unique advantages. FR/CPR has a differential formulation, involving no explicit numerical quadratures. Also, FR/CPR offers a general framework for other high-order methods including DG and SD, and makes it possible to implement several different kinds of high-order methods without much code modification.

Here we present a brief introduction of the FR/CPR method. Starting from a hyperbolic conservation law:

\[
\frac{\partial U}{\partial t} + V \cdot F(U) = 0
\] (1)

with initial and boundary conditions, where the vector \( U \) consists of conservative variables, and \( F \) is flux vector.降rnetize the computational domain with non-overlapping elements, and introduce an arbitrary test function \( W \) in each element, the weighted residual formulation of Eq. (1) on element \( V_i \) can be expressed as:

\[
\int_{V_i} \left( \frac{\partial U}{\partial t} + V \cdot F(U) \right) W d\Omega = 0
\] (2)

In the FR/CPR method, the conservative variables inside one element are assumed to be polynomials, and expressed by nodal values at certain points called solution points (SPs). After applying integration by parts to the divergence of flux, replacing the normal flux term with a common Riemann flux \( F_{com}^n \), and integrating back by parts, we get

\[
\int_{V_i} \frac{\partial U}{\partial t} W d\Omega + \int_{V_i} W V \cdot F(U_i) d\Omega + \int_{\partial V_i} W \left[ F_{com}^n - F^n(U_i) \right] dS = 0
\] (3)

Here, the common Riemann flux is defined as:

\[ F_{com}^n (U_i, U_j) = \frac{F(U_j) + F(U_i)}{2} - \frac{1}{2} \frac{d}{dt} \left( U_j - U_i \right) \]
\[ F^a_{com} = F^a_{com} \left( U_i, U_{i+}, n \right) \]  

(4)

where \( U_{i+} \) stands for the solution outside the current element, and \( n \) denotes the normal direction of the interface. The normal flux at the interface is:

\[ F^n (U_i) = F(U_i) \cdot n \]  

(5)

It is noticeable that if the face integral in Eq. (3) can be transformed into a element integral then the test function will be eliminated. In order to do so, a “correction field” \( \delta_i \) is defined in each element as:

\[ \int_{V_i} W \delta_i d\Omega = \int_{\partial V_i} W \left[ F^n \right] dS \]  

(6)

where \( \left[ F^n \right] = F^n_{com} - F^n (U_i) \) is the normal flux difference. Eqs. (3) and (6) results in:

\[ \int_{V_i} \left( \frac{\partial U_i}{\partial t} + \nabla \cdot F(U_i) + \delta_i \right) W = 0 \]  

(7)

Since we are solving Navier-Stokes equations, the flux is a nonlinear function of conservative variables. Therefore, the flux divergence term does not belong to the solution polynomial space, and a projection operator is needed. One choice is:

\[ \int_{V_i} \nabla \cdot F(U_i) W d\Omega = \int_{V_i} \nabla \cdot F(U_i) W d\Omega \]  

(8)

Because the test function \( W \) is arbitrary, we can extract a differential formulation:

\[ \frac{\partial U_i}{\partial t} + \nabla \cdot F(U_i) + \delta_i = 0 \]  

(9)

As mentioned before, the solution polynomials are expressed by solution points in each element. According to Huynh’s original paper, the location of the SPs does not affect the solution of linear equations, while they do affect nonlinear equations. Here, we choose Gauss quadrature points to be the SPs. The final formulation is:

\[ \frac{\partial U_{i,j}}{\partial t} + \nabla \cdot F(U_i) + \delta_{i,j} = 0 \]  

(10)

where subscript \( j \) denotes the \( j \)-th solution point in a certain element.

There are several choices for projection operators, including the Lagrange polynomial (LP) and the chain rule (CR) formulation. For the LP approach, the flux is assumed to belong to the same polynomial space of \( U_{i+} \), and can be expressed as

\[ \Pi (\nabla \cdot F) = \nabla \left( \sum_j L_j F_j \right) \]  

(11)
where \( L_j \) denotes the Lagrange polynomial corresponding to the \( j \)-th solution point. For the CR approach, the flux divergence is assumed to belong to the same polynomial space of \( U_i \), as follows:

\[
\Pi(\nabla \cdot F) = \sum_j L_j \left( \frac{\partial F}{\partial Q} \cdot \nabla Q \right)_j
\]  

