Mechanics of flag – computational aspects of fluid-structure interaction problems

The project is concerned with computational methods used for computer simulations of fluid-structure interaction problems. Phenomena of this type are commonly observed in everyday life - an example might be motion of a flag or bending of a tree in the wind. Technically important examples are vibrations of aircraft wings, wind turbine blades, bridges or high-rise buildings due to the air flow. The correct solution to the fluid-structure interaction problem is often necessary for desing and miscalculations could be catastrophic. One of examples is collapse of the bridge in Tacoma in 1940 due to vibrations caused by wind. In the case of aircraft wings or helicopter rotor blades self-oscillation can occur (flutter) leading to the rapid destruction of the structure. Models of fluid-structure interaction have also applications in biomechanics, the most popular example is the flow of blood in the elastic blood vessels. Studies of the biomechanical flows are important in the design of medical devices (eg. stents).

In the project, computational methods for fluid-structure interaction will be studied. Due to the increasing complexity of the actual models, it is necessary to use more accurate simulation methods, which also leads to an increasing computational complexity. This generates a need for using high performance computers, and thus development of special computational methods is important to optimal usage of the available resources. Architecture of high performance computers makes it difficult to apply the algorithms used in smaller problems. In contrast to standard personal computers, high-performance computers used for scientific computations don't have shared memory, which makes exchange of information between different processors expensive and requiers special procedures. An important feature of an algorithm is its scalability – the ability to maintain productivity of software with simultaneus increasing size of the problem and the number of processors (size of problem per processor remains constant).

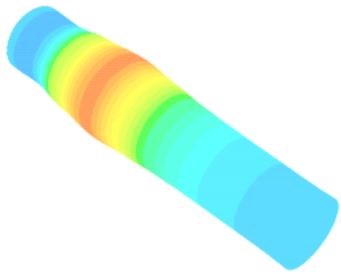

Figure 1: Propagation of a pressure wave through an incompressible fluid in a flexible tube (source: Wikipedia)

Coupling multiple physical models is known as multiphisics problems. In fluid structure interaction, there are two different phases - one described by Navier-Stokes equations (fluid) and the other by the equations of elasticity (structure). Combining all those equations, there are coupling conditions on the boundary between two phases - force equilibrium and velocity continuity. By combining all those equations, we obtain a complete system of equations describing the dynamics of the system.

After discretizing equations describing the model, a system of equations is obtained, linear for linear problems, nonlinear otherwise. Applying the Newton method to the nonlinear equations leads to a system of linear equations. The number of unknowns is proportional to the number of discrete points at which the approximate solution is determined, hence to obtain the required accuracy it is necessary to solve large system of equations (millions or more uknowns). At such sizes it is not possible to use direct solution methods due of their memory complexity. Iterative methods seem a natural choice, but to get sufficiently rapid convergence, they require construction of an aproximate inverse of a matrix - so-called preconditioner. The method is said to be optimal if the number of iterations needed to obtain requied accuracy does not depend on problem parameters or number of equations.

The goal of the project is implementation and testing of scalable computational methods for fluid-structure interaction problems. Development of an adequate program will allow testing of effectiveness experimentally and compare it with literature. Theoretical studies will allow estimation of the convergence rate of the used methods, in particular to test efficiency depending on the parameters of the problem.