Validating Direct Numerical Simulation of Channel Flow Using ANSYS Fluent

LIBERTY UNIVERSITY

Reid Prichard and Dr. Wayne Strasser

Background

- Computational Fluid Dynamics (CFD) is a tool to simulate moving fluids, such as liquid or gas.
- Turbulent fluid flow is innumerably complex, with physically significant motion occurring at microscopic length scales.
- Even supercomputers struggle to perform CFD at this resolution, so the effects of these small turbulent motions are generally approximated.
- Sometimes, however, we need to examine those small motions.
- Direct Numerical Simulation (DNS) is a type of CFD that simulates all significant length scales.

Introduction

- Because the equations that describe fluid flow are nonlinear, they cannot be solved directly.
- Instead, a fluid region must be broken down into small pieces ("elements") that are simple enough for a solution to be approximated.
- There are multiple approaches to this decomposition.
- DNS typically uses an approach called the "spectral element method". While very efficient, this method can only be applied to simple simulations.
- The "finite volume method" (FVM) is much more flexible, but its computational cost is high, requiring millions or billions of elements to produce correct results.
- The School of Engineering's FLUID Group plans to use DNS to study flexible structures found on sharkskin (Figures 1 & 2).
- To accomplish this, it will be necessary to use FVM DNS.
- Before we can do this, it is necessary to validate that the software we use, ANSYS Fluent, is capable of producing correct results.
- No published research has attempted to validate Fluent or similar tools.
- In validation studies, it is customary to study simple canonical cases.
- We chose a "channel flow", which consists of water flowing between two flat, parallel surfaces.

Methods

- We used ANSYS Fluent 2023R2's double precision solver.
- Numerical methods used:
  - Pressure-velocity coupling: SIMPLE
  - Momentum discretization:
    - Bounded Central Differencing
    - Central Differencing
    - Second-Order Upwind
  - Pressure interpolation: Second Order
  - Transient formulation: Second Order Implicit
  - Gradient reconstruction: Least-Squares
- To analyze our results, a custom program was written in C++ to compute high-order statistics, discretized partial differential equations, and power spectral densities
- We considered three different domain decompositions:
  - 8 million hexahedral elements
  - 70 million hexahedral elements
  - 20 million polyhedral elements
- We also considered a domain 2x as long and 2x as wide

Motivation

Figure 1: A Mako shark [4]

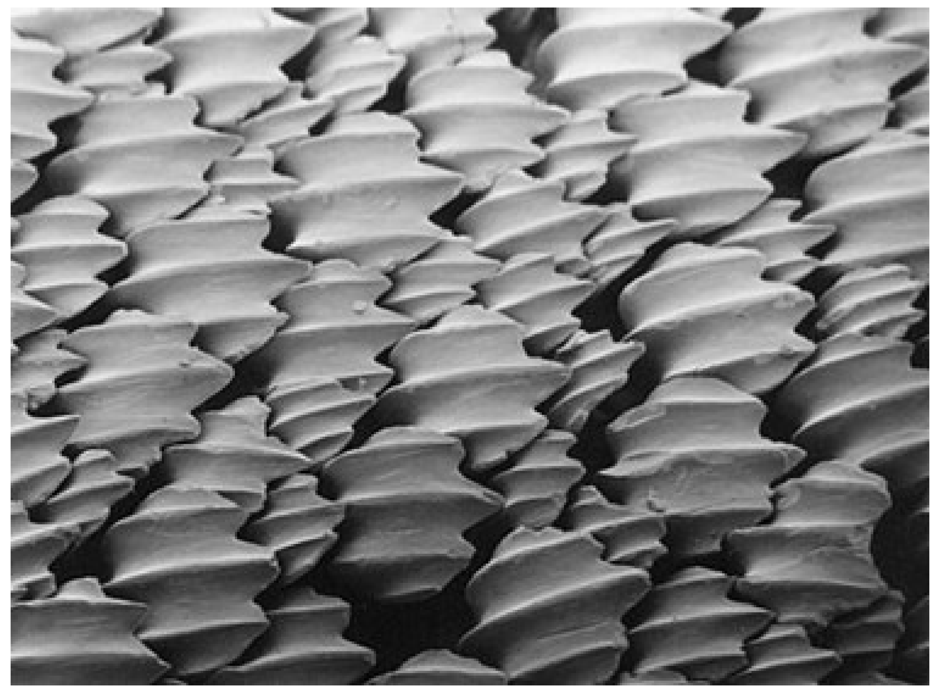

Figure 2: "Denticles" on the shark's skin, which passively prevent flow separation [5].

