# Naga Nitish Chamala

Blacksburg, VA 24060 • naga@vt.edu • (248) 212-3516 • naganitish.com • LinkedIN

## Technical Summary

Highly motivated researcher with expertise in CFD, multiphase flows, and thermal analysis, demonstrated through advanced numerical simulations, model development, and experimental validation. Proficient in industry-standard software (ANSYS Fluent/CFX/Icepak, OpenFOAM, StarCCM+) and programming languages (C++, Python). Proven ability to optimize designs and improve performance in complex fluid and thermal systems.

## Education

**Ph.D. in Aerospace and Ocean Engineering | Virginia Tech | Jan 2021 - Dec 2025 | CGPA: 3.7**
Notable Courses: Verification and Validation in Scientific Computations, Data Analysis in Fluid Dynamics, Turbulence Modeling and Simulation

**M.Tech. in Thermal & Fluids | Amrita Vishwa Vidyapeetham | 2017-19 | CGPA: 8.88/10**
Honors: Graduated with Silver Medal for academic excellence.
Notable Courses: Aerodynamics, Computational Methods in Thermal & Fluids, Advanced Heat Transfer, Design of Heat Exchange Equipments

**B.Tech. in Mechanical engineering | JNTUH | 2017**
Honors: Winner of Vishnu Karting Championship, 2016, lead the team as Design Lead.

## Skills

| | | | | | |
|---|---|---|---|---|---|
| C++ | ICEM-CFD | MATLAB | StarCCM+ | ANSYS Fluent | Thermal Analysis |
| Python | ANSYS CFX | Solidworks(CAD) | ANSYS Icepak | OpenFOAM | Machine Learning |
| Bash | Linux | Data Analysis | | | |

## Projects

**CFD Analysis of Through-Chip Microchannel Cooling for 3D Integrated Circuits | Virginia Tech**
- Executed a Computational Fluid Dynamics (CFD) analysis to optimize thermal management in high-power 3D ICs using a novel Through-Chip Microchannel (TCMC) architecture.
- The baseline simulation confirmed the design's capability to dissipate 14.1 kW/cm² while keeping temperature rise below 60 K at a 1 MPa inlet pressure.
- Extended the analysis to square microchannels, which yielded a 12% improvement in thermal performance but resulted in an 18% increase in required pumping power.
- Identified a key performance trade-off, concluding that square channels provide superior heat dissipation where a moderate increase in parasitic energy consumption is acceptable.

**Comparative Thermal Analysis of EV Battery Cold Plates using Ansys Fluent | Virginia Tech**
- Conducted a CFD project in Ansys Fluent to evaluate the thermal-hydraulic performance of convex vs. U-shaped liquid cooling plates for an EV battery.
- Analysis was run at a constant heat flux of 1000 W/m² and a coolant mass flow rate of 0.025 kg/s to simulate a 1C discharge rate.
- The convex structure demonstrated superior hydraulic performance, achieving a 22.7% reduction in pressure drop compared to the U-shaped design while maintaining similar peak temperatures.
- Concluded the convex structure is more efficient for balancing cooling effectiveness with system efficiency, noting a similar design is used in the Lucid Air.

**Optimization of Heat Sink for Cooling a Chip | Amrita Vishwa Vidyapeetham**
- Optimized a heat sink for a chip with 100W/cm^2 by varying base thickness, fin height, fin thickness, number of fins, and fan proximity using Ansys Icepak, obtaining a 15% drop in max temperature.
- Identified and addressed high thermal spread resistance by implementing a vapor chamber and heat pipes.
- Achieved a total of ~40% drop in max temperature using a vapor chamber as a heat spreader.

**Advanced Flow Diagnostics for Multiphase Flows Using Machine Learning | Virginia Tech**
- Pioneered a novel application of RAFT (Recurrent All-Pairs Field Transforms) optical flow to achieve the first-ever simultaneous velocity measurements of both liquid and vapor phases in cloud cavitation.

- Developed and utilized synthetic datasets to fine-tune a deep learning model to overcome the challenge of sparse particle seeding, resulting in a 50% reduction in End-Point Error (EPE).
- The custom-tuned RAFT model outperformed traditional velocimetry algorithms (PIVLab, Farneback), which failed in these conditions, by a factor of 30x.
- Established critical guidelines concluding that high spatial and temporal data resolution are paramount for the successful application of optical flow in complex multiphase systems.

**Comprehensive CFD Modeling of Cloud Cavitation in a Venturi | Virginia Tech**

- Performed a systematic evaluation of CFD methodologies for simulating cloud cavitation in a venturi geometry using OpenFOAM (interPhaseChangeFoam solver with Merkle cavitation model).
- Compared the predictive accuracy of various approaches, including incompressible vs. compressible solvers, 2D RANS vs. 3D RANS, and high-fidelity 3D Large Eddy Simulations (LES).
- Assessed the influence of geometric scaling on flow physics by rigorously benchmarking simulation strategies against experimental observations.
- Identified the key capabilities and critical limitations of current-generation CFD tools in capturing complex, transient cavitation physics.

**Implementing Verification and Validation Framework on an OpenFOAM solver | Virginia Tech**

- Verified and validated an OpenFOAM solver for 2D steady-state incompressible laminar flow using the Roy-Oberkampf framework.
- Estimated the predictive uncertainty of the solver by considering aleatory and epistemic inputs separately.
- Quantified numerical uncertainty using Richardson extrapolation and Roache's Grid Convergence Index (GCI).
- Achieved a model form uncertainty (d) of 0.8472 using the Area Validation Metric and demonstrated the importance of considering all sources of uncertainty.

---

## Experience

**CFD Engineer | Hydro Pneumatic Engineers Pvt Ltd | Hyderabad, India | Aug 2019 - Nov 2020**

- Contributed to the development of an in-house Scramjet engine by optimizing the intake to minimize spillage for operating speeds.
- Significantly improved nozzle design by employing the Method of Characteristics, resulting in a 10% increase in thrust and a 3% reduction in length.
- Optimized fuel injectors position for better combustion efficiency and lower drag.
- All simulations were performed using ICEM-CFD and ANSYS Fluent software.

---

## Conferences

- *Unraveling Cloud Cavitation: Role of Rayleigh-Taylor Instability*, presented at 77th Annual Meeting of the Division of Fluid Dynamics 2024.
- *Numerical uncertainty due to parallelization in an unsteady cloud cavitation*, presented at 76th Annual Meeting of the Division of Fluid Dynamics 2023.
- *A Comparative Study of Transport Equation Models for Prediction of Cloud Cavitation in a Venturi*, presented at The 8th World Congress on Momentum, Heat and Mass Transfer 2023. DOI: 10.11159/icmfht23.121
- *Numerical simulation on flow through a double forward-facing step*, accepted in International Conference on Applied Mechanics and Optimisation 2019 – AIP Proceedings. https://doi.org/10.1063/1.5120212
- *Simulation of Scramjet Combustor using Quasi-1D and 3D CFD models*, presented at 21st Annual CFD Symposium, NAL, 2019

---

## Journal

- *High-Fidelity, Two-Phase Velocimetry in Cloud Cavitation using a Fine-Tuned Recurrent All-Pairs Field Transforms Optical Flow Network*. Submitted to Experiments in Fluids. **Under Review.**
- *Study of Shedding Mechanisms in Cloud Cavitation and the scale effects on dynamics*. Submitted to Journal of Fluid Mechanics. **Under Review.**