On the dynamic computation of the model constant in delayed detached eddy simulation

Zifei Yin
_Iowa State University_, zifeiyin@iastate.edu

K.R. Reddy
_Iowa State University_

Paul A. Durbin
_Iowa State University_, durbin@iastate.edu

Follow this and additional works at: [http://lib.dr.iastate.edu/aere_pubs](http://lib.dr.iastate.edu/aere_pubs)

Part of the Aerodynamics and Fluid Mechanics Commons, Other Aerospace Engineering Commons, and the Structures and Materials Commons

The complete bibliographic information for this item can be found at [http://lib.dr.iastate.edu/aere_pubs/62](http://lib.dr.iastate.edu/aere_pubs/62). For information on how to cite this item, please visit [http://lib.dr.iastate.edu/howtocite.html](http://lib.dr.iastate.edu/howtocite.html).
On the dynamic computation of the model constant in delayed detached eddy simulation

Abstract
The current work puts forth an implementation of a dynamic procedure to locally compute the value of the model constant CDES, as used in the eddy simulation branch of Delayed Detached Eddy Simulation (DDES). Former DDES formulations [P. R. Spalart et al., “A new version of detached-eddy simulation, resistant to ambiguous grid densities,” Theor. Comput. Fluid Dyn. 20, 181 (2006); M. S. Gritskevich et al., “Development of DDES and IDDES formulations for the k-ω shear stress transport model,” Flow, Turbul. Combust. 88, 431 (2012)] are not conducive to the implementation of a dynamic procedure due to uncertainty as to what form the eddy viscosity expression takes in the eddy simulation branch. However, a recent, alternate formulation [K. R. Reddy et al., “A DDES model with a Smagorinsky-type eddy viscosity formulation and log-layer mismatch correction,” Int. J. Heat Fluid Flow 50, 103 (2014)] casts the eddy viscosity in a form that is similar to the Smagorinsky, LES (Large Eddy Simulation) sub-grid viscosity. The resemblance to the Smagorinsky model allows the implementation of a dynamic procedure similar to that of Lilly [D. K. Lilly, “A proposed modification of the Germano subgrid-scale closure method,” Phys. Fluids A 4, 633 (1992)]. A limiting function is proposed which constrains the computed value of CDES, depending on the fineness of the grid and on the computed solution.

Keywords
navier stokes equations, eddies, large eddy simulations, viscosity, eddy viscosity closure

Disciplines
Aerodynamics and Fluid Mechanics | Other Aerospace Engineering | Structures and Materials

Comments
The following article appeared in Physics of Fluids 27, 025105 (2015); 1 and may be found at doi:10.1063/1.4907746.

Rights
Copyright 2015 American Institute of Physics. This article may be downloaded for personal use only. Any other use requires prior permission of the author and the American Institute of Physics.
On the dynamic computation of the model constant in delayed detached eddy simulation

Z. Yin, K. R. Reddy, and P. A. Durbin

Citation: Physics of Fluids 27, 025105 (2015); doi: 10.1063/1.4907746
View online: http://dx.doi.org/10.1063/1.4907746
View Table of Contents: http://scitation.aip.org/content/aip/journal/pof2/27/2?ver=pdfcov
Published by the AIP Publishing

Articles you may be interested in

Dynamic global model for large eddy simulation of transient flow
Phys. Fluids 22, 075106 (2010); 10.1063/1.3459156

Vector level identity for dynamic subgrid scale modeling in large eddy simulation
Phys. Fluids 14, 3616 (2002); 10.1063/1.1504450

Dynamic testing of subgrid models in large eddy simulation based on the Germano identity
Phys. Fluids 11, 245 (1999); 10.1063/1.869873

A dynamic similarity model for large eddy simulation of turbulent combustion
Phys. Fluids 10, 1775 (1998); 10.1063/1.869696

Large eddy simulation of turbulent front propagation with dynamic subgrid models
Phys. Fluids 9, 3826 (1997); 10.1063/1.869517
On the dynamic computation of the model constant in delayed detached eddy simulation

Z. Yin, K. R. Reddy, a) and P. A. Durbin
Department of Aerospace Engineering, Iowa State University, Ames, Iowa 50011, USA

(Received 20 November 2014; accepted 27 January 2015; published online 10 February 2015)

