RANS Modeling of Turbulent Flow in Compound Open Channels

Supervisor: Philip A. Ott
Email: philip.ott@tum.de

Keywords: compound-channel flow; RANS; turbulence models

1 Background

One encounters shallow flows in open channels both in civilized areas and in the wild. Typical examples that affect our daily lives are sewers, irrigation canals, and developed or artificial rivers for recreational use or transport of goods. A special shallow flow appears in a so-called compound channels. These consist of a main channel (MC) and one or more floodplains (FPs), which differ from the MC by their geometry, bottom roughness or the like, and thus produce a different flow. In times of increasing weather extremes a particularly important case of a compound channel is the activation of retention areas in the event of a flood.
Numerous simulation studies (e.g., [Sibel Kara et al., 2012, Zhihua Xie et al., 2013]) have been conducted to replicate the fully resolved flow field observed in the experiments of [Tominaga and Nezu, 1991], which continue to serve as benchmark cases to this day. A schematic representation of these findings is shown in fig. 1 and fig. 2. As in open-channel flow, longitudinal secondary currents are formed due to turbulent anisotropy. Additionally, the geometry causes the formation of interface vortices in the transition region. This has significant influence on the flow field and shear stresses.

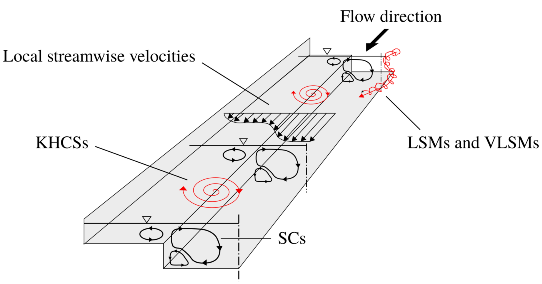

Figure 1: Figure is taken from [Proust and Nikora, 2020].

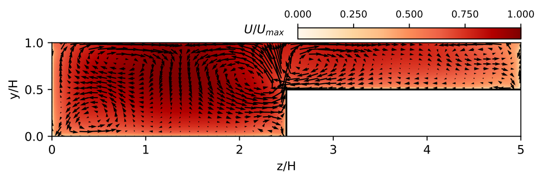

Figure 2: Cross sectional view on lateral vector field and mean streamwise velocity (color).

2 Tasks

The core objective of this project is to perform a Reynolds averaged numerical simulation (RANS) of a prescribed compound open-channel geometry using `OpenFOAM` ([OpenCFD Ltd, 2025b]), and to evaluate the results against experimental data from [Tominaga and Nezu, 1991] and provided large-eddy simulation (LES) reference data. The focus lies on understanding the predictive capabilities of RANS and limitations of turbulence models in representing the complex compound-channel flow.

The specific tasks are structured as follows:

- Conduct a concise literature review on RANS modeling of open-channel and compound-channel flows with a focus on modeling strategies.
- Become familiar with `OpenFOAM` ([OpenCFD Ltd, 2025a, Keysight OpenCFD, 2025]) and establish a working simulation workflow.

- Perform a systematic grid-study.
- Post-process, visualize, and interpret simulation results, focusing on velocity profiles, shear-stress distribution, flow structures, and turbulence characteristics.
- Compare RANS predictions against experimental and LES reference data.

Depending on progress there are many options for additional directions of deeper investigation:

- Turbulence modeling: Implement and compare different turbulence closures (one-equation models, two-equation models, Reynolds-Stress-Transport Modeling (RSTM)). Analyze which model best represents near-wall behavior, free-surface effects, secondary currents, ...
- Error and uncertainty analysis: Quantify deviations using statistical measures (e.g., RMSE, relative error). Discuss potential error sources: numerical resolution, turbulence closure assumptions, experimental uncertainties.
- Boundary condition sensitivity: Explore free-surface modeling assumptions (rigid lid, slip wall, Volume-of-Fluid Method (VOF)). Study the influence of wall roughness and inflow conditions.

3 What you should bring

3.1 Required skills

- Fundamental knowledge of fluid dynamics and turbulence.
- Willingness to work in a Linux environment and to learn simulation workflows in complex frameworks.
- Good communication skills in English or German.
- Basic experience in scripting languages (e.g., Python, MATLAB, R).

3.2 Recommended skills

- Interest in open-channel hydraulics.
- Previous experience with Computational Fluid Dynamics (CFD) concepts or tools, e.g. `OpenFOAM`.
- Experience with High Performance Computing (HPC) or Linux shell scripting.

4 What you will learn / Benefits for the applicant

- Hands-on CFD experience with a popular flow solver (`OpenFOAM`).
- Practical training in numerical simulation, post-processing, and validation of turbulent flows.
- Insights into turbulence modeling strategies and their applicability to environmental hydraulics.
- Skills in HPC and scientific workflow management.
- Exposure to ongoing research in environmental fluid mechanics.

References

[Keysight OpenCFD, 2025] Keysight OpenCFD (2025). Openfoam v2506 release announcement. https://www.openfoam.com/current-release. Accessed: 2025-08-19.

[OpenCFD Ltd, 2025a] OpenCFD Ltd (2025a). Openfoam documentation overview. https://www.openfoam.com/documentation/overview. Accessed: 2025-08-19.

[OpenCFD Ltd, 2025b] OpenCFD Ltd (2025b). Openfoam: The open source cfd toolbox. https://www.openfoam.com/. Accessed: 2025-08-19.

[Proust and Nikora, 2020] Proust, S. and Nikora, V. I. (2020). Compound open-channel flows: effects of transverse currents on the flow structure. *Journal of Fluid Mechanics*, 885.

[Sibel Kara et al., 2012] Sibel Kara, T. Stoesser, and T. Sturm (2012). Turbulence statistics in compound channels with deep and shallow overbank flows. *Journal of Hydraulic Research*, 50:482–493.

[Tominaga and Nezu, 1991] Tominaga, A. and Nezu, I. (1991). Turbulent structure in compound open–channel flows. *Journal of Hydraulic Engineering*, 117(1):21–41.

[Zhihua Xie et al., 2013] Zhihua Xie, B. Lin, and R. Falconer (2013). Large-eddy simulation of the turbulent structure in compound open-channel flows. *Advances in Water Resources*, 53:66–75.