Transactions of the Korean Nuclear Society Virtual Spring Meeting
July 9-10, 2020

Application of Computational Fluid Dynamics for Generation of Hydraulic Forcing Functions on Reactor Internals

Dong Hwa Jeong [a] [*], Kyong In Ju [a], and Seong Chan Park [a]
[a]*KEPCO E&C, Inc., Fluid System Eng. Dept., 989-111, Daedeokdaero Yuseong-gu, Daejeon, 34057, Korea*
[*]*Corresponding Author: dhjeong@kepco-enc.com*

1. Introduction

In the reactor design, evaluating the structural vibration of the reactor internals is a very important task. Turbulent coolant flow is a main source causing the Flow-Induced Vibration (FIV) of the reactor internals during operation of nuclear power plants. Reynolds number of the reactor internal flow is of the order of 10^5 to 10^7, and pressure fluctuations in the high Reynolds number turbulent coolant flow exert random forces in a wide forcing frequency range on the surfaces of reactor internals. The turbulence-induced vibration of the reactor internals could lead to structural fatigue damage which is related to nuclear safety. United States Nuclear Regulatory Commission Regulatory Guide (RG) 1.20 [1] requires the vibration testing on the reactor internals for at least 10^6 cycles based on the lowest frequency for which each critical component has a significant structural response.

In addition, since fluctuations of pressure signal due to turbulence might be troublesome in terms of monitoring process, turbulence is a phenomenon that shall be considered from the development stage of the nuclear power plants. However, nature of turbulence (e.g. randomness and high frequency flow oscillations etc.) makes analysis of the phenomenon difficult in almost all of engineering fields, especially including high Reynolds number turbulent flows.

RG 1.20 [1] provides guidance for evaluating structural vibrations of the reactor and presents various methodologies applicable to evaluating FIV (including turbulence-induced vibration) such as experiment, computational fluid dynamics (CFD). Most engineers in the reactor design have employed a methodology combining experimental and analytical methods by the scaling law. The scaling law always requires basic reference data obtained in advance. Performing experiments or testing was the most practical way to obtain the basic data, and today it is still valid. Recently, on the other hand, the CFD has appeared to be a probable option to simulate turbulent flow in time with the improvement of computer performance and analysis techniques. Especially the latest revision of RG 1.20 (Revision 4) provides detailed guidance for the application of CFD, e.g. validation of results, turbulence models, and numerical methods, for the purpose of quantifying hydraulic forcing functions.

This study presents suggestions for selecting a proper turbulence model in order to simulate turbulent pressure fluctuations forcing structures. A turbulence model properly selected can help determine the appropriate level and frequency range of forcing due to turbulent flow. It goes without saying that numerical methods pertinent to each modelling approach shall be applied. In Korea the application of CFD has rapidly spread in various industries. However, it is still questionable whether analysis by the CFD is being done properly.

Nowadays research and development of various types of nuclear reactors are being conducted according to market demands. It is important to properly evaluate structural vibrations of the reactor internals for safe operation of nuclear reactors. Although an experiment or testing is even more practicable means than the CFD because of its excessive calculation cost and time, the CFD would be a useful means to quantify hydraulic forcing functions in years to come. Whatever the means, choosing the right way to obtain the forcing functions will reduce trial and errors in research and development of the reactor, and help make works easier.

2. Generation of Hydraulic Forcing Functions

Regulatory guidance in Revision 4 of RG 1.20 extends to FIV, Acoustic Resonance (AR), Acoustic-Induced Vibration (AIV) and Mechanical-Induced Vibration (MIV). Of these, FIV, AR and AIV are related to phenomena in fluid region, which are due to loads inherent in flow or interaction between fluid and structures. Coolant flow inside the reactor is a high Reynolds number turbulent flow and the reactor internals are subjected to the turbulence-induced loads. Cross flows on the structures could cause vibrations by vortex shedding or Fluid-Elastic Instability (FEI) in addition to turbulence. AR is induced by a separated flow above opening of a side branch with a closed end, and AIV is caused by the reactor coolant pumps.

It might not be proper to define forcing functions for some of these vibrations. For example, avoidance strategies on AR and FEI are available because criteria for assessment of possibility of these phenomena has already been established through various researches. However, the forcing functions for turbulent pressure fluctuations shall be determined everywhere in the reactor where the coolant flows. Vibrations by vortex shedding and acoustic wave are not reviewed herein since there is no need to use the CFD thanks to their simplicity.

