Ganesh Borde

:ganeshyng789@gmail.com | +1 (737)-326-0702 | Austin, Texas
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: Ansys Fluent/CFX/Icepak, Star CCM+, Flotherm, SolidWorks, Paraview, Catia, Open FOAM, Cantera, Chemkin, HPCC, OpenMP, MPI, Comsol, VS Code, GIT, SVN, Advance Excel, and MS tools.
Programming Languages: MATLAB, C/C++, Python, Julia, Fortran, and LaTeX.
Operating Systems: Linux OS, Mac-OS, and Windows.

Experience

Research Assistant | Georgia Southern University, Mechanical Engineering *March 2023 – Present*
- Calculated flame properties for vented gas–air mixtures from Lithium-Ion Batteries (LIB) at different SoC, equivalence ratios, pressures, and temperatures using Cantera.
- Currently developing a surrogate machine learning model to predict flame properties rapidly for a wide range of conditions.
- Future work includes integrating surrogate predictions into an in-house Explosion Vent Analyzer (EVA) to estimate peak pressure and minimum vent area for enclosure safety design.

Research Assistant | University of Texas at Austin, Aerospace Engineering *Sep 2023 – Dec 2024*
- Worked with Dr. Fabrizio Bisetti in the Reactive Flow Modeling Laboratory on turbulent premixed flame analysis.
- 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.
- Extended solver capabilities by incorporating new physics to better capture laminar flame characteristics.

Summer Research | University of Texas at Austin, Computational Engineering and Sciences *June 2023 –Dec 2024*
- Conducted DNS studies on separation in channel flow under Dr. Robert Moser,
- Fourier-Galerkin method in x & z directions and solving 1D time-dependent PDEs in Fourier space.

Teaching Assistant | University of Texas at Austin *June 2023 – Dec 2024*
- Assisted in Differential Calculus, Business Calculus, and Introductory Physics Lab courses.
- Led decision sessions to guide students through problem-solving and clarify core concepts.
- Provided one-on-one support during office hours to address difficulties and boost comprehension.

Intern | KITE, AMTDC-IIT Madras, Ministry of Heavy Industry, Chennai, India *March 2022 - July 2022*
- Led the development of an Automatic Pallet Changer (APC), utilizing CAD modeling and FEA simulations.
- Designed the Human-Machine Interface (HMI) for the APC, ensuring seamless integration into CNC systems.
- Worked with cross-functional teams to integrate features that increase the effectiveness of APC operation.

Undergraduate Research | Centre for Computational Aerodynamics and Shock Waves *Dec 2021*
- Designed an efficient scramjet inlet for improved pressure recovery using Hypersonic Aerodynamics and JP-7 fuel.
- Modeled inlet geometry in CATIA V5 and applied structured meshing for precise control over shock interactions.

- Simulated supersonic flow in Ansys Fluent; analyzed shock structure, Mach number profiles, and temperature gradients.
- Presented detailed findings at ICES 2022, showcasing ignition feasibility and fuel efficiency in high-speed regimes.

Projects

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.

2D Steady -State Lid Driven Cavity Using Simple Algorithm - Finite Volume Method.

- Solved the steady-state Navier-Stokes (NS) equations for continuity and momentum using the FVM.
- Employed the SIMPLE algorithm for pressure-velocity coupling and simulated results for Reynolds numbers 100, 400, and 1000, validating the outcomes. Analyzed streamlines and vorticity of the flow field.

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

- Designed a Tesla-inspired serpentine aluminum cooling plate for 18650 battery packs with two configurations: hollow and multi-channel.
- Modeled an 8-cell battery pack in SolidWorks with thermal contact along one face of each battery cell.
- Simulated thermal behavior in ANSYS Fluent and observed improved cooling in the discrete channel configuration.
- Compared temperature distributions and pressure drops to evaluate thermal performance and flow uniformity.

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 channel heatsink & 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 an axial flow pump with helical blades, studying six impeller variations to analyze fluid motion.
- Optimized impeller performance using CFD by comparing pressure and velocity distribution across designs.
- Modeled impeller in Catia and simulated flow using Ansys CFX, targeting compact medical applications.
- Presented findings at LAKSHYA-2K21, a national-level symposium, among top selected student projects.