



PROBLEM SPECIFICATION

A simplified CFD solver is developed based on the SOLA code [1]. Special attention is paid to calculation efficiency, so the code runs fast for simulations of large models with many grid points. As a test case, flow around a complex geometry is tested with spacial attention to the forces. Therefore, a method for simulations of fluid-structure interactions is developed. The main objectives of the thesis are:

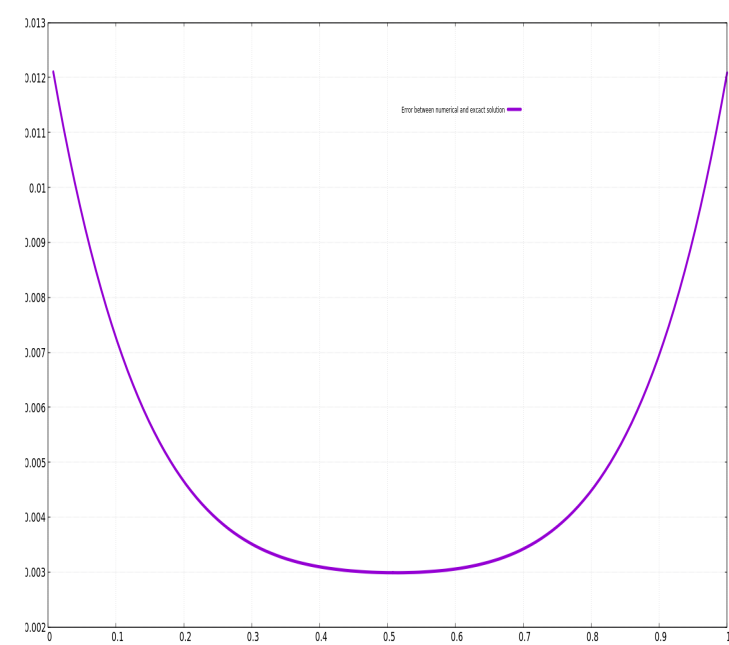
- *Multigrid methods* are developed to reach convergence faster than a iterative scheme.
- *Immersed boundary method* is implemented for simulations of fluid-structure problem.

SOLVER CONSTRUCTION

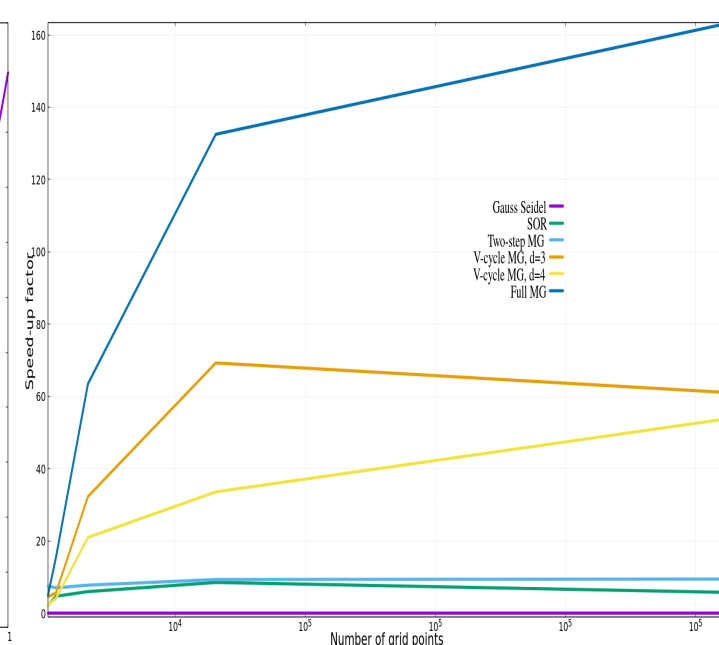
A simplified Navier-Stokes solver for two-dimensional incompressible viscous flow is the base for this master thesis. The solver is constructed by use of:

- A equidistant *staggered grid*.
- The *finite difference method* as spatial discretization, described in [2].
- The *explicit Euler method* for temporal discretization.
- The *projection method* for decoupling the pressure and velocity coupling by [3].
- A method for solving the Poisson equation, future described under METHOD DESCRIPTION.

