http://dx.doi.org/10.5516/NET.02.2014.001

FLUID-STRUCTURE INTERACTION IN A U-TUBE WITH SURFACE ROUGHNESS AND PRESSURE DROP

GYUN-HO GIM[1], SE-MYONG CHANG[1*], SINYOUNG LEE[1], and GANGWON JANG[2]
[1]Department of Mechanical Engineering, Kunsan National University
558 Daehak-ro, Gunsan-shi, Jeonbuk, 573-701, Korea
[2] Faculty of Mechanical and Aerospace Engineering, Sejong University
98 Gunja-dong, Gwangjin-gu, Seoul, 143-747, Korea
[*]Corresponding author. E-mail : smchang@kunsan.ac.kr

Received January 06, 2014
Accepted for Publication April 24, 2014

In this research, the surface roughness affecting the pressure drop in a pipe used as the steam generator of a PWR was studied. Based on the CFD (Computational Fluid Dynamics) technique using a commercial code named ANSYS-FLUENT, a straight pipe was modeled to obtain the Darcy frictional coefficient, changed with a range of various surface roughness ratios as well as Reynolds numbers. The result is validated by the comparison with a Moody chart to set the appropriate size of grids at the wall for the correct consideration of surface roughness. The pressure drop in a full-scale U-shaped pipe is measured with the same code, correlated with the surface roughness ratio. In the next stage, we studied a reduced scale model of a U-shaped heat pipe with experiment and analysis of the investigation into fluid-structure interaction (FSI). The material of the pipe was cut from the real heat pipe of a material named Inconel 690 alloy, now used in steam generators. The accelerations at the fixed stations on the outer surface of the pipe model are measured in the series of time history, and Fourier transformed to the frequency domain. The natural frequency of three leading modes were traced from the FFT data, and compared with the result of a numerical analysis for unsteady, incompressible flow. The corresponding mode shapes and maximum displacement are obtained numerically from the FSI simulation with the coupling of the commercial codes, ANSYS-FLUENT and TRANSIENT_STRUCTURAL. The primary frequencies for the model system consist of three parts: structural vibration, BPF(blade pass frequency) of pump, and fluid-structure interaction.

KEYWORDS : U-tube, Steam Generator, Inconel 690, FSI, Fretting Wear, CFD

1. INTRODUCTION

In Korea, since the first nuclear power plant was built in 1978, 22 plants have been running, playing an important role by generating about 23% of total domestic electric power. In the case of the steam generator of Gori-1, which was exchanged in 1989, various types of corrosion have been experienced such as pitting, SCC (stress corrosion cracking), and wear, etc. The succeeding plants also have had various types of corrosion problems though this is now better managed [1].

Vibrations generated by fluid have three causes: fluid-elastic instability, forced vibration with unsteady pressure fluctuation originating from turbulence, and the periodic vibration with vortex shedding on the heat pipe around the steam generator can cause the wear or the fatigue fracture, finally resulting in the failure of SG (steam generator) [2]. Cracks of heat pipe and TSP (tube support plate) caused by various microscopic factors like chemicals and sludge deposits [3], but one of the primary factors to consider is fretting wear [4] in the combination of the tube and TSP.

In this research, we studied a simplified U-tube model for further simulation in the future. An experimental reduced scale model was constructed, and a fully coupled multi-physical analysis of fluid-structure interaction with commercial codes based on ANSYS workbench was carried out. The primary water circulation system was focused on as a source of the forced vibration, and the present numerical method is validated for this simplified model.

2. METHODS OF RESEARCH

2.1 Numerical Method

The pressure drop is directly related to surface roughness, which is a function of the Reynolds number and the roughness ratio, defined as the ratio between roughness and tube diameter. For the analysis of primary water, three-dimensional unsteady incompressible Navier-Stokes equations are used:

$$\nabla \cdot \mathbf{V} = 0 \tag{1}$$

$$\rho\left(\frac{\partial \mathbf{V}}{\partial t} + \mathbf{V}\cdot\nabla\mathbf{V}\right) = -\nabla p + \mu\nabla^2\mathbf{V} \tag{2}$$

In Eqs. (1~2), $\mathbf{V}$ is velocity vector; p is pressure; ρ and μ are density and viscosity, respectively. No-slip boundary condition is applied at the tube wall; the inlet condition is specified as a mean flow rate and a given fluctuation; the outlet boundary is set as the ambient pressure. Additionally, k-ω SST (Shear Stress Transport) turbulence model is used for the turbulent intensity of 5% for the incident flow [5].

For the analysis of the structural dynamics of the tube, the following equations are used [6]:

$$-\widetilde{\mathbf{C}} : \nabla \cdot \widetilde{\boldsymbol{\varepsilon}} = \mathbf{f}_V \tag{3}$$

where the elastic stiffness matrix is defined as

$$\widetilde{\mathbf{C}} = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.5-\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5-\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 0.5-\nu \end{bmatrix} \tag{4}$$

In Eqs. (3~4), $\mathbf{f}_V$ is the external force per unit volume, which is integrated from the fluid pressure at the wall; E and ν are Young's modulus and Poisson's ratio (which should be constant in this study) respectively. The time rate of strain is expressed with the velocity components of tube elements.

$$\frac{\partial \widetilde{\boldsymbol{\varepsilon}}}{\partial t} = \frac{1}{2}\left[(\nabla\mathbf{V})^T + \nabla\mathbf{V}\right] \tag{5}$$

In Eq. (5), the velocity components on the right hand side can be obtained from the structural deformation from the strain field. Fig. 1 is the procedure for the computation of fluid-structure interaction. The finite element model uses shape and node information in common. Under the boundary conditions, the flow field is computed, and then the effective mass distribution is exerted onto the tube structure for the consideration of the mass of internal and external fluid. After the common nodal points are set for the data exchange between fluid and structure, the system coupling to both sides is used in every time step in Eqs. (1~5) to describe the deformation of structure.

2.2 Surface Roughness

For a straight tube with a constant inner diameter and a cross-sectional shape, the correlation of the Darcy friction factor, f with the Reynolds number and roughness ratio, ε/d [7].

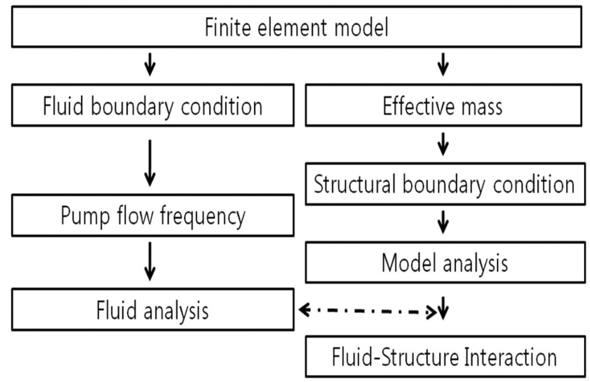

Fig. 1. Procedure of Multi-Physical Computation

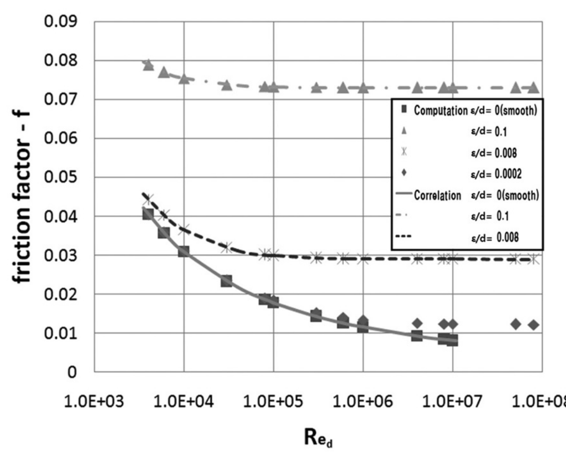

Fig. 2. Darcy Friction Factor vs. Reynolds Number

$$\frac{1}{\sqrt{f}} = -2.0\log_{10}\left(\frac{\varepsilon/d}{3.7} + \frac{2.51}{\mathrm{Re}_d\sqrt{f}}\right) \tag{6}$$

The plot of Eq. (6) is given in given in the Moody chart in Fig. 2. This correlation is valid only for the rigid pipes without vibration or deformation of the wall. By changing the parameters, we can obtain the numerical values of Fig. 2 where ANSYS-FLUENT is used for numerical computation.

The result is very sensitive to the grid scale, especially the vertical size of the first grid, Δy, which is expressed with a dimensionless parameter:

$$y^+ = \Delta y\sqrt{\frac{\rho}{\mu}\frac{du_T}{dy}\bigg|_{y=y_w}} \tag{7}$$

where u_T is the transformed velocity component of the tangential direction of the wall. To get the proper values coinciding with Eq. (6) in the parametric plane of Fig. 2, the dimensionless wall distance should be guaranteed as $y^+<1$ in the whole computational domain. For example, when the computational domain is a pipe of 20 mm di-