Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 113 (2015) 306 – 311
Procedia Engineering
www.elsevier.com/locate/procedia

International Conference on Oil and Gas Engineering, OGE-2015

Developing of computational investigation methodology of Newtonian fluid in the crescent-shaped gap of turbogenerator oil-free bearing

Raykovskiy N.A.[a]*, Yusha V.L.[a], Abramov S.A.[a], Potapov V.V.[a], Zyulin D.V.[a]

[a]*Omsk State Technical University, 11, Mira Pr., Omsk 644050, Russian Federation*

Abstract

The paper suggests the methodology of computational investigation of fluid and gas flow in the crescent-shaped gap of turbogenerator oil-free bearing on the basis of ANSYS CFX package. Numerical computation data have good convergence with empirical results obtained by other scientists for the same objects. Verification data prove the possibility of computational methodology implementation, at the first stage, in a limited sphere of engineering applications, particