Tribology International 115 (2017) 546–556

Contents lists available at ScienceDirect

Tribology International

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

Application of a wall function to simulate turbulent flows in foil bearings at high rotational speeds

Kan Qin [a,*], Rowan J. Gollan [b], Ingo H. Jahn [b]

[a] Queensland Geothermal Energy Centre of Excellence, School of Mechanical and Mining Engineering, The University of Queensland, St Lucia, Brisbane 4072, Australia
[b] Centre for Hypersonics, School of Mechanical and Mining Engineering, The University of Queensland, St Lucia, Brisbane 4072, Australia

ARTICLE INFO

Keywords:
Foil thrust bearings
Turbulent simulation
Checker-boarding
Wall function

ABSTRACT

For turbulent simulations in hydrodynamic applications, a fine mesh close to wall boundaries is required to correctly predict the friction and heat fluxes. This results in a remarkable increase of computational cost due to the high aspect ratio cells and reduced allowable time step. To reduce the computational cost, a compressible wall function is introduced and validated with the representative flows. Moreover, high aspect ratio cells lead to simulation instabilities. These spurious oscillations are smoothed through the addition of a fourth-order artificial dissipation term. To verify that the solution accuracy is not affected, the method of manufactured solution is applied. These two additions result in a fast and stable solver for turbulent simulations of foil thrust bearings.

1. Introduction

The supercritical CO_2 (sCO_2) power cycle has been selected as next generation power cycle because of the high thermal conversion efficiency and lower capital cost [1]. It is considered a suitable power cycle for various applications, including nuclear [2], coal power [3], and renewable energy [4]. To mature this promising power cycle, all major turbomachinery components, including compressor, turbine, heat exchangers, seals and bearings need to be re-designed to work with the higher pressures and densities and lower volume flow rates. Wright et al. [5] from Sandia National Laboratories (SNL) tested a prototype sCO_2 compressor in a small scale test loop at conditions typical for a Brayton cycle. Gas foil bearings were chosen to support the long duration tests [5]. Typical operating conditions for the foil bearing at SNL are around 1.4 MPa and 300 K [5,6]. When using the high pressure CO_2 as the operating fluid, the flow regime within foil thrust bearings becomes turbulent and results in significant inertia forces, non-ideal gas properties, and reduction of rotordynamic damping coefficients [7–9]. These phenomena are different to our conventional knowledge of foil thrust bearings, which is mostly based on foil bearings using air as the operating fluid.

Foil bearings can be categorised as foil journal and thrust bearings depending on the load direction. Only the thrust-type bearings are studied in this paper. Foil thrust bearings usually consist of a top foil and a corrugated bump foil. The top foil is welded to the bearing housing (attachment edge) and presents a partially convergent shape. A foil thrust bearing consists of multiple pads in the circumferential direction, e.g. 6 pads as shown in Fig. 1(a). A more detailed diagram of the arrangement of bump foils is depicted in Fig. 1(b and c). The corrugated bump foil structure provides the stiffness to support the top foil. At the same time, the structure generates damping due to the relative motion between top and bump foils.

To accurately model this CO_2-specific foil thrust bearings, several studies have been carried out. Conboy [6] used the modified turbulent Reynolds equation to include real gas effects for foil thrust bearings by coupling to REFPROP [10]. They found that turbulent effects can augment the load capacity and power loss simultaneously. Kim [11] developed a modified three-dimensional Reynolds equation for foil bearings. This model has been applied to investigate the three dimensional performance of a hybrid thrust foil bearing [12]. However, these computational models do not account for all of challenges introduced by high density CO_2. Recently, the transient compressible fluid dynamic code Eilmer [13,14] coupled with a two dimensional structural model was adapted to model their elasto-hydrodynamic performance [7,8]. In these works, the effects of centrifugal inertia forces and turbulent flows on CO_2 foil thrust bearings have been identified. For turbulent flow, in the previous work by the authors [7], Wilcox's 2006 low-Reynolds $k-\omega$ turbulence model [15] was selected.

