Transactions of the Korean Nuclear Society Virtual Autumn Meeting
October 21-22, 2021

Numerical Calculation of Forcing Functions of Turbulence Induced Vibration by the RCP of APR1400

S.W. Kim [a*], K.W. Yi [a]
[a]*Fluid System Engineering Department, NSSS, KEPCO E&C, 111, 989 Beon-gil, Daeduck-daero, Daejeon 34057*
[*]*Corresponding author: sangwonkim@kepco-enc.com*

1. Introduction

The pressure pulsations induced by rotation of the Reactor Coolant Pump (RCP) impeller of APR1400 is one of the main causes for vibration-induced load. These pressure disturbance components, that is, periodic pressure fluctuations propagate through discharge leg of the RCP, leading to structural damage from fatigue [1]. It is thereby of importance to better understand the pressure fluctuations and the forcing functions.

A numerical calculation was conducted in this study to obtain the pressure fluctuations in the RCP discharge leg. The pressure fluctuations were transformed into Power Spectral Density (PSD) in order to be utilized as turbulence-induced load, that is, forcing function and compared with the design data to validate the methodology used in the calculation.

2. Methods and Results

Numerical simulations of the RCP flow paths including discharge leg were carried out. Steady Reynolds-Averaged Navier-Stokes (RANS) simulation was firstly conducted using Moving Reference Frame method (MRF) to be input as initial field velocity distribution, turbulence length scale and so on. Large Eddy Simulation (LES) with rigid body motion method was then conducted in order to obtain pressure fluctuation data under unsteady condition. The numerical simulation procedure was carried out using commercial flow analysis software, STAR-CCM+.

2.1 Numerical Analysis Model

The geometry model of numerical simulation is the RCP of APR1400, which is a vertical single-stage centrifugal pump. Reactor coolant flows vertically from below into the pump with 6-blade impellers and 11-blade guide vanes. The geometry of fluid domain for the RCP is visualized in Fig. 1.

The pressure fluctuations can be obtained using unsteady-state calculation, that is, LES methodology with high computational expense, but this expense can be reduced by providing appropriate initial condition [2]. The steady-state calculation was firstly performed and provided as initial condition for unsteady-state calculation. MRF method was used for steady RANS to approximate the effect of a rotating structure on the surrounding flow field without moving grids. Realizable k-ε two-layer model with good convergence was used for turbulence model. With regard to the unsteady LES, rigid body motion that involves mesh motion was used with Wall-Adapting Local-Eddy viscosity (WALE) model that automatically gives accurate scaling at walls without requiring any form of wall-damping, which can be universally applicable with reasonable computational resources. The numerical analysis models used in this study are described in Table 1.

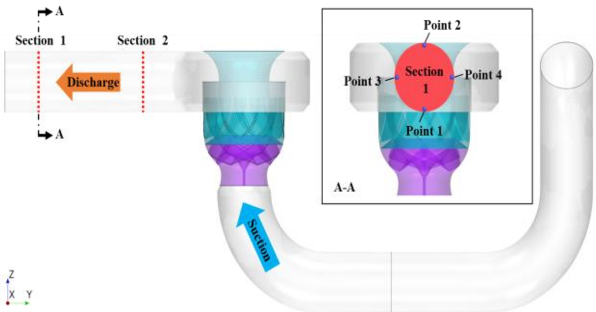

Fig. 1. Geometry of the RCP fluid domain.

Table 1: Numerical analysis model

	RANS	LES
Turbulence Model	Realizable k-ε Two-Layer	WALE
Time Domain	Steady	Transient (Unsteady)
Impeller Motion	MRF	Rigid Body Motion

2.2 Grid and Boundary Conditions

Such a complex structure as the RCP fluid model requires sufficient number of grids to assure grid independence. Fig. 2 shows the total pressure coefficient with grid sensitivity, indicating that the grids with highest number of cells over 20 million is reliable. Since polyhedral grid has higher accuracy than tetrahedral grid in order to obtain accurate solutions for complicated shapes, polyhedral grids amounting 24 million were adopted for the unsteady calculation, with a part of grids refined in the impeller and especially in the discharge leg.

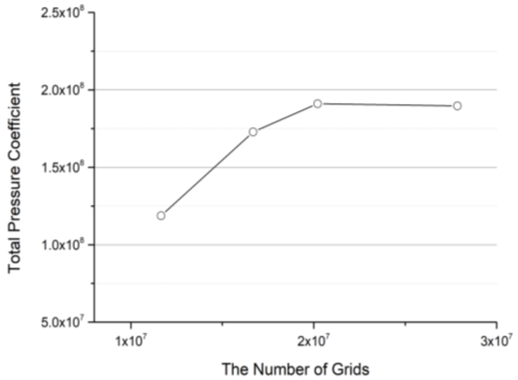

Fig. 2. Grid dependency test.

