ASSESSMENT OF HYBRID TURBULENCE MODELS FOR THE SIMULATION OF RIBBED CHANNEL FLOWS WITH HEAT TRANSFER

W. Sadowski, F. Lo Presti, F. di Mare

Chair of Thermal Turbomachines and Aeroengines, Ruhr University Bochum
wojciech.sadowski@rub.de

ABSTRACT
Manufacturing periodic ribs in cooling channels is a widely used technique to enhance turbulent heat transfer. Currently, the improvement of such systems is a relevant topic of industrial research and numerical simulations, including heat transfer, are of great importance for their development. Hybrid RANS/LES approaches are an important tool in this context, combining the advantages of the classical LES and RANS models. The determination of an adequate resolution, however, requires finding balance between the accuracy and computational cost, and must be targeted to the specific application. In the current work, an LES simulation of the flow through a ribbed duct including heat transfer is presented. The LES is used as a baseline reference for the study of the performance of two hybrid models, \( k-\omega\)-SST-SAS and \( k-\omega\)-SST-IDDES. The mesh resolution requirements for the use of each model are discussed and tested on a series of block-structured, hexahedral meshes. The accuracy of the models is judged based on the prediction of friction coefficient and Nusselt number. The results gathered in the study indicate that the IDDES model predicts flow field parameters and heat transfer well, while the SAS model remains inaccurate irrespectively of the mesh resolution. Additionally, a computation on an unstructured mesh, optimised w.r.t. cell count and computational effort, has been performed with the IDDES model, assessing the ability of the hybrid model to work on lower quality grids.

KEYWORDS
RIBBED CHANNEL, TURBULENCE MODELLING, HYBRID TURBULENCE MODELS

NOMENCLATURE

\( u \) velocity
\( T \) temperature
\( \rho \) density
\( p \) pressure
\( q \) heat flux through the heated wall
\( U_{\text{bulk}} \) bulk velocity
\( \nu \) viscosity
\( k_{\text{sgs}} \) subgrid-scale viscosity
\( C_f \) skin friction coefficient
\( \alpha \) heat transfer coefficient
\( W \) channel width
\( L \) channel length
\( e \) rib size
\( H \) channel height
\( Re_{\text{bulk}} \) bulk Reynolds number
\( Nu \) Nusselt number
\( M \) resolved turbulent kinetic energy
\( \eta \) Kolmogorov length scale
INTRODUCTION

The accurate design of cooling systems is of paramount importance for gas turbines. A balance must be sought between the desired reduction of blade metal temperature, manufacturing complexity and the detrimental effect of the cooling on the performance of the whole machine. Rib turbulated cooling is an established method of enhancing heat transfer in cooling channels of turbine blades. The ribs have the effect of breaking the boundary layer and creating separation and recirculation regions between the ribs, thus enhancing convective mixing. A wide number of parameters such as shape, orientation, aspect-ratio and arrangement of the ribs influence the flow field around them and the resulting heat transfer. The effects of the above, as well as the influence of rotation, have been studied over the years in a considerably high number of scientific works. An extensive review is provided by Han et al. (2015).

Despite the wide knowledge accumulated, the actual design and further improvement of turbulated cooling systems requires a deep understanding of the underlying physics, involving turbulent flows with heat transfer. The ideal tool for such investigations is scale-resolving Computational Fluid Dynamics simulations which, however, come at a very high computational cost. Performing a Large Eddy Simulation (LES) of a single cooling channel might be feasible nowadays, but it is still a very time- and resource-consuming activity. Moreover, in an industrial context, the possibility of performing a time-accurate numerical simulation of a whole machine, with its cooling system, would be desirable to fully comprehend interactions between different parts. Yet, this is hardly possible with both the spatial and temporal resolutions required for an LES. The industrial standards for the numerical simulation of turbomachinery flows are still Reynolds Averaged Navier-Stokes (RANS) simulations. While they are fairly accurate in the prediction of the velocity field, most RANS models, depending on the configuration, tend to over- or under-predict heat transfer phenomena in ribbed channels (Wright & Han 2006).

Ribbed channels also provide an opportunity to study the influence of surface roughness on turbulent flow, an important effect in flows over particle assemblies, rivers beds and in fixed-bed reactors. In this context, rough surfaces are usually also permeable, with both parameters having a combined influence on the flow properties. Scale-resolving simulations are perfect for the investigation of such flows as well, as they provide an ability to investigate turbulence at the scale of the roughness element (e.g. Kuwata & Suga 2016). In both contexts, hybrid RANS/LES models seem to be a reasonable compromise between the desired level of accuracy and the need for a moderate computational cost.

Numerous hybrid models have been developed over the years, with the basic idea of resolving a wider range of the energy spectrum in chosen regions of the flow, without the need for a prohibitive grid resolution. Detached Eddy Simulation (DES), pioneered by Spalart et al. (1997) and Scale Adaptive Simulation (SAS) by Egorov & Menter (2008) are the two most popular approaches. The first one combines RANS method in the near-wall region with subgrid-scale model in separated areas, while the latter relies on the additional sink term in the RANS equations to introduce unsteady fluctuations to the flow field.

For both RANS and LES, the numerical mesh is of prime importance for obtaining accurate prediction of flow parameters. Generating an adequate grid for RANS models is a relatively straightforward task governed by a set of easy-to-understand principles (appropriate wall spacing for the chosen model, resolution of gradients etc.) and a priori quality indicators. Additionally, mesh convergence of the simulation results is the most powerful a posteriori tool for the verification of the grid and simulation quality, remaining still relatively computationally inexpensive. For scale-resolving simulations, it is much more difficult to establish guidelines for mesh quality, as most of the modelling approaches are directly tied to the mesh spacing. There-
fore, mesh convergence is not possible in the case of LES (until the limit of Direct Numerical Simulation is reached) and only costly *a posteriori* methods are available to assess the quality of both the mesh and the simulation together. Conversely, for hybrid models, a certain amount of engineering intuition and experience is required as the mesh influences not only the model performance but also which part of the model is active.

In the context of the DES approach, for the RANS region standard meshing conditions for Reynolds Averaged models should be applied (Spalart & Streett 2001). What is difficult, is ensuring an LES-like grid in the so-called focus region (where the scale-resolving part of the model is active). This is crucial, as in this part of the domain, the model resembles standard LES-type closure (Spalart et al. 2006). Fortunately, the new versions of the model (Spalart et al. 2006, Gritskevich et al. 2012), make the gridding process far easier, owing to an improved treatment of the grey areas, i.e. where the transition between the two modelling approaches occurs, and of the classical pitfalls of the DES models: Grid Induced Separation and Modelled Stress Depletion. This version of the model is called Improved Delayed Detached Eddy Simulation or IDDES. The increase in accuracy is achieved by re-calibrating the model constants and changing the formulation of RANS/LES blending (Gritskevich et al. 2012).

In the case of the SAS model, the grid requirements are much less clear. The model itself is more akin to RANS model than its other hybrid counterparts, as it does not contain explicit dependence on mesh spacing (Spalart 2008). Egorov & Menter (2008) compare results obtained from standard URANS and SAS on the same grids. Even the name of the model seems to suggest, that it should be able to “adapt” to the chosen mesh resolution and provide an improvement versus URANS model.

Within this work, we test the performance of the hybrid models in the simulations of ribbed channels. The basis for the study is the channel with periodic ribs, described in the next section, simulated at increasing levels of grid resolution with both the $k$-$\omega$-SST and the hybrid models $k$-$\omega$-SST-IDDES and $k$-$\omega$-SST-SAS.

While computing resources remain limited, it is instructive to develop a guideline for creating an “optimal” grid, balancing the high fidelity and accuracy of a chosen model with the computation effort required to compute the results. Our second goal is therefore to assess, by comparison with classical, fully hexahedral, block-structured mesh, the loss of accuracy while using a type of mesh generated with the following criteria: low total cell count, local refinement based on expected results, obeying only strictly necessary quality requirements. The mesh has been generated with an automatic unstructured mesher developed for processing complex geometries, as would be in a typical use-case.

