Inflow turbulence generation for eddy-resolving simulations of turbomachinery flows

Sunil K. Arolla
Sibley School of Mechanical and Aerospace Engineering
Cornell University
Ithaca, NY 14853
Email: ska62@cornell.edu

ABSTRACT

A simple variant of recycling and rescaling method to generate inflow turbulence using unstructured grid CFD codes is presented. The method has been validated on large eddy simulation of spatially developing flat plate turbulent boundary layer. The proposed rescaling algorithm is based on the momentum thickness which is more robust and essentially obviates the need of finding the edge of the turbulent boundary layer in unstructured grid codes. Extension of this algorithm to hybrid RANS/LES type of approaches and for wall-bounded turbomachinery flows is also discussed. Results from annular diffuser with different inflow boundary layer characteristics is presented as an example application to show the utility of such an algorithm.

1 Introduction

High-fidelity eddy-resolving simulations require specification of accurate and realistic inflow conditions. The inflow boundary layer thickness can have significant influence on the flow characteristics downstream. For example, in inter-turbine or inter-compressor diffuser configurations relevant to turbomachinery, the inlet boundary layer thickness determines the flow behavior through the diffuser [1]. Hence, addressing this issue with a robust approach that can be used within general unstructured CFD
codes is critical.

Numerical simulations of fully developed, time-evolving flows are often performed using periodic boundary conditions in which the downstream flow can be directly re-applied at the inlet. However, these boundary conditions are not appropriate for spatially developing flows, such as turbulent boundary layers. In simulating such flows, the flow downstream is highly dependent on the conditions at the inlet, making it necessary to specify a realistic time series of turbulent fluctuations that are in equilibrium with the mean flow. The inflow data should satisfy the Navier-Stokes equations to be accurate.

The most straightforward approach to simulate a spatially developing turbulent boundary layer is to start the calculation far upstream with a laminar profile plus random disturbances and then allow for natural transition to turbulence to occur. This method is not generally applicable for turbulent flow simulations as it requires a long development section to simulate natural transition and hence is prohibitively expensive. The other simple procedure for specifying turbulent inflow conditions is to superimpose random fluctuations on a desired mean velocity profile. The amplitude of the turbulent fluctuations can be adjusted to satisfy a desired set of one-point second order statistics. However, the velocity derivative skewness is zero and hence inflow condition is void of nonlinear energy transfer and the flow lacks realistic turbulent structure. Also, a fairly lengthy development section is required to allow for development of organized turbulent motion. In addition, it is often hard to control the skin friction and integral boundary layer thickness at the end of the development section.

The method of using an auxiliary simulation to generate inflow boundary conditions is commonly used for internal flows [2]. A similar approach can be used for turbulent boundary layers as well. To account for spatial growth, Spalart (1988) developed a method by adding source terms to the Navier-Stokes equations [3]. This method is capable of producing equilibrium turbulent boundary layer with direct control on skin friction and integral boundary layer thickness. However, it requires a coordinate transformation that minimizes the streamwise inhomogeneity and hence cannot be adopted into general purpose CFD codes.

Lund et al. (1998) proposed the widely used recycling and rescaling method in which the velocity at the inflow plane is estimated using the flow downstream [4]. The velocity field extracted at a downstream location is rescaled and reintroduced at the inlet. This method proved to be very successful in generating accurate inflow data with specific boundary layer thickness. Some of the numerical issues reported in
the literature with the Lund et al. method are: spurious spanwise structures are recycled that can grow in time and disrupt the numerical stability, sensitivity to the initialization. Several different strategies have been adopted to address these issues such as using dynamically shifting the recycling location [5], constant spanwise shift [6], constant spanwise reflection [7], dynamic shifting and reflection using a random-walk method [8]. Ferrante and Elghobashi (2004) presented a modified method by imposing a specific energy spectrum to insure the statistical correlation between the streamwise and wall-normal fluctuations a non-vanishing magnitude [9].

