

Abhinav Anand

Research Progress (Spring 2025)

Research title: Computational Fluid Dynamics (CFD) based fine-scale modeling of methane flows around O&G equipment at the Methane Emissions and Technology Evaluation Centre (METEC)

Research overview:

Recent studies estimate that emissions from oil and gas production facilities contribute between 20 to 50% of the total methane emitted in the US; therefore, quantifying and reducing these emissions is crucial for achieving national and international climate goals. Methane quantification depends on both measuring methane concentrations and converting these to emissions through a modeling framework. Currently, simple atmospheric dispersion models are primarily used to quantify emissions, but these emission estimates are highly uncertain when quantifying emissions from complex aerodynamic sources, such as oil and gas facilities. This investigation uses a CFD modeling approach, which can account for aerodynamic complexity but has hitherto not been used to model methane concentrations downwind of a methane release of known rate and compared against in-situ measurements.

Activities (Jan 2024-Present)

Data Collection: Data samples were collected at METEC between February 21, 2024, and July 18, 2024.

Model set-up: CONVERGE CFD was run multiple times (~30 simulations) to find a working model set-up for the experiments. Each model run took a few hours, totaling over 100 hours of computational time utilizing the CU Boulder high-performance computing facility.

Simulation and model evaluation: Multiple CFD runs resulting in 8 final simulations. These simulations are the CFD-based model of methane observations made at the METEC facility on 2 days- 21st March 2024 and 11th July 2024. The modeling was done for distances between 20-70 meters from two different points of release in two separate controlled release experiments, one from a separator and another from a wellhead. The METEC site configuration, measured wind data, and controlled emission rates at the METEC site were used as input for the CONVERGE CFD to model downwind methane concentration. The fractional bias in maximum modeled concentration was under 32%, and the fractional bias in time-averaged mean methane concentration was under 41%.

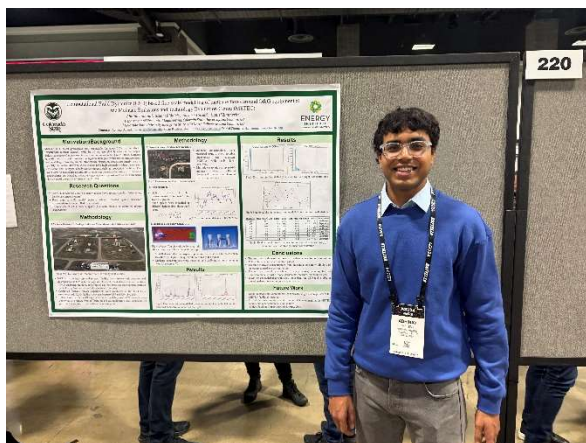


Figure 1 Presenting poster at the American Geophysical Union, 2024, Washington DC, USA

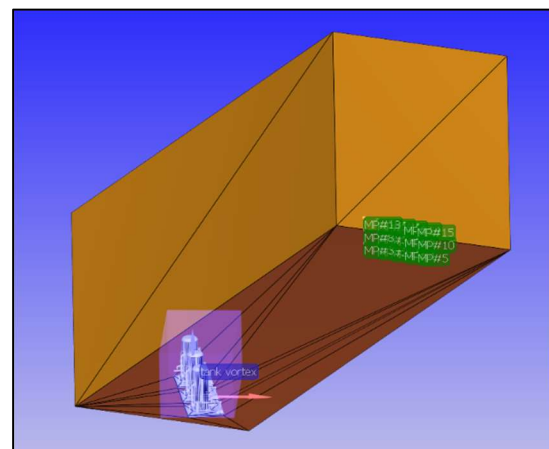


Figure 2 CFD Model set-up for one of the simulations