Adaptive RANS Solution with the High-order Correction Procedure via Reconstruction Method

Lei Shi, Cheng Zhou and Z. J. Wang

Department of Aerospace Engineering, University of Kansas, Lawrence, KS, 66045

In this paper, we present an adjoint-based mesh adaptation method for the Reynolds-averaged Navier-Stokes (RANS) Equations to minimize the output error. A dual-consistent high-order correction procedure via reconstruction method (CPR) is utilized to discretize the RANS equations with the modified Spalart-Allmaras (SA) model. The wall distance of each solution point in the computing domain is computed by solving the Eikonal equation using the CPR framework. The mesh refinements are driven by the automated output-based adaptation. The adaptive results of the turbulence flow over a flat plate problem and the turbulence flow over the NACA0012 airfoil problem demonstrate the ability of the present method to efficiently reduce the functional errors in terms of the number of degrees of freedom (DOFs).

I. Introduction

High-order methods have the potential to achieve higher accuracy at lower cost than lower order methods. This potential has been demonstrated conclusively for smooth problems in several recent International Workshops on High-Order Methods [47]. The use of high-order methods to compute turbulent flows governed by the Reynolds-averaged Navier-Stokes (RANS) equations is an active research topic in the computational fluid dynamics (CFD) community. However, due to the numerical stiffness, the high-order methods for the RANS equations are difficult to converge to the steady-state [35, 37, 38]. For those problems, solution based hp-adaptations offer the best promise. It can ensure the reliability and increase the robustness of the high-order methods for the RANS equations, which has received considerable attentions in the high-order CFD community. [15, 16, 21, 43–45, 52].

In this paper, a dual-consistent high-order correction procedure via reconstruction method (CPR) is utilized to discretize the RANS equations with the modified Spalart-Allmaras (SA) one-equation model. To alleviate the stability issues caused by the negative turbulence working variable, several researchers have proposed a variety of modifications for the SA turbulence model [14, 34, 36]. The modified SA turbulence model investigated in [36] has been demonstrated to significantly improve the robustness of high-order simulations, which is used in the present study. In this model, the r closure function depends on the distance to the nearest wall. To compute the distance of each solution point in the domain to the nearest curved polynomial wall boundaries, the CPR high-order discretization is extended to solve the Eikonal equation.

The effectiveness of adaptive methods highly depends on the accuracy of the error estimation. The dual-weighted residual method proposed by Becker and Rannacher [5] relates a specific functional output directly to the local residual by solving an additional adjoint equation. It can capture the error propagation effects inherent in the hyperbolic equations. This kind of adjoint-based error indicator has been shown very effective in driving a hp-adaptation procedure to obtain a very accurate prediction of the functional outputs [6, 7, 10, 13, 20, 25, 38]. In the present method, a dual-consistent high-order CPR is utilized to compute the discrete adjoint solution associated to the engineer interested outputs, e.g. the lift coefficient, and derive the output-based local error indicator. The mesh refinement procedures are driven by the automated output-based adaptation.

*Post-doctoral researcher, Department of Aerospace Engineering, 2120 Learned Hall, Lawrence, KS 66045, AIAA Member.
†Ph.D. candidate, Department of Aerospace Engineering, 2120 Learned Hall, Lawrence, KS 66045, AIAA Member.
‡Spahr Professor and Chair, Department of Aerospace Engineering, 2120 Learned Hall, Lawrence, KS 66045, Associate Fellow of AIAA.
The marked candidate elements for adaptation can be modified by enriching its solution order or subdividing its element or resizing its grid. Thus, the ways to increase the discretization resolution can be generally classified into 3 categories: h-refinement, r-refinement and p-refinement. For h-refinement, subdivision is performed locally for each candidate element to increase the total DOFs. R-refinement or the moving mesh method keeps the total number of nodes the same but moves the location of the grid locally or globally [22]. With p-refinement, the local degree of approximation polynomial is modified. The moving mesh method with curved elements in 3D is still an on-going research. In this work, we only subdivide elements hierarchically for adaptations.