(12)

To compute \( \delta_i \), a series of points called flux points are defined along each interface, where the normal flux differences are calculated. For elements with straight edges, the correction at solution points can be written as:

\[
\delta_{i,j} = \frac{1}{|V_i|} \sum_{j \in V_i} \sum_j \alpha_{j,f,i} \left[ F^n_j \right] S_f
\]  

(13)

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:

\[
R = \nabla U
\]  

(14)

In this paper, we use a Bassi-Rebay 2 (BR2) scheme to solve for the gradient variable \( R \), then Eq. (14) can be expressed in an algebraic way, giving both corrected local gradient inside one cell and corrected common gradient on interfaces of cells.

### III. New Hybrid RANS-Implicit LES Approach

#### A. General Framework

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:

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

(15)

Here, \( y^+ \) is non-dimenional 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 LLM phenomenon.

#### B. A Near-Wall algebraic version of SA model

Traditional algebraic turbulence models, such as the Baldwin-Lomax model and the Cebeci-Smith model, often involve too many non-local variables. This makes 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.

The formulation is

\[
\mu_{t,\text{RANS}} / \mu = \tilde{\nu}^* f_{v1}, \quad \tilde{\nu}^* = \kappa y^*, \quad f_{v1} = \left( \tilde{\nu}^* \right)^3 / \left[ \left( \tilde{\nu}^* \right)^3 + C_v^3 \right].
\]  

(16)
where, $f_{v1}$ is a damping function as defined in the original Spalart-Allmaras model, and $c_{v1}$ is a constant equal to 7.1. In the limit of the wall, where $y^+$ goes to zero, this function is proportional to $(y^+)^4$. This means that the eddy viscosity itself grows with the fourth power of distance to the wall, which is consistent with the theory of wall turbulence. In the log-layer, $f_{v1}$ returns to one, and the eddy viscosity is proportional to $y^+$, which makes the velocity profile agree with the log-law.

When this model is combined with the new hybrid approach, the relationship between the eddy viscosity and $y^+$ is illustrated in Fig. 1. In the viscous sublayer and buffer layer, the eddy viscosity equals to the algebraic version of the Spalart-Allmaras model. As $y^+$ approaches 20, the hyperbolic tangent function takes effect, and the eddy viscosity drops smoothly to zero at $y^+ = 30$.

![Figure 1. Relationship between the Eddy Viscosity and the Nondimensional Wall Distance](image)

C. 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 $y^+$.

$$u^+ y^+ = \rho uy / \mu,$$

$$u^+ = a_1 \tan^{-1} \left( b_1 y^+ + c_1 \right)$$

$$+ a_2 \log \left[ \left( y^+ + b_2 \right) y^+ + c_2 \right]$$

$$+ a_3 \log \left[ \left( y^+ + b_3 \right) y^+ + c_3 \right]$$

$$+ a_4 \tan^{-1} \left( b_4 y^+ + c_4 \right)$$

$$+ a_5,$$

where

$$a_1 = -3.63975, a_2 = -1.33017, a_3 = 2.54968, a_4 = 3.59946, a_5 = -6.33792$$
The sampling method works in the following way:
1) Get the flow variables at a certain solution point inside the first wall cell, including density, tangential velocity, and physical viscosity. Here, like Lodato, we choose the farthest solution point from the wall corresponding to each flux point on the wall, typically at $y^+$ close to 10.
2) Using the aforementioned $u^+$-$y^+$ relationship to solve $y^+$ at this chosen solution point.
3) When $y^+$ at all the chosen solution points are solved, compute the $y^+$ at all other solution points and flux points in the first wall cell. Then use the eqs. (15) and (16) to compute the eddy viscosity. Since eqs. (15) and (16) represent a strong nonlinear relation, interpolating the eddy viscosity introduces too much numerical error.
4) Check the maximum $y^+$ value corresponding to each flux point at the wall cell. If it reaches 30, nothing else needs to be done. Otherwise, a finite eddy viscosity is computed for the 2nd cell away from the wall. This process is repeated if necessary for the 3rd cell away from the wall until the maximum $y^+$ equals to or exceeds 30. In the actual implementation, we may limit the number of RANS elements to at most 2 to 3 in the wall normal direction to make sure it does not give wrong values of eddy viscosity in the initial stage of the simulation.

