Chinese Journal of Aeronautics, (2018), 31(5): 1085–1094

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

cja@buaa.edu.cn
www.sciencedirect.com

A transient bulk flow model with circular whirl motion for rotordynamic coefficients of annular seals

Peng XIA *, **Zhansheng LIU**, **Xiangyu YU**, **Jingming ZHAO**

School of Energy Science and Engineering, Harbin Institute of Technology, Harbin 150001, China

Received 1 March 2017; revised 5 June 2017; accepted 4 August 2017
Available online 20 February 2018

KEYWORDS

Annular seals;
Bulk flow;
Computational Fluid Dynamics (CFD);
Rotordynamic;
Turbulence

Abstract Bulk flow model with perturbation simplification has been used to calculate rotordynamic coefficients in annular seals which have significant influences on the dynamic behavior of rotors in turbomachinery. In this work, a transient bulk flow model with arbitrary rotor motion is developed, and the boundary conditions and friction factor in the model are calibrated with steady Computational Fluid Dynamics (CFD) analysis. The numerical solution scheme is developed based on the finite element method to obtain the transient reaction force in the seal clearance. With a periodic circular rotor orbit, the transient forces at multiple whirling frequencies are used to evaluate the rotordynamic coefficients. The leakage flowrate of CFD analysis has good agreement with experimental results and the calibrated parameters in bulk flow model are dependent on operating conditions. Although CFD calibration improves the accuracy of the perturbed bulk flow model, the direct damping is overestimated and the cross-coupled damping is underestimated. Compared with the perturbed model, the predictions of the transient bulk flow model are more agreeable with the experiment.

1. Introduction

Rotordynamic characteristics of high-performance turbomachinery significantly depend on the hydrodynamic forces developed by annular pressure seals.[1] Accordingly, a large number of studies have been conducted to theoretically predict and to experimentally measure the hydrodynamic forces in a number of seal configurations under various operating conditions.

The bulk flow model introduced by Hirs[2] has been widely used to predict the dynamic coefficients for turbulent annular seals. In order to simplify the Navier-Stokes equations and reduce substantial computational costs, the bulk flow theory associated bulk mean flow in the seal clearance with averaged turbulence forces. Childs[3] firstly used 1D bulk flow model with fluid inertia effect to calculate the rotordynamic coefficients for smooth annular seals concentric with rotors. San Andres[4] developed a Computational Fluid Dynamics (CFD) solution

* Corresponding author.
E-mail address: xiapeng19860610@163.com (P. XIA).
Peer review under responsibility of Editorial Committee of CJA.

Production and hosting by Elsevier

https://doi.org/10.1016/j.cja.2018.02.011
1000-9361

for eccentric smooth annular seals. A finite difference scheme was implemented to solve 2D bulk flow model and the numerical solution procedure was based on the Semi-Implicit Method for Pressure Linked Equation (SIMPLE) to accelerate convergence. To improve the accuracy of bulk flow model, simplified turbulence models for different surface textures were developed based on a large number of experimental data in the Refs.[5,6] Although the turbulent models were proved to be effective in those studies, it was difficult to capture the full nature of turbulence flow based on analogical experiments. In the Refs.[7,8], the analytical results and experiment data had good agreement by manually adjusting the friction factors in the turbulence models. Saba et al.[9] evaluated the friction factors by minimizing leakage derivation between analytical results and experiment data, and the prediction of rotordynamic coefficients had good agreement with experiment results. However, empirical boundary conditions used in the Refs.[7–9] were another important error source. As the boundary conditions of bulk flow model also had an influence on the leakage flowrate, the friction factors could not be determined independently in the Ref.[9] Actually, the boundary conditions such as pressure loss coefficient and pre-swirl ratio could be measured in experiments, but the measurements were very expensive and seldom have been conducted. In the Refs.[3–9], the bulk flow model was a static model and a perturbation method was implemented to calculate rotordynamic coefficients with a mathematical simplification that the amplitude of rotor whirl motion was much smaller than the clearance thickness, but the amplitude in experiments and engineering hardly fulfilled the perturbation simplification. In addition, there were many other cases where the perturbation condition did not fulfill. In the Refs.[10,11], the large-scale dynamic responses of floating ring seals were investigated and the hydrodynamic forces were calculated by the perturbed bulk flow model.

