A NUMERICAL STUDY ON A 4KW CENTRIFUGAL PUMP USING COMMERCIAL CFD CODE AND OPENFOAM

MYUNGSUNG LEE[1], JOO-HAN KIM[2]
[1]*Korea Electronics Technology Institute, ms.lee@keti.re.kr*
[2]*Korea Electronics Technology Institute, kimjh@keti.re.kr*

Keywords: *Turbomachinery, Centrifugal Pump, Impeller, Blade, Volute Casing, Tongue, Cutoff*

In the present study, the Navier-Stokes equation for three-dimensional flow in a 4kW single-suction volute-type centrifugal pump (See Fig. 1) was numerically simulated by using CFX (commercial CFD code) and OpenFOAM [1]. The two equations 'k-epsilon' was adopted for turbulence model in the simulation. The multi-reference frame technique was used, where the impeller was in the rotating reference frame, and the volute casing was in the fixed reference frame, and they are related to each other through the frame change method 'Frozen Rotor'. In this analysis, the definition of geometry covers the five pump sections of inlet pipe, curved pipe, rotating impeller, volute casing, and outlet pipe, which are connected together as shown in Fig. 2.

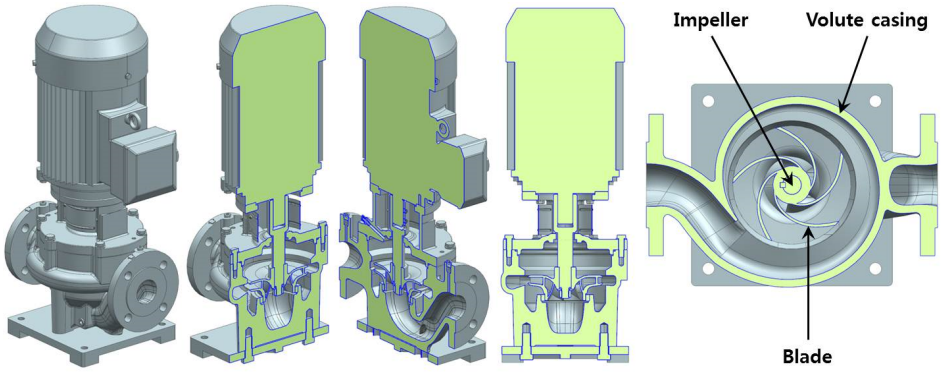

Figure 1: Centrifugal pump assembly showing impeller and volute casing

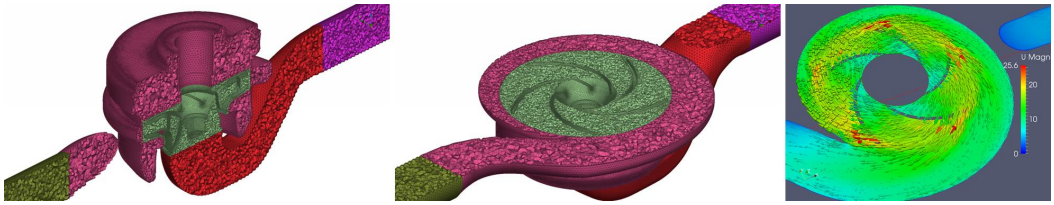

Figure 2: Computational meshes and predicted flow field in centrifugal pump

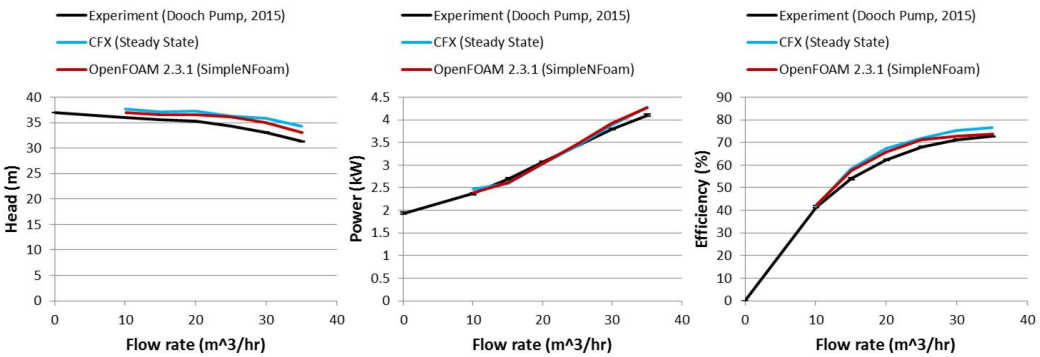

Figure 3: Comparisons of overall pump performance between experiment and simulation

In order to check the capability of the numerical model, its predictions were compared to the experimental data, which include head, power, and efficiency with variation of flow rate. As shown in Fig. 3, the numerical predictions are in good agreement with the experiment.
The 3-D unsteady flow behaviour will be presented with the pimpleDyMFoam solver to investigate the secondary flow along the volute casing and the interaction between impeller blade and volute cutoff. Sensitive analyses of the numerical model also will be performed in order to impose appropriate parameters regarding grid density, time step size, and turbulence model.

Acknowledgements
The authors thank all those involved in the organisation of OFW11 and to all the contributors that will enrich this event.

References

[1] OpenFOAM: The Open Source CFD Toolbox. User Guide Version 2.3.1, OpenCFD Limited, Dec. 2014.