The rest of the paper is organized as follow: In section 2, we briefly review the governing equations and the modified SA model. The high-order CPR formulation and the solution method are presented in Section 3. Section 4 describes the adjoint-based error estimation and the h-adaptation strategies. In Section 5, adaptive results of the turbulence flow over a flat plate problem are presented. Finally, conclusions and some possible future work are given in Section 6.

II. The Modified Spalart-Allmaras Turbulence Model

The compressible RANS equations with the modified one-equation SA turbulence model [14, 34, 36] can be written in the conservative form as

$$\frac{\partial Q(t)}{\partial t} + \nabla \cdot (F(Q) - F_t(Q, \nabla Q)) = S(Q, \nabla Q).$$

(1)

Here, $Q$ is the conservative variables, $F$ is the inviscid flux, $F_t$ is the viscous flux vector and $S$ is the source term, which are given by

$$Q = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ E \\ \rho \dot{\theta} \end{pmatrix}, \quad F^x = \begin{pmatrix} \rho u \\ \rho u^2 + P \\ \rho u v \\ u(E + P) \\ \rho u \dot{\theta} \end{pmatrix}, \quad F^y = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + P \\ v(E + P) \\ \rho v \dot{\theta} \end{pmatrix},$$

$$F^z = \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ u\tau_{xx} + v\tau_{xy} + c_p(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t}) \frac{\partial T}{\partial x} \\ \frac{1}{\sigma}(\mu + \mu\Psi) \frac{\partial \dot{\theta}}{\partial x} \end{pmatrix}, \quad F^y = \begin{pmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ u\tau_{yx} + v\tau_{yy} + c_p(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t}) \frac{\partial T}{\partial y} \\ \frac{1}{\sigma}(\mu + \mu\Psi) \frac{\partial \dot{\theta}}{\partial y} \end{pmatrix},$$

$$S = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ c_b1\tilde{S}\rho\nu\Psi + \frac{1}{\sigma}[c_{t2}\rho(\nabla \dot{\nu} \cdot \nabla \dot{\nu}) - c_{w1}\rho f_{w}(\nu\Psi)^2 - \frac{1}{2}\nu(1 + \Psi)\nabla \dot{\nu} \cdot \nabla \dot{\nu}] \end{pmatrix},$$

where the $\rho$, $P$, $E$ are respectively the density, pressure and specific total energy per unit mass, $u, v$ denote the Cartesian velocity. $\nu$ denotes the kinematic viscosity and $\dot{\nu}$ represents the turbulence working variable of the SA model. Then the pressure $P$ is given by,

$$P = (\gamma - 1)(\rho E - \frac{1}{2}\rho(u^2 + v^2)), \quad (3)$$

where $\gamma = 1.4$ is the ratio of specific heats. Define velocity vector $u = (u, v)$, then the fluid viscous stress tensor for Newtonian fluid $\tau$ is defined as

$$\tau_{ij} = (\mu + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right), \quad (4)$$

American Institute of Aeronautics and Astronautics
where $\delta_{ij}$ is the Kronecker delta. $\mu$ refers to the fluid dynamic viscosity, and $\mu_t$ is the turbulence eddy viscosity defined by the SA model reads

$$\mu_t = \begin{cases} \rho \tilde{v} f_{v1} & \text{if } \tilde{v} \geq 0 \\ 0 & \text{if } \tilde{v} < 0. \end{cases}$$  

(5)

$$f_{v1} = \frac{\Psi^3}{\Psi^3 + C_{v1}^3}$$

For the source term $S$ of Eq. (2), the production term of the modified SA model $\tilde{S}$ is given as

$$\tilde{S} = \begin{cases} S + \dot{S} & \text{if } \dot{S} \geq -c_{v2} S \\ \frac{S(c_{v2}^2 S + c_{v3})}{(c_{v3} - 2c_{v2})S - \dot{S}} & \text{if } \dot{S} < -c_{v2} S \end{cases}$$

$$S = \sqrt{\tilde{\omega} \cdot \tilde{\omega}}$$

$$\dot{S} = \frac{\nu \Psi}{\kappa_2 d^2} f_{v2}$$

$$f_{v2} = 1 - \frac{\Psi}{1 + \Psi f_{v1}}$$

where $\tilde{\omega} = \nabla \times \mathbf{u}$ is the vorticity vector. The destruction term coefficients are given by

$$r = \min \left[ \frac{\nu \Psi}{S \kappa_2 d^2}, 10 \right]$$

$$g = r + c_{w2}(r^6 - r)$$

$$f_w = g \left( \frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right)^{1/6}$$

(7)

where $d$ denotes the distance to the nearest wall at a specific location. The parameter $\Psi$ is designed for high-order discretization schemes to remove the effects of negative turbulence working variable on the robustness of the turbulence model. This parameter is given as

$$\Psi = \begin{cases} 0.05 \ln(1 + e^{20 \chi}) & \text{if } \chi \leq 10 \\ \chi & \text{if } \chi > 10 \end{cases}$$

(8)

When $\tilde{v}$ goes negative, the parameter $\Psi$ can prevent instabilities by turning off the production, destruction and dissipation terms. Finally, the constants in the modified SA model are given as

$$c_{b1} = 0.1335, \ c_{b2} = 0.622, \ \sigma = 2/3, \ \kappa_t = 0.41, \ P_r = 0.72, \ P_r t = 0.9,$$

$$c_{w1} = c_{b1}/\kappa^2 + (1 + c_{b2})/\sigma, \ c_{w2} = 0.3, \ c_{w3} = 2, \ c_{o1} = 7.1, \ c_{o2} = 0.7, \ c_{o3} = 0.9$$

(9)

### III. The CPR Discretization

Eqn (1) is discretized using the correction procedure via reconstruction (CPR) method [23]. The CPR formulation has some remarkable properties. The framework is easy to understand, efficient to implement and recover several well known methods such as the discontinuous Galerkin (DG) [2 4 - 12, 13, 39, 40, 51], the spectral volume method (SV) [32, 46, 50] and the spectral difference methods (SD) [27, 30, 31, 33, 42]. For recent development with CPR, interested readers can refer to [9, 10, 17, 18, 24, 26, 39, 54].

Assume that the computational domain $\Omega$ is discretized into $N$ non-overlapping elements $\{V_i\}_{i=1}^{N}$. And let $Q_i$ be an approximate solution to the analytical solution $Q$ on $V_i$. The CPR discretization of Eq. (1) for each solution point $j$ of cell $i$ can be expressed as

$$\frac{\partial Q_i}{\partial t} + \sum_{j}(\nabla \cdot F(Q_i)) - \sum_{j}(\nabla \cdot F^v(Q_i, R_i)) + \frac{1}{|V_i|} \sum_{f \in \partial V_i} \sum_{l} \alpha_{j,f,l}([F^n]_{f,l} - [F^{v,n}]_{f,l}) S_f = S(Q_i, R_i),$$

(10)
\[ \mathbf{R}_{i,j} = (\nabla (Q_i))_j + \frac{1}{|V_i|} \sum_{f \in \partial V_i} \sum_{l} \alpha_{j,f,l} [\mathbf{F}^{\text{com}} - Q_i]_{f,l} \mathbf{n}_f S_f, \]  

where \( \alpha_{j,f,l} \) are the correction coefficients due to the jumps at the flux point \( l \) of face \( f \), \( S_f \) is the face area, \( |V_i| \) is the cell volume, \( [\mathbf{F}^n] \) is the inviscid flux jump and \( [\mathbf{F}^{v,n}] \) is the viscous flux jump. The inviscid and viscous flux jumps defined as

\[
[F^n] = F^n_{\text{com}} - F(Q_i) \cdot \mathbf{n} \\
[F^{v,n}] = F^v(Q_{\text{com}}, \nabla Q^{\text{com}}) \cdot \mathbf{n} - F^v(Q_i, \mathbf{R}_i) \cdot \mathbf{n}
\]

Here, \( Q^{\text{com}} \) and \( \nabla Q^{\text{com}} \) are the common solution and the common gradient on each interface respective. \( F^n_{\text{com}} \) denotes a numerical flux on the interface. In the current study, we use the Roe Riemann flux to compute the inviscid common flux \( F^n_{\text{com}} \) and the Bassi and Rebay (BR2) \[3\] for the the common solution \( Q^{\text{com}} \) and the common gradient \( \nabla Q^{\text{com}} \). \( \Pi \) is the projection operator for the inviscid and viscous flux divergence terms. The Lagrange polynomial approach (LP) \[45\] is used.

