Applied Computational Fluid Dynamics (CFD) of Unsteady Flows for Naval Aviation

About NAWCAD:

The Applied Aerodynamics Branch at the Naval Air Warfare Center Aircraft Division supports development of new and improved aerodynamic technologies to fixed wing and rotary wing aircraft. Provides aerodynamic research required by the Navy; develops and validates methods for aerodynamic and performance analysis; derives, maintains, and improves engineering criteria, techniques, and methodologies for predicting the aerodynamic characteristics for which other aeromechanics competencies are responsible; and aerodynamic analysis as required by outside competencies dealing with external aerodynamic interaction effects. Defines and conducts aerodynamic Science and Technology research programs. Provides advanced modeling of aircraft/ship air-wake interaction. Provides the development and application of computational fluid dynamics while conducting research into fundamental aerodynamic principles and advanced aerodynamic concepts. Conducts analytic investigations to derive, maintain, and improve engineering criteria, techniques, and methodologies for predicting aerodynamic characteristics; reducing the technical risk associated with the design and development of aircraft. Develops and validates concepts and methods for reducing aerodynamic drag or increasing usable aerodynamic lift in flight vehicle configurations. Provides computational fluid dynamics analysis, lower level theoretical and empirical analysis, and experimental test support to characterize the aerodynamics of air vehicles.

RESEARCH LOCATION: Patuxent River, MD

PROJECT DESCRIPTION:

The intern will investigate impacts and best practices for modeling unsteady detached flows using HPCMP software and hardware, including representative problems for Naval Aviation. Typical predictions of this nature for Naval applications utilize single domain meshes. However, increasing simulation complexity has required the adoption of multi-domain meshing approaches, which have been observed to diminish the quality of the predicted flow field. The intern will learn and apply single- and multi-domain CFD best practices to model problems. The intern will identify sensitivity of prediction quality to meshing approach and numerical options. The intern will apply these findings to predictions of the air wake of a generic destroyer, which is representative of a typical Naval application of HPCMP tools.

The primary goal of this project is to execute a multi-domain meshing strategy study focusing on the accurate prediction of unsteady flow physics in bluff-body wakes. The representative problem will start simple and build to a representative Navy ship. Where applicable, experimental validation data will be utilized. The project will examine the impact of meshing strategy and CFD modeling parameters such as grid density, orientation of overset domain boundaries, turbulence modeling, numerical schemes, etc. on prediction accuracy.

Best practices and findings established during this study will be transitioned to the Applied Aerodynamics branch's standard workflows when supporting the many aircraft and ship programs that seek CFD modeling of shipboard operations.

Intern Activities:

- Familiarization
- Literature survey / review of current state-of-the-art
- Helios/Kestrel tutorials
- Grid Generation / Model Development
- Single and multi-domain mesh development for various geometries
- Mesh manipulation for oversetting studies
- CFD Execution & Mesh Sensitivity
- CFD case setup and execution
- Flow data extraction and post-processing
- Data Analysis
- Analyze time-varying flow physics to determine impact of meshing strategy on prediction quality
- Documentation and Presentation

The intern will have access to tour and training opportunities available to all NAWCAD Patuxent River intern hangar tours, lab tours, intern challenges, organization team building activities, and Patuxent River base amenities.

ANTICIPATED START DATE:

May/June 2026 – Exact start dates will be determined at the time of selection and in coordination with the selected candidate.

QUALIFICATIONS:

Bachelor's degree or higher in Aerospace or Mechanical Engineering. Candidates must be familiar with aerodynamics and numerical methods, preferably with a knowledge of computational fluid dynamics.

Skills:

- Familiar with at least 1 CFD code (e.g. Kestrel, Helios, Overflow, OpenFOAM, STAR-CCM+)
- Familiar with post-processing software (e.g. Ensight, Tecplot, Paraview, Matlab)
- Proficient in at least 1 programming language (e.g. Python, C/C++, Fortran, Matlab)

ACADEMIC LEVEL:

Degree received within the last 60 months or currently pursuing:

- Bachelor’s
- Master’s
- Doctoral

DISCIPLINE NEEDED:

- Engineering
- Science & Engineering-related