Tribology International 126 (2018) 9–15

Contents lists available at ScienceDirect

Tribology International

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

Predicting aerodynamic resistance of brush seals using computational fluid dynamics and a 2-D tube banks model☆

Yuchi Kang[a,b], Meihong Liu[a,*], Sharon Kao-Walter[b,c], Wureguli Reheman[b], Jinbin Liu[a]

[a] Faculty of Mechanical and Electrical Engineering, Kunming University of Science and Technology, Kunming, China
[b] Department of Mechanical Engineering, Blekinge Institute of Technology, Karlskrona, Sweden
[c] College of Engineering, Shanghai Polytechnic University, Shanghai, China

ARTICLE INFO

Keywords:
Brush seal
Aerodynamic resistance
Euler number
Reynolds number*2010 MSC:*
00–01
99-00

ABSTRACT

Aerodynamic resistance of a brush seal was mainly studied. The velocity distribution along three specified lines was presented. By considering the pressure differential, Reynolds number and Euler number (*Eu*) were modified. The effect of geometric arrangements and pressure differentials on Eu and leakage were analyzed. Two correlations were fitted based on the numerical results. The results reveal the velocity distribution is almost flat, asymmetric along the specified lines. The velocity increases and decreases almost linearly at centerlines. *Eu* decreases gradually less with the increase of pressure differential and trends towards a fixed value. A larger *Eu* indicates stronger resistance but not necessarily less leakage. Finally, two fitted correlations are developed and one is exponential to the row number fits better.

1. Introduction

A brush seal, a novel type contact seal, has been widely used in recent decades. It is characterized by excellent leakage performance, low friction and rotor excursion capability. A typical brush seal consists of a back plate, a front plate and thousands of bristles, as illustrated in Fig. 1. Generally, a brush seal is used to reduce parasitic leakage between turning and stationary components of a turbomachinery.

Many literature have been focused on numerical simulation to analyze fluid flow in brush seals. One numerical model for assessing brush seals is the tube bank model, which contains in-line or staggered arranged circles. Dai and Liu developed a compact, staggered tube array numerical model and analyzed resistance which revealed an exponential relationship between energy loss and Reynolds number. In addition, the extent of resistance increased dramatically with a more compact geometry [1]. Huang et al. analyzed the fluid flow in a brush seal based on a two-dimensional (2-D) staggered tube bundle model. They argued that the sealing effect can be significantly enhanced by reducing the inter-tube spacing of the bristles for normal pressure differentials and the number of axial bristle rows [2]. Tan et al. proposed a meshing method which is able to achieve a better mesh independence and studied the separation points in tube banks models of a brush seal. They found that the separation point may move to the rear part of the bristle when the brush seal is subjected to a high pressure differential [3]. Wang et al. analyzed the flows of two-stage brush seals by staggered tube bank model [4].

In addition to analyze flow across tube banks in the brush seal, several studies over the years have adopted tube bank models in heat exchangers. Aerodynamic or hydraulic resistance is one of the primary quantities of heat exchangers and is characterized by the total pressure drop. This is a form of energy loss partially caused by neighboring tubes and service conditions. Jin et al. performed numerical studies about the pressure drop in H-type finned tube banks with 10 rows and presented the correlation of Euler number (*Eu*) for the 10-row tube banks model [5]. Chen et al. analyzed the relationship between pressure drop and geometric parameters. The predictable correlations of *Eu* were developed based on the experimental results [6]. Studying the pressure loss of tube banks with bimetallic fins, Kuntysh et al. found that tight-finned tube banks have better thermal efficiency than a loose one [7]. The study about the pressure drop in tube banks has been conducted in terms of geometry parameters such as longitudinal and transverse pitch ratios and working conditions such as Reynolds number.

A brush seal is used for secondary system sealing performance in a gas turbine, where a higher pressure ratio can improve global efficiencies. So the pressure differential for a brush seal is a key design factor. According to the literature, brush seal leakage increases exponentially with excessive pressure loads [9]. Bristle deformation, pressure stiffening effect, and hysteresis are associated with the

☆ Fully documented templates are available in the elsarticle package on CTAN.

* Corresponding author.
E-mail address: 64115737@qq.com (M. Liu).

https://doi.org/10.1016/j.triboint.2018.04.023
Received 21 February 2018; Received in revised form 20 April 2018; Accepted 20 April 2018
Available online 30 April 2018
0301-679X/

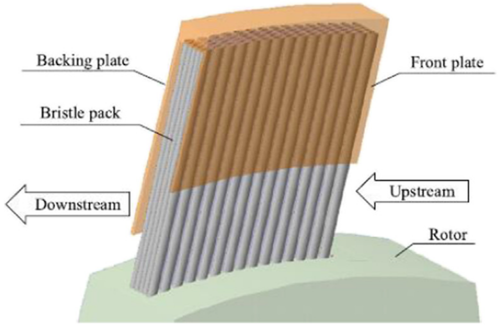

Fig. 1. Schematic diagram of a typical brush seal [8].

pressure differential across a brush seal. Zhu argued that 0.3 ∼ 0.35MPa is the maximum load for these seals [10]. By contrast, Chupp et al. reported that the typical operation limit is 0.21 MPa [11].