In the SA model, the working variable \( \tilde{\nu} \) can have different order of magnitude than the other components of the state. In order to improve the floating point precision, the dynamic scaling of \( \tilde{\nu} \) in Ref. is used. To compute the distance of each solution point in the domain to the nearest curved polynomial wall boundaries, the CPR high-order discretization is extended to solve the Eikonal equation [Ref].

The first order backward Euler scheme is used for the time integration. The system of linear equations is solved using the preconditioned GMRES (Generalized Minimal RESidual) solvers from the PETSc library \[1\] . The ILU(1) preconditioner and Line-searches are used to speed up the convergence. More solution details can be found in \[8\].

### IV. Adjoint-based Error Estimation and H-adaptation

Adjoint-based error estimation relates a specific functional output directly to the local residuals by the adjoint solution, which can be used to construct a very effective error indicator to drive an adaptive procedure toward any engineering output. Let \( Q_h \) denotes an approximate solution to the analytical solution \( Q \). The difference between them can be interpreted as a solution perturbation \( \delta Q = Q - Q_h \). The output error defined as \( \delta \mathcal{J} = \mathcal{J}(Q_h) - \mathcal{J}(Q) \) can be estimated by the adjoint weighted residual method

\[
\delta \mathcal{J} \approx - \int_{\Omega} \psi(\mathcal{N}(Q_h) - \mathcal{N}(Q))d\Omega = - \int_{\Omega} \psi \mathcal{N}(Q_h)d\Omega.
\]

Since the CPR method is not in a variational form, the discrete adjoint formulation for the CPR method utilizes an explicitly defined variational form to obtain the dual-consistent adjoint solution. Assume the adjoint solution belongs to the same space of the primal solution, the adjoint variable \( \psi_i \) of cell \( i \) can be approximated using the Lagrange basis \( L_j \)

\[
\psi_i = \sum_j L_j \psi_{i,j}.
\]

The discrete adjoint equation for the CPR method reads

\[
- \sum_i \sum_j \frac{\partial r_{i,j}}{\partial Q_l} \omega_j |J_{i,j}| \psi_{i,j} = \frac{\partial \delta \mathcal{J}}{\partial Q_l},
\]

where \( r_{i,j} \) is a pointwise residual defined on each solution point \( j \) of cell \( i \) arising from a CPR scheme and \( \omega_j \) and \( |J_{i,j}| \) are the quadrature weight and the element Jacobian at the solution point. More detailed discussion about the dual-consistency of the CPR method can be found in Ref. \[31\].

Based on Eq. \[13\] the output error estimate can be expressed as

\[
\delta \mathcal{J} \approx - \sum_i \sum_j r_{i,j}(Q_h) \omega_j |J_{i,j}| \psi_{i,j}.
\]

Also, we can define a corrected output using the functional error estimate

\[
\mathcal{J}_{\text{corr}} \equiv \mathcal{J}(Q_h) + \delta \mathcal{J}.
\]
From the Eq. [13] the output error can be estimated by performing a quadrature rule as

\[ \delta J_h(Q_h) \approx - \sum_i \sum_j \omega_j |J_{i,j}| \psi_{i,j} r_{i,j}(Q^H_h). \]  

(18)

The continuous adjoint solution \( \psi \) is approximated by solving \( \psi_h \) on the finer space through enriching the degree of the solution polynomial. The finer solution \( Q_h \) is obtained by performing several steps of GMRES relaxation after prolongating from the coarse solution \( Q^H_h \)

\[ Q^H_h = I^H_h Q_H \]  

(19)

with an injection operator \( I^H_h \). The adjoint-based local error indicator \( \eta \) used in this paper is defined by taking an absolution value of the elemental output error contribution

\[ \eta_i = | \sum_j \omega_j |J_{i,j}| (\psi_h - I^H_h \psi^H_h)_{i,j} r_{i,j}(Q^H_h) |. \]  

(20)

Here, to achieve a better estimates, the adjoint defect between the coarse level and fine level \( \psi_h - I^H_h \psi_H \) is used. For systems of equation, the local error indicators are formed by summing together every component’s contribution to the functional error estimate.