**CASE DESCRIPTION**

The investigated configuration is the turbulent flow in the ribbed channel visible in Fig. 1. The bulk Reynolds number is equal to $Re_{bulk} = U_{bulk}H/\nu = 4174$. The Reynolds number chosen for the study is slightly lower than that of the typical application (see, for example, the work by Han 1984). This is a deliberate choice, so that a number of simulations could be carried out to derive grid-sensitivity information. Nevertheless, the presented workflow is generalisable to turbulent ribbed channels at higher Reynolds numbers, as the flow field in these configurations presents the same features over a wide range of $Re$ (Ahn Joon & Lee 2022). The lower wall of the channel is heated with a constant heat flux assuming Prandtl number equal to $Pr = 0.71$. The domain is periodic in the spanwise and the streamwise directions. The channel width is given as $W = 1.6H$, which has been demonstrated in the DNS study by Marocco & Franco (2016) to be enough to capture all relevant scales of the turbulent flow.
First, a highly-resolved LES simulation has been conducted, providing reference results for testing the performance of the hybrid models, as well as the actual mesh resolution required for accurate SRS of the chosen configuration. Since hybrid turbulence models work effectively in two modes (URANS in regions where cell size is far greater than the smallest eddies and LES mode in regions of adequate resolution) it is also important to investigate the performance of classical URANS models. The results obtained with URANS approach can be considered as a baseline for defining a mesh for a hybrid model, i.e. the mesh chosen for the hybrid model should not result in a worse prediction than the URANS approach.

Mathematical model

Assuming constant thermophysical parameters, low-Mach conditions and streamwise periodicity, the fluid flow and heat transfer are governed by the following equations:

\[ \frac{\partial \vec{u}_i}{\partial x_j} = 0, \quad \text{(1)} \]

\[ \frac{\partial \vec{u}_i}{\partial t} + \frac{\partial \vec{u}_i \vec{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \hat{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu_{\text{eff}} \left( \frac{\partial \vec{u}_i}{\partial x_j} + \frac{\partial \vec{u}_j}{\partial x_i} \right) \right] + \Pi \delta_{i1}, \quad \text{(2)} \]

\[ \frac{\partial \hat{T}}{\partial t} + \frac{\partial \vec{u}_i \hat{T}}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \alpha_{\text{eff}} \frac{\partial \hat{T}}{\partial x_j} \right] - \vec{u}_i \gamma \delta_{i1}. \quad \text{(3)} \]

In the above \( \rho, \ u, \ T \) and \( p \) stand for fluids density, velocity, temperature and pressure, respectively. The present study employs both URANS and LES turbulence modelling approaches, hence the operator \( \cdot \) denotes either the time average or the spatial filtering. Similarly, effective viscosity \( \nu_{\text{eff}} \) and effective heat transfer coefficient \( \alpha_{\text{eff}} \) combine the fluids properties with the contribution from the chosen turbulence model (e.g. turbulent/subgrid-scale viscosity). The Prandtl number is \( \text{Pr} = 0.71 \) and the turbulent Prandtl number is \( \text{Pr}_t = 0.85 \). The accent \( \hat{\cdot} \) denotes the streamwise periodic component of the accented quantity.

The flow is driven by the source term \( \Pi \). Its value is adjusted to achieve the intended bulk velocity in the domain. The temperature field is decomposed into the periodic and linear parts (Patankar et al. 1978), with the latter resulting in the presence of a sink term \( \gamma \) in Equation (3).
given by
\[ \gamma = \frac{q(L - e)}{\rho c_p U_{\text{bulk}} H} \],

where \( H \) is the average height of the channel.

The LES simulation conducted in the study is performed with the WALE model (Nicoud & Ducros 1999). The URANS computation is conducted with the standard version of \( k-\omega \)-SST model as it remains one of the most popular choices among industry professionals and it is a base for all of the hybrid models investigated in this work: \( k-\omega \)-SST-SAS and \( k-\omega \)-SST-IDDES. However, it is important to note that Marocco & Franco (2016), conducting steady state RANS simulations, found out that the \( k-\varepsilon \) realisable model performs better in this configuration.

