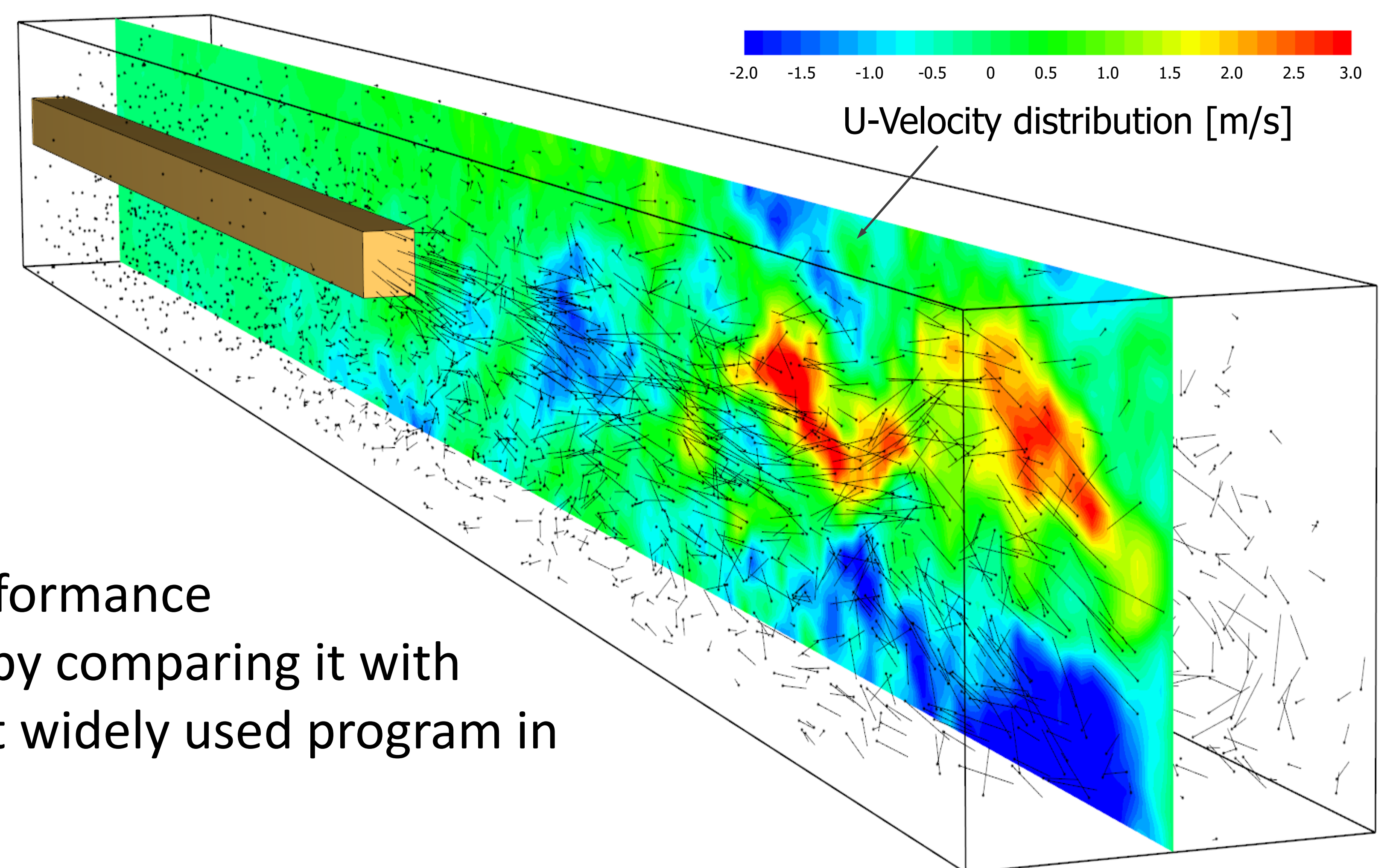


Aliaksei Patsekha, Robert Galler

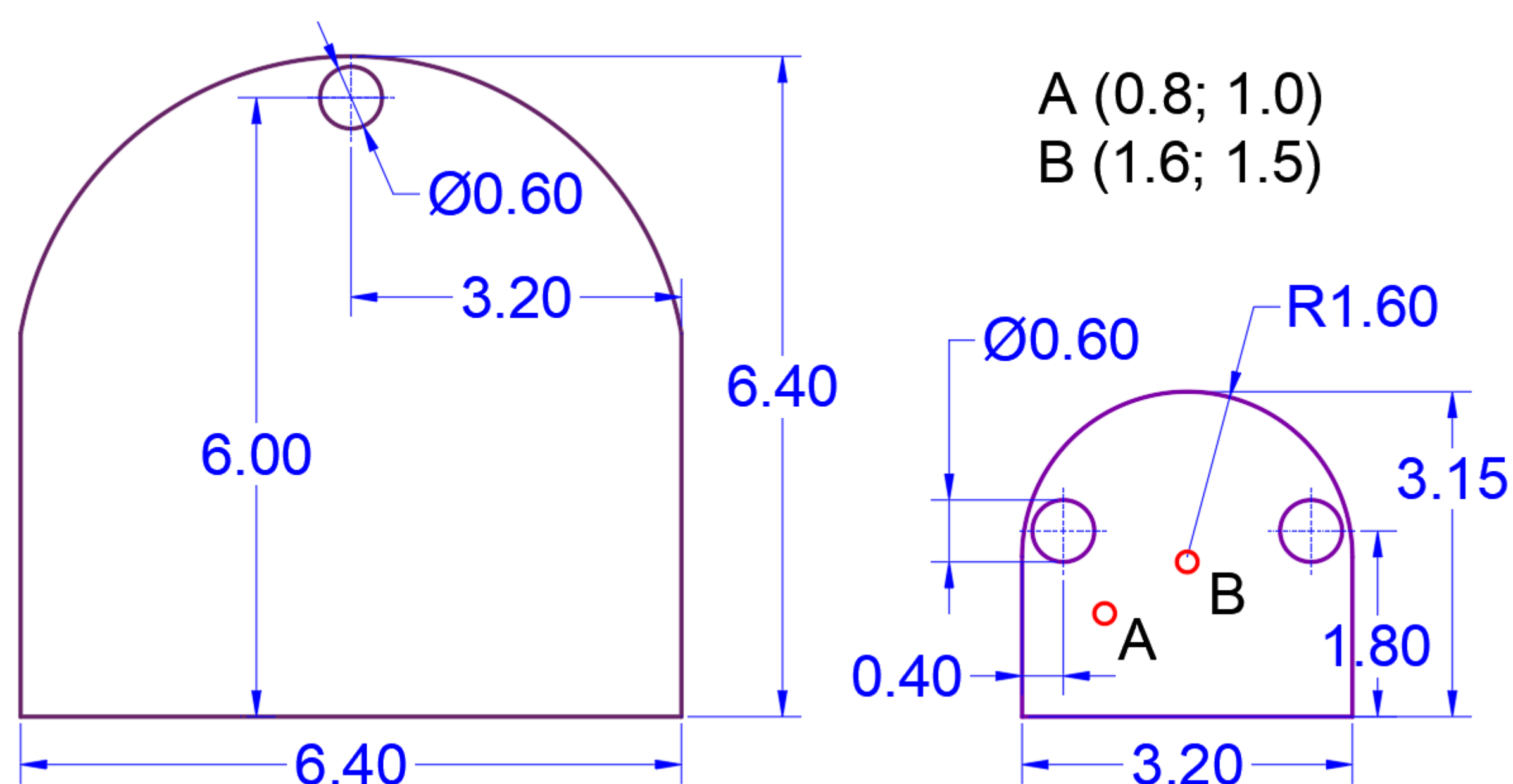
Montanuniversität Leoben, Chair of Subsurface Engineering

Computational fluid dynamics (CFD) models are generally applied to analyse the flow behaviour within an underground facility and determine optimal solutions for the ventilation system arrangement and pollution monitoring.



## Objective

The objective of this study is to assess the performance of the Fire Dynamic Simulator (FDS) software by comparing it with the field study results and outputs of the most widely used program in the field, Ansys Fluent.