The current work puts forth an implementation of a dynamic procedure to locally compute the value of the model constant $C_{DES}$, as used in the eddy simulation branch of Delayed Detached Eddy Simulation (DDES). Former DDES formulations [P. R. Spalart et al., “A new version of detached-eddy simulation, resistant to ambiguous grid densities,” Theor. Comput. Fluid Dyn. 20, 181 (2006); M. S. Gritskevich et al., “Development of DDES and IDDES formulations for the $k - \omega$ shear stress transport model,” Flow, Turbul. Combust. 88, 431 (2012)] are not conducive to the implementation of a dynamic procedure due to uncertainty as to what form the eddy viscosity expression takes in the eddy simulation branch. However, a recent, alternate formulation [K. R. Reddy et al., “A DDES model with a Smagorinsky-type eddy viscosity formulation and log-layer mismatch correction,” Int. J. Heat Fluid Flow 50, 103 (2014)] casts the eddy viscosity in a form that is similar to the Smagorinsky, LES (Large Eddy Simulation) sub-grid viscosity. The resemblance to the Smagorinsky model allows the implementation of a dynamic procedure similar to that of Lilly [D. K. Lilly, “A proposed modification of the Germano subgrid-scale closure method,” Phys. Fluids A 4, 633 (1992)]. A limiting function is proposed which constrains the computed value of $C_{DES}$, depending on the fineness of the grid and on the computed solution. © 2015 AIP Publishing LLC.

I. INTRODUCTION

Detached eddy simulation (DES) was put forth as a method to couple Reynolds averaged (RANS) models and eddy resolving simulation. It is an idea for using a single turbulence model in both the RANS and the eddy simulation branches. Some fundamental issues were identified with the original formulation, such as modeled stress depletion, and log-layer mismatch. This led to modifications such as delayed DES (DDES) and Improved DDES (IDDES). These have led to an operational methodology. The successes to date argue for further advances.

A natural desire would be to employ a dynamic model on the eddy simulation branch, analogous to the dynamic Smagorinsky model (DSM). To some degree, this was explored previously by using 2 different models—the Spalart-Allmaras RANS model and DSM—and interpolating between them. Yet another method is the use of a hybrid-filter, which leads to a set of filtered Navier-Stokes equations with additional terms. However, these are quite different from the present approach. DES utilizes a single turbulence model throughout the whole domain. We retain that feature. In most formulations, it is not obvious how a dynamic procedure can be implemented—the primary reason being uncertainty about the form of the eddy viscosity on the eddy simulation branch. This difficulty with DES models has been pointed out previously.

The uncertainty arises because the original DES models were based on enhancing dissipation, using the grid spacing as the dissipation length when it became smaller than the RANS length scale.

---

a)Author to whom correspondence should be addressed. Electronic mail: kreddy@iastate.edu

1070-6631/2015/27(2)/025105/15/$30.00 27, 025105-1 ©2015 AIP Publishing LLC

This document is copyrighted as indicated in the article. Reuse of AIP content is subject to the terms at: http://scitation.aip.org/termsconditions. Downloaded to IP: 129.186.176.188 On: Tue, 08 Dec 2015 16:01:49
The same approach of enhancing dissipation was followed when DDES was adapted to the \( k - \omega \) SST (Shear Stress Transport) RANS model\(^{11}\) (\( k \) is the turbulent kinetic energy, and \( \omega \) is the specific dissipation rate). Here again, it is not clear what the functional form of the eddy viscosity is in terms of the DDES/IDDES length scale.

We recently put forth an alternate formulation of DDES\(^{12}\) based on the \( k - \omega \) (or \( k - \omega \) SST) RANS model, which uses the DDES length scale \( \ell_{DDES} \) to define the eddy viscosity as \( \nu_T = \ell_{DDES}^2 \omega \). It follows that the length scale limiter can be interpreted as limiting the production term, rather than enhancing the dissipation term. This alternate formulation bears a similarity to the Smagorinsky model. Thus, an \textit{a priori} estimate of the model constant \( C_{DES} \approx 0.12 \) was made from the Smagorinsky constant \( C_s \). However, when the model was calibrated by channel flow simulations, a range of values of about 0.05 \( \leq C_{DES} \leq 0.15 \) was found to be satisfactory.

It is known that the best value of the Smagorinsky constant \( C_s \) depends on the flow configuration.\(^{13}\) The dynamic procedure allows it to adapt to the flow, and to the particular grid. This suggests that the leeway in the calibration of \( C_{DES} \) can be exploited in the same way. Because the eddy viscosity is specified directly in this alternate formulation,\(^{12}\) the dynamic procedure is immediately apparent.

The model formulation will be described in Sec. II. The open source code OpenFOAM\(^{14}\) was used for all the present computer simulations. Gaussian finite volume integration with central differencing for interpolation was selected for spatial discretization of equations. Time integration was by the 2nd order, backward difference method. The resulting matrix system was solved using the Pre-conditioned Bi-conjugate gradient algorithm, with the simplified, diagonal-based, incomplete-LU (Lower Upper) preconditioner. Solution for the matrix system at each time step was obtained by solving iteratively, to a specified tolerance of the residual norm.