Strictly speaking, sampling instantaneous values and using them in an averaged equation is questionable. But here, we have some reasons to support this practice. In our hybrid RANS-ILES approach, the vicinity of the wall is designed to be a RANS region. With the eddy viscosity introduced, most of the turbulence fluctuations are modelled, or numerically suppressed, making the near wall region a RANS one. Therefore, the sampled values can be regarded as averaged ones with small oscillations in space and time.

IV. Test Cases

A. Cylinder Flow at Re=3900
Flow over a cylinder at a Reynolds number of 3,900 based on the diameter is a classical test case for LES codes, and many researchers such as Kravchenko [16] use this case to validate their LES codes. This Reynolds number lies in the subcritical region, which means that the attached boundary layer is all laminar, and transition to turbulence occurs in the wake. The detached shear layers become unstable in the wake, shedding vortices, as seen in Figure 3. Since the boundary layer is laminar, the hybrid approach is not used. Thus this case serves as a test for ILES.

Figure 2. Overview of p3 calculation of cylinder flow at Re=3900.

Figure 3. Force History comparison between p2 and p3 calculation at initial stage.
Figure 4. Mean pressure distribution on the cylinder.
Figure 5. Mean streamwise velocity profiles at different streamwise locations.

e) $x/D=7$

a) Normal Reynolds stress at $x/D=6$  
b) Normal Reynolds stress at $x/D=7$
c) Shear Reynolds stress at x/D=6  d) Shear Reynolds stress at x/D=7

Figure 6. Reynolds stress distributions in the far wake

The flow is calculated with the initial condition for a non-dimensional time of 100 before averaging, and then another 900 non-dimensional time is used to compute the time-averaged quantities. After that, a spanwise averaging is also employed to obtain more reasonable statistics.

The pressure distribution on the cylinder wall is shown in Fig. 4. The results of P2 and P3 FR/CPR schemes both agree well with the experiment data of Norberg [17]. The pressure distribution on the windward side of the cylinder matches the experimental data well, whereas there is a little difference on the leeward side where the flow is fully separated. It is worth mentioning that the experiment of Norberg is at a slightly higher Reynolds number (Re=4020), which may contribute to the difference observed here.

These results confirm that the FR/CPR method with ILES can provide reliable predictions for complicated massively-separated flows. The fact that the results from the P2 and P3 computations agree reasonably well with each other is a clear indication that the simulations are nearly p-independent. In addition, both the mean velocity and the Reynolds stresses in the wake showed good agreement with experimental data. These results serve as a foundation for the new hybrid RANS-ILES approach.

B. Turbulent Channel Flows

Turbulent Channel Flows at three different Reynolds numbers are calculated, ranging from Re_c=395 to Re_c=1140.

The total number of DOFs and resolution in wall units are summarized in Table 1. These include cases without the hybrid RANS-ILES approach to show the requirement of spanwise and streamwise resolution for ILES, and cases with the hybrid RANS-ILES approach to show its capability and insensitivity to spanwise and streamwise mesh resolutions. The viscosity coefficients in different cases are calculated according to Dean’s semi-empirical relationship of \( \Re \) and \( \Re_\theta \).

Firstly, two cases with a pure ILES approach are presented here. In these two cases, the mesh meets the requirement in the wall normal direction with a \( \gamma^+ \) value at the first cell of 12 with a 4th-order scheme. Due to the insufficient resolution in the streamwise and spanwise directions in case 1, the velocity profile is much higher than the log-law in the log layer, and the skin friction coefficient is much lower than the estimation with Dean’s law, as shown in Figure 7. It is observed that the slope of the velocity profile in the log layer is correct in both cases. This means that the turbulence is resolved well in the log layer, and the problem for the incorrect velocity profile lies in the viscous sublayer and buffer layer. This result leads to the introduction of the new hybrid RANS-ILES approach.

Turbulent channel flows at three different Reynolds numbers are calculated in case 3, case 4 and case 5 using the hybrid approach, as shown in Fig. 9. The skin friction coefficients of these cases all match Dean’s law very well, with a maximum error of only 3.3%. The velocity profiles in all three cases match the log-law very well, and the intercept for the highest Reynolds number case goes higher to 5.5 from 5.0, but still this lies in the scope of the classical log-law theory. In the velocity profile, a small kink is observed at about \( \gamma^+=25 \), where the transition from RANS to ILES occurs.

