

CFD Modelling of Pollutant Transport from a Docked Marine Vessel

S Madsen¹, A Zubiaga², H Khawaja¹, G Boiger²

1. UiT The Arctic University of Norway

2. ZHAW Zurich University of Applied Sciences



Abstract

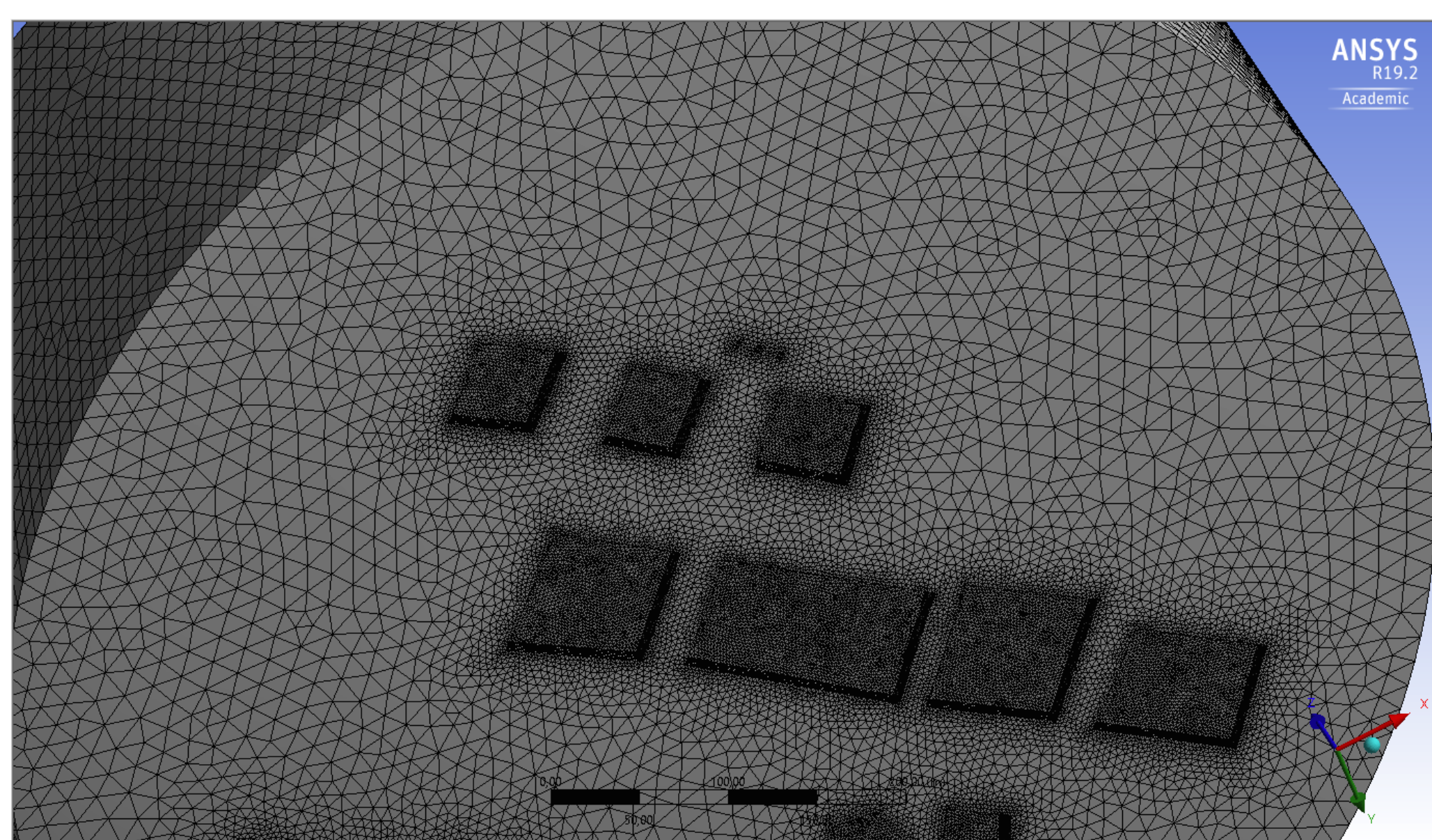
With the increase of the human activities in the Arctic, including land and sea, is resulting in higher release of pollutants to the environment. These pollutants can be categorised as gases released in the air such as carbon dioxide, sulphur dioxide, NO_x, etc., and waste released in seas such as chemical toxins, microplastic, etc.