Pressure inlet and mass flow outlet were set as boundary conditions, considering physical meaning for turbomachinery flow simulation. The operating pressure, temperature and flow rate are considered as the RCP rated condition, 15.24 MPa, 290.56 ℃ and 7.67 m^3/s (5740.73 kg/s), respectively. Segregated fluid isothermal model was applied, assuming temperature variations are small and negligible [3]. The LES calculation was initialized with the steady RANS solution with the time step set to 1.4E−4 s. Considering the impeller rotational speed (1190 rpm), this time step is small enough to get time resolution for the grid movement and the dynamic analysis. The residual was set to below 1E-4, which is an acceptable value for each time step. The last 3 impeller revolutions were used to obtain the pressure fluctuation data out of 5 impeller revolutions, that is, 9000 time steps calculated.

2.3 Results and Discussion

Fig. 3 shows the pressure distribution for impeller center section plane, indicating that pressure is increased through the impeller since the kinetic energy of the rotating impeller is added to the flow and transformed into the pressure energy by the diffuser.

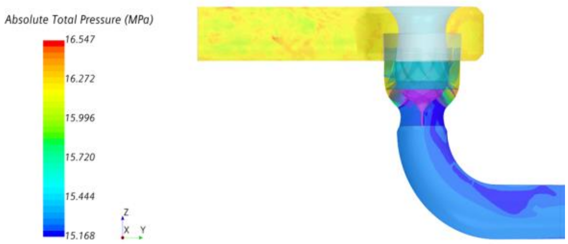

Fig. 3. Pressure distribution on the impeller center section plane.

Total 8 measuring points were selected to obtain pressure fluctuation as depicted in Fig. 1, with 4 points respectively aligned on 2 measuring sections along the discharge path. Random turbulence-induced vibration such as pressure fluctuation is not deterministic in time window, so PSD was applied in order to obtain the time-independent result with amplitude units of squared pressure per frequency and used as forcing functions. Figs. 4 and 5 show the PSD distribution for pressure fluctuation on sections 1 and 2. In Fig. 4, PSD design data at Core Support Barrel (CSB) of APR1400, to which the measuring points in section 1 are nearest, were compared with the numerical result of the PSD distribution in section 1. The PSDs from the numerical result were calculated slightly higher than the design data in the low frequency region below 200 Hz, but showed a good agreement with similar trends.

The PSD distributions in sections 1 and 2 were also following the similar trend but the peak value of section 2 were 225E2 kPa^2/Hz, twice higher than that of section 1, 114E2 kPa^2/Hz.

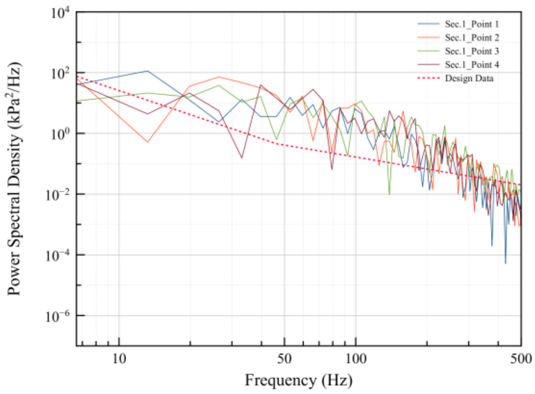

Fig. 4. Power spectral density distribution for pressure fluctuation on the RCP discharge leg in Section 1.

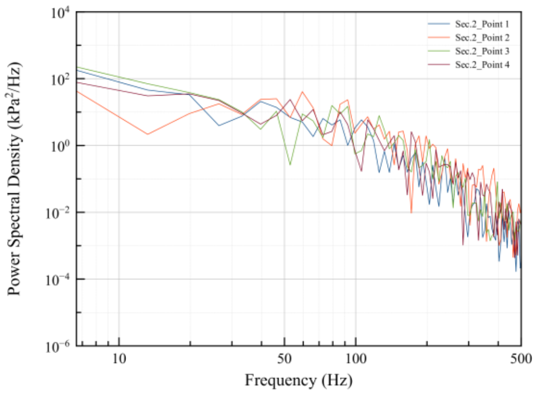

Fig. 5. Power spectral density distribution for pressure fluctuation on the RCP discharge leg in Section 2.

3. Conclusions

A numerical simulation of the unsteady flow in the RCP of APR1400 was carried out. The pressure distribution was obtained through the steady RANS calculation. The pressure fluctuations were subsequently obtained by utilizing the steady results as initial condition to the LES calculation. The results showed that the PSD distributions, that is, forcing functions were similar in Figs. 4 and 5 but with different peaks. The result was compared with the design data of

APR1400 and showed a good agreement. Although the peaking value at RCP blade passing frequency has not been obtained, the results presented in this study may give guidelines to the numerical analysis of RCP induced pressure fluctuations, associated design data generation and investigation of forcing functions for further optimal design of the RCP.

REFERENCES

[1] A. G. Backman, M. J. Cervantes, A. R. Zobeiri, M. Farhat, CFD Validation of Pressure Fluctuations in a Pump Turbine, Luleå University of Technology, 2008.
[2] R. B. Tajadura, S. V. Suárez, J. P. H. Cruz, C. S. Morros, Numerical Calculation of Pressure Fluctuations in the Volute of a Centrifugal Fan, Journal of Fluids Engineering, Vol.128, pp. 359-369, 2006.
[3] Siemens, Simcenter STAR-CCM+ User Guide, Siemens Digital Industries Software, pp. 32-7380, 2020.