

Shrey Rajesh Waghmare

ShreyRajesh.Waghmare@colorado.edu | Boulder-80303 | 303-961-2256 | [LinkedIn](#)

SUMMARY

- 4 years of industry experience in design, development and analysis of mechanical components, leveraging CFD and FEA in a highly collaborative engineering environment.
- Proficient in numerical modelling and high fidelity CFD simulations using MATLAB, Python and ANSYS Fluent, with strong foundation in thermofluids, heat transfer and experimental validation for performance optimization.

EDUCATION

MS. Mechanical Engineering – 3.91/4.0

University of Colorado Boulder

Boulder, Colorado, US

May 2025

B. Tech. Aerospace Engineering

Indian Institute of Space Science and Technology (IIST)

Thiruvananthapuram, Kerala, India

Aug. 2015 – May 2019

SKILLS

- **Application Skills:** SolidWorks/ Catia, ABAQUS, ANSYS - Fluent, STAR-CCM+, LabVIEW, Simulink
- **Programming languages:** Python, MATLAB, C, Shell scripting for HPC, Linux Scripting, LATEX
- **Design/ Analysis:** CAD, DFM & DFA, GD&T, DfR, FMEA, RCA

WORK EXPERIENCE

Research Assistant – Labbe Lab, University of Colorado Boulder

Sept. 2023 – Present

Thesis: Deep learning algorithm for identification of combustion relevant species from infrared (IR) spectra

- Developed ML model pipeline for identification of combustion relevant species from IR spectra
- Automated web scrapping, data management and data labelling using python for processing large dataset comprising 70,000 compounds.
- Demonstrated high model performance with an average F1- Score of 0.9 for efficient chemical detection and speciation.

Project: Microreactor Combustion Optimization

- Optimized a novel microreactor geometry using CFD simulations to stabilize thermodynamic conditions and improve combustion reaction studies.
- Developed high fidelity CFD models using ANSYS-Fluent to analyze compressible, high-speed flow in a confined microreactor environment, ensuring numerical stability and accuracy
- Conducted Design of Experiments (DOE) simulations to optimize reactor choke diameters, nozzle geometries, and reactor lengths, improving reaction control, species transport and thermal efficiency.

Scientist/Engineer – Mechanical Design & Analysis

Indian Space Research Organization (ISRO)

Aug. 2019 – June 2023

- Planned, set up and executed multiphysics simulation using ANSYS Fluent for 100+ pump fed rocket engine components (valves, regulators, turbopump, nozzles, fluid lines etc.), validating model against benchmarked results.
- Conducted failure mode and effect analysis (FMEA) using CFD for defective engine components and subsystems, ensuring reliability and integrity for 35+ engines nominal operations.
- Designed and conducted FEA on engine hardware fixture/supports for mechanical and pressure test.
- Performed numerical simulations using MATLAB to investigate transient phenomena in startup of pump-fed liquid rocket engine, validating results against hot test data for design and qualification.
- Performed root cause analysis (RCA) of hardware failure using FEA and CFD to replicate failure conditions.
- Numerically modelled opening characteristics of pneumatic poppet valve and validated the results experimentally.
- Conducted test campaign of 14 hot tests, to evaluate extreme operating conditions, off-nominal mixture ratios and thrust level conditions for human rating and qualification of liquid rocket engine for India's maiden human spaceflight mission.

Research Intern - National Centre for Combustion Research and Development (NCCRD)

May 2018 – July 2018

Indian Institute of Technology (IIT), Madras

Thesis: Experimental Investigation of stationary flat flames

- Conducted multiphysics analysis on one-dimensional adiabatic planar methane-air flames, analyzing heat transfer and combustion characteristics inlet conditions.
- Designed and fabricated a specialized test apparatus to facilitate experimental testing, ensuring high quality data acquisition for combustion analysis.
- Modelled 3D premixed flat flame using CFD (Fluent) with reduced chemical kinetics, validating predictions through experimental testing.

PROJECTS RELEVANT TO CFD

- Simulated steady, incompressible Navier-Stokes flow in curved 2D channels and 3D pipes using finite element-based computation model in Python/FEniCS, performing a parametric study on pressure drop variations across different bend angles and validating results theoretically. Examined the formation and evolution of Dean vortices in 3D pipes, emphasizing their development along the curvature.
- Developed a custom CFD framework in python/FEniCS to simulate transient flow, including Von Kármán vortex formation, across square, elliptical, and circular cavities. Conducted Lagrangian post-processing using ParaView to analyze vortex dynamics.
- Developed a custom CFD code in python/FEniCS to investigate the impact of heat transfer on the flow field inside a cavity, studying temperature-induced flow variations.

RELEVANT COURSES

Combustion, Computational fluid dynamics (CFD), Finite Element Analysis (FEA), reacting flows, fluid mechanics, rocket propulsion, IC engines, renewable fuels, aerodynamics, statistical machine learning, numerical analysis, gas dynamics, heat transfer, thermodynamics, optimization techniques.