A review of the literature presents that almost all studies about brush seal pressure capability are based on experience. Although no model can be comprehensive enough to be applicable for all geometric arrangements, a few cases can be used to test a hypothesis. A quantity, *Eu*, has been used to express the aerodynamic resistance of tube banks in heat exchangers. Therefore, to analyze the aerodynamic resistance of the brush seal, *Eu* is adopted in the present study. In addition, the effect of parameters, such as longitudinal and transverse pitch ratios, and pressure differentials, on aerodynamic resistance are studied. Furthermore, the correlation of *Eu* is developed based on numerical results. The results of this study can provide a theoretical basis for measuring resistance in brush seals and serve as a reference for industrial application.

2. Model description and numerical method

2.1. Physical model

A schematic diagram of a multiple-row brush seal tube bank model is presented in Fig. 2. The geometric parameters of the tube bank model are listed in Table 1.The geometry in the present study is by refering Huang's aforementioned model [2]. They found that the 1-column model and the 6-column model are equivalent for analyzing intertube flow.

To simplify the expression, $(A, B, C)_p^N$ stands for the different cases according to the geometric parameters in Table 1. A, B and C stands for the angle θ of 30°, 45°, 60°, respectively, where subscript *p* is the pressure differential and superscript *N* is the total row number of bristles in a brush seal. For example, $A_{0.1}^{10}$ denotes the case $\theta = 30°$, pressure differential 0.1 MPa and row number 10.

2.2. Governing equations and boundary conditions

Before starting a computational fluid dynamics (CFD) numerical simulation, the flow status must be assessed. Previous studies reveal different opinions about the flow status in a brush seal. Some researchers believe that the fluid flow is turbulent [3,8,12] while others believe it becomes laminar. Huang et al. adopted the laminar model in compact tube banks and the turbulent model in spacious tube banks [2,13]. Zhang et al. argued that fluid flow is turbulent in the upstream and downstream regions but becomes laminar in the intertube due to slow fluid flow [14]. According to the states of flow around a circular cylinder in different flow regimes, the transition (from laminar to turbulent) occurs at low Reynolds number [15]. A turbulent model was adopted in this research.

Table 1
Geometric parameters of a brush seal based on tube banks model.

Case	θ/°	S_D/d	S_L/d	S_T/d	δ/mm	d/mm	N
A	30	1.1	0.953	1.1	0.007	0.07	10, 14, 18
B	45	1.1	0.778	1.556	0.007	0.07	10, 14, 18
C	60	1.1	0.55	1.905	0.007	0.07	10, 14, 18

The fluid was assumed to be an ideal gas in a steady state. The governing equations of this research includes those of mass, momentum and energy conservation, all of which have been previously described elsewhere. The turbulent model is presented below. The standard $k-\varepsilon$ turbulent model was used and expressed as

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k+G_b-\rho\varepsilon-Y_M \tag{1}$$

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{2}$$

where G_k is the formation of turbulent kinetic energy because of the average velocity gradients, G_b is the formation of turbulent kinetic energy due to buoyancy, and Y_M is the contribution of the fluctuating dilatation on compressible turbulence to the overall dissipation rate. μ_t is the eddy viscosity, which was modeled as $\mu_t=\rho C_\mu\frac{k^2}{\varepsilon}$ with $C_\mu = 0.0845$. The standard model yields $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$ [16].

To guarantee inlet uniformity, the upstream domain was extended by fifteen times of the bristle diameter for the entrance section. To reduce the effect of reverse flow, the downstream domain was extended by twenty times of the bristle diameter for the exit section.

There are four different boundary conditions in this model. At the inlet boundary, pressured air enters the computational domain with the turbulent intensity 5%. The outlet boundary pressure was set as 0.101 MPa. The bristle surface was assumed to be fixed and set as wall. The upper surface and lower surface were set as symmetric to simulate a more authentic flow domain. A detailed description of these boundary condition are presented in Table 2.

2-D steady-state numerical simulations were solved by commercial code FLUENT. A hybrid discretization scheme (the default settings in FLUENT) was used until a preliminary convergence is obtained. Then the second-order discretization scheme was adopted to gain a better solution efficiency. The residual criteria for each variable was 1.0×10^{-6}, except for continuity. By monitoring values between iterations is another criterion of the FLUENT tutorial [17]. The velocity difference, between different iteration steps at Line CD was monitored (Fig. 2).

2.3. Grid meshing and independent solution

By reviewing the previous method [3], an inflation tool was used to generate four layers of quadrilateral cells in contact with bristle

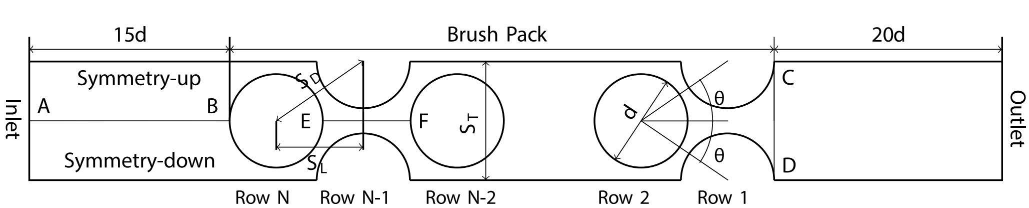

Fig. 2. A schematic configuration of a brush seal using a two-dimensional tube banks model.