DIRECT NUMERICAL SIMULATIONS OF TURBULENCE SUPPRESSION IN ROTATING PIPE FLOWS

J. DAVIS1, S. GANJU, S. BAILEY AND C. BREHM

Department of Mechanical Engineering, University of Kentucky, Lexington, KY 40508, USA

Key words: Turbulence Suppression, Relaminarization, Direct Numerical Simulations

Abstract. Although relaminarization has been observed and documented for many decades, very little knowledge has been accumulated about the relevant physical mechanisms. In this study, the rotating pipe serves as a prototypical flow to investigate the core mechanisms behind turbulence suppression and relaminarization. Direct numerical simulations are employed on high performance computing systems with sufficient grid resolution of $O(10^9)$ grid points to capture all relevant temporal and spatial scales.

1 INTRODUCTION

In past experiments and analysis [1, 2] rotation of turbulent flows has been shown to provide a stabilizing effect, reducing drag and causing the mean velocity profile to appear laminar. The mechanisms causing turbulence suppression are currently not well understood, and a deeper understanding of these mechanisms is of great value for many practical examples involving swirling or rotating flows, such as swirl generators, wing-tips, axial compressors, hurricanes, etc. The turbulent flow through a rotating pipe is an excellent prototypical case and can be used for detailed investigations of these physical mechanisms. While models, such as RANS and wall-resolved LES, fail to accurately reproduce the flow physics involved in turbulence suppression, Direct Numerical Simulation (DNS) can be used to effectively study rotation effects on turbulent structures.

Existing DNS studies of this phenomenon have been restricted to relatively low Reynolds numbers ($<7,400$) [3] and low rotation numbers. The limited range of these parameters prohibit the observance of the full relaminarization process. It is the goal of the current study to fully capture the relaminarization process using high fidelity simulations across a wide range of Reynolds numbers so that low Reynolds number effects can be isolated. DNS at higher Reynolds numbers require significantly computational resources to resolve the wide range of turbulent scales.

2 COMPUTATIONAL METHODOLOGY

The computational domain size and grid of the rotating pipe (with a length of 15D) was carefully chosen to be sufficient to capture all relevant flow structures observed in turbulent suppression. Three different Reynolds numbers, $Re_D=5300, 11700, 19000$ were

1e-mail correspondence: jefferson.davis@uky.edu
Jefferson Davis, Sparsh Ganju, Sean Bailey and Christoph Brehm

run each at a range of rotation numbers, \( N = \Omega D/2U = 0 - 5 \), where \( \Omega \) is the angular velocity, \( D \) is the diameter, and \( U \) is the mean bulk velocity.

A scaling study was conducted for our in-house higher-order accurate incompressible Navier-Stokes solver to ensure efficient utilization of the computational resources. Strong and weak scaling studies as shown in Figs. 1a-b were performed on the Stampede 2 HPC system with the number of nodes ranging from 1 to 64 (or 68 to 4,352 cores). The strong scaling was conducted on a grid consisting of approximately 140 million points. The weak scaling study was conducted at 32,000 and 125,000 grid points per processor.

3 CONCLUSIONS AND OUTLOOK

The final paper will provide details about the numerical methods used to conduct DNS and code performance studies will be shown and analyzed to demonstrate the efficiency of these methods on a large HPC system considering up to 10,000 cores. Finally, DNS results of turbulent flow through a rotating pipe considering a wide range of Reynolds numbers, \( \text{Re}=5,300-19,000 \), and rotation numbers, \( \text{N}=0-5 \), will be presented including a detailed analysis of the flow physics involved in turbulence suppression, such as turbulent budgets, higher-order spectral analysis, etc.

REFERENCES

