A numerical study on air squeeze-film damping based on structure-fluid co-simulation technique

Mingzhang Ren, Miaki Tanoue, Keiichiro Tsutsui

ESTECH Corporation

Abstract: CAE applications in dealing with multiphysics problems have been drawing much attention in product development in recent years. In particular, structure-fluid interaction (FSI) problems are of major concern. In this article, a numerical simulation on air squeeze-film damping which is important in MEMS design is presented. The study employs Abaqus and STAR-CD to perform a structure-fluid co-simulation. The squeeze-film damping phenomenon of a simple plate structure is demonstrated and its mechanism investigated. An equivalent modeling method to approximately represent the squeeze-film effects is also examined. It is considered that this coupling analysis method based on Abaqus Co-simulation could be an effective approach to deal with a broad range of FSI problems.

Keywords: Squeeze-Film Damping, Fluid-Structure Interaction, Co-Simulation, MEMS, Vibration

1. Introduction

In recent years, CAE tools have been seeing increasing applications for dealing with multiphysics problems, such as fluid-structure interaction (FSI). A typical FSI example is air squeeze-film damping (SFD) phenomenon, which is characterized by the fact that when a vibrating plate moves close to a fixed surface, the air in the thin film will demonstrate resistances to the moving plate. This damping phenomenon is known from old times, but intensive studies on it are conducted only in recent years (Bao and Yang, 2007; Bicak and Rao, 2010; Pandey et al., 2007; Veijola and Lehtovuori, 2009). The background behind is the extending applications of MEMS (micro-electro-mechanical system) devices, where SFD plays an important role in improving the dynamics and functions of these devices. There is also report that SFD phenomenon is utilized to reduce noise and vibration of general mechanical systems (Ishihama and Hayashi, 2010).

It is considered that the studies on SFD can be classified into analytical (or theoretical) studies and numerical simulations. The former focuses on the basic characteristics of SFD subject to simple geometry and boundary conditions. One the other hand, the latter mainly makes use of finite element method (FEM) to solve the general Navier-Stokes equations or specialized Reynolds equations. Although the FEM approach is suitable for more complex geometry and boundary conditions and strong coupling problems, there are various difficulties in coding and solving the system equations which couple solid mechanics and fluid dynamics in terms of unified matrices. In this article, it is presented another approach, where commercial FEM code and computing fluid