Hybrid Couette and Poiseuille is typically found in the thin films of bearings. It is challenging to correctly model this hybrid pressure and shear driven flow. The average velocity-based friction terms for pressure wave propagation introduced by Gretler [16] might lead to some errors.

---

[*] Corresponding author.
E-mail address: jasonqk@hotmail.com (K. Qin).

http://dx.doi.org/10.1016/j.triboint.2017.06.018
Received 13 February 2017; Received in revised form 9 April 2017; Accepted 13 June 2017

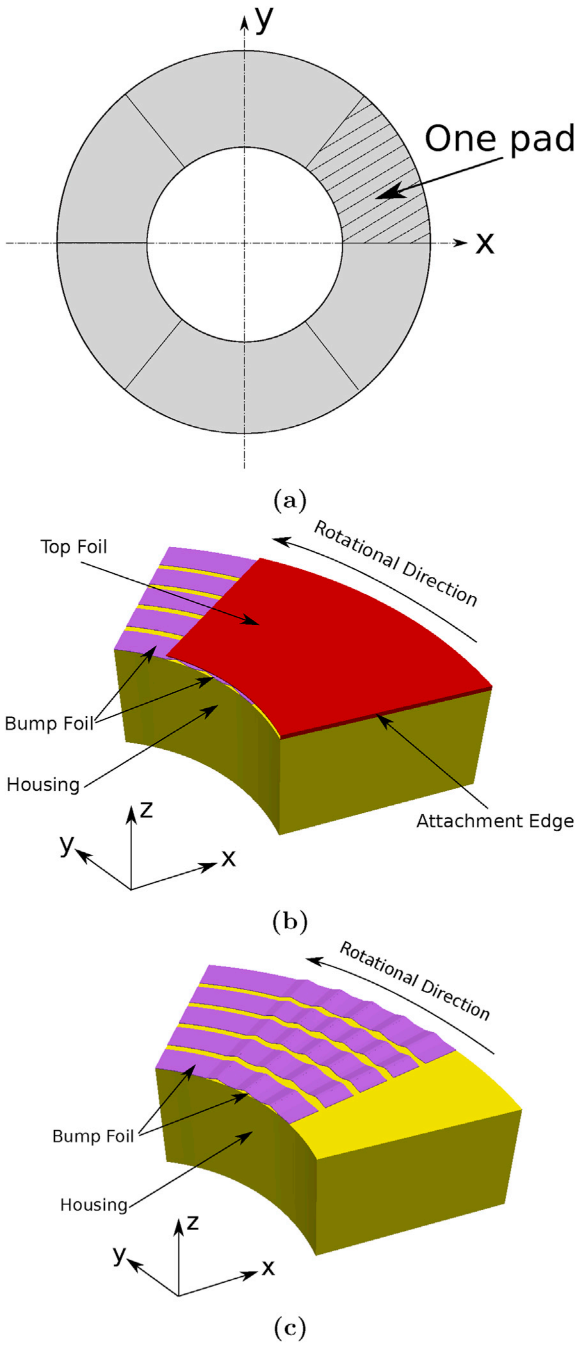

Fig. 1. Schematic diagram of one pad of foil thrust bearings, not in scale, (a): front view; (b): one stator pad in three dimensional; (c): one stator pad with top foil removed in three dimensional.

The $k-\varepsilon$ model, used by Henry and Reynolds [17], shows uniform turbulent kinetic energy across the flow, which may not capture all effects appropriately. Direct numerical simulations might be attractive, but they are currently limited to low Reynolds number [18], which is not applicable for the possibly sonic flow within foil thrust bearings [19]. Reynolds-averaged Navier-Stokes (RANS) modelling approach is still a very useful predictive tool for the hybrid Couette and Poiseuille flow. However, because of the thin film (order of microns) and high rotational speed operation of foil thrust bearings, RANS model presents challenges in computational cost due to the requirement for fine meshes close to wall boundaries. This creates small timestep due to stability constraint, and subsequently increased simulation time to reach the steady state condition. The checker-boarding effect is also observed due to a very high aspect ratios cell within the foil thrust bearings geometry.

