Unlocking the Secrets of Turbulence: Instabilities in Flows Over Bluff Plates

Hemant K. Chaurasia* and Professor Mark C. Thompson

*Corresponding author: hemant.chaurasia@gmail.com

The Problem

• To design fuel-efficient and high-performance technologies, engineers need to accurately predict turbulent flow – e.g. aerodynamic drag forces on cars, aircraft and Space Shuttles.
• For such designs, computational fluid dynamics (CFD) software is widely used by industry to predict fluid flow.
• Due to a limited understanding of turbulence, present-day CFD produces typical errors of ~5%, placing hard limits on achievable design efficiency.

Project Goals

Simulate the transition to turbulence for flow over a flat-faced rectangular plate

Characterise the underlying “instability modes” governing the onset of this transition

This knowledge may then be applied to devise CFD models which predict turbulent flow more accurately

Method

• In-house direct numerical simulation CFD code (“SE2D”)
• Spectral element method to simulate the flow; Floquet stability analysis technique5 to examine flow transitions
• Results verified by independent 2D and 3D simulations
• Faculty of Engineering Beowulf Computer Cluster: converts 100 PCs in student computer labs into a powerful supercomputer after hours

Key Findings

• Identified two instability modes with wavelengths ~1H and 4-5H (dominant).
• Dominant mode shows characteristics of the “elliptical instability”: a well-known theoretical mechanism for fluid instability.
• These same characteristics are seen in cylinder and bluff body wake flows4.
• This suggests that the elliptical instability mechanism may play an important role in the process of turbulent transition itself, independent of flow geometry.
• Such knowledge may enable CFD models that better predict turbulent flow.

References: