Nuclear Engineering and Design 322 (2017) 547–562

Contents lists available at ScienceDirect

Nuclear Engineering and Design

journal homepage: www.elsevier.com/locate/nucengdes

Flow-induced vibration analysis of a helical coil steam generator experiment using large eddy simulation

Haomin Yuan[a,*], Jerome Solberg[b], Elia Merzari[a,c], Adam Kraus[a], Iulian Grindeanu[c]

[a] Nuclear Engineering Division, Argonne National Laboratory, Lemont, IL, USA
[b] Lawrence Livermore National Laboratory, Livermore, CA, USA
[c] Mathematics and Computer Science Division, Argonne National Laboratory, Lemont, IL, USA

ARTICLE INFO

Keywords:
Nek5000
DIABLO
SHARP
CFD
Flow-induced vibration (FIV)
Large eddy simulation (LES)
Helical coil steam generator
Spectral-element method

ABSTRACT

This paper describes a numerical study of flow-induced vibration in a helical coil steam generator experiment conducted at Argonne National Laboratory in the 1980 s. In the experiment, a half-scale sector model of a steam generator helical coil tube bank was subjected to still and flowing air and water, and the vibrational characteristics were recorded. The research detailed in this document utilizes the multi-physics simulation toolkit SHARP developed at Argonne National Laboratory, in cooperation with Lawrence Livermore National Laboratory, to simulate the experiment. SHARP uses the spectral element code Nek5000 for fluid dynamics analysis and the finite element code DIABLO for structural analysis. The flow around the coil tubes is modeled in Nek5000 by using a large eddy simulation turbulence model. Transient pressure data on the tube surfaces is sampled and transferred to DIABLO for the structural simulation. The structural response is simulated in DIABLO via an implicit time-marching algorithm and a combination of continuum elements and structural shells. Tube vibration data (acceleration and frequency) are sampled and compared with the experimental data. Currently, only one-way coupling is used, which means that pressure loads from the fluid simulation are transferred to the structural simulation but the resulting structural displacements are not fed back to the fluid simulation.

1. Introduction

The reliability of the steam generator (SG) is one of the most significant safety issues in nuclear power plants. In the operating history of nuclear power plants, a significant number of steam generators failed or were found to be defective and removed from service or repaired each year (MacDonald et al., 1996). A large fraction of the failures were due to tube rupture, which usually caused complex plant transients and induced damage to the whole system. The most important driver of tube rupture is flow-induced vibration (FIV) inside SGs. FIV leads to both fretting-wear and fatigue, both of which lead to the growth of pre-existing flaws that eventually result in severe tube failures (Jo and Jhung, 2008). Therefore, the improved safety and operating performance of nuclear power plants depend on the ability to assess a particular SG design for reliability, especially with regard to the tube rupture problem. Simple design changes that minimize flow-induced vibration, such as decreased span length, larger tubes, thicker walls, and greater tube spacing, also decrease the steam generator efficiency and increase the plant cost and footprint. If the lifetime of a steam generator can be predicted, preventive measures and/or design changes can be implemented to decrease the possibility or severity of tube rupture failure. Since the leading driver of tube rupture is flow-induced vibration, a complete FIV analysis for the tubes in SGs must be performed.

The FIV problem has been studied for decades, with a special emphasis on steam generators and other heat exchangers incorporating tube arrays. The authors Paidoussis (1980, 1982), Pettigrew et al. (1998), Pettigrew and Taylor (2003a,b), Lowdon et al. (1990), Chen et al. (1983), Chen (1989), Tanaka and Takahara (1980, 1981), and Weaver and Grover (1978) all did comprehensive work studying this problem. However, the methodology used in those papers is primarily a mix of analytical and empirical methods.

Regarding the numerical approach, most of the researchers used CFD/FEM method to coupled CFD and FEM code to simulate the response of structure under flow condition. Jo and Jhung (2008) performed a numerical simulation of helical coil steam generator using CFX. He simulated flow in both primary side and secondary side considering a single tube. Kuehlert et al. (2008) studied the FIV problem for a tube bank, a single tube, and a hydrofoil in cross flow using FLUENT with RNG k-epsilon model and a dynamic subgrid LES model. His numerical simulation considered a full scaled coupled CFD/FEA FSI analysis, but limited again to a single unit cell.

* Corresponding author.
E-mail address: hyuan@anl.gov (H. Yuan).

http://dx.doi.org/10.1016/j.nucengdes.2017.07.029
Received 16 March 2017; Received in revised form 19 July 2017; Accepted 24 July 2017
0029-5493/

Nomenclature

u	velocity, m/s
ρ	density, kg/m^3
t	time, s
P	pressure, Pa
τ	stress tensor
μ	dynamic viscosity, kg/(m s)
y+	non-dimensional wall unit
Rij(r)	two point velocity correlation
V_{gap}	projected gap velocity, m/s
D	tube diameter, m
Re	Reynolds number
z	stream-wise direction coordinate, m
r	radial direction coordinate, m
φ	azimuthal direction coordinate, radian
alpha, beta	coefficients for Newmark time integration algorithm
x	structure motion displacement, m
M	mass matrix
C	damping matrix
K	stiffness matrix
F	external forcing
ξ_i	Rayleigh damping ratio for mode i
ω_i	baseline frequency for mode i
α,β	Rayleigh damping factor
f_{air}	natural frequency in air, Hz
f_{water}	natural frequency in water, Hz
m	mass of structure, kg
m_{added}	added mass of structure, kg
a_{tube}	tube acceleration, m/s^2
a_{rms}	rms acceleration, m/s^2