The conventional modelling approaches that use Reynolds equation, may not be adequate for foil bearings operating with highly dense CO_2 [7,8]. Computational fluid dynamics is proposed to capture the high centrifugal inertia and turbulence effects. However, using the explicit time-stepping solver causes problems of increased computational cost and spurious checker-boarding effects. This paper discussed numerical methods, a compressible wall function and artificial dissipation to resolve these issues. The paper is organised as follows. Section 2 recalls the governing equations for fluid flow. Section 3 details the implementation of the compressible wall functions and the corresponding test cases. Here, the suitability of Wilcox's 2006 $k-\omega$ turbulence model for the hybrid pressure and shear driven flow in hydrodynamic applications is discussed. Section 4 documents the fourth-order artificial dissipation approach used to stabilise turbulent simulations of foil thrust bearings. The verification of this extended solver using the method of manufactured solution is discussed. The selection of the minimum coefficients for the artificial dissipation is then investigated. With these two additions, the in-house computational fluid dynamics (CFD) solver Eilmer is able to provide fast and stable turbulent simulations of foil thrust bearings.

2. Details of numerical approach

The compressible fluid flow solver, Eilmer [13,14] is used to simulate the flow within foil bearings. It has been extended to turbomachinery applications [7,8,20,21]. The compressible Navier-Stokes equations implemented in Eilmer are expressed as,

$$\frac{\partial}{\partial t}\int_V \mathbf{U}dV = -\oint_S (\mathbf{F}_i - \mathbf{F}_v)\cdot\hat{\mathbf{n}}dA + \int_V \mathbf{Q}dV\,, \qquad (1)$$

where $\mathbf{U}$ is the conserved quantities. The inviscid fluxes $\mathbf{F}_i$ are discretised with Wada and Liou's AUSMDV scheme [22]. The flow derivatives for the viscous fluxes $\mathbf{F}_v$ at the cell interfaces are approximated by the Green-Gauss theorem.

For the turbulent simulations, the Wilcox's 2006 $k-\omega$ turbulence model [23] is implemented in Eilmer. The suitability of this turbulence model for scramjet flowfield simulations is provided in Ref. [24]. For the simulations of real gases, a look-up table approach is utilised to speed up the computation of real gas properties without reducing the computational accuracy. A thermodynamic mesh is generated with the NIST database REFPROP [10] according to the defined temperature and density ranges, and a bilinear interpolation is used to obtain the local properties from the generated thermodynamic mesh. For CO_2 properties, the model in REFPROP is based on the work from Span and Wagner [25].

3. Wall modeled turbulence model for foil bearing applications

For turbulent simulations with Eilmer, the low Reynolds Wilcox's 2006 $k-\omega$ turbulence model [15] is used. As shown by Chan et al. [24], the y^+ (non-dimensional wall distance) value at the first cell from the wall, has a significant effect on the friction coefficient and heat fluxes prediction. At least one cell has to be within the viscous sublayer for this turbulence model. This results in very fine cells close to wall boundaries. For explicit time-stepping solvers such as Eilmer [14], the fine cells close to the wall boundaries result in prohibitively small time steps due to Courant-Friedrichs-Lewy (CFL) stability criterion that is imposed.

Wall functions are typically utilised as an efficient approach to reduce the computational cost. Several wall functions exist in the literature and are implemented by various CFD codes. The piecewise wall function is typically utilised [26], and different velocity profile is activated depended on the local y^+ value at the first cell from the wall. In OpenFOAM [27], an automatic near-wall treatment is used in conjunction with SST $k-\omega$ turbulence model, the formulation for the velocity profile near the wall is defined as