The computations are performed with OpenFOAM-v2012 package, with a modified version of incompressible, pressure-based \textit{pimpleFoam} solver including Equation 3. Time derivatives were treated with implicit second-order backward scheme. For LES, a second-order central differencing scheme was used for convective terms, whereas URANS employed a second-order upwind scheme. Following Menter (2015), for \( k-\omega \)-SST-SAS central scheme was blended with upwind for all convective terms. Finally, in the case of \( k-\omega \)-SST-IDDES, a specialised scheme developed by Spalart et al. (2012) was employed for velocity and temperature, blending second-order upwind in RANS regions with central difference in LES areas. Before starting the time-averaging, the simulations were run until the domain was free of initial transients. Statistics were gathered over a total time corresponding to 30 flowthrough times \( U_{\text{bulk}}/L \). To boost statistical convergence, additional spanwise averaging was performed.

**Mesh**

For the initial set of computations, investigating the performance of the different turbulence modelling strategies, fully hexahedral and orthogonal meshes have been employed. The number of cells was scaled by the parameter \( K \) (a higher value corresponds to more cells). The total number of cells for each mesh is listed in Table 1, while a sample mesh topology is shown in Fig. 2a. Hybrid and RANS model computations have been run with grids from \( K = 3 \) to \( K = 9 \). In the text and graph labels, these meshes will be simply denoted by the value of \( K \) (e.g. SAS/3 denotes computation with the SAS model on mesh with \( K = 3 \)).

The generated meshes should obey the requirements for scale-resolving simulations far from the wall (Menter 2015). Hence, great care has been taken to ensure that cells in the core of the

![Figure 2: Meshes used in the investigation.](image)
channel are nearly cubical, as anisotropic meshes should not be used with the hybrid models (Spalart & Streett 2001). In Table 1, cell size expansion ratios and the resulting $y^+$ are reported. For all of the meshes, expansion ratios are mostly limited to $\sim 1.4$, while Spalart & Streett (2001) recommend a value of 1.25 for the boundary layer mesh. The maximum $y^+$ is lower than 5, thus fulfilling the standard RANS guidelines and allowing for the use of low-Re boundary conditions for the turbulence models.

A first, rough estimation of the cell size required for an LES can be given as $\Delta = 25\eta$ (Celik et al. 2005). The Kolmogorov length scale $\eta$ is estimated from the bulk Reynolds number:

$$\frac{H}{\eta} \propto \text{Re}^{3/4}_{\text{bulk}} \approx 520 \rightarrow \eta \approx \frac{H}{520}. \quad (5)$$

Adopting a more conservative estimation, $\Delta = 10\eta$, the resulting cell size of $\Delta \approx 0.02H$ is the typical size of grid 5. However, in order to eliminate the numerical instabilities of the central scheme (especially in the temperature field), the LES computation had to be carried out on the mesh 9, which also allowed to satisfy the quality criteria that will be introduced together with the results in the next section.

As mentioned in the introduction, generating high-quality grids which fulfil the criteria of high mesh quality relevant for scale-resolving simulation (cubic hexahedral cells, smooth change of cell volume, low face non-orthogonality etc.) is not always feasible. Even if block-structured mesh can be created for complex geometry, it will usually result in a significantly higher cell count than unstructured mesh. Therefore, automatically generated tetrahedral or polyhedral meshes with prism boundary layer and local refinement in the regions of interest are the de-facto standard of industrial CFD nowadays. Hence, for the current study, the meshing package snappyHexMesh was employed to generate a more realistic mesh. It has been refined in areas where a high level of turbulence is expected while attempting to keep the total number of cells at a low level. The topology of the mesh is shown in Fig. 2b. Grid spacing, $y^+$ values, cell counts and mesh properties are reported in Tab. 1. The performance of the IDDES model has been evaluated again on this mesh while comparing the results to the ones obtained on the hexahedral meshes.

### RESULTS AND DISCUSSION

One of the most important factors for the accuracy of the LES is ensuring a proper resolution of the boundary layer. According to Chapman (1979) the $x^+ < 100$, $y^+ < 2$, $z^+ < 20$ requirements need to be fulfilled to obtain an appropriate representation of structures inside the boundary layer. The plots of all of those qualities for computed LES are visible in Fig. 3. The
conditions are met everywhere, apart from the 0.7% of wall-adjacent cells near the edge of the rib, where $y^+ > 2$.

The quality of the LES can be also judged by the indicator based on the resolved turbulent kinetic energy (TKE) (di Mare et al. 2014), given by

$$M = \frac{\text{resolved TKE}}{\text{total TKE}} = \frac{\langle u'_i u'_i \rangle}{2k_{sgs} + \langle u'_i u'_i \rangle}.$$  

The subgrid-scale TKE $k_{sgs}$ was estimated as $k_{sgs} = \nu^2/(C_k^2 \Delta^3)$, with $C_k = 0.094$. A proper LES is characterised by resolving more than 80% of the energy spectrum (di Mare et al. 2014, Celik et al. 2005), therefore, the statistical average of Equation 6, denoted as $\langle M \rangle$ should be greater than 0.8. The plot of $\langle M \rangle$ at the symmetry plane of the channel is visible in Fig. 4. Apart from a limited number of cells ($\sim 0.02\%$) located near the edges of the ribs with $\langle M \rangle < 0.8$, the LES provides an excellent energy resolution, with $\langle M \rangle \geq 0.95$ in more than 99% of cells, therefore it approaches DNS limit. The parts where energy-containing eddies are under-resolved coincide with the near wall region of $y^+ > 2$. Based on Fig. 3 and Fig. 4 it can be assumed that the effect of insufficient resolution at the wall remains localised, and does not influence the overall quality of the simulation.

Mean velocities are compared with the DNS results by Marocco & Franco (2016) in Fig. 6. An inspection of the velocity fields reveals the presence of two separated regions downstream of the ribs and two smaller recirculation regions upstream. LES results are in very good agreement with the reference data. It is further showcased by comparing the friction coefficient, computed as

$$C_f = \frac{\tau_{wall}}{(0.5 \rho U_{bulk}^2)},$$

at the lower wall of the channel.

The plot of $C_f$ for LES is visible in Fig. 5. The computation with the WALE model predicts accurately both the magnitude of shear stresses and the size of the two recirculation regions located over the bottom wall, upstream and downstream of the rib.

For heat transfer problems, the most important parameter is the Nusselt number, quantifying the ratio of convective to conductive heat transfer, computed as

$$\text{Nu} = H \nabla T \cdot n/(T - T_{bulk}),$$

where $n$ denotes an outwards directed wall normal vector. The distribution of $\text{Nu}$ along the lower wall is visible in Fig. 5. The results from LES follow closely the DNS profile, with small discrepancies present in the region of flow reattachment downstream of the rib. Results obtained from the LES resemble the reference DNS data to a good extent. As the LES represents the theoretical limit of accuracy of the hybrid models, it will be used hereafter to assess their capabilities.
The geometry has been also simulated with \( k-\omega \)-SST model on meshes 3, 5 and 7. In each case, the unsteady simulation produced nearly two-dimensional solutions. Mesh independence has been reached at mesh 5.

Classically, RANS turbulence models are not always accurate in this type of configurations, as pointed out by Han et al. (2015), due to the separated regions dominating the flowfield. This is reflected in the predicted friction coefficient profile depicted in Fig. 8. The size of the recirculation regions is computed inaccurately and, furthermore, the magnitude of the shear stress is grossly underpredicted after the flow reattaches downstream.

Similarly, the predicted Nusselt number compares poorly with the profile computed from the LES. The values are severely underestimated along the whole heated wall, indicating a drastically decreased influence of convective heat transfer. This seems to be caused by the inaccurate prediction of the turbulence mixing, due to the lack of resolved fluctuations in the flow field. Marocco & Franco (2016) reported similar results with both the \( k-\omega \)-SST and the \( k-\varepsilon \) realisable models. The average errors of prediction of both parameters, for each mesh has been gathered in Tab. 2.