The error indicators defined above are used to drive a fixed-fraction anisotropic h-adaptation. In this approach, a certain fraction \( f \) of the current elements with the largest local error indicators \( \eta \) are marked for h-refinements. Figure 1 shows the procedure of the adjoint-based h-adaptation for the CPR method. Non-conforming interfaces between cells with different h levels are created during the adaptations. In order to ensure the solution smoothness, only one level difference of h-refinement between neighboring cells are allowed. Special treatment is required when computing the common numerical flux on those non-conforming interfaces with hanging nodes. The “mortar” element method developed by Kopriv[28] is used here. For a non-conforming interface, a “mortar” face is introduced to link the unmatched elements, whose space are always chosen as the higher h or p space of the two sides. First, the solution from the left and right sides of the face are prolongated to the mortar surface by a simple interpolation process (see Figure 2a on the following page). Then, the common flux are computed by solving the Riemann problem on the mortar surface. The last step is to project the common flux on the mortar surface back to the original space. Here, the standard \( L^2 \) projection is utilized to preserve the average.

Figure 1: The procedure of the adjoint-based h-adaptation.
Mesh refinement is performed in the original element’s polynomial space using the reference coordinates. So the refined elements inherit the same geometry approximation order. However, for elements on the geometry boundaries, the newly generated vertex on the boundary edge may not be exactly on the real geometry. An extra remapping process is employed to snap the boundary points to the truth geometry during each adaptation level.

V. Numerical Results

V.A. Turbulence Flow over a Flat Plate

We consider subsonic, turbulence flow over a flat plate. This test case is from the NASA’s Turbulence Modeling Resource (TMR) website \[29\]. The problem is solved on a rectangular domain of size \([-1/3, 2] \times [0, 1]\). The plate length is 2, which spanning from \(x = 0.0\) to \(2.0\). The free stream Mach number is \(M_0 = 0.2\), and the Reynolds number based on the plate length of 1 is \(Re = 5 \times 10^6\). The adiabatic no-slip wall boundary condition is enforced along the plate, and a symmetry boundary condition is specified on the first part of the lower boundary. Thus, the leading edge of the plate is a singularity point between the symmetry boundary condition and the no-slip boundary condition. The total pressure and static pressure are fixed respectively on the left and right boundaries. Farfield characteristic boundary condition is enforced on the upper surface. For the laminar viscosity, Sutherland’s law with \(T_s = 110K\) and \(T_{ref} = 300K\) is used.

The 2nd and 3rd order CPR schemes \((k = 1, 2)\) with the Gauss points as the SPs/FPs and the LP approach are tested. The initial mesh, as shown in Figure 4a, consists of 34 \(\times\) 24 quadrilateral elements, which has an approximate average \(y^+ \approx 1.7\) over the plate. In this case, isotropic h-adaptions driven by the drag adjoint error indicator are tested. Figure 4 shows the adapted mesh and the eddy viscosity contours from the finest adaptation stage with \(k = 2\). The leading edge and the elements around the lower boundary are refined repeatedly on each adaptation level.

Figure 6a shows the convergence history of the drag coefficient with CFL3D and FUN3D results using the SA model. The results show that the adaptive values converge much faster than the uniform refinements, and all try to converge to the same value. The converged results agree with CFL3D and FUN3D within 0.1
count. The truth $C_D = 0.00285875$ is chosen from the finest $k = 2$ adaptive result. Figure 5 compares the extracted non-dimensional eddy viscosity at $x = 0.97$. The $p = 1$ result shows some oscillations, while the $p = 2$ results shows excellent agreement with the CFL3D and FUN3D results. Figure 6b compares the CD error for all the tested adaptation strategies. With h-adaptation, effective convergence rates of 1.2 and 5.2 were achieved for $C_D$ with $k = 1$ and $k = 2$ respectively, as shown in the Figure 6b. Again it is shown that the adjoint based h-adaptation approach can reduce the number of DOFs by orders of magnitude.