To test the grid sensitivity of the new hybrid approach, two other cases are calculated at the medium Reynolds number. In case 6, the mesh size in the wall normal direction is clustered in the vicinity of the wall and coarsened at the center of the channel, with a \( \gamma^+ \) value of 8 instead of 12 in the first wall cell, and the same size in the spanwise and streamwise directions. In case 7, the wall normal mesh is the same as in case 4, while the number of elements in both the spanwise and streamwise directions is doubled. The skin frictions in both case 6 and 7 are close to that in case 4, with a maximum difference of 0.5%. Also, the velocity profiles of case 4, 6 and 7 almost lie on top of each other, as shown in Fig. 10. From these results we can conclude that the new hybrid RANS-ILES is capable of producing reasonable results for a variety of mesh resolutions in all three directions.

<table>
<thead>
<tr>
<th>Label</th>
<th>Nominal ( \Re_c )</th>
<th>( N_x \times N_z \times N_c )</th>
<th>Min ( \gamma^+ )</th>
<th>Using Hybrid Approach</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case1</td>
<td>640</td>
<td>16 × 24 × 16</td>
<td>12</td>
<td>No</td>
</tr>
<tr>
<td>Case2</td>
<td>640</td>
<td>32 × 24 × 32</td>
<td>12</td>
<td>No</td>
</tr>
<tr>
<td>Case3</td>
<td>395</td>
<td>16 × 24 × 16</td>
<td>12</td>
<td>Yes</td>
</tr>
<tr>
<td>Case4</td>
<td>640</td>
<td>16 × 24 × 16</td>
<td>12</td>
<td>Yes</td>
</tr>
</tbody>
</table>

Table 1 Grid size for channel flow computations
<table>
<thead>
<tr>
<th>Case</th>
<th>Resolution</th>
<th>Re_{b}</th>
<th>Number of Iterations</th>
<th>Validation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case5</td>
<td>1140</td>
<td>16×24×16</td>
<td>12</td>
<td>Yes</td>
</tr>
<tr>
<td>Case6</td>
<td>640</td>
<td>16×24×16</td>
<td>8</td>
<td>Yes</td>
</tr>
<tr>
<td>Case7</td>
<td>640</td>
<td>32×24×32</td>
<td>12</td>
<td>Yes</td>
</tr>
</tbody>
</table>

Figure 7. Skin friction coefficient of calculation and Dean’s empirical formula

Figure 8. Velocity profiles for cases with a pure ILES approach
Figure 9. Velocity profiles of cases at different Reynolds numbers using the hybrid RANS-ILES approach

Figure 10. Velocity profiles of cases with different mesh resolutions at the same Reynolds number, using the hybrid RANS-ILES approach
C. Periodic Hill

The periodic hill problem is a classical test case to evaluate the capability of turbulence simulation approaches for massively separated flows. The flow is restricted by a series of smooth symmetric hills of height $h$, and the distance between the crest of two adjacent hills is $9h$. On the opposite side of the hills, the flow is constricted with a flat plate, and the normal distance at the crest is $2.035h$. The whole flowfield is dominated by alternative adverse and favourable streamwise pressure gradients, which make the case a great challenge for the simulation of turbulence.

A P3 curved mesh is generated with Gmsh. The mesh size is $32 \times 32 \times 16$ in $x$, $y$, $z$ directions respectively, which means for a 4th-order FR/CPR calculation the number of DOFs is $128 \times 128 \times 64$. As for the two cases in our simulations, this mesh is a relatively fine one for the low Reynolds number case (Re$_b=2800$), whereas for the higher Reynolds number case (Re$_b=10595$) this mesh is similar in size to the coarse mesh of the ATAAC project. Reference data for these two cases can be found in [18](wall resolved LES) and [19](experimental data) respectively.

In our simulation, we start statistics for turbulence when the transitional phase is completed at a non-dimensional time of 500, and then calculate another non-dimensional time of 1500 to ensure converged turbulence statistics.

