Automatic Design Optimization using Fluent and Epogy Softwares in Linux cluster environment.

Johan Åkerholm, Björn Jernström and Kenneth Eriksson*

Process Flow Ltd Oy, Puolalanpuisto 1 b A 21, FIN-20100 Turku, Finland

In this paper we will present results of a study, where two commercial software packages have been used to minimise head losses caused by an obstacle inside a channel (see Figure 1). The main goal with this presentation is not been to study this test problem but to integrate an optimisation software together with a general-purpose CFD software and thus to develop a design tool, which can be utilised in various CFD modelling problems we have in industry. The Epogy software [1] has been used in optimisation and the fluid flow has been modelled by using a commercial CFD software FLUENT [2].

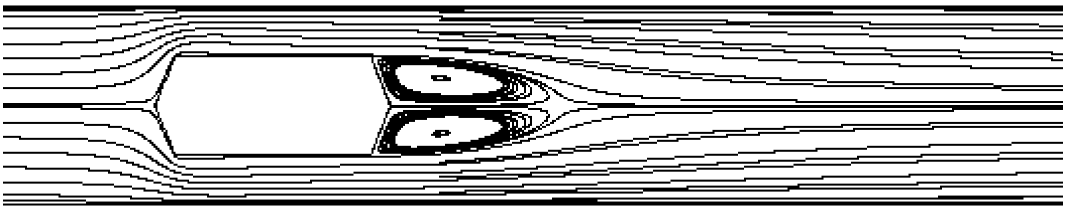

Figure 1. Initial geometry, Δp = 1600 Pa.

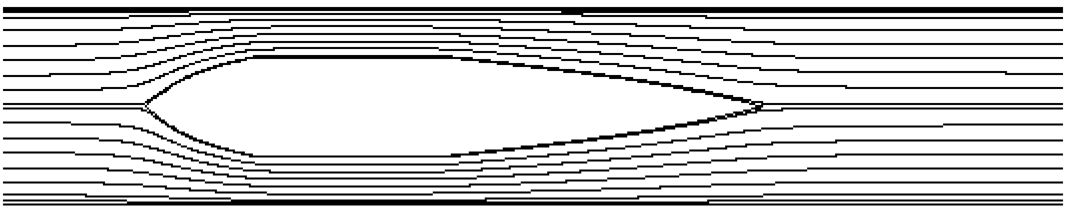

Figure 2. Optimised geometry, Δp = 260 Pa.

The presentation shows how the programs are coupled and what the possibilities, strengths and weaknesses for similar design optimisation problems are. The main aim is to have a flexible process optimisation tool, in which the new features of the continuously developing commercial codes can be utilized.

Fluent is the world leading CFD software house and the software can be used in CFD-simulations of almost any process from combustion to turbo machinery simulations.

Epogy provides possibilities to optimise all sorts of problems including different in-house codes due to its very effective coupling possibilities. The state of the art optimisation engine in Epogy optimises the use of different optimisation algorithms during the solution. This speeds up the otherwise very time expensive CFD optimisation. There is still the possibility to choose the optimisation algorithm manually, if one algorithm is known to be efficient for a certain problem.

A gradient-based method and a Genetic Algorithm were used in the optimisation of the test problem. In this case, the gradient-based method found only a local minimum close to an initial design and reduced a cost function value just a little. But, GA converged to a much better solution giving a remarkable smaller cost function value. The converged solution (by using GA) is given in Figure 2.

The CFD-analyses where carried out in parallel Fluent in order to reduce the calculation times. Fluent was run in a PC-cluster, with Linux as operating system. The calculation times can

* E-mails: Johan.Akerholm@processflow.fi, Bjorn.Jernstrom@processflow.fi, Kenneth.Eriksson@processflow.fi

be reduced almost linearly with the number of used computational nodes. More detailed discussion concerning the test simulations are presented below.