\section*{II. MODEL FORMULATION}

The alternate DDES formulation\(^{12}\) is reproduced here for convenience,

\[ \ell_{DDES} = \ell_{RANS} - f_d \max(0, \ell_{RANS} - \ell_{LES}), \]
\[ \ell_{RANS} = \frac{\sqrt{k}}{\omega}, \]
\[ \ell_{LES} = C_{DES} \Delta, \]
\[ \Delta = f_d \nu^{1/3} + (1 - f_d) h_{max}, \quad C_{DES} = 0.12, \quad \nu_T = \ell_{DDES}^2 \omega, \]

where \( V \) is the cell volume, \( h_{max} = \max(dx, dy, dz) \) is the maximum cell spacing, and \( f_d \) is the DDES shielding function,

\[ f_d = 1 - \tanh([8r_d]^3), \]
\[ r_d = \frac{k/\omega + \nu}{k^2d_w^2\sqrt{U_{i,j}U_{i,j}}}, \]

where \( \nu \) is the kinematic viscosity, \( \kappa \) the Von Kármán constant, \( d_w \) the wall distance, and \( U_{i,j} \) the velocity gradient tensor.

Note, especially, that \( \nu_T = \ell_{DDES}^2 \omega \). This \( \nu_T \) defines the production term of the \( k \) equation in the \( k - \omega \) RANS model,\(^{15}\) leaving all the other terms unaltered.

\[ \frac{Dk}{Dt} = 2\nu_T |S|^2 - C_{\mu} k \omega + \nabla \cdot [(\nu + \sigma_k(k/\omega))\nabla k], \]
\[ \frac{D\omega}{Dt} = 2C_{\omega 1}|S|^2 - C_{\omega 2} \omega^2 + \nabla \cdot [(\nu + \sigma_\omega(\omega/\omega))\nabla \omega]. \]

The standard constants are invoked,

\[ C_{\mu} = 9/100, \quad \sigma_k = 1/2, \quad \sigma_\omega = 1/2, \quad C_{\omega 1} = 5/9, \quad C_{\omega 2} = 3/40. \]

For future reference, we will cite this formulation\(^{12}\) as “Model 1.”

Thus, on the eddy simulation branch (\( f_d = 1, \ell_{LES} < \ell_{RANS} \)), we have

\[ \nu_T = (C_{DES} \Delta)^2 \omega, \]
which is similar to the Smagorinsky sub-grid viscosity expression,

\[ \nu_{SGS} = (C_s \Delta)^2 |S|. \]  

(5)

In LES, the dynamic procedure evaluates a local value of \( C_s \) as follows:

\[ C_s^2 = 0.5 \frac{L_{ij} M_{ij}}{M_{ij} M_{ij}}, \]  

(6)

\[ L_{ij} = -\hat{u}_i \hat{u}_j + \hat{u}_i \hat{u}_j, \]  

(7)

\[ M_{ij} = (\hat{\Delta}^2 \hat{S}_{ij} \hat{S}_{ij} - \Delta^2 \bar{S}_{ij} \bar{S}_{ij}). \]  

(8)

The notations used in Eqs. (7) and (8) are the same as in Lilly.\(^7\) The hat denotes explicit, test filtering where the test filter width is twice the grid scale. The test filtering is carried out via a spatial average of the face neighbour cells weighted by the surface area of the common face.

It is rather apparent that for the eddy viscosity definition in (4), this same dynamic procedure gives

\[ C_{DES}^2 = 0.5 \frac{L_{ij} M_{ij}}{M_{ij} M_{ij}}, \]  

(9)

\[ M_{ij} = (\hat{\Delta}^2 \hat{\omega} S_{ij} - \Delta^2 \bar{\omega} \bar{S}_{ij}). \]  

(10)

Essentially, \( \omega \) plays the role of the filtered rate of strain \( |S| \). So the only change occurs in the definition of \( M_{ij} \) (Eq. (10)) due to the difference in the eddy viscosity definition. In the first of Eq. (1), \( C_{DES} \) determines the switch from the RANS to LES length scales. By submitting this coefficient to the dynamic procedure, the switching criterion becomes adaptive.

The dynamic procedure can yield locally negative values of \( C_{DES}^2 \), which is not acceptable—this problem already exists in LES. It is resolved by clipping the right side of (9) at 0.

Indeed, there is yet another issue related to the mesh resolution. In order for the test filter to be valid, a significant portion of the inertial range needs to be resolved. But the coarse meshes that sometimes are used in DES do not capture enough of the small scales. Figure 1 highlights this, where the power spectral density (PSD) of the streamwise velocity component \( u \) obtained in the simulation of a backward facing step is shown. The coarse mesh results in rather little inertial range and a rapid falloff at high frequency. Then, formula (9) yields spuriously low values of \( C_{DES} \). In such circumstances, avoiding the dynamic procedure altogether might be best. For anything but these
very coarse meshes, there is a good prospect for dynamic DES. Indeed, if the mesh resolution is close to that of wall resolved LES, utilizing the dynamic procedure might be favorable, even in the near-wall region.

For DES, there is an additional issue related to the near-wall RANS region. Based on the model formulation described thus far, it would seem that the extent of the RANS region would remain unaffected since the shielding function \( f_d \) would make the model to follow RANS behaviour. However, \( f_d \) is a function of \( k \) (via Eq. (2)), which in turn depends on \( C_{DES} \) (due to its appearance in the production term of the \( k \) equation).