The turbulence-induced forcing functions have been mainly determined through experiments and testing for a scale or full-sized model. It is practical to use data measured in a precedent reactor or an experimental system similar to the reactor. Meanwhile, engineers are always interested in the analytical method, because of the difficulties, expense and effort that they could face while testing. Revision 4 of RG 1.20 mentions the application of CFD to obtain hydraulic forcing functions. According to RG 1.20, the CFD can become suitable for use in the future, but not yet, because of insufficiency of current capability to compute pressure fluctuation spectra on large complex structures subject to high Reynolds number turbulent flows. In addition, the turbulence-induced forcing functions obtained using the CFD shall be validated by test results. Otherwise, it could be a risk in business. RG 1.20 provides guidance required to use the CFD analysis results in generation of hydraulic forcing functions on the reactor internals

3. Turbulence Models for Turbulence-Induced Forcing Functions

3.1. Turbulence models recommended in RG 1.20

There are various methods established for the analysis of turbulent flows, which are as follows:

- Direct Numerical Simulation (DNS),
- Reynolds Averaged Navier-Stokes (RANS) or Unsteady RANS (URANS)
- Large Eddy Simulation (LES),
- Hybrid RANS-LES Simulation.

Revision 4 of RG 1.20 addresses that all methods except RANS (or URANS) are acceptable for transient simulations of high frequency flow oscillation at high Reynolds number flows. In other words, the RANS model is unacceptable to simulate turbulent fluctuations at a high frequency. Reynolds introduced a time averaging concept for the sake of describing random turbulence, which is a basis of modelling turbulence by RANS. Thus, analysis results by RANS represent time averaged physical quantities. Here the fluctuating velocity components are involved in the turbulent kinetic energy or Reynolds stresses time-averaged. To make matters worse, the fluctuating pressure components are not presented in its results because only the mean pressure is computed. Thus, it is impossible to quantify pressure fluctuations. In the analysis by RANS, relatively high level of eddy viscosity (or Reynolds stresses) modelling all amount of the fluctuating velocity results in the effect as if the diffusion term of the Navier-Stokes equation is strengthened, and accordingly quantities of the fluctuating velocity components are reduced. It is the main reason why RG 1.20 does not recommend RANS (or URANS) for simulation of the high frequency flow oscillations. In analysis by URANS, the RANS model plays the same role as in the steady state flow analysis except for the effect of deterministic low-frequency forcing like fluctuations by vortex shedding. So, forcing functions generated from the results of URANS are less than actual values.

While the DNS simulating all fluctuating information without any turbulence model is considered to be a method to obtain results with almost similar level of accuracy with the exact solution, the DNS has been used only for studies of low-Reynolds-number turbulent flows in simple calculation domains because of a huge calculation cost and time beyond computing ability currently available in engineering fields. Thus, the DNS is considered impractical to simulate high frequency fluctuations for the reactor design since very fine grids are required to simulate the smallest eddies that includes very low turbulent kinetic energy.

Among the simulation methods for turbulent flows, LES and hybrid RANS-LES simulation are considered to be applicable to determining the hydraulic forcing functions due to turbulence in the future if not present. Thus, applicability of these two methods is reviewed in the following sections.

3.2. Large eddy simulation for turbulence-induced forcing functions

From the technical perspectives, the LES is the most appropriate method to determine the turbulence-induced loads among the simulation methods. The LES can compute turbulent fluctuations including information at relatively high frequency. The LES replaces low turbulent energy of small eddies in a high frequency range with the subgrid scale model. So, the fluctuating components computed by the LES contain most of the turbulent kinetic energy in a wide frequency range including relatively high frequency. Turbulent kinetic energy modelled by the subgrid scale model can be discerned in the frequency domain by the cutoff wave number related to an eddy size (or a filtering mesh size).

Despite such a theoretical background discussed above, the LES is still limited to researches in academic level and some industries which are interested in relatively low-Reynolds-number turbulent flows since the LES requires relatively many grids and lengthy calculation time. In the LES, the first node near a wall is generally located at y+ equal to or less than 1, and thus the grids become relatively fine in the whole calculation domain. The time interval for transient analysis becomes shorter as per the mesh size. Accordingly, relatively many grids and iteration steps are required. It should be noted that the ambiguous wording ‘relatively’ can cause misunderstanding on the applicability of LES such that the LES doesn’t need so many grids for simulating larger eddies than the smallest eddies. Merzari et al. tried to assess the applicability of LES to the nuclear engineering [2]. In his study, for MASLWR (Multi-Application Small Light Water Reactor) integral test facility with 1/3 scale height, analysis for 1 second by the LES was estimated to take up to 1,000,000 CPU-hours. He estimated trillions of CPU-hours for the full-sized small modular reactor. As shown in the case of MASLWR, it is still impractical to use the LES for this application.