Figure 4: The initial and adapted results for the turbulence flow over a flat plate problem at $M_0 = 0.2$, $Re = 5 \times 10^6 (k = 2)$. Red circle indicates the leading edge.

V.B. Turbulence Flow over the NACA0012 Airfoil

The next test case is a turbulent flow over the NACA0012 airfoil at Mach number $Ma = 0.15$, Reynolds number $Re = 6 \times 10^6$, with angles of attack $\alpha = 0^\circ, 10^\circ$. This case is used as a validation case of CFD codes
Figure 5: The non-dimensional eddy viscosity of turbulent flow over a flat plate at \( Ma = 0.2, Re = 5 \times 10^6, x = 0.97 \) with 136 \times 96 elements.

Figure 6: \( C_D \) convergence for the turbulent flow over a flat plate problem at \( M_0 = 0.2, Re = 5 \times 10^6 \).
on the TMR webpage[29], by comparing all the CFD results with the experimental results.

The farfield boundary is located almost 500 chords away from the airfoil. The initial mesh is a C-type grid with $54 \times 23$ fourth-order curved elements generated using gmsh. The first layer grid gives $y^+ \approx 10$. H-adaptations with $k = 1, 2, 3, 4$ are driven by the output-based error indicator. The lift coefficient is considered as the outputs of interest. Additionally, uniform h-refinement is performed to compare those adaptation strategies. Figure 7 compares the Mach contours and presents the adapted meshes at angles of attack $\alpha = 0^\circ$ and $10^\circ$. Note that regions near the stagnation streamlines and inside the boundary layer are targeted for refinements. The trailing edge is also refined repeatedly to reduce the effect of the geometry singularity.

Figure 9 displays the convergence history of the drag coefficients along with the CFL3D and FUN3D results. The results show that the adaptive values converge much faster than the uniform refinements, and the adaptation with $k = 3$ and $4$ try to converge to the same value between the CFL3D and FUN3D results with much finer grid. The truth outputs is chosen from the the final stage of the adaptive simulations with $k = 4$. Figure 10 compares the $C_D$ errors of all tested adaptation strategies with results from the uniform h-refinements.

VI. Conclusions

In this paper, we apply an output-based h-adaptation method with the high-order CPR formulation to the Reynolds-averaged Navier-Stokes (RANS) Equations to minimize the functional error. A dual-consistent high-order correction procedure via reconstruction method (CPR) is utilized to discretize the RANS equations with the modified Spalart-Allmaras (SA) model. The wall distance of each solution point in the computing domain, which is required in the SA model, is computed by solving the Eikonal equation using the CPR framework. The mesh refinements are driven by the automated output-based adaptation. The adaptive results of the turbulence flow over a flat plate problem and the turbulence flow over the NACA0012 airfoil problem demonstrate the ability of the present method to efficiently reduce the functional errors in terms of the number of degrees of freedom (DOFs).

References


Figure 7: Adjoint-based h-adaptation for the NACA 0012 airfoil at $M_0 = 0.15$, $\alpha = 0^\circ$, $Re = 6 \times 10^6$. 

(a) The initial mesh

(b) The adapted mesh

(c) Mach number contours on the initial mesh

(d) Mach number contours on the adapted mesh
Figure 8: Adjoint-based h-adaptation for the NACA 0012 airfoil at $M_0 = 0.15$, $\alpha = 10^\circ$, $Re = 6 \times 10^6$
Figure 9: $C_D$ convergence for turbulence flow over the NACA 0012 airfoil at $M_0 = 0.15$, $\alpha = 1^\circ$ and $10^\circ$, $Re = 6 \times 10^6$

Figure 10: $C_D$ error for turbulence flow over the NACA 0012 airfoil at $M_0 = 0.15$, $\alpha = 1^\circ$ and $10^\circ$, $Re = 6 \times 10^6$