This is highlighted in Figure 2(a) which shows \( f_d \) profiles obtained from 2 simulations of channel flow using Model 1, with different values of \( C_{DES} \). We observe that the extent of the shielded region reduces when \( C_{DES} \) is reduced, which stems from the reduced production of \( k \). This means that on a coarse mesh, the spuriously low values of \( C_{DES} \) returned by formula (9) would lead to a drastic reduction in the extent of the RANS region, leading to incorrect predictions of near-wall properties such as the wall shear stress, and subsequently, the mean velocity. This behaviour is highlighted in Figure 2(b), which shows profiles of \( f_d \) and \( U^+ \) obtained in a channel flow simulation using the dynamically evaluated constant \( C_{DES} \) (from Eq. (9)). Negative values for \( C_{DES}^2 \) were clipped to zero.

The mesh used here has a non-dimensional cell spacing of \( \Delta x^+ = 400 \) and \( \Delta z^+ = 200 \) with \( \Delta y^+ < 1 \) at the wall. For the same grid and flow conditions, Model 1 was able to produce a good estimate for the mean velocity profile. Hence, it is quite clear that using the dynamic procedure on coarse meshes can actually prove to be detrimental.

To address these caveats, we introduce a limiting function which acts as a bound on the computed value of \( C_{DES} \). It is described as follows:

\[
C_{DES} = \max(C_{lim}, C_{dyn}),
\]

\[
C_{dyn}^2 = \max\left(0, 0.5 \frac{L_{ij} M_{ij}}{M_{ij} M_{ij}}\right),
\]

\[
C_{lim} = C_{DES}^0 \left[1 - \tanh\left(\alpha \exp\left(-\frac{\beta h_{max}}{L_k}\right)\right)\right],
\]

\[
C_{DES}^0 = 0.12, \quad L_k = \left(\frac{y^+}{\epsilon}\right)^{1/4}, \quad \alpha = 25, \quad \beta = 0.05.
\]

\[
\epsilon = 2(C_{DES}^0 h_{max})^2 \omega |S|^2 + C_\mu k\omega.
\]

Equation (12) is the same as Eq. (9), except that it is now clipped at 0, avoiding negative values for \( C_{dyn}^2 \). The right side of Eq. (9) is averaged over the face neighbor cells, weighted by the surface area of the common face, before it is clipped. No other averaging, such as along homogeneous
directions, or Lagrangian dynamic averaging\textsuperscript{16} is performed. As will be shown, the results obtained using such an approach yield satisfactory results, although it is possible that the incorporation of some form of averaging might lead to additional robustness.

The idea behind Eq. (13) is to gauge the mesh resolution\textsuperscript{17} and subsequently, its suitability for invoking the dynamic procedure. The constants $\alpha$ and $\beta$ were calibrated via channel flow simulations with various mesh resolutions.

The right side of Eq. (14) represents the contribution to the total turbulent kinetic energy dissipation of the sub-grid and the modeled component to $\epsilon$. $L_k$ is representative of the Kolmogorov length scale. If $h_{\text{max}}$ represents the size of the smallest eddies being resolved, then $h_{\text{max}}/L_k \to 0$ represents a mesh resolution where a large portion of the inertial range has been resolved, and $h_{\text{max}}/L_k \to \infty$ represents a coarse mesh where using a constant $C_{\text{DES}}$ might be more suitable. That constant value has been set to 0.12. Equation (13) interpolates between $C_{\text{lim}} = 0$ and $C_{\text{lim}} = 0.12$.

Figure 3 reflects this idea, where for a coarse mesh, $C_{\text{DES}} = C_{\text{lim}}$ and the model and the dynamic procedure cannot produce low values. For the other extreme, where the mesh is fine enough to run LES even in the near-wall regions, the dynamic procedure would be utilized almost everywhere.

As pointed out in the Model 1 formulation,\textsuperscript{12} away from the wall, the average values of $\omega^2$ and $|S|^2$ are proportional. In the near-wall region $\omega$ increases more rapidly than $|S|$ as $y \to 0$, because of its boundary condition, leading to large $\epsilon$. Hence, there will be a thin RANS region even for a wall-resolved, LES mesh, although the extent of the RANS region can be much smaller than that would be obtained with the native Model 1, or any other DDES formulation. Thus, the limiting function takes advantage of the fineness of the mesh, by not imposing a mandatory, large near-wall RANS region. This behavior will be highlighted for some test cases.

The $C_{\text{DES}}$ value obtained from Eq. (11) is used to evaluate $\ell_{\text{LES}}$ in Eq. (1), and subsequently, $\nu_T$ and the turbulent kinetic energy production. This completes the new dynamic DDES model formulation. The new model with the limiting function described above will be referred to as “Model 2” in the remaining portions of this article.