Test case description

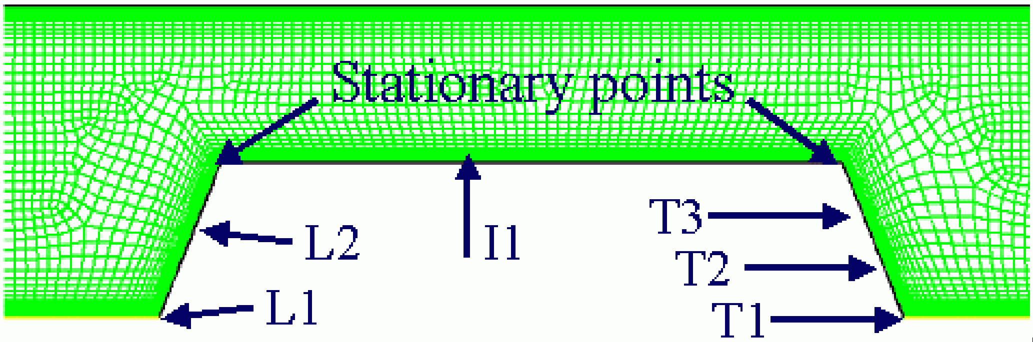

Figure 3. The lay-out of the obstacle is defined by three NURBS-curves. The leading edge is defined by parameters L1 and L2 and on of the stationary points. The intermediate edge is defined by parameter I1 and both stationary points and the trailing edge is defined by parameters T1, T2, T3 and on of the stationary points.

The calculation domain was 2D, 23 m long and 2 m high. The fluid in the simulations was water, which had a uniform velocity of 1 m/s at the inlet. The calculation grid was non-consistent, the original cell count was 10 600 and the number of cells for the optimized shape was 15 100.

The simulations were carried out with parallel CFD analyzes with CPU:s (about 700 MHz processors) running with the Linux operating system. The optimization was carried out with a genetic algorithm, which iterated 10 populations with three parents and a step size of 0.02 (90 CFD runs), and after that with a SQP algorithm for 14 iterations with the step size 0.001 (170 CFD runs). The calculation time was about 60 h and the head loss in the calculation domain dropped from 1600 Pa to 260 Pa, i.e. with 84 %.