Ichioka et al. (1997) simulated the vibration of two tubes and a tube row in under flow condition with a two-way coupling setup. Robinson-mosher and Su (2008), Pittardrobert et al. (2004) and Merzari et al. (2016) proposed used a two-way coupling approach to perform FIV analysis for several example problems. None of the studies mentioned above presented work at the scale described in this manuscript. In fact, simulation packages that take advantage of high-performance computing platforms have only recently advanced to the point that such packages can now be used to analyze FIV for practical nuclear engineering applications. These packages now enable whole-scale simulation and they can provide detailed information on both the fluid dynamics and the thermal performance; for example, turbulent scales, vortex shedding, and secondary flows.

Detailed understanding of the fluid dynamics of tube arrays is not sufficient for predicting FIV. Also essential is the structural response of the tubes with loading induced by the flow condition. In the most general case, the structural response of the tubes affects the flow characteristics, and vice versa. Hence the most comprehensive analysis would involve a two-way coupled approach, where the flow simulation affects the structural simulation, and vice versa. From the fluid dynamics, the loading on the structure from the mean and fluctuating components of the flow can be predicted (Naudascher and Rockwell, 2012) by extracting transient surface pressures from the computational fluid dynamics (CFD) simulation. The response of the structure, such as stress, deformation, and oscillation, can then be predicted from the known fluid dynamics loading and structure characteristics. In our current study, a one-way coupling method is used, which means that the CFD simulation provides loading information to the structural simulation but assumes that the structure remains fixed: it does not utilize the displacements calculated by the structural simulation. This methodology is valid only when flow velocity and tube displacement are both small. As flow velocity increases, tube displacement is enhanced; and at the same time the structural motion starts to affect the flow significantly. As the flow rate increases, the one-way coupling method becomes less and less accurate, finally becoming completely invalid at the onset of fluid-elastic instability.

In the 1980s, Chen et al. (1983) performed an experiment on a half-scale sector model of a steam generator helical coil tube tank at Argonne National Laboratory. This test was designed to study the structural motion of the whole tube bundle under the conditions of still and flowing air and water. Details of the experiment are discussed in section 1.1. Data from this test is used to validate SHARP (Yu et al., 2016), which is an advanced modeling and simulation toolkit for the analysis of nuclear reactors (Yu et al., 2016). Results demonstrate that for low velocity, where the one-way coupling assumption is likely to hold, SHARP can be used successfully to simulate flow-induced vibrations.

SHARP's thermal-hydraulic code Nek5000 (Fisher et al., 2008) is used to simulate the flow using large-eddy simulation (LES) for turbulence modeling. Lai et al. (2016) performed a study applying LES to models of a helical coil steam generator using Nek5000. Even though that study was based on one subsection of a tube bank, the simulation results are in excellent agreement with the experimental data, confirming LES's applicability to this problem. Fig. 1 shows the computational domain used in the Nek5000 fluid simulation for the helical steam generator tested in Chen's experiment. The Nek5000 simulation setup and results are discussed with more detail in Section 2. We also discuss turbulence modeling approaches and justify the need for a large eddy simulation approach in this context and highlight the need of further research in RANS and hybrid modeling approaches.

SHARP's structural mechanics code DIABLO (Solberg et al., 2014) is used to simulate the response of the tube bank subjected to surface loadings predicted by fluid simulation. Fig. 2 presents the geometry in the DIABLO simulation with a magnified tube displacement. The DIABLO simulation setup and results are discussed in more detail in Section 3.

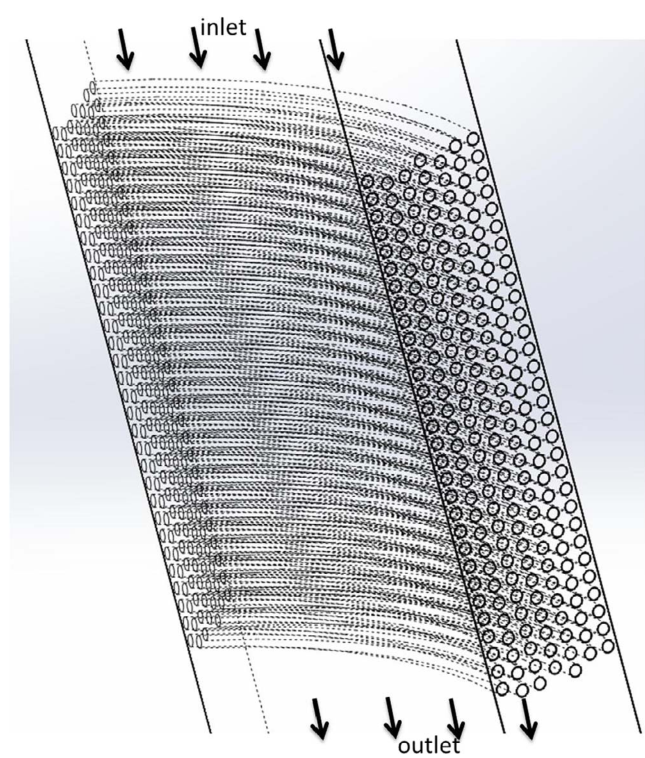

Fig. 1. Computational domain in Nek5000 fluid simulation (flow goes downward).