While experimenting or testing, people can face many difficulties. However there are many obstacles in computational analysis as well. The computational burden in analysis by the LES is the reason why RG 1.20 points out that the applicability of CFD would be acceptable in the future rather than the present. Therefore, the applicability of CFD for generating hydraulic forcing functions depends entirely on the improvement of computer equipment performance.

3.3. Hybrid RANS-LES simulation for turbulence-induced forcing functions

Hybrid RANS-LES simulation is a method that takes advantages of RANS and LES in a computational domain. In general, hybrid simulation employs the time-averaging concept in the flow region close to a wall where small eddies exist, and resolves relatively large eddies detached from the wall. It is done in order to lessen a computational burden by simulating only relatively large eddies.

Hybrid simulation methods have been developed with two different concepts according to the way of dividing the computational domain, which are zonal and non-zonal hybrid approaches. The non-zonal (seamless) hybrid approach sets up one simulation model (named universal model) in a whole computational domain analyzed by RANS and LES (hereinafter, RANS and LES domains, respectively), and it is thought to be theoretically robust. But the seamless approach has a disadvantage that one cannot apply the LES directly to the flow region of interest, especially close to a wall. In contrast, the zonal hybrid approach uses different models in each of RANS and LES domains clearly divided by a certain boundary. Though it has some difficulties to connect the two regions in a rational way, the zonal hybrid approach has the advantage of applying LES directly into a small flow region of interest. In the zonal approach, it is important to locate the LES domain near a wall and set up the RANS domain outside the LES domain to obtain an acceptable spatial resolution in the whole computational domain. Embedded LES showed the possibility of successfully implementing the practice [3], and the method was included in a commercial CFD software, ANSYS Fluent.

Detached Eddy Simulation (DES) introduced in RG 1.20 is the most famous hybrid RANS-LES simulation. The DES may be classified as the zonal or non-zonal hybrid model depending on researchers' perspective on its nature. The DES employs the time-averaging concept in the flow region close to a wall and resolves large eddies detached from the wall. So, the DES is unsuitable to simulate the wall pressure fluctuations in order to generate the turbulence-induced loads on the reactor internals.

On the whole, it can be said that selecting a turbulence model for obtaining the turbulence-induced forcing functions depends on how much amount of turbulent kinetic energy is resolved, not on how they are modelled. The more turbulent kinetic energy is modelled, the more fluctuations of calculated velocity are diminished. It could lead to misunderstanding of turbulent flow phenomena and incorrect estimation of turbulence-induced loads.

3.4. Additional considerations for application of CFD results

In addition to the turbulence simulation methods, RG 1.20 presents various guidance, e.g. similarity of computational domain, conservative condition of flow, sensitivity tests of results and so on. Finally, it should be noted that the analysis results by the CFD shall be validated by in-plant measurements and/or scale model testing. Since the accuracy of analysis results by the LES might be very sensitive to the mesh size, numerical method and so on, the simulation must be carefully performed.

4. Conclusion

In this study, the applicability of CFD is reviewed as a means of quantifying the turbulence-induced forcing functions for evaluating the flow-induced vibration of reactor internals, and some suggestions are presented on the application of the turbulence model. Revision 4 of RG 1.20 states that the CFD may suitable for generating the hydraulic forcing functions only in the future because of the limiting capacity of current computation equipment available in the industry. Although experiments and testing are still more practical than the CFD to obtain the turbulence-induced forcing functions, it is necessary and worthy to try to develop and apply the analysis methodology by the CFD for the reactor design including vibration assessment.

Among various CFD application options, the LES is considered the most appropriate to generate the turbulence-induced forcing functions including high frequency flow oscillations. However, it would be a hasty choice yet when considering the present technical limitations. If the hybrid RANS-LES simulation is applied, the zonal approach is more appropriate choice than the non-zonal approach. It is important that the LES domain should be located near a wall in order to simulate wall pressure fluctuations. Adequacy of analysis results by the CFD shall be validated by comparison with measurement data.

REFERENCES

[1] RG 1.20, Rev. 4, Comprehensive vibration assessment program for reactor internals during preoperational and startup testing, 2017.

[2] E. Merzari, A. Obabko, P. Fischer, N. Halford, J. Walker, A. Siegel, and Y. Yu, Large-Scale Large Eddy Simulation of Nuclear Reactor Flows: Issues and Prospectives, Nuclear Engineering and Design, Vol. 312, pp. 86-98, 2017.

[3] D. Cokljat, D. Caridi, G. Link, R. Lechner, and F. Menter, Embedded LES Methodology for General-Purpose CFD Solvers, Proceedings of Turbulence and Shear Flow Phenomena Conferences, 2009.