This work presents a computational fluid dynamics (CFD) study presenting a two-phase Euler model for pollutant transport from a docked marine vessel at the Breivika harbor, Tromsø, Norway. The model is built by collecting the regional GPS data and converting into localized spatial coordinates. ANSYS® fluent is used to solve the CFD simulations including a plume of CO₂ and air in a two-phase Euler model. A free mesh is built for the solution using ANSYS® meshing tools with increased mesh intensity in critical regions. Mesh sensitivity study is conducted to ensure the correctness and reliability of the results. The results are checked qualitatively with a similar OpenFOAM® CFD model and found to be in a good agreement.

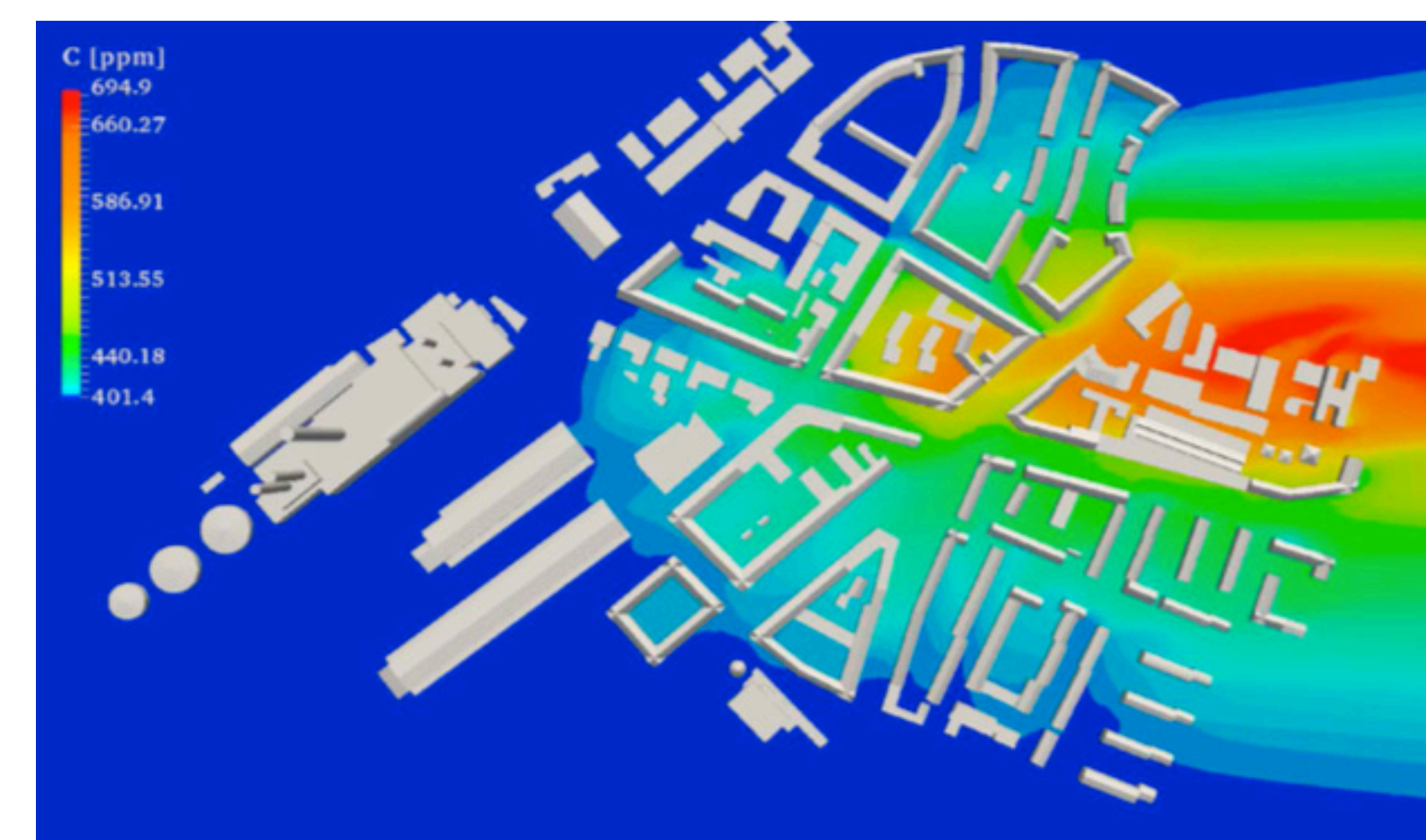
The range of test cases varying the vessel's height, wind velocities and directions are performed. It is found that the impact of plume towards the local buildings reduces with the increase of the vessel's height. Also, the plume rises high and diffuses in the air at lower wind velocities and vice versa.