The predicted profiles of \( C_f \) and \( Nu \) computed from the IDDES simulation are visible in Fig. 7. The computation on mesh 3 results in a good qualitative prediction of the skin friction coefficient. Overall, the character of the main recirculation region, as well as the magnitude of the wall shear stress are reproduced well. However, the separated region downstream of the rib is visibly too big compared to the LES results. The Nusselt number (the figure is clipped below \( Nu = 80 \) to improve readability) is highly overestimated over most of the wall, with a qualitative agreement between the LES and the presented results visible only in the immediate upstream vicinity of the rib.

On the other hand, the results from meshes 5, 7 and 9 are in excellent agreement with LES data, both in the case of the skin friction coefficient and the Nusselt number, as indicated by low values of RMS differences between the fields listed in Tab. 2. This indicates two important points. First, the use of \( k-\omega \)-SST-IDDES allows for an accurate prediction of the flow field and heat transfer at a potentially lower resolution than LES. This does not necessarily mean that this model allows for resolving a lesser portion of the energy spectrum than LES. In fact, already on mesh 5, the model resolves more than 0.8 of the energy in the whole domain. However, the
ability to use the RANS part of the model near the walls allows for higher values of $y^+$ than usually required, therefore, the simulations could be seen as wall-modelled LES. In addition to that, the use of the blended scheme enables more aggressive mesh refinement without numerical instabilities that would be generated by a central scheme on a similar mesh.

Secondly, it is useful to note that the IDDES model results in comparable results to WALE model on mesh 9. This confirms that the resolved part of the model really acts as a proper sub-grid scale closure, and when mesh resolution is adequate, the solution is not polluted with an overabundant amount of artificial viscosity. The performance of the SAS model was assessed with the same quantities, plotted in Fig. 9. Overall, the results exhibit poor accuracy w.r.t. the reference LES. However, similarly as in the case of $k$-$\omega$-SST-IDDES, they offer substantial improvement in comparison to the URANS predictions. The solutions seem to be converging to an inaccurate result with increasing mesh resolution. In the case of the Nusselt number, the curves approach the reference profile, although even on the finest mesh the model does not predict $\text{Nu}$ accurately in the rib downstream region. Similarly, the friction coefficient remains underpredicted. In general, on more refined meshes, the model overpredicts the length of the separation bubble downstream, while reproducing the flow parameters upstream of the rib to a satisfactory degree. Comparing the results on the coarsest grid, predictions with SAS model appear to be more accurate than those with IDDES. At this resolution, $C_f$ is predicted with similar accuracy as the other hybrid model, with a significant improvement over IDDES in the case of the Nusselt number computation.

An additional test has been performed with the unstructured mesh shown in Fig. 2b. It is important to point out that this mesh here has approximately three times less cells compared to mesh 5. The simulation was performed with the IDDES model and compared with the LES, URANS and IDDES results shown before. Results are shown in Fig. 10. The prediction of the friction coefficient is particularly good in the recirculation zone before the rib. On the other hand, downstream of the rib, the size of the recirculation bubble is bigger than the one predicted by LES. A negligible improvement is reached in comparison to mesh 3. Looking at the Nusselt number, the prediction is fairly good both upstream and downstream of the rib, even though the accuracy is affected by the incorrectly reproduced recirculation region. In comparison to mesh
Figure 9: Nusselt number and friction coefficient along heated wall. SAS simulations.

Figure 10: Nusselt number and friction coefficient along heated wall. Comparison of unstructured grid with LES, IDDES, URANS.

3, the increased fidelity of the prediction can be noticed especially for Nu. Overall, the performance of the hybrid model on such a coarse mesh is impressive, as also indicated by the values listed in Tab. 2. The RMS error of Nusselt number and skin friction coefficient prediction is much lower than estimations provided by the URANS model and IDDES simulation performed on mesh 3.