A comment needs to be made regarding the choice for the form of Eq. (14). The $\epsilon$ estimate is based on $C_{\text{DES}}^0$ and $h_{\text{max}}$, rather than $\nu_T$ directly. This yields a conservative estimate, wherein a slightly larger $\epsilon$ is obtained, leading to a smaller value of $L_k$. That provides a more stringent requirement on the mesh resolution needed to achieve $h_{\text{max}}/L_k \to 0$. It acts as a safeguard against invoking the dynamic procedure on relatively coarse meshes.

III. TEST CASES

A. Channel flow

Several channel flow simulations were carried out for a range of Reynolds numbers. All the channel flow cases were simulated using Model 2 and the results obtained are compared with DSM or $k-\omega$ RANS. For simulations with sufficient grid resolution, we expect a large portion of the domain to utilize the dynamic procedure. The grid and the extent of the computational domain are the same as in Reddy \textit{et al.} (2014).\textsuperscript{12} The corresponding grid resolution in wall units for each Reynolds number is listed in Table I. In all the cases, $\Delta y^+ < 1$ for the near-wall cells. The time step $\Delta t$ is chosen to ensure that the maximum local Courant-Friedricks-Lewey (CFL) number is approximately 0.5.

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{fig3.png}
\caption{Variation of $C_{\text{lim}}$ with $h_{\text{max}}/L_k$.}
\end{figure}
TABLE I. Grid resolution for channel flow cases with different Reynolds numbers.

<table>
<thead>
<tr>
<th>$Re_{\tau}$</th>
<th>$\Delta x^+$</th>
<th>$\Delta z^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>50</td>
<td>25</td>
</tr>
<tr>
<td>1200</td>
<td>120</td>
<td>60</td>
</tr>
<tr>
<td>2000</td>
<td>200</td>
<td>100</td>
</tr>
<tr>
<td>6000</td>
<td>600</td>
<td>300</td>
</tr>
</tbody>
</table>

Figure 4 shows the non-dimensionalized velocity profiles obtained for different values of $Re_{\tau}$. The results show good agreement between the dynamic DDES model (Model 2) and DSM/RANS. The limiting value for $C_{DES}$ reduces to 0 for the lower $Re_{\tau}$ cases (when the mesh in the eddying region is fine) and retains a larger value for the higher $Re_{\tau}$ cases (when the mesh is coarse). For $Re_{\tau} = 500$, the limiting function takes advantage of the mesh and allows the dynamic procedure to be utilized in the near wall region, with the entire log-layer located in the eddy simulation region. However, as pointed out in Sec. II, we still have a thin RANS region close to the wall, due to $\omega$ growing more rapidly than $|S|$ as $y \to 0$. The large $\omega$ results in a large $\epsilon$, which activates the limiting function, and the RANS branch replaces the eddy simulation branch.

The difference between the performance of Model 2 and Model 1 is highlighted in Figure 5. Model 1 and Model 2 data correspond to a channel flow simulation with $Re_{\tau} = 500$, while the DNS data correspond to $Re_{\tau} = 590$. Profiles of resolved $u'\tau$, $v'\tau$, and $w'\tau$ are shown in Figure 5(a). The trend observed in the Model 1 predictions for this $Re_{\tau} = 500$ case is similar to that observed...
for $Re_\tau = 2250$. This is primarily due to the presence of a significant RANS region for Model 1 as shown in Figure 5(b), where the shielding function $f_d$ is shown, along with $k^+$—the non-dimensional total turbulent kinetic energy.

$$k^+ = \frac{(k_m + k_r)}{u_\tau^2},$$

$$k_m = \text{modeled component of } k,$$

$$k_r = 0.5(u'^2 + v'^2 + w'^2) = \text{resolved component}.$$  

Notice that the extent of the RANS region is similar for Model 1 with $Re_\tau = 500$ and $Re_\tau = 2250$, despite the fine mesh for the lower $Re_\tau$. Model 2 however was able to “detect” that the mesh has sufficient resolution to employ the dynamic procedure. This leads to lower $C_{DES}$ and subsequently, lower $k$ and $\ell_{LES}$ values, resulting in a smaller shielded region. Thus, the eddy simulation branch is active over a larger region, which gives a better prediction of the velocity fluctuations and the turbulent kinetic energy.

B. Backward facing step

The flow over a backward facing step is an excellent case to test the performance of any hybrid RANS/LES method due to the abrupt change in flow features across the sharp edge. The model must be capable of switching from RANS to eddy simulation at the step, where the flow separates. The experimental setup of Vogel and Eaton$^{19}$ was simulated. The Reynolds number at the inflow boundary is 28,000 based on the bulk velocity $U_b$ and the step height $H$. Simulation details such as the grid used, the boundary conditions specified and the extent of the computational domain are the same as in Reddy et al.$^{12}$ Overall, a good agreement between the simulation and the experimental data is observed. Figure 6 shows the normalized mean streamwise velocity profiles and rms profiles at several streamwise locations, and the variation of the skin friction co-efficient $C_f$ along the bottom wall. The $C_f$ is computed from the wall shear stress obtained using a first order interpolation. The near-wall cells have $\Delta y^+ < 1$. Since the velocity varies linearly with the wall distance within the viscous sublayer ($y^+ \lesssim 5$), a first order interpolation is sufficient to accurately calculate the velocity gradient, and subsequently, the shear stress at the wall.