Synthetic methods form another class of generating inflow conditions. These methods are characterized by the use of some model to prescribe turbulent fluctuations about a mean flow profile. Yao and Sandham (2002) proposed one of the first synthetic methods in which the observed features in a turbulent boundary layer such as near-wall and lifted streaks are semi-analytically prescribed by enforcing perturbation velocities according to the superposition of several waveform functions [10]. These waveform modes have amplitudes and phase shifts that correspond to desired streak lengths and thicknesses. Klein et al. have developed a digital filtering approach which is also widely used [11]. These methods however require sufficiently long domain lengths for the turbulent flow to recover from modeling errors. In addition, a priori knowledge of mean flow, Reynolds stresses is required for using these methods. Additional synthetic methods exist that offer shorter recovery lengths [12, 13]. A more detailed review of different inflow generation methods can be found in [14]. This article is concerned with recycling and rescaling type of approach in which a priori knowledge of mean flow and turbulence statistics is not required.

Spalart et al. (2006) proposed a variant of recycling and rescaling method where several physical arguments have been used to simplify the algorithm [6]. Lund et al. (1998) method uses different scaling laws for inner and outer layers. It also involves decomposition of the velocity field into mean and fluctuating components. Spalart et al. argue that the near-wall turbulence regenerates itself much faster than the outer region and hence proposed to use outer layer scaling throughout. Also, the rescaling is applied only the streamwise velocity components as corrections to the wall-normal components have very little effect. The advantage of this method is the spatially developing simulation generates its own inflow conditions and a short recycling distance leads to a reduction in computational cost. This method has been used in investigating Spalart-Allmaras model based detached eddy simulation (DES) models.

Use of such inflow generation methods with unstructured grid CFD codes in the context of turbomachinery applications has not been reported in the literature. In this article, a simple variant of recycling and rescaling method for generating inflow turbulence for unstructured grid CFD codes is presented and validation on large eddy simulation (LES) of flat plate turbulent boundary layer is reported. Extension of the method to hybrid RANS/LES type of approaches is discussed. As an example for such an approach, a recently proposed simplified version of Improved Delayed Detached Eddy Simulation (IDDES) of [17] is implemented and applied for spatially developing turbulent boundary layer using the proposed inflow generation method. For applying this method for turbomachinery applications, the required modifications are presented and validated on annular diffuser problem.

2 Computational framework

The computational framework used in this research is that of OpenFOAM finite volume based incompressible flow solver. The filtered Navier-Stokes equations solved in the context of LES are:

\[
\begin{align*}
\partial_t \hat{u}_i &= 0 \\
\partial_t \hat{u}_i + \partial_j \hat{u}_j \hat{u}_i &= -\frac{1}{\rho} \partial_i \rho + \nabla \hat{u}_i - \partial_i \tau_{ij}^{SGS}
\end{align*}
\]

where \( \hat{u}_i \) is the filtered velocity field. Note that symbol used to denote filtered variable is dropped in rest of the paper for simplicity. The unclosed term that arises due to filtering operation are the subgrid scale stresses given by \( \tau_{ij}^{SGS} \). The equations are close by employing a dynamic Smagorinsky model [18] with modification by Lilly (1992) [19].

As an example for hybrid RANS/LES approaches, a recently proposed simplified version of IDDES for \( k - \omega \) SST model [17] has been implemented within OpenFOAM framework. Stated briefly, the transport equations for turbulent kinetic energy \( (k) \) and specific dissipation rate \( (\omega) \) are of the following form:
\[
\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = P_k - \sqrt{k^3/l_{IDDES}} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_T \frac{\sigma_k}{\sigma_T} \right) \frac{\partial k}{\partial x_j} \right]
\]

(3)

\[
\frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} = P_\omega - D_\omega + CD_\omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_T \frac{\sigma_\omega}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right]
\]

(4)

where \( P_k \) is the production of turbulent kinetic energy, \( P_\omega \) is the production of specific dissipation rate, \( D_\omega \) is the dissipation of \( \omega \), and \( CD_\omega \) is the cross-diffusion term in \( \omega \). The term \( l_{IDDES} \) is the length scale that operates the switch between RANS and LES. The eddy viscosity is of the form \( \nu_T = k/\omega \) with a limiter for separated flows. For detailed description of these terms and empirical constants, see [17]. The governing equations solved are similar to that of LES, but subgrid scale stress term is replaced by a modeled Reynolds stress.

