Ganesh Borde

✉ :ganeshyng789@gmail.com | 📞 +1 (737)-326-0702 | 🏠 Madison, Wisconsin

linkedin.com/in/ganesh-borde-928701217 | 🖥 :ganesh-borde.github.io

Education

Cockrell School of Engineering, The University of Texas at Austin   *Jan 2023 – Dec 2024*

Master of Science in Aerospace Engineering

Courses - CFD, Combustion, Compressible Flow, Viscous Flow, Dynamics of Turbulent Flow, Applied Numerical Methods, Adv Plasma, Tools/Tech Comp Sci, Computational Methods and Hypersonic Aerodynamics.

Vel Tech Rangarajan Dr. Sagunthala R&D Institute of Science and Technology   *July 2018 - June 2022*

Bachelor of Technology in Aeronautical Engineering

Courses - Aerodynamics, Fluid Mechanics, FEA, Turbomachinery, Thermodynamics, Heat transfer and Gas Propulsion.

Technical Skills

Applications: CONVERGE, ANSYS Fluent/CFX/Icepak, OpenFOAM, SolidWorks, AutoCAD, ParaView, CATIA, Cantera, CHEMKIN, MPI, VS Code, Git, Advanced Excel, and Microsoft Office tools.

Programming Languages: MATLAB, C/C++, Python, Julia, Fortran, and LaTeX.

Operating Systems: Linux OS, Mac-OS, and Windows.

Experience

CFD Application Engineer (Academics) | Convergent Science Inc   *Jan 2026 - Present*

- Provide advanced CFD modeling, simulation setup, and technical guidance to professors, PhD researchers, and graduate students using CONVERGE CFD for Multiphysics research applications.
- Lead end-to-end CFD case setup, including geometry preparation, meshing strategy, boundary condition definition, solver configuration, and post-processing for research-grade simulations.
- Support and advise academic users on numerical methods, turbulence modeling, multiphase flows, rotating machinery, AMR and transient simulations.
- Attend academic and research conferences to stay current with advances in CFD and engage with faculty on emerging modeling methods and applications.

Mechanical Engineer | De Nora Water Technologies   *June 2025 - Dec 2025*

- Contribute to mechanical engineering projects including pump/blower selection, fluid mechanics calculations, thermal load evaluation, and structural sizing/calculations.
- Support a high-stakes project by analyzing CFD results and conducting small-scale experimental validation to ensure product performance for customers.
- Support new PFAS product line development by generating P&IDs, BOM, performing design calculations, and creating vessel drawings per customer specifications.

Part Time Research Assistant | Georgia Southern University, Mechanical Engineering   *March 2025 – Sep 2025*

- Calculated flame properties for vented gas–air mixtures from Lithium-Ion Batteries at different condition using Cantera.
- Worked on developing integrating surrogate predictions into an in-house Explosion Vent Analyzer (EVA) to estimate peak pressure and minimum vent area for enclosure safety design.
- Worked on developing an FAA fire certification test for fiber-reinforced plastics (FRP) materials using a multiphysics solver integrating combustion, pyrolysis, and diffusion flame models to calculate ignition time, fluxes, and flammability limits.

Research Assistant | University of Texas at Austin, Aerospace Engineering   *Sep 2023 – Dec 2024*

- Simulated unsteady, reactive Navier-Stokes equations using a finite-difference in house CFD solver.
- Ran large-scale simulations on TACC (HPC), utilizing over 1,000 Service Units to study premixed flame behavior.
- Conducted 1D, 1D cylindrical and 3D CFD simulations of H_2-O_2 laminar flames across equivalence ratios and pressures.
- Computed Markstein numbers and laminar flame speeds for unity Lewis number mixtures to support turbulence studies.

Projects

CFD-Based Study of 18650 Battery Pack Cooling Using Serpentine Flow Paths

- Designed a serpentine aluminum cooling plate for an 8-cell 18650 battery pack.
- Simulated thermal performance in ANSYS Fluent and compared temperature distribution and pressure drop.

Analytical and CFD&FEA Study of Active Chip Cooling with Optimized Heatsink Design.

- Developed a thermal resistance model (RM) for a chip-heatsink-fan system with forced convection by using DOE.
- Optimized heatsink channels & identified 30 mm fin height &12 fins to maintain temperature below 85 °C for fixed CFM.
- Used PyAEDT automated Ansys Icepak simulations to optimize distance between chip-heatsink and fan.
- Results from simulations & resistance model had small differences in temperatures and velocities, due to simplicity in RM.

Design and analysis of small-scale axial flow pump impeller using CFD (Ansys CFX)

- Designed and modeled an axial flow pump with helical blades for compact medical applications.
- Evaluated multiple impeller designs using CFD (ANSYS CFX) to compare pressure and velocity performance.

Plasma Fluid Models for DC Glow Discharges using the Drift-Diffusion Model (FVM)

- Studied drift-diffusion models, basic models in industrial applications like material (semi-conductor) processing.
- Solving coupled equations for electron and ion momentum and second-order ODEs for electric potential.