One of the main challenges is to predict both the separation point and re-attachment point in the time-averaged field. Unlike other cases with sharp corners in the geometry, the separation point is determined by the flow instead of the geometry. Through the results of the European ATAAC project, the periodic hill case is beyond the ability of RANS simulations. All RANS results predicted the separation bubble to be much larger than reference data, no matter which turbulence models (Spalart-Allmaras, Menter-SST, or Reynolds stress models) were selected. However, with our new hybrid approach, both the separation point and reattachment point are predicted close to the reference data, and the difference is no larger than 0.14$h$, as shown in Figure 12. In comparison, if we use a pure ILES approach on the same mesh, the difference is much larger, especially for the re-attachment location.

The streamwise velocity profiles are plotted next to demonstrate the new hybrid approach in resolving details in the flow field. In the low Reynolds number case, the velocity profiles can match the reference data of Fröhlich’s wall resolved LES[18] at all four locations very well. In the high Reynolds number case, the difference between our result and experiment data is larger. However, the results with the hybrid approach compare favorably to results with a pure ILES and other DES results from the ATAAC database. Especially, at $x/h=6$ after the reattachment location, the velocity profile computed with the hybrid approach is close to the experimental data. It is noticeable that at all 4 locations ILES always significantly underpredicts the velocity gradient at the upper wall where there is no separation. Like the channel flow case, this is because the turbulence structures are resolved insufficiently in the upper wall boundary layer.

Figure 11. Overview of the Periodic Hill test case. Isosurface of $Q=0.1$ coloured by $x$-velocity
Figure 12. Separation and reattachment locations
Figure 13. Streamwise velocity contours and streamline sketches
Figure 14. Streamwise velocity profile at different stream locations at $Re_b=2800$

a) $x/h=0.5$

b) $x/h=2$

c) $x/h=4$

d) $x/h=6$
V. Conclusions

In this paper, a new hybrid RANS-ILES approach for the FR/CPR method is developed. It combines an algebraic eddy viscosity RANS model near solid walls with ILES elsewhere. This approach involves no additional turbulence model equations, and no sub-grid stress models in the LES regions. Therefore the formulation is very simple. In addition, contrary to traditional algebraic RANS models such as the Baldwin-Lomax model or the Cebeci-Smith model, the RANS model in the new hybrid approach does not require any non-local variables, so it is more appropriate for solvers on unstructured mesh.

The basic idea for this approach is as follows. Wherever it is possible to resolve most of the turbulence with the high-order FR/CPR method, ILES is employed. Where the cost to resolve the turbulence is too high, an eddy viscosity model is used to model most of the turbulence in the viscous sublayer and buffer layer inside a turbulent boundary layer near a wall. In between, a simple hyperbolic tangent function is applied to ensure a smooth transition. Therefore, the hybrid approach can also be categorized as a wall modelled LES approach. Comparing with the DES approach, this new hybrid approach does not involve an explicit mesh size, thus avoids such phenomenons as grid induced separation. Moreover, the RANS-LES interface location in the new approach is fixed in the buffer layer, eliminating another drawback of DES, the log-layer mismatch problem.

Several test cases are simulated to evaluate the capability of this new approach. Firstly, the flow over a cylinder at a relatively low Reynolds number of 3900 is calculated to test the high-order FR/CPR method with a pure ILES approach. Both the third and fourth order FR/CPR schemes can accurately predict the streamwise velocity profiles in the wake, as well as the Reynolds stresses. Then, the turbulent channel flow at three different Reynolds numbers are simulated. It is found that without the new hybrid RANS-ILES approach, the mesh requirement for a pure ILES simulation is too high. The hybrid approach can correctly calculate the velocity profiles of the attached turbulent flow, without a strong dependence on the mesh size or the Reynolds number. Finally, a classical test case of flow over a periodic hill at two different Reynolds numbers is calculated. This challenging case contains alternative favourable and adverse pressure gradients and a massively separated flow region. The new hybrid approach is capable of predicting a much better recirculation length than a pure ILES, as well as the streamwise velocity profiles.

These encouraging results show the potential of the new hybrid RANS-ILES approach. Further work will be devoted to applying this approach to more complicated turbulent flows of engineering interest, such as delta wings or wing-body configurations. Also, the RANS model applied here is based on a zero pressure gradient assumption, which may deteriorate the performance of the hybrid approach in flows with massively separations. To make further improvements, algebraic turbulence models considering pressure gradients such as Duprat’s will be implemented in the future.

Figure 15. Streamwise velocity profile at different stream locations at Reₗ=10595

c) x/h=4

d) x/h=6
References