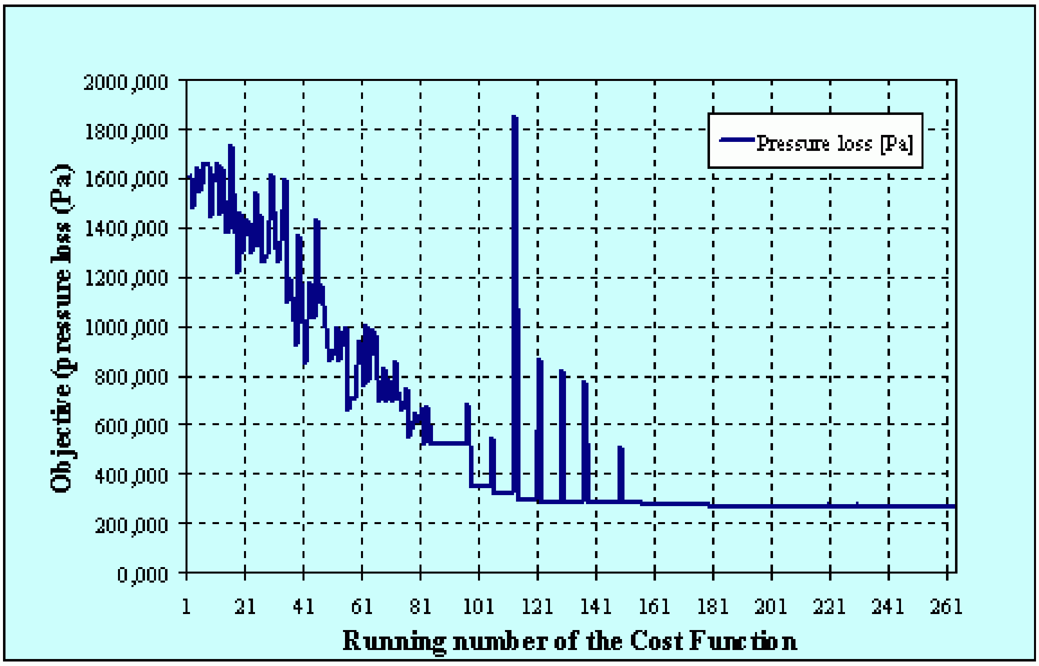

Figure 4. The figure shows how the value of the objective function has changed during the optimisation run.

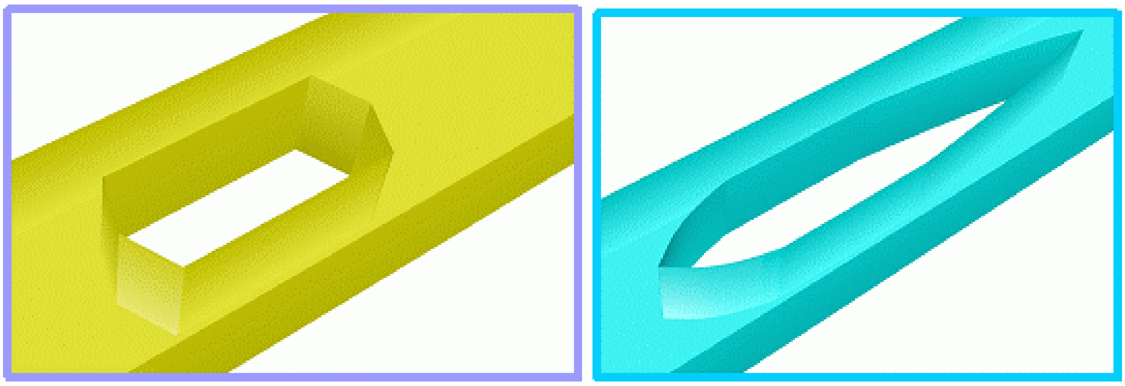

Figure 5. The original shape of the obstacle is shown to the left and to the right the optimised shape is illustrated.

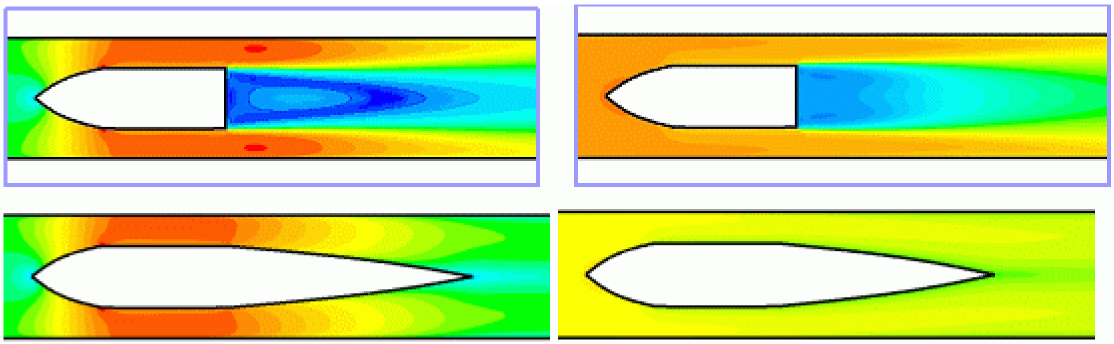

Figure 6. The two figures to the left show the velocity magnitude taken from a parameter scan. The corresponding total pressures are shown to the right. The figure illustrates the sudden decrease in head loss as the flow attaches to the trailing edge.

References

1. http://www.synaps-inc.com/pointer.html
2. http://www.fluent.com/