

ANDI ZHOU

Dear ANSYS Hiring Team,

I hope your day is going well! My name is Andi Zhou, a master's student from the University of Michigan Ann Arbor studying aerospace engineering, and I am writing to express my interest in applying for the **Fluids Intern** position for this summer!

I have a combined 3-year worth of CFD and thermal modeling experience in university project teams, aerospace, and the automotive industry. In particular, I am proficient at using ANSYS Fluent and Structure, and is familiar with different system coupling techniques.

During my time as the aerostructure lead at MASA (University Rocketry Team), I led a team of 12 in designing, simulating, and prototyping a 3-ft wide, 4-ft tall aluminum fin assembly capable of taking on Mach 5 aerothermodynamic load. I led the aerothermal analysis effort using ANSYS and Star CCM+, where we performed a fully transient convective heating simulation to ensure that our fins would not undergo aero-structural failure during flight. During the later stage of the project, I also led full-body aerothermal simulations on our rocket, where we leveraged ANSYS Fluent's aerothermal coupling abilities to investigate shock behaviors and their impact on other external rocket components such as raceways and the engine.

As a CFD Engineer Intern at Volvo Trucks North America, I spent 4 months in Greensboro, North Carolina, and became proficient with Multiphysics modeling using industry-level software such as ANSA and PowerFLOW. Using Star CCM+, I single-handedly set up a multi-phase Eulerian-Eulerian workflow to evaluate air-water separation performance within a cyclone separator. In the end, I was able to design a separator capable of maintaining a 99% separation efficiency while reducing the mass from the original concept by 40%. I also assisted in the thermal management of internal electronics and the HVAC system.

To add to my passion for aerodynamics and numerical modeling, I single-handedly coded a CFD solver using MATLAB that employs the famous SIMPLE method to solve the incompressible Navier-Stokes equations. Furthermore, I was able to verify the solver using the classic lid-driven cavity test case.

As an aerospace engineer with an unrelenting passion for numerical methods, I believe my skill sets align perfectly with your listed opening. Thank you so much for your consideration, and don't hesitate to contact me if you have any questions.

Thank you for your consideration!

Sincerely,
Andi Zhou