Velocity Profiles Are Correct

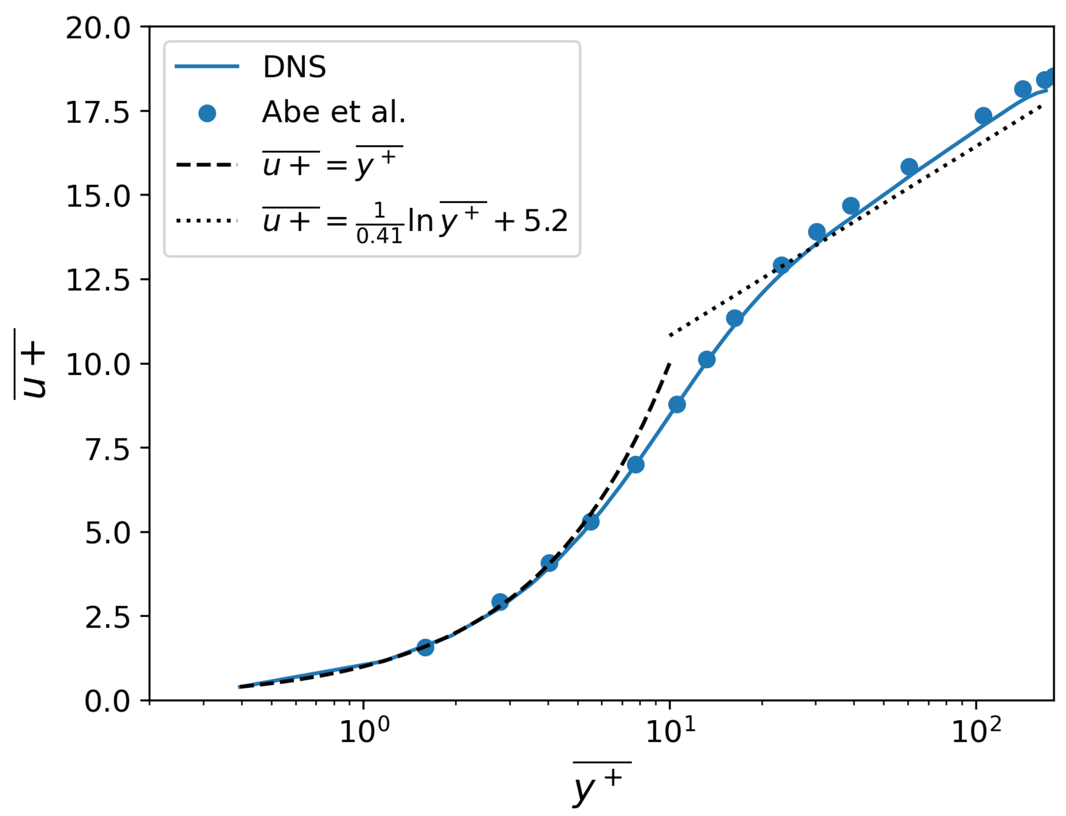

Figure 3: Mean streamwise velocity profile. Our data (the line) agrees well with prior published data (points).

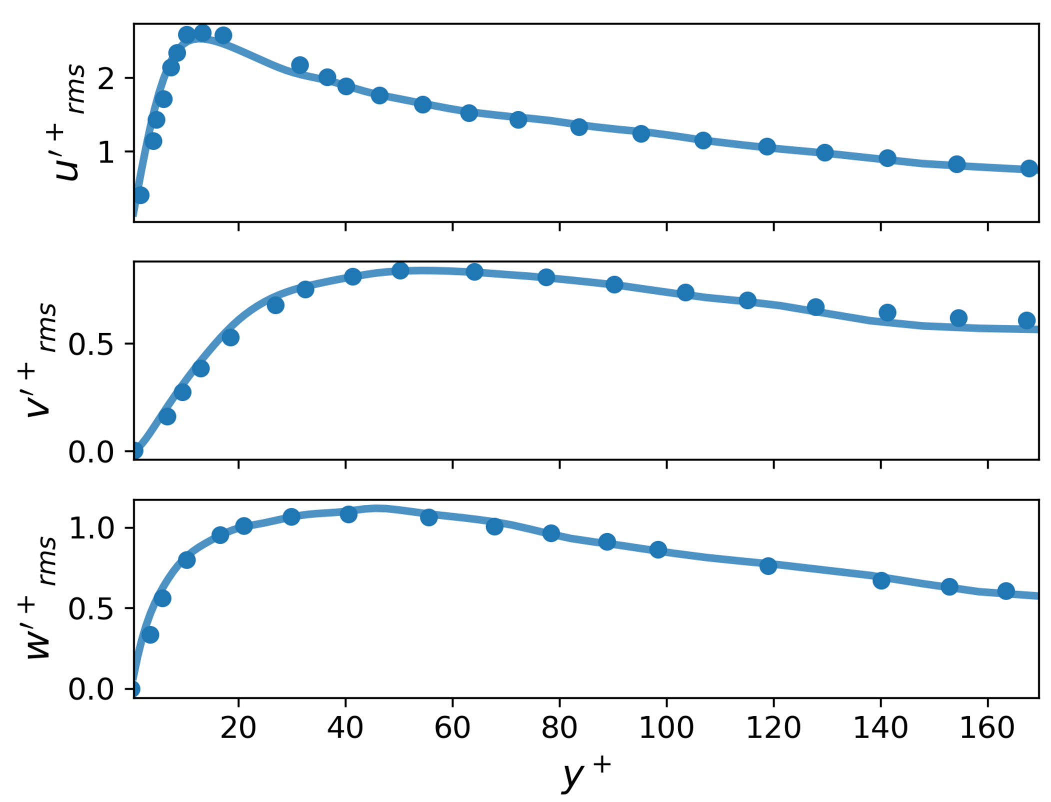

Figure 4: Mean velocity fluctuations. Again, our data (lines) agree well with published data (points).

Higher-Order Quantities Mostly Match Validation

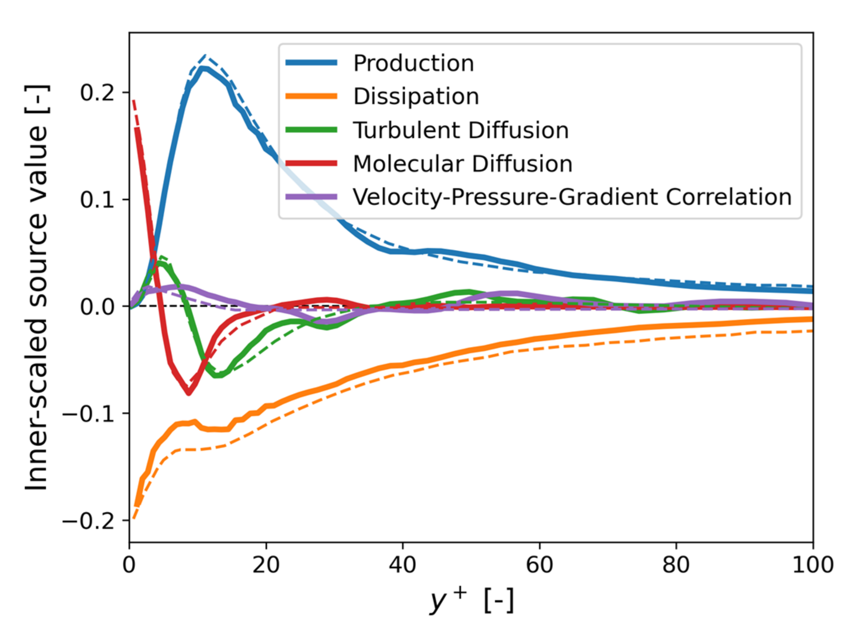

Figure 5: Turbulence Kinetic Energy (TKE) budget. The TKE represents the effects of turbulence on the bulk flow, so it is an important quantity to consider. Our data (solid lines) match the validation data (dashed lines) reasonably well, although we underpredict dissipation.

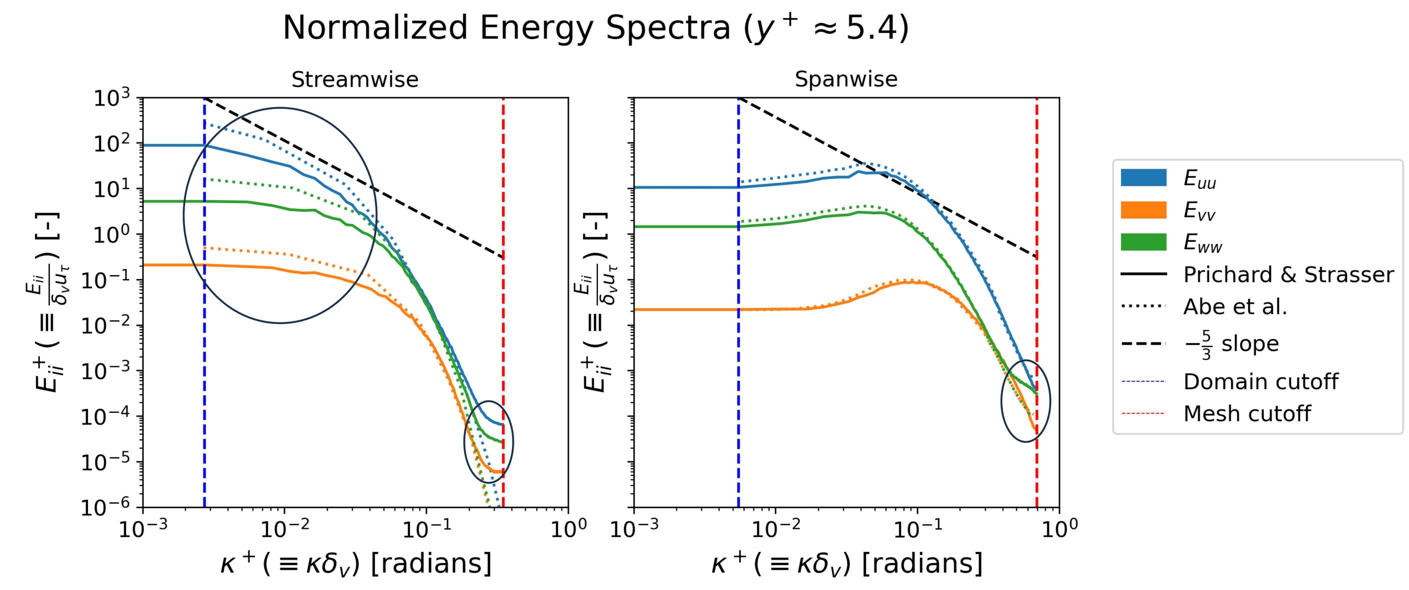

Figure 6: Power spectral densities (PSDs). Much like an audio visualizer shows the energy levels across a range of temporal audio frequencies, the PSD of a turbulent flow shows the amount of energy contained across a range of spatial frequencies. This is an extremely important metric in DNS, and it is the most difficult to match. We match the validation data generally, but energy is underpredicted at low frequencies and overpredicted at high frequencies.

Results

- The velocity profiles (mean, Figure 3, and RMS, Figure 4) match validation data from [1] extremely well.
- This indicates that at the very least, we are resolving a turbulent velocity profile.
- The turbulence kinetic energy budget (Figure 5) matches validation data well, though dissipation is underpredicted across the board.
- This suggests that we are capturing most of the energy-containing eddies, which means Fluent can produce useful (though perhaps not 100% correct) DNS results.
- So far, we have been unable to perfectly match the power spectral densities seen in Figure 6.
- While it is possible the discrepancy at low wavenumbers (left side of plot) can be explained by errors in the data we are validating against, the shape of the right-hand sides of our curves indicates insufficient resolved dissipation (which agrees with the underprediction of dissipation in Figure 5).

Conclusions & Future Work

- We have confirmed that ANSYS Fluent is capable of resolving a turbulent velocity profile in DNS of channel flow.
- However, some discrepancies exist when comparing to validation data.
- Turbulence kinetic energy dissipation is underpredicted across the entire channel.
- The effects of this are seen in the power spectral density plots, where too much energy has built up at large wavenumbers.
- We are in the process of assessing different numerical methods to correct this issue.
- Reducing cell size and/or switching numerical schemes should increase resolved dissipation.
- Increasing the domain size and/or switching numerical schemes should correct any issues with low-wavenumber energy, though it remains to be seen whether our validation data is correct there.
- We are also broadening our validation to include other metrics and other sources [2].
- Once we are satisfied with Fluent's correctness, we will proceed to validating a more complex type of flow [3].

References

[1] Abe, H., Kawamura, H., and Matsuo, Y., 2001, "Direct Numerical Simulation of a Fully Developed Turbulent Channel Flow With Respect to the Reynolds Number Dependence," Journal of Fluids Engineering, 123(2), pp. 382–393.

[2] Vreman, A. W., and Kuerten, J. G. M., 2014, "Comparison of Direct Numerical Simulation Databases of Turbulent Channel Flow at Re τ = 180," Physics of Fluids, 26(1), p. 015102.

[3] Cherukat, P., Na, Y., Hanratty, T. J., and McLaughlin, J. B., 1998, "Direct Numerical Simulation of a Fully Developed Turbulent Flow over a Wavy Wall," Theoretical and Computational Fluid Dynamics, 11(2), pp. 109–134.

[4] "A 300 lb. 8-foot female Short-fin Mako Shark (Isurus oxyrinchus) off the coast of San Diego, California, USA.", Obra Shalom Campo Grande, Flickr.

[5] "Denticules cutanés du requin citron Negaprion brevirostris vus au microscope électronique à balayage.jpg", Wikimedia Commons.