The grid used is relatively coarse ($\Delta x^+ \approx 200$ and $\Delta z^+ \approx 100$ away from the step), so we expect the limiting function to impose lower bounds on $C_{DES}$. Figure 7 shows contours of time-averaged $C_{lim}$. We observe that almost throughout the entire eddying region, $C_{lim} > 0.06 \Rightarrow C_{DES} > 0.06$.

$C_{DES}$ hits the limiter at 0.12 where the flow separates from the step. Due to wall resolution requirements, the cell at the separation corner has very large aspect ratio, which deviates from
FIG. 6. Flow over backward-facing step: Comparison with experimental data. (a) Normalized $\bar{U}$ profiles, (b) normalized $u_{rms}$ profiles, (c) post-step $C_f \times 1000$ distribution along the bottom wall. Profiles taken at $x/H = 2.2, 3.3, 3.7, 4.5, 5.2, 5.9, 6.7, 7.4$, and $8.9$. Solid lines—Model 2 results, Symbols—Experimental data.\textsuperscript{19}

typical LES grid resolution. Also, the rate of strain is large, which means that dissipation is high. As a result, the values of $L_k$ are relatively low, causing the bound on the value of $C_{DES}$ to be invoked.

C. Periodic hills

This case shows flow separation from a smooth surface, unlike the backward-facing step. The geometry and flow conditions are as described in Fröhlich et al.\textsuperscript{20} The extent of the computational domain is $9H$ and $4.5H$ along the streamwise and spanwise directions, respectively, where $H$ is the hill height at the crest. The Reynolds number based on the hill height and the bulk velocity at the crest is 10,595. The grid used has $106 \times 100 \times 90$ points in the streamwise, wall normal, and spanwise directions. Periodic boundary conditions are enforced along the streamwise and spanwise directions. The flow is driven by a pressure gradient source term which is adjusted to sustain the required bulk velocity at the inflow boundary. A maximum local CFL number $< 0.5$ is maintained throughout the entire domain.

Figure 8 compares the skin friction distribution along the bottom wall, mean streamwise velocity profiles, and rms profiles from Model 2 to LES data.\textsuperscript{20} Overall, there is a good agreement. Additionally, Figure 8(a) also shows the $C_f$ prediction obtained from Model 1. We notice that Model 1 predicts a larger $C_f$ than LES data near the inlet ($x/H = 0$), compared to the more accurate prediction of Model 2. The mean and rms velocity predictions of Model 1 are, however, very similar.

FIG. 7. Time averaged $C_{lim}$ contours.
FIG. 8. Flow over 2D periodic hills: (a) Variation of the skin-friction coefficient along the bottom wall, (b) normalized mean velocity profiles, (c) normalized $u_{rms}$ profiles. Profiles taken at $x/H = 0.05, 2, 6, \text{and} 8$.

to those of Model 2 for the current grid, and hence, those profiles have not been shown in order to avoid clutter.

D. 3D diffuser

As an example of a 3D geometry, the flow through a 3D diffuser was simulated. The geometry and flow conditions correspond to the “diffuser 1” of Cherry et al.\textsuperscript{21} The grid and boundary conditions are the same as in Jeyapaul.\textsuperscript{22} The grid is nearly LES-quality. Three simulations were carried out for this geometry, each corresponding to a different turbulence model—the $k - \omega$ RANS model,\textsuperscript{15} Model 1,\textsuperscript{12} and Model 2 (the current dynamic DDES model).

Figure 9 shows contours of the time-averaged streamwise velocity component obtained from all three simulations at the diffuser exit ($x/H = 15$, where $H$ is the height of the inlet section). The RANS result (Figure 9, top left) is qualitatively incorrect since it predicts separation along the side wall, as opposed to experiments\textsuperscript{21} and DNS\textsuperscript{23} where separation is along the top wall. Model 1 does predict separation along the top wall (Figure 9, top right)—an improvement over RANS—but, the separation region is much thinner than the DNS data. Figure 10 compares the separation contours and mean velocity profiles (at $x/H = 0, 2, 6, 8, 12, 14, 15.5,$ and 17) along the midplane obtained for Model 1 with DNS data,\textsuperscript{23} showing the deviation of Model 1 predictions from DNS.