For the numerical simulations presented in this article, Pressure Implicit with Splitting of Operator (PISO) algorithm is employed. A second order accurate backward implicit scheme for time discretization and a second order central scheme (with filtering for high-frequency ringing) for spatial discretization is used. The initial and boundary conditions are discussed for each validation problem in the subsequent sections.

3 A variant of recycling and rescaling method for inflow turbulence generation

The recycling and rescaling method by [4] uses scaling laws by dividing the boundary layer into inner and outer regions. A composite profile is derived using a weighting function based on hyperbolic tangent function. The scaling operation requires computation of friction velocity and to obtain required momentum thickness, one must iteratively adjust the boundary layer thickness until the target value is reached.

The essential idea presented in this paper is based on the work by [6] to simplify the inflow generation algorithm based on the following physical arguments:

1. The near-wall turbulence regenerates itself much faster than the outer region turbulence \( \rightarrow \) Apply outer layer scaling throughout.

2. When the recycling station is located quite close to the inflow, which is desirable in terms of comput-
ing cost, the conflict between inner and outer region scaling essentially vanishes \( \rightarrow \) Short recycling distance

3. Corrections to the wall normal velocity component \( v \) have very little effect \( \rightarrow \) Omitted

Fig. 1. A schematic of the computational domain used for flat plate turbulent boundary layer simulation. The recycling plane is located at \( x_r = 5\delta_0 \) from the inflow boundary.

In the current work, momentum thickness based scaling is used in place of 99% boundary layer thickness. This avoids the need of locating the edge of the boundary layer thickness. Moreover, using integral quantities like momentum thickness (or displacement thickness) is numerically robust. Most experiments report the momentum thickness Reynolds number at the inflow and hence back-to-back simulations can be set-up easily. A spanwise mirroring method [7] is adopted as it was found to be adequate in the current work for disorganizing unphysical spanwise durable structures. It should be noted that more advanced strategies like random-walk based dynamic shifting and reflection might be more efficient, but those have not been tried out in the present work. The steps involved in the inflow generation algorithm are:

1. Extract the velocity field, \( u(x_r, y, z, t) \), at the recycling plane located at \( x_r \) and project on to the inflow boundary (see figure 1).

2. Perform spanwise averaging to get \( U(y, t) = \langle u(x_r, y, t) \rangle_z \). A simple indexing algorithm is used for the averaging. It involves looping over all the faces and index faces with the same wall-normal coordinate. Since the recycling plane is fixed, this indexing can be stored at the preprocessing step itself and reused at each timestep.

3. Find the freestream velocity \( U_{\infty} = U(y_{\text{max}}, t) \).
4. Integrate the velocity profile to compute the momentum thickness:

\[ \theta_r = \int_0^{y_{\text{max}}} \frac{U(y,t)}{U_{\infty}} \left( 1 - \frac{U(y,t)}{U_{\infty}} \right) dy \]  

(5)

5. Compute the rescaling factor, \( \gamma = \theta_r / \theta_{\text{in}} \), where \( \theta_{\text{in}} \) is the desired momentum thickness at the inflow.

6. Rescale only the x-component of the velocity field:

\[ u(x_{\text{in}}, y, z, t) = u(x_r, y \gamma, z, t - \Delta t) \]  

(6)

where \( t - \Delta t \) means the velocity from the previous time step is used for convenience. A linear interpolation is used to compute velocity at the rescaled y-coordinate.

7. Apply spanwise mirroring to disorganize unphysical structures which would otherwise be recycled and take much time to get dampened by the spanwise diffusion.

\[ u(x_{\text{in}}, y, z, t) = u(x_{\text{in}}, y, \Delta z - z, t) \]
\[ v(x_{\text{in}}, y, z, t) = v(x_{\text{in}}, y, \Delta z - z, t) \]
\[ w(x_{\text{in}}, y, z, t) = -w(x_{\text{in}}, y, \Delta z - z, t) \]  

(7)

where \( \Delta z \) is considered to be equal to the spanwise period. Note that \( w \) has to be negative to ensure spatial coherence once mirrored [7].