The study was also in agreement with the observations of the local port authority (Tromsø Havn KF).

ANSYS® Mesh

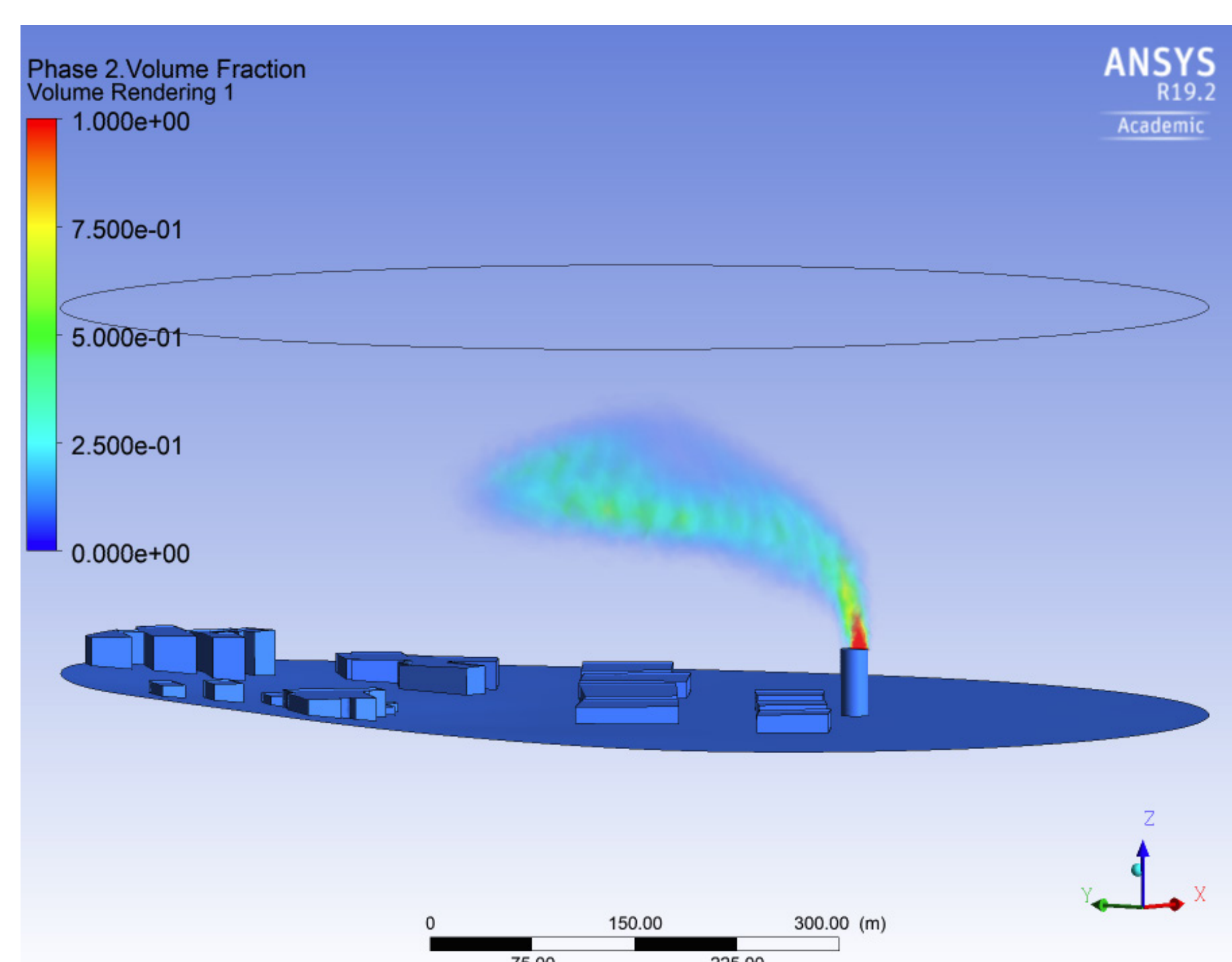


CO₂ concentration in the urban area*

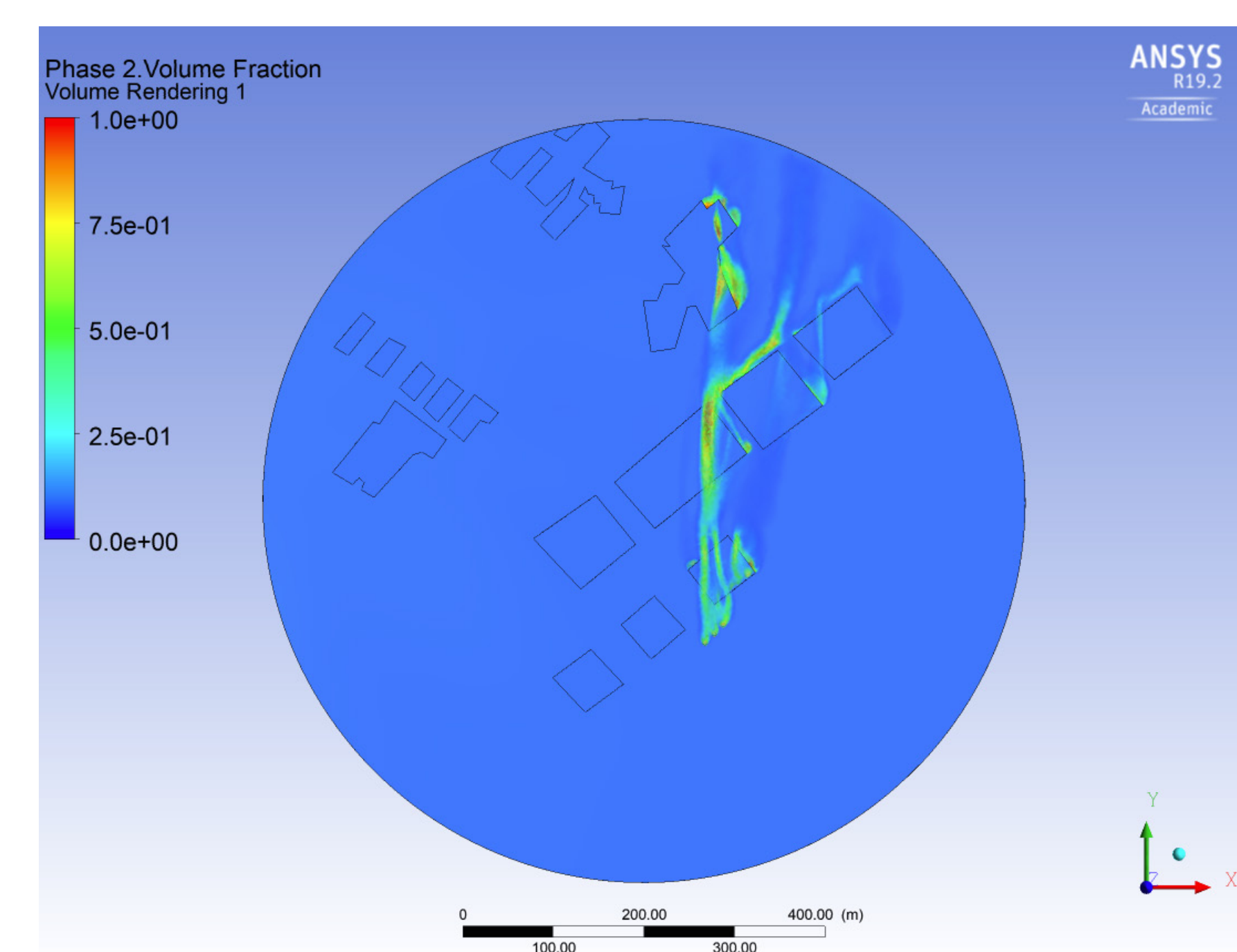


* Toja-Silva, F., Chen, J., Hachinger, S., & Hase, F. (2017). CFD simulation on CO₂ dispersion from urban thermal power plant: Analysis of turbulent Schmidt number and comparison with Gaussian plume model. Journal of Wind Engineering & Industrial Aerodynamics

ANSYS® Simulation Results (Heavy Discharge)



ANSYS® Simulation Results (Around the Buildings)



Conclusion & Future Work

The presented work demonstrated the impact of plume from the docked vessels in Tromsø Breivika harbour. Two key factors were varied namely; wind (velocity & direction) and height of the vessel. It was found that lower wind velocities and taller vessel heights reduces the instantaneous impact of the plume to the local buildings (region).

Contact:

Hassan Khawaja
University of Tromsø,
Tromsø, Norway
hassan.a.khawaja@uit.no