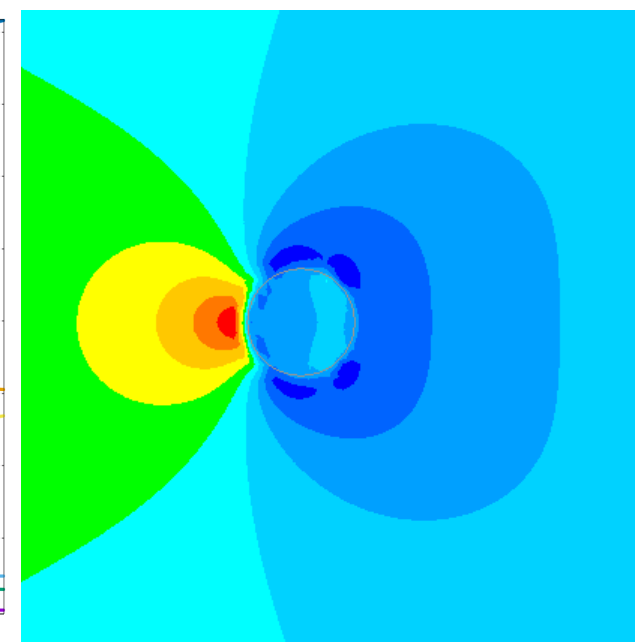
SIMULATIONS



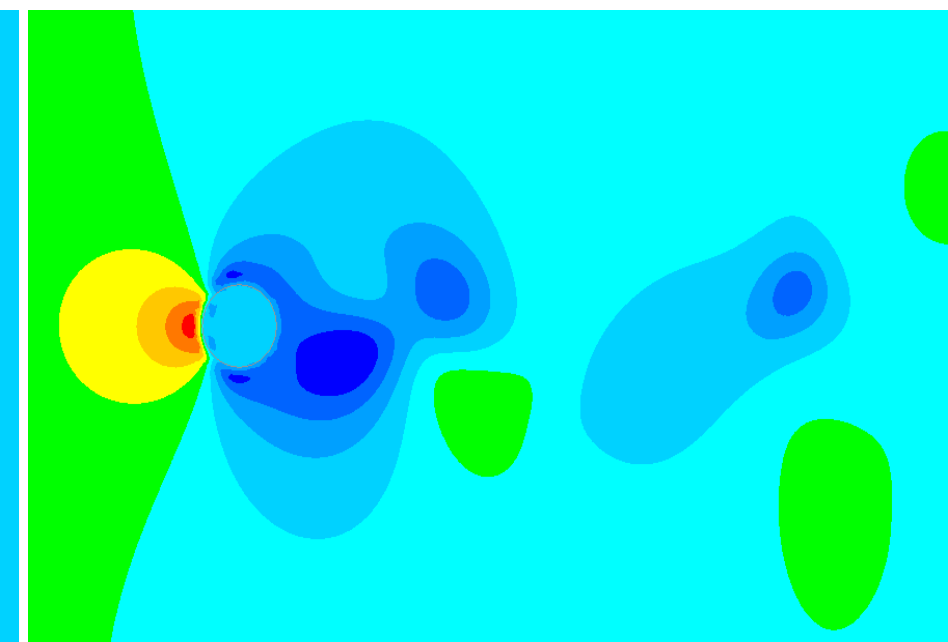
Error in the Poisson solver



Speed-up factor



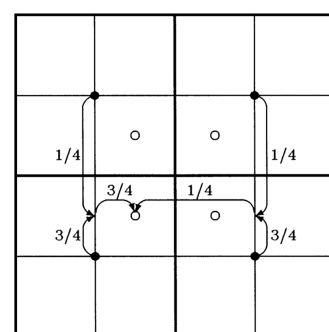
Pressure for Re = 20



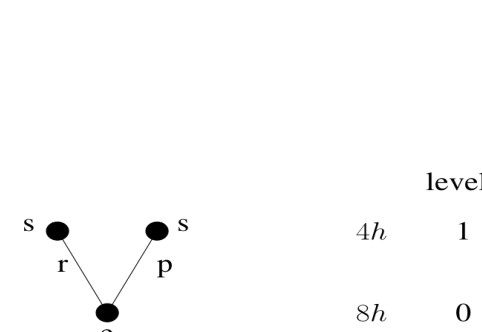
Pressure for Re = 100

METHOD DESCRIPTION

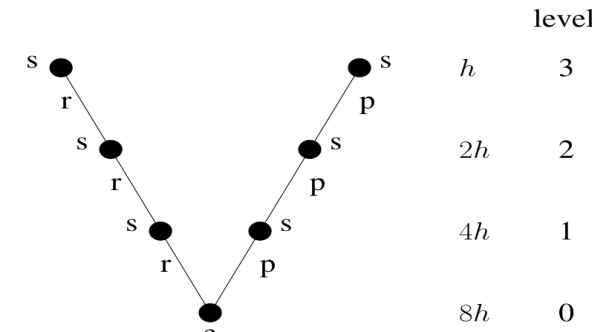
The *multigrid method* for solving elliptic partial differential equations, like the Poisson equation, introduced by Brandt [4] accelerate the convergence of an iterative method, like *Gauss Seidel* which is used here, from $\mathcal{O}(n^2)$ to $\mathcal{O}(n)$, when the partial differential equation is discretized on n grid points. The main purpose of a multigrid method is reducing the different error components by transferring the domain to different grid levels and iterate by a iterative method. To transfer from a fine grid to a coarser grid is called *restriction* and done by a restriction matrix, $I_{2h}^{2h} : \mathcal{R}^n \rightarrow \mathcal{R}^{n/2}$. To transfer from a coarse to a finer grid is called *prolongation* and done by a prolongation matrix, $I_{2h}^h : \mathcal{R}^{n/2} \rightarrow \mathcal{R}^n$. n is number of grid points in one direction of the domain. When transferring, the number of grid points n and the grid spacing h is always changing with a factor of 2. The transferring is always a local averaging or a direct transferring of a cell value. The *prolongation* is preformed by a bi-linear interpolation matrix and the *restricting* is preformed by *injection* (direct transferring of even-numbered fine grid values).



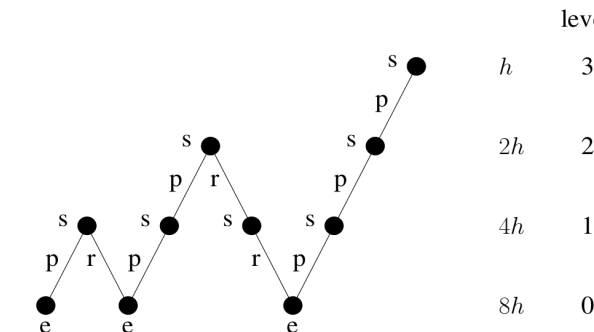
Bilinear interpolation



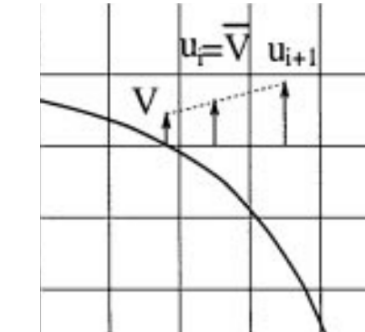
Two-step multigrid



V-cycle multigrid



Full multigrid



Linear interpolation (from [6])

The foundation of the *immersed boundary method* was introduced by Peskin [5]. He provided a method where it is possible to replace a mesh-conforming boundary description with a force replicating the boundary in a fixed computational domain. The governing equations of the immersed boundary is transformed from a Lagrangian representation onto the Eulerian grid of the computational domain as a force. The forces are found by a *direct forcing method*, i.e. *linear interpolation* of the velocities closest to the immersed boundary with respect to the immersed velocity boundary conditions. The immersed pressure boundary condition, $\frac{\partial(\delta p)}{\partial x_n} = 0$ is not corrected, according to [6].

RESULTS AND CONCLUSION

The chosen grid spacing for the test case is $h = 0.0078$, i.e. 128 cells in each direction on a domain $[0:1] \times [0:1]$.

- The efficiency is improved by a speed-up factor, with respect to the *Gauss Seidel method*, equals to 63.5.
- The root-mean square of the error relative to an exact solution is 7.34×10^{-7} , i.e. the accuracy of the resolution in the domain is good enough.
- Force coefficients from the cylinder test: $Re = 20$, $C_L = 0.007$ and $C_D = 2.282$. $Re = 100$, $C_L = 0.333$ and $C_D = 1.499$. The coefficients coincides with existing studies.

C_L for $Re = 20$ should be equals to zero, i.e. there is an small offset in the solver. By analyzing different results, it is possible to see an asymmetry in the error of the Poisson solver, witch probably causes the offset.

For future work:

- Correct the offset in the Poisson solver.
- Implement *weighed restriction* for better performance of the *multigrid method*.
- The temporal discretization method should be evaluated. The *explicit Euler* method needs too small Δt to achieve the optimal efficiency increase.

REFERENCES

- [1] R. Kristofferesen, SOLA - Solution Algorithm for 2D Incompressible Laminar Transient Flow
- [2] F.H. Harlow and J.E. Welch, et al., Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface , 1965.
- [3] J.A. Chorin, Numerical Solution of the Navier-Stokes Equation , 1968.
- [4] A. Brandt, Multi-Level Adaptive Solution to Boundary-Value Problems, 1977.
- [5] C.S. Peskin, Flow Pattern Around Heart Valves: A Numerical Method, in Journal of Computational Physics , 1972.
- [6] E.A. Fadlun, R. Verzicco, P. Orlandi and J. Mohd-Yusof, Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations , 2000.