Chair of Aerodynamics and Fluid Mechanics
TUM School of Engineering and Design
Technical University of Munich

Implementation of static mesh refinement in JAX-FLUIDS: A fully-differentiable high-order CFD solver for compressible two-phase flows

Semester / Master's Thesis

Flows with immersed boundaries (e.g., the flow around an airfoil) or material interfaces (e.g., a helium bubble in air, see Fig. 1 C)) have varying spatial resolution requirements. The flow field around the region of interest should be highly refined so that all important flow features can be accurately represented. To increase computational efficiency, regions with less complex dynamics can be resolved much coarser. A simple but effective way for static mesh refinement is a multi-block based strategy, see Fig. 1 A). For example, this refinement strategy is often used in the study of shock-bubble interactions, see Fig. 1 B). I.e., the region in which the shock interacts with the bubble is highly refined compared to the periphery of the computational domain.

In this work, we will implement a multi-block based mesh refinement strategy in our in-house computational fluid dynamics (CFD) code JAX-FLUIDS. JAX-FLUIDS is a differentiable high-performance CFD code written entirely in the JAX Python package and runs on CPU/GPU/TPU. The array-based programming paradigm in JAX poses novel and exciting challenges for CFD practitioners.

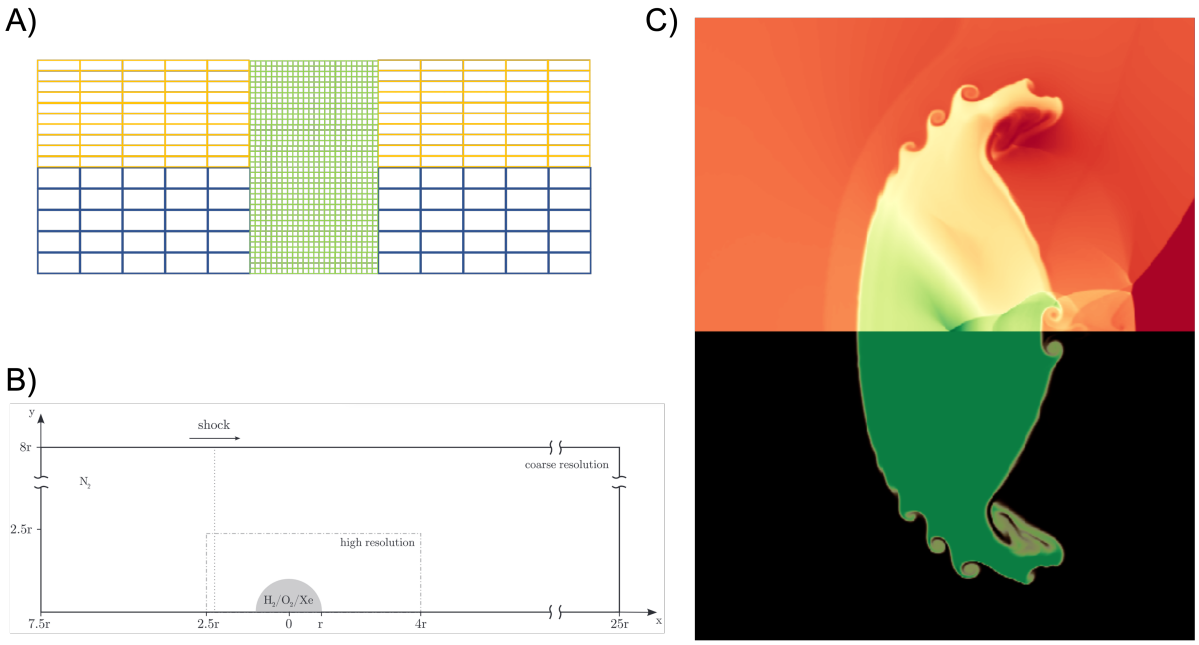

Figure 1: A) Schematic of multi-block based mesh refinement. B) Schematic of the computational domain of a shock-bubble interaction. Taken from Diegelmann et al. 2016. C) Flow field of a shock-bubble interaction.

Tasks

- Familiarize yourself with the JAX-FLUIDS CFD code.
- Implement multi-block mesh refinement.
- Run single and two-phase fluid simulations and analyze the computational performance benefit.
- Optional: Couple the mesh refinement with GPU parallelization of JAX-FLUIDS.

Requirements

- Programming experience in Python.
- Interest in computational fluid dynamics / partial differential equations.
- Beneficial: Applied CFD, Turbulent Flows, Numerical Methods for Conservation Laws.

Contact

Deniz Bezgin deniz.bezgin@tum.de and Aaron Buhendwa aaron.buhendwa@tum.de.