December 4, 2026 – CFD Post-Processing

Topics: Extracting and interpreting CFD results, flow visualization, data analysis, and presenting simulation results effectively.

Speaker: Tayna Chang (M.A.Sc. Candidate)

Time: 6:30–8:30 PM for all sessions

Location: ME 3235 (Mackenzie Building)

Registration is required for students wishing to attend the seminar. Please register using the following link:

https://forms.cloud.microsoft/r/Nkrqyedfhr

Speaker Bio – Tanya Chang

Tanya Chang is a MASc student in Aerospace Engineering at Carleton University, specializing in computational fluid dynamics (CFD) and aerodynamics. Her master's research focuses on reducing aerodynamic noise in low-speed wind tunnels through turning vane design optimization, CFD simulations, and experimental testing.

During her undergraduate studies, she worked as a Research Assistant (RA) with Prof. Ella and the Royal Military College of Canada (RMC) on CFD research related to aircraft engine performance, using ANSYS CFX and ICEM CFD . She also served as a Rear Wing Engineer for Ravens Racing (2024–2025), applying aerodynamic principles and CFD techniques to race car design and performance evaluation. Her experience includes ANSYS Fluent, CFX, Fluent meshing and ICEM CFD, with a focus on meshing, turbulence modeling, and practical CFD applications.

Guest Speaker: Behzad Amirsalari

Behzad is a PhD researcher in Aerospace Engineering at Carleton University, specializing in CFD, aeroacoustics, and aerodynamic optimization. His research focuses on using high-fidelity CFD simulations to investigate airflow-induced noise and improve airfoil performance through innovative design and optimization techniques.

His session will focus on practical considerations for Large Eddy Simulation (LES), including meshing strategies, pre- and post-processing techniques, and working with CFD applications in Linux environments. The workshop will provide students with insights into more advanced simulation practices and computational workflows.

The workshops are intended for graduate students as well as interested undergraduate students, particularly those involved in engineering design teams or CFD-related research. Rather than focusing only on basic CFD theory, the sessions will emphasize practical workflows, useful techniques, and lessons learned from working with ANSYS.