8. Check for constant mass flow rate at the inflow by verifying the bulk velocity.

The simplicity of this algorithm makes it amenable for extending to more complex applications as discussed further in this paper.
4 Extended algorithm for hybrid RANS/LES type of approaches

The algorithm presented in the previous section is mainly intended for direct numerical simulation (DNS) or large eddy simulation (LES). To extend this for hybrid RANS/LES type of approaches, the rescaling operation has to be modified depending on the the underlying RANS model.

For the Improved Delayed Detached Eddy Simulation (IDDES) model considered in this work, the rescaling operation on the underlying SST variant of $k – \omega$ model requires the following:

\[
\begin{align*}
k(x_{in}, y, z, t) &= k(x_r, y\gamma, z, t - \Delta t) \\
\omega(x_{in}, y, z, t) &= \omega(x_r, y\gamma, z, t - \Delta t)
\end{align*}
\]  \(8\)

with $\gamma = \theta_r / \theta_{in}$.

To verify the accuracy, this recycling and rescaling method has been applied for the RANS simulation of flat plate turbulent boundary layer. As shown in figure 2, the method was able to generate a turbulent velocity profile accurately. So, no special care is taken to account for the location where the switching from RANS to LES takes place. The grid sensitivity of such hybrid RANS/LES approaches is still an open question and hence any errors associated with using hybrid methods for spatially developing boundary layers might be due to the underlying modeling assumptions and not the inflow generation method \textit{per se}.

![Fig. 2. RANS simulation of flat plate boundary layer using recycling and rescaling procedure: Mean velocity profile in wall units](image-url)
The modifications required for using this algorithm for wall-bounded turbulent flows is discussed in the following subsection. Accuracy of this method for RANS approach means that, this could be used for imposing asymmetric velocity and turbulence profiles at the inlet. This is especially useful where the experimental data has inherent asymmetry due to the wind tunnel sidewall effects. This algorithm provides a means to set-up simulations consistent with the experiments and hence is useful in robust evaluation of the turbulence closure models used in the design.

5 Extended algorithm for wall-bounded turbulent flows applied to turbomachinery

Wall-bounded turbulent flows are often simulated using a streamwise periodic boundary condition to achieve fully developed turbulence condition at the inlet. But, in realistic turbomachinery applications such as inter-turbine or inter-compressor diffusers, it is important to specify a specific boundary layer thickness.

Since there is a variation in pressure in the streamwise direction, the validity of the scaling laws used in the inflow generation method becomes questionable. The momentum thickness based scaling used in the proposed algorithm can be extended to wall-bounded flows by making the following assumptions:

2. Velocity at the half the height of annular diffuser is considered to be freestream velocity for computing momentum thickness.
3. Effect of transverse curvature is assumed negligible for the inflow generation purpose.
4. Effect of streamwise pressure gradient is ignored as a short recycling distance is used.

To adapt the inflow generation algorithm for annular diffuser type of applications, rescaling operation is applied separately for the hub and casing boundary layers. The momentum thickness for the hub and casing boundary layers is calculated as:

\[ \theta_r = \int_0^{r_{0.5}} \frac{U(r,t)}{U_\infty} \left( 1 - \frac{U(r,t)}{U_\infty} \right) dr \]  

(9)

where \( U_\infty = U_{0.5} \) and \( r_{0.5} = (r_o - r_i)/2 \). For the casing boundary layer, the velocity profile is integrated down to the half of the annulus height. A schematic of the computational domain is shown in figure 3.
The accuracy of the algorithm for realistic turbomachinery applications is discussed in the following section.

![Diagram](image1)

(a) Schematic of the computational domain

![Diagram](image2)

(b) Nomenclature used in the algorithm

Fig. 3. LES of 30° sector of the annular diffuser

6 Results

The proposed algorithm has been validated on eddy resolving simulations of flat plate turbulent boundary layer. As an example for a practical application, inlet conditions generated for LES of flow through annular diffuser are also presented and compared with the available experimental data. The mesh employed for these problems is structured, but is stored in an unstructured grid format for OpenFOAM, and hence the proposed algorithm is applicable for general unstructured CFD codes.

6.1 LES of Spatially developing flat plate turbulent boundary layer

As a baseline validation, results from LES of flat plate turbulent boundary layer with inflow momentum thickness Reynolds number of $R_\theta = 1520$ are presented. The computational domain has dimensions $12\delta_0 \times 3\delta_0 \times 3\delta_0$ in the streamwise, wall-normal, and spanwise directions, respectively where $\delta_0$ is the 99% boundary layer thickness at the recycling plane. The mesh contains $182 \times 96 \times 164$ points in the streamwise, wall-normal, and spanwise directions, respectively. In terms of the wall units, the mesh resolution is $\Delta x^+ \approx 45$, $\Delta y_{wall}^+ \approx 1$, $\Delta y_{max}^+ \approx 20$, and $\Delta z^+ \approx 12$. Uniform mesh is used in the streamwise and spanwise directions while a hyperbolic tangent stretching is used in the wall-normal direction to cluster points close to the wall. The recycling station was located at $5\delta_0$ downstream of the inlet and the simulation provides its own inflow. The bottom wall is treated as a no-slip wall, top boundary is a slip
wall, and at the outflow an advective boundary condition is used.

As noted in [6], the initialization is important when using such inflow generation algorithms. The mean velocity profile given by Spalding law with random fluctuations with a maximum amplitude of 10% of the freestream value superimposed on the mean value. The time step used is approximately two viscous time units ($\Delta t \approx 2\nu/u_\tau^2$). The simulation was run for 1000 inertial timescales ($\delta_0/U_\infty$) to eliminate transients and the statistics are collected over another 1000 timescales.

![Graph](image)

(a) Mean velocity in wall units

(b) Reynolds stress components in wall units

Fig. 4. LES of spatially developing turbulent boundary layer: one-point statistics

Figure 4 presents comparison of the mean streamwise velocity and three Reynolds stresses plotted in wall units with the experimental data of [20] for a flat plate boundary layer at $R_\theta = 1430$. The mean velocity profile is in good agreement with the experimental profile as well as the DNS of [3]. The normal Reynolds stresses also show good agreement for the current grid resolution chosen. The shear stress shows much better agreement than that published in the earlier literature with LES.

In the context of this flat plate boundary layer problem, some features of the numerical algorithm are briefly discussed below:

### 6.1.1 Robustness

In the recycling and rescaling method proposed by [4], boundary-layer thickness $\delta_{99\%}$ is used. It is defined as the wall-normal location at which velocity equals 99% of the freestream velocity, $U_\infty$. This
is a poorly conditioned quantity as it depends on the measurement of a small velocity difference. Integral quantities such as momentum thickness or displacement thickness are more reliable measures [21]. To demonstrate the robustness of using momentum thickness for rescaling operation, numerical experiments are conducted by keeping the grid resolution constant. Normalized mass flux across the inflow boundary using the two methods is plotted in figure 5. When 99% thickness is used, the normalized mass flux across the inflow boundary fluctuates as much as 5%. This becomes a critical issue especially when the initial conditions are not carefully chosen. Using integral thickness ensures a stable inlet mass flux.

![Fig. 5. Comparison of normalized mass flux across the inflow boundary. The dashed line represents a method using 99% boundary layer thickness, solid line for a method using momentum thickness.](image)

### 6.1.2 Statistical convergence

The statistical convergence is ensured by looking at time evolution of the friction velocity at the inflow boundary. As shown in figure 6, friction velocity decreases as the turbulence decays during the initial transient. It takes about 100 inertial timescales before reaching a stable value. The statistics are extracted by averaging in time over another 100 inertial timescales. In terms of convergence, using the momentum thickness for rescaling operation did not show much difference.
6.1.3 Sensitivity to the initial conditions

The sensitivity of such recycling and rescaling methods to initial conditions is recognized in the literature, for example [5,6]. Random perturbations that are too weak might die out resulting in laminar flow. Numerical experiments with two different initializations are conducted to demonstrate the sensitivity of the proposed algorithm to the initial conditions. One with a random fluctuations over a laminar velocity profile, the other using velocity profile given by Spalding law with random fluctuations with a maximum amplitude of 10% of the freestream value superimposed on the mean value. The fluctuations die out in the freestream and towards the wall. The velocity profiles extracted from these initializations are shown in figure 7. The time evolution of the friction velocity shows that the flow nearly laminarizes in the case 1 before the perturbations grow and sustain the turbulence. This is a good example to show the robustness of proposed algorithm and moreover, if one were to iterate on the friction velocity to obtain required momentum thickness as in the case of [4], one is likely to get a laminar flow with such initialization.

6.1.4 Sensitivity to the grid resolution

To study the sensitivity to the grid resolution, numerical experiments are performed by reducing the number of grid points by half in the streamwise and spanwise directions. For wall-resolved large eddy simulation, Chapman suggested a grid resolution in wall-units of $\Delta x^+ \approx 100, \Delta z^+ \approx 20$ [22]. Typically,
Δx^+ ≈ 50 ~ 130, Δz^+ ≈ 15 ~ 30 have been used in wall-resolving LES [23]. The baseline resolution is chosen to be within this range. As shown in figure 8, by coarsening the grid in the spanwise direction, the mean velocity profile is retained. When the grid is coarsened in the streamwise direction, the mean velocity profile departs from the log-law behavior. This behavior is also observed by [4] with their recycling and rescaling method. The proposed rescaling method retains this.

6.1.5 Sensitivity to the location of the recycling plane

The location of recycling plane is, in general, chosen to be approximately at 5δ_{99%} from the inflow boundary. This depends on the Reynolds number at which the simulation is performed. At higher
Fig. 8. Sensitivity to the spanwise grid resolution on mean velocity. In wall units, solid line shows simulation with $\Delta x^+ \approx 45, \Delta z^+ \approx 12$ in the streamwise and spanwise directions respectively. The dashed line represents $\Delta x^+ \approx 45, \Delta z^+ \approx 24$, and the dotted line represents $\Delta x^+ \approx 90, \Delta z^+ \approx 12$.

Reynolds numbers, there will be wide range of scales and hence the location of the recycling plane can have spurious effects on the results.

Within the scope of this work, three locations of the recycling plane are tested: $1\delta_{99\%}$, $10\delta_{99\%}$, $5\delta_{99\%}$. As shown in the figure 9, the mean flow shows that if the recycling plane is too short, it does not reproduce correct log-law behavior. It is best to chose the location based on an estimate of the largest energy containing scale.

Fig. 9. Sensitivity to the location of the recycling plane. The solid line represents recycling plane placed at $5\delta_{99\%}$, dashed line is for $1\delta_{99\%}$, and dotted line is for $10\delta_{99\%}$.
6.2 IDDES of spatially developing flat plate turbulent boundary layer

The mesh used for IDDES for the same computational domain has $140 \times 96 \times 116$ points in the streamwise, wall-normal, and spanwise directions, respectively. In terms of wall units, the mesh resolution is $\Delta x^+ \approx 60$, $\Delta y_{wall}^+ \approx 1$, $\Delta y_{max}^+ \approx 20$, and $\Delta z^+ \approx 16$. It was found that the recycling plane needs to be much closer to the inflow boundary for IDDES to sustain turbulence. In the current work, $2 - 3\delta_0$ was found to be optimal recycling distance for IDDES.

The mean velocity and Reynolds stress profiles obtained with IDDES approach are shown in figure 10. The mean velocity profile is in excellent agreement with the DNS and experimental data. The peak in the Reynolds stresses are not predicted accurately. This is because of the near-wall RANS model used in the IDDES approach. It is a well-known issue with SST $k - \omega$ model that the near-wall peak in turbulent kinetic energy is under-predicted, but it gives accurate mean velocity [24].

![Fig. 10. IDDES of spatially developing turbulent boundary layer: one-point statistics](image)

6.2.1 Comparison of vortical structures predicted by LES and IDDES

The skin friction variation along the bottom wall predicted by LES and IDDES approaches is plotted in figure 11. As the momentum thickness increases along the bottom wall, the skin friction decreases and it is predicted well by both the approaches. The quantitative discrepancy between the predicted skin friction is due to the different near-wall behavior of the underlying closure models used.
The vortical structures resolved using LES and IDDES are shown in figure 12. As expected, LES predicts finer scale near-wall structures. In IDDES, LES is activated away from the wall and hence only large scale vortical structures are resolved. This evidence of vortical structures shows the effectiveness of the inflow generation methodology for eddy resolving simulations.
6.3 LES of flow through annular diffuser

To show the utility of the inflow generation algorithm for wall-bounded turbulent flows applied to turbomachinery, LES of flow through annular diffuser is performed. The geometry used is that of [1]. Two cases are considered with momentum thickness of about 4% and 0.3% of the height of the annulus at the inflow to the diffuser. The Reynolds shear stress profiles predicted by LES inflow generation method are compared with the available experimental data. The mesh used has $348 \times 128 \times 128$ points in the streamwise, radial, and azimuthal directions, respectively. The simulation has been run for 50 timescales ($r_{0.5}/U_{0.5}$) and the statistics are collected over another 50 inertial timescales.

The cross plane velocity contours in figure 13 show that the growth of the boundary layer is much rapid when the inflow momentum thickness is larger. It is also evidenced from the vortical structures presented in figure 14. The velocity and Reynolds shear stress profiles in these two cases are plotted in figure 15. The normalized shear stress shows good agreement with the experimental data at the inflow to the diffuser. The velocity profile with the lower momentum thickness looks to be uniform, but it is critical to provide the turbulence quantities to predict the flow behavior through the annular diffuser. The proposed inflow generation method proves to be effective for this purpose. The profiles of mean velocity and Reynolds shear stress are plotted within the diffuser section in figures 16-19. During the initial stage of diffusion near the entrance section given by $X/(R_o - R_i) = 8$, the turbulence enhances due to adverse pressure gradient and as the flow recovers downstream the turbulence intensity decreases. The mean velocity of the flow decreases through the diffuser. It can be observed that these profiles show significant differences in the two cases with different inflow conditions, near the inner and outer walls as well as through the core of the diffuser.
(a) Case 1: $\theta_r = 4\%$ of $(r_o - r_i)$

(b) Case 2: $\theta_r = 0.3\%$ of $(r_o - r_i)$

Fig. 13. Cross plane velocity contours

(a) Case 1: $\theta_r = 4\%$ of $(r_o - r_i)$

(b) Case 2: $\theta_r = 0.3\%$ of $(r_o - r_i)$

Fig. 14. Vortical structures predicted in the annular diffuser using Q-criterion
Fig. 15. LES of 30° sector of the annular diffuser. Profiles are extracted at the inlet to the diffuser. Solid lines: Case1, Dotted lines: Case2
Fig. 16. Profiles are extracted within the diffuser at the $X/(R_o - R_i) = 8$, near the inlet section. Solid lines: Case1, Dotted lines: Case2
Fig. 17. Profiles are extracted within the diffuser at the $X/\left( R_o - R_i \right) = 10$. Solid lines: Case1, Dotted lines: Case2
Fig. 18. Profiles are extracted within the diffuser at $X/(R_o - R_i) = 12$. Solid lines: Case 1, Dotted lines: Case 2.
7 Conclusions

A simple variant of recycling and rescaling method to generate inflow turbulence is presented for unstructured grid CFD codes. This method contains a momentum thickness based rescaling algorithm combined with a mirroring method to disorganize spanwise durable structures. The mean streamwise velocity and turbulence profiles predicted by LES and IDDES proves the accuracy of the methodology. For annular diffuser, it has been demonstrated that the algorithm presented can be used to obtain required turbulent boundary layer characteristics at the inflow.

8 Acknowledgements

Thanks to Professor Paul Durbin for helpful discussions on this work. This work used computational resources of Stampede under the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation.

References