(a) Xie et al. (2021)

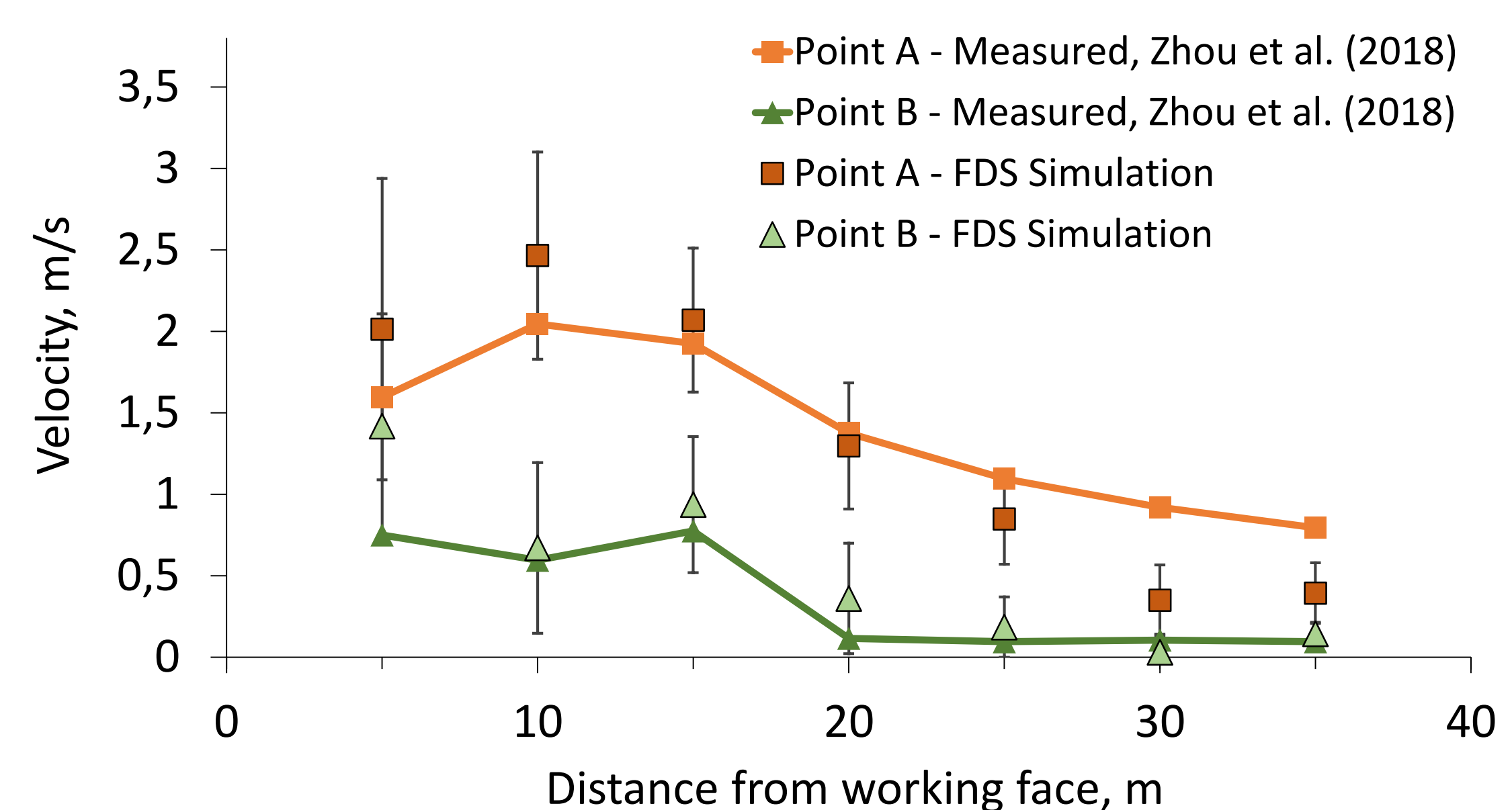
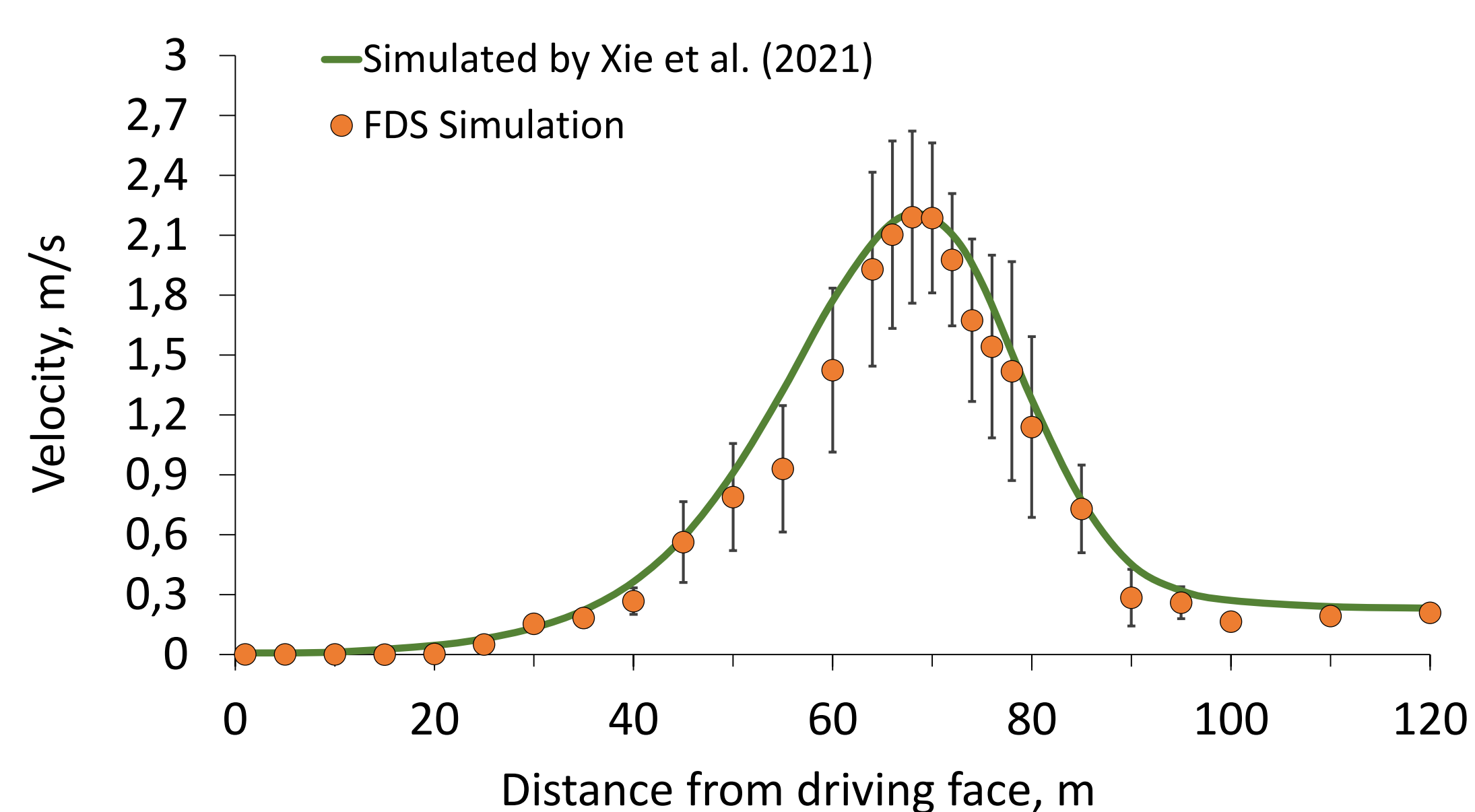
(b) Zhou et al. (2018)

Several investigations with the obtained by Ansys Fluent and verified through field measurements findings are analysed.

The reported tunnel geometry, ventilation conditions and experimental setups are used to design FDS models and predict the airflow pattern and parameters.

## Results

The calculated air velocity values show that FDS provides comparable to Fluent output while certain adjustments to the settings allow obtaining data (magnitude and behaviour trends) that are in good agreement with the experimental results.



## Conclusion and Outlook

FDS can be considered a valuable tool for evaluating the flow distribution in tunnel areas, though further examinations are needed to use it for more complicated problems, in particular for dust dispersion analysis in order to significantly raise health and safety performance in construction and operation of underground infrastructure systems.