Simulation of Compressible Flows using Python

Niklas Andersson
Division of Fluid Dynamics, Mechanics and Maritime Sciences (M2)

Background and Project Description

The Chalmers-developed Computational Fluid Dynamics (CFD) framework G3D::Flow is a multi-purpose parallel C^{++} code for simulation of compressible fluid flows. G3D::Flow has its roots in a long tradition of development of in-house CFD codes at Chalmers and during the past ten years it has been used in research projects investigating generation of noise in high-speed jets, optimization of compressor blades, highly-resolved simulations of the fluid flow in compressor ducts, noise generated by blade interaction and by turbulent boundary layers in turbomachinery applications, and side load generation during startup of rocket nozzles. The wide range of applications studied using the code means that it has grown extensively over the years. Therefore, a new code has been developed where the core functionality from the G3D::Flow framework has been transferred into a more focused code keeping the numerics and some of the models from G3D::Flow. The new code is written as a Python library and the intention is to use the new code for isolated investigations of numerical techniques and models for compressible flow simulation. This means that the python library will serve as a test bench in the development of, for example, turbulence models and numerics for compressible flow simulation that later can be transferred back into the G3D::Flow framework. There's also an interest to investigate the possibility to do simulations of more fundamental nature such as for example nozzle flows or turbulent jets using the multi-threading capability of Graphics Processing Units (GPU:s).

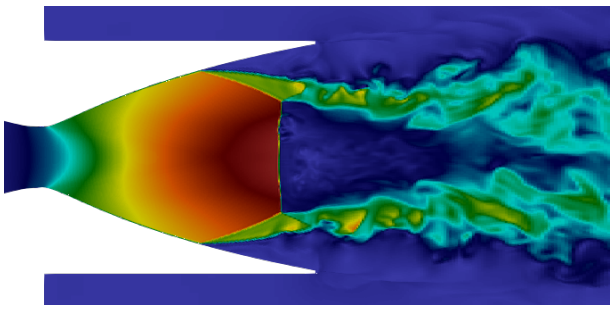

Figure 1: Mach number contours obtained from a simulation of a start-up sequence using the G3D::Flow solver

The proposed bachelor thesis project will be focused on

1. benchmarking of the Python library for canonical compressible flow cases
2. highly resolved simulation of the flow in a convergent-divergent nozzle under non-ideal operating conditions
3. investigation of the capability of GPU:s for speed up of compressible flow simulations

Intended Learning Outcomes

During the proposed bachelor thesis project, the students will

- get to know techniques for simulation of fluid flows in general and techniques specific for numerical simulation of compressible flows
- gain knowledge about the nature of compressible flows and get a better understanding of flow phenomenas such as compression shocks and expansions
- get to experience the development of simulation software

Target Group

The proposed project is suitable for a group of up to four students from any of the following programmes:

- Maskinteknik
- Teknisk fysik
- Teknisk matematik
- Kemiteknik med fysik
- Teknisk design

Pre-requisities

A basic course in fluid mechanics is required and it is meritorious to have taken a course in programming.

Supervision and Examination

Supervisor	Debarshee Ghosh	Division of Fluid Dynamics
Examiner	Niklas Andersson	Division of Fluid Dynamics