Introducing the dynamic procedure improves the results appreciably. The bottom portion of Figure 9 shows the mean velocity contours obtained using Model 2, and the corresponding separation contours and mean velocity profiles along the midplane are shown in Figure 11. The agreement with DNS data is much better than with Model 1. The dynamic DDES model was able to take advantage of the grid resolution, utilizing the dynamic procedure almost everywhere in the domain, leading to a marked improvement in the prediction.

E. Rotating channel

The flow through a fully developed rotating turbulent channel was simulated as another illustration of the advantage of the dynamic procedure over a constant $C_{DES}$. In pure RANS mode, $k - \omega$ would require some kind of curvature correction to handle rotating flows.\textsuperscript{24} No such corrections are used here. This means that simulations based on Model 1 would likely be subject to errors due to the
presence of a thick RANS region near the walls. In the eddy-simulation region, rotation effects are captured by the Navier-Stokes equations. Thus, we expect to get better results using Model 2 since the RANS region will be smaller, provided the mesh is fine enough.

The non-dimensional measure of rotation is the rotation number, \( \text{Ro} = 2\Omega \delta / U_b \), where \( U_b \) is the bulk velocity, \( \delta \) the channel half-width, and \( \Omega \) the rate of coordinate system rotation. Four different simulations were carried out, corresponding to four different Ro values. These simulations correspond to previous DNS studies of Grundestam et al. \( (\text{Ro} = 0.98, 1.5) \) and Kristoffersen and Andersson \( (\text{Ro} = 0.1, 0.5) \).

In the DNS studies, a constant pressure gradient was prescribed, which forces constant total \( u_r \) and \( Re_r \) values. The bulk velocity, \( U_b \) and \( Re_b \) (Reynolds number based on the bulk velocity) then vary with \( \text{Ro} \). In our simulations, \( U_b \) was specified, for each \( \text{Ro} \), and the resulting \( u_r \) and \( Re_r \) values were computed.
Figure 11 shows mean velocity profiles obtained with both Model 1 and Model 2, compared with DNS data. Model 2 results are more in line with the data, especially near the right wall, at higher $Ro$, where the turbulence is suppressed by rotation.

Due to the asymmetry in the velocity profile, there are 2 different friction velocities, $u_{\tau u}$ and $u_{\tau s}$, corresponding to the unstable and stable sides. An average friction velocity $u_{\tau}$ is defined as

$$ u_{\tau} = \left[0.5(u_{\tau u}^2 + u_{\tau s}^2)\right]^{1/2}. $$

For the specified bulk velocity $U_b$, the predicted $Re_{\tau}$ values for Model 1 and Model 2 are shown in Table II, along with the reference DNS values. Model 2 predicts more accurate values for the wall shear stress than Model 1. The grid used for these cases has a non-dimensional cell spacing $\Delta x^+ = \Delta z^+ \approx 30$ for Model 2 (the corresponding numbers evaluated when using Model 1 $\approx 50$ due to the larger predicted $u_{\tau}$), with $\Delta y^+ < 1$ for the near wall cells in all the simulations. This leads to
TABLE II. Predicted $Re_T$ for different $Ro$ values.

<table>
<thead>
<tr>
<th>$Ro$</th>
<th>$Re_T$</th>
<th>Model 1</th>
<th>Model 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>194</td>
<td>229</td>
<td>196</td>
</tr>
<tr>
<td>0.5</td>
<td>194</td>
<td>206</td>
<td>199</td>
</tr>
<tr>
<td>0.98</td>
<td>180</td>
<td>215</td>
<td>179</td>
</tr>
<tr>
<td>1.5</td>
<td>180</td>
<td>330</td>
<td>187</td>
</tr>
</tbody>
</table>

A smaller RANS region while using Model 2, and subsequently, a smaller error stemming from the absence of any curvature correction terms.

At large $Ro$, we observe that Model 2 starts to deviate from the DNS results, especially on the right wall (Figure 12(d)). That is the wall where rotation is stabilizing. A likely explanation for the discrepancy is that the RANS model does not include a curvature correction. Hence, as long as there is a thin RANS region, it cannot laminarize. Regions of negative production were observed for $Ro = 1.5$, and that certainly cannot be captured by the $k - \omega$ eddy viscosity model. For lower $Ro$ values, the predictions are in good agreement with DNS.

F. Fundamental Aero Investigates The Hill (FAITH) geometry

As an illustration of the model performance for a complex flow configuration, a simulation of the flow over a 3D axisymmetric hill was carried out. The geometry is the FAITH. The variation of the hill height $h$ with the radius $r$ is

$$h = 3 \cos \left( \frac{\pi r}{9} \right) + 3, \quad 0 \leq r \leq 9,$$

where $r$ and $h$ are in inches. The total radius of the hill is $R = 9'$, with the hill height at the centroid $H = 6'$. The Reynolds number based on $H$ is $Re_H = 500,000$, with a mean inflow velocity $U_\infty = 50.3$ m/s. More details regarding the experimental setup, and available data can be found in Bell et al. and Husen et al.