Moreover, the simulation on the unstructured mesh needed only 25 core-hours per one flow through time, while IDDES and SST simulations performed on mesh 5 required 136 and 51 core-hours, respectively. With regards to the computational times mentioned, we point out that to ensure consistent results while timing the computations, each run was performed on the same machine with the same linear solvers and $p-U$ coupling settings: 4 PISO loops and up to 15 PIMPLE loops if the residuals did not decrease below $10^{-5}$. Each mesh was also decomposed in such a way to ensure a similar amount of communication overhead between the processors, i.e. approximately 50000 cells per processor. As a final comment, we remark that the quality of the solution could, of course, be improved by gradually changing the mesh to have a higher refinement in the recirculation region, similarly to mesh 5. However, the distribution achieved and shown here provided a much faster solution, while sacrificing only a fraction of accuracy.

Table 2: Mean error of estimation of $Nu$ and $C_f$ vs. results from LES. The error is computed as a root mean square of the difference between both fields.

<table>
<thead>
<tr>
<th>mesh</th>
<th>$(Nu - Nu^\text{LES})_{\text{rms}}$</th>
<th>$(C_f - C_f^\text{LES})_{\text{rms}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SST</td>
<td>SAS</td>
</tr>
<tr>
<td>3</td>
<td>13.727</td>
<td>10.121</td>
</tr>
<tr>
<td>5</td>
<td>15.719</td>
<td>5.68</td>
</tr>
<tr>
<td>7</td>
<td>16.171</td>
<td>4.062</td>
</tr>
<tr>
<td>9</td>
<td>2.873</td>
<td>0.9</td>
</tr>
<tr>
<td>unstructured</td>
<td>3.728</td>
<td>0.005</td>
</tr>
</tbody>
</table>
CONCLUSIONS

The performance of two different hybrid RANS/LES turbulence models’ (\(k-\omega\)-SST-SAS and \(k-\omega\)-SST-IDDES) has been benchmarked for the simulation of ribbed channels. Such geometries are particularly relevant for turbomachinery cooling applications and investigations of the influence of surface roughness for turbulent flows. Four different block-structured, hexahedral meshes, with increasing resolution, have been employed, to test the sensitivity of the model to grid refinement. Additionally, highly resolved LES of the same configuration has been performed and used as a reference for the discussion of the models accuracy. To further facilitate the comparison, the URANS computation of the same flow with \(k-\omega\)-SST was conducted. The \textit{a priori} and \textit{a posteriori} criteria to judge the quality and appropriateness of the meshes have been discussed and employed.

The first conclusion drawn, in line with the results reported by other authors, is the failure of the adopted RANS model to predict accurately the length of the recirculation region and the magnitude of the heat transfer occurring through convection. Secondly, both hybrid models offer a substantial improvement over the \(k-\omega\)-SST. The IDDES model is able to reproduce the LES results with excellent accuracy on a coarser mesh, in a simulation that might be regarded as wall-modelled LES. Additionally, the model achieves similar accuracy to the WALE model on the mesh employed for the LES, showcasing that it truly behaves like an SGS model when resolution is sufficient. The SAS modelling approach produced less accurate results as both the friction coefficient and Nusselt number profiles were mostly underpredicted, even on finer meshes. Interestingly, SAS provided far better results than IDDES on the coarsest mesh, indicating that this model might be a better choice when sufficient resolution cannot be guaranteed.

The IDDES model has also proved to be fairly accurate on the unstructured mesh not meeting the strictest resolution criteria for SRS. The mesh exhibits a high cell stretch ratio and anisotropy, and provides coarser resolution in the core flow, leading to three times fewer cells than the coarsest hexahedral mesh which captured the flow features accurately. Despite that, both the profiles of \(C_f\) and \(Nu\) were reproduced with satisfactory accuracy, showcasing that for such configurations a reduction of the computational effort is possible without sacrificing simulation fidelity.

ACKNOWLEDGEMENTS

The first author gratefully acknowledges financial support from the Deutsche Forschungsgemeinschaft (DFG) through SFB/TRR287, Project Number 422037413. The second author gratefully acknowledges financial support of the Bundesministerium für Wirtschaft und Klimaschutz under Grant No. 03EE5013M.

References


