A Study of Flow inside a Centrifugal Pump: High Performance Numerical Simulations Using Graphics Cards and Experiments

Alessandro Nocente, Department of EPT, NTNU, Trondheim, Norway

Daniel Jasiński, Simflow, Warsaw, Poland

Tufan Arslan, IT Division, NTNU, Trondheim, Norway

Torbjørn K. Nielsen, Department of EPT, NTNU, Trondheim, Norway

Long Abstract

Introduction

Computational Fluid Dynamics (CFD) simulations are nowadays very accurate in predicting flow conditions in turbomachinery. The development and diffusion of CFD codes and the growth of available computer power has hugely increased the use of CFD in turbomachinery [1,2]. It is less expensive and less time consuming than an experimental investigation, and allows to test new parts design before the actual realization of a prototype. Less time-consuming simulations are to be preferred if they are able to give a satisfactory level of accuracy. The reduced simulation time for a simulation on GPU cards, can free resources which can be used to investigate deeper the internal flow by mean of a more accurate simulation (i.e. a finer mesh for the same domain). In this work, the pump is a three-staged centrifugal pump with closed impeller and a vaned diffuser specifically designed for offshore produced water treatment. It processes a dispersed phase flow which requires the least possible shear and a proper internal circulation [3] in order to avoid the use of chemicals [4]; hence the importance to obtain data on the velocity and pressure fields inside diffuser channels.

1. Methods

The open-source software RapidCFD [5] has been used. The code is based on OpenFOAM solvers which are running on GPU cards rather than CPUs. The performance gain of GPUs over CPUs has been studied for the simulated case which is a steady three dimensional (3D) flow past the diffuser channels of a centrifugal pump. MRF approach is used to simulate steady flow.

The considered domain is one of the three stages, since each has exactly the same design. The assumption is that each of the stages contributes for one third of the head and the torque. The return vanes between consecutive stages are not part of the calculation because of computational resources, and the results demonstrate that they have negligible influence on head or efficiency. In the first part of the work, simulations are carried out at the best efficiency point (BEP), single-phase flow and realizable k-e turbulence model. The simulations have been repeated for several different mass flows and the results for integral quantities were compared with the data from the manufacturer. On the other hand, preliminary LDV tests showing the flow field inside one diffuser passage are available as seen in Fig. 1. Those results will be used to validate the numerical results.

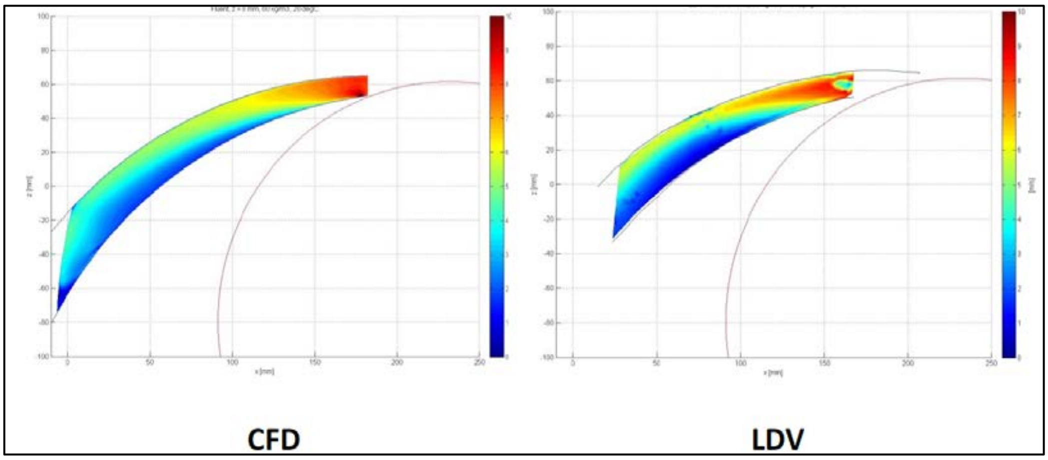

Figure 1. Mean velocity magnitudes in mid-section of the diffuser: CFD (left), LDV (right)

2. Results

The comparison at different discharge levels shows that the steady model that a steady model can approximate within 4% error the real flow characteristics (Fig. 2). The steady state simulation results are then acceptable and the minor computational effort needed makes it attractive to be used for further investigations on the same pump which will involve two-phase flow studies on the same machine.