The extent of the computational domain used is $20H \times 5.3H \times 8H$ along the streamwise, wall normal and spanwise directions. The hill is centered at $x/H = z/H = 0$. These dimensions correspond to the wind tunnel test section used in the experiments. A plug flow is specified at the inflow and the boundary layer develops along the streamwise direction. The length of the inlet section ensures that the required boundary layer thickness is obtained at $x/H = 0$ in the absence of the hill. The grid used has $\approx 3$ million cells. At the hill, $130 \times 130$ cells are distributed uniformly along the streamwise and spanwise directions along its diameter, with the cell spacing stretched out towards the inflow and outflow boundaries, and along the remaining spanwise portions. The maximum value of the local CFL number $\approx 0.5$.

Figure 13 shows simulation results obtained using Model 2. Figure 13(a) shows contours of the magnitude of skin friction coefficient $C_f$ over a square region around the hill (the circular edge of the hill is the incircle of the square) and is in good agreement with experimental data. Normalized time-averaged streamwise velocity components are compared with experimental data in Figure 13(b).

Figure 14 shows contours of $\overline{U}$, $k$, $u_{rms}$, and $u'v'$ along the spanwise centerplane on the lee side of the hill. Here, $k$ represents the total turbulent kinetic energy, which is the sum of the modeled and resolved components ($k_m + k_r$). Overall, the trends observed in the PIV (Particle Image Velocimetry) data are captured by the simulation. However, the peak values of $k$ and $u_{rms}$ are slightly overestimated.

One possible explanation for this would be the coarseness of the mesh used—$\Delta x^+ = \Delta z^+$ is large (as high as 1000 in some regions, depending on the local friction velocity $u_\tau$). The fact that the mesh is coarse can also be inferred from Figure 15(d) which shows that $C_{DES} = C_{0DES} = 0.12$ over the entire region behind the hill, where we observe most of the relevant unsteady phenomena.
FIG. 13. (a) Contours of magnitude of skin friction coefficient, (b) mean streamwise velocity profiles behind the hill at $x/H = 0, 0.4, 0.8, 1.2, 1.6, \text{ and } 2$.

FIG. 14. Contours of (a) mean streamwise velocity $\overline{U}$, (b) total turbulent kinetic energy $k$, (c) $u_{rms}$, and (d) $u'v'$ in the $z/H = 0$ plane behind the hill.

FIG. 15. Contours of (a) $k_m$, (b) $k_r$, (c) $f_d$, and (d) $C_{DES}$ in the $z/H = 0$ plane behind the hill.
Hence Model 2 essentially functions as Model 1 for simulations involving very coarse meshes. Figure 15(c) shows the extent of the RANS region ($f_d = 0$), and from Figure 15(a), we can observe that the magnitude of the modeled turbulent kinetic energy $k_m$ in the LES region is comparable to that in the RANS region. This is another indication that the mesh being used is coarse. Better agreement with experimental data could likely be achieved by increasing the mesh resolution such that the dynamic procedure is employed in the eddy simulation regions.

IV. CONCLUSION

The previously proposed, DDES formulation\(^{12}\) opened the possibility to develop a dynamic DDES formulation. The model constant $C_{DES}$ is computed locally via a well-established procedure. This requires a test filter that captures the small scales. Coarse grids are sometimes used for DES, and these small scales are not present. A limiting function was introduced in order to estimate the validity of utilizing the dynamic procedure on the given mesh. The function compares grid spacing to a Kolmogorov scale. Based on this, $C_{DES}$ becomes a default value if the dynamic procedure is likely to fail. Simulations showed improved predictions when employing the dynamic procedure, rather than using a constant $C_{DES}$. That was especially true when simulations were carried out on LES-quality meshes.

The dynamic procedure yields superior performance over the constant coefficient model for 2 reasons. The first reason is similar to the case of LES: the coefficient adapts to how well the turbulence is resolved; if it is well resolved, $C_{DES}$ becomes very small. The second reason is peculiar to detached eddy simulation: using a locally computed $C_{DES}$ in $\ell_{LES}$ causes the RANS region to become thinner when the mesh is fine. By maximizing the size of the eddy simulation region, the dynamic DDES model is able to reduce any drawbacks in the RANS model (such as the absence of curvature corrections while simulating rotating turbulent channel flow).

A key observation is how obvious it was to implement a dynamic procedure into our alternate DDES formulation.\(^{12}\) That is because it was designed to be similar to the Smagorinsky model. It is likely that other improvements/modifications made to the original Smagorinsky formulation can also be implemented. This could lead to additional robustness of this DES formulation, capable of handling a wide range of flow configurations.

ACKNOWLEDGMENTS

Computing resources were provided by Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation Grant No. OCI-1053575. Funding was provided by NASA Grant No. NNX12AJ74A and by Pratt and Whitney.

15 D. C. Wilcox, Turbulence Modeling for CFD (DCW Industries, 1993).