In recent years, CFD has been increasingly used to predict the performance of annular seals. CFD method solves the complete Navier-Stokes equations and is able to predict turbulence flow at a highly detailed level in the solution. Moore and Palazzolo[12] conducted CFD simulations to investigate the liquid mean velocity of a grooved liquid annular seal. Numerical predictions of the mean velocity at different positions across the seal showed good agreement with experimental results. With the development of computational technology, CFD method was used to calculate rotordynamic coefficients for annular seals. Moore[13] introduced a steady-state CFD method to calculate rotordynamic coefficients of a labyrinth seal. The method used a steady-state flow field and a rotor whirl motion with small eccentricities. The axisymmetric geometry of the labyrinth seal allowed frequency-independent rotordynamic coefficients to be evaluated with impedance calculation at three different whirl frequencies. Untaroiu et al.[14] calculated rotordynamic coefficients for a circumferentially grooved liquid seal with the steady-state method. To study pre-swirl effect, the upstream region for the seal was explicitly modeled in CFD study. Chochua and Soulas[15] developed a 3D transient, dynamic mesh method to calculate dynamic characteristics of a hole-pattern gas seal. Yan et al.[16] improved the dynamic mesh method with a small circular rotor whirl motion, and the computational costs for a simulation needed 15 days on an Intel Core2 Quad6600 2.4 GHz CPU. Generally, full 3D CFD method demonstrated superior capacity in predicting leakage and rotordynamic coefficients over bulk flow model while the CFD method required long solution time for calculating the dynamic coefficients. In addition, for wall-bounded flow in liquid seals, turbulence intensity in the clearance was much lower than requirements of the wall function in CFD theory and was inevitable to exponentially increase computational costs to solve the wall-bounded flow in 3D CFD analysis.

Several investigations were conducted to take advantage of the two methods. Arghir et al.[17] modified the turbulence model in bulk flow model with a great deal of CFD results for a typical cell extracted from seal textured surfaces. Migliorini et al.[18] substituted wall shear stress predicted by steady-state CFD analysis into bulk flow model to calculate rotordynamic coefficients. However, the wall shear stress was only part of the turbulence resistance in seal clearance.

2. Analytical model

2.1. Bulk flow model with arbitrary rotor motion

Under high pressure drop and rotational speed, the flow of low viscosity is characterized by high levels of turbulence. By neglecting velocity variation in the radial (y) direction and incorporating simplified turbulence models into the Navier-Stokes equations, the bulk flow continuity and momentum equations (Eqs. (1)–(3)) in the axial (z) and circumferential (x) directions are given by the Ref.[4] within each control volume of 2D flow field in seal clearance.

$$\frac{\partial \rho h}{\partial t}+\frac{\partial(\rho h u)}{\partial x}+\frac{\partial(\rho h w)}{\partial z}=0 \tag{1}$$

$$-h\frac{\partial p}{\partial z}=\frac{\rho}{2}wu_{\mathrm{s}}f_{\mathrm{s}}(n)+\frac{\rho}{2}wu_{\mathrm{r}}f_{\mathrm{r}}(n)+\rho\left(\frac{\partial hw}{\partial t}+\frac{\partial hwu}{\partial x}+\frac{\partial hww}{\partial z}\right) \tag{2}$$

$$-h\frac{\partial p}{\partial x}=\frac{\rho}{2}uu_{\mathrm{s}}f_{\mathrm{s}}(n)+\frac{\rho}{2}(u-R\Omega)u_{\mathrm{r}}f_{\mathrm{r}}(n)+\rho\left(\frac{\partial hu}{\partial t}+\frac{\partial huu}{\partial x}+\frac{\partial hwu}{\partial z}\right) \tag{3}$$

where t is the time, w is the axial averaged velocity, u is the circumferential averaged velocity, u_{s} is the velocity relative to the stator, u_{r} is the velocity relative to the rotor, h is the clearance thickness, p is the fluid pressure, ρ is the fluid density, R is the rotor radius, Ω is the rotor speed, f_{r} and f_{s} are Moody's turbulence formulas at rotor and stator surfaces, n is the friction factor.

To directly consider rotor whirl motion, the squeeze velocity V_{s} in the control volume is explicitly represented as the time derivative of film thickness.

$$\frac{\partial h}{\partial t}=V_{\mathrm{s}}=V_{\mathrm{radial}}\cos\theta+V_{\mathrm{tangential}}\sin\theta \tag{4}$$

where θ is the circumferential angle of a control volume, V_{radial} is the radial velocity and $V_{\mathrm{tangential}}$ is the tangential velocity in the local coordinate system shown in Fig. 1.