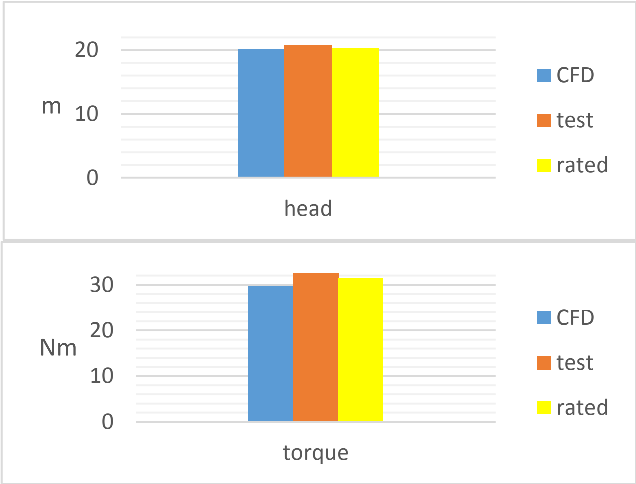

Figure 2. The pump head (top) and the torque (bottom) for BEP (60 m^3/h)

This work is also aiming to show that the new hardware platforms can be used for simulations at the industry and they will be more important for future of CFD. The speed-up gained from GPU is showed in Fig. 3.

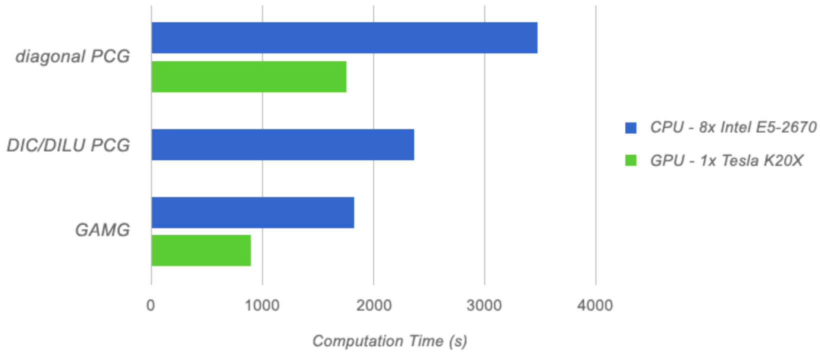

Figure 3. Performance of GPU versus CPU with different matrix solvers.

References

[1] Eric Dick, Jan Vierendeels, Sven Serbruyns, and John Vande Voorde. Performance prediction of centrifugal pumps with CFD-tools. Task quarterly, 5(4):579–594, 2001.

[2] John Vande Voorde, Erik Dick, Jan Vierendeels, and S Serbruyns. Performance prediction of centrifugal pumps with steady and unsteady cfd-methods. In 4th International on Advances in Fluid Mechanics, pages 559–568. WIT Press, 2002.

[3] Mark J. van der Zande and W.M.G.T. van den Broek. Break-up of oil droplets in the production system. page 7pp, Houston, TX, USA, 1998. ASME, Fairfield, NJ, United States.

[4] Johan Sjöblom, Narve Aske, Inge Harald Auflem, ystein Brandal, Trond Erik Havre, ystein Sther, ArildWestvik, Einar Eng Johnsen, and Harald Kallevik. Our current understanding of water-in-crude oil emulsions.: Recent characterization techniques and high pressure performance. Advances in Colloid and Interface Science, 100102(0):399 – 473, 2003.

[5] RapidCFD, https://sim-flow.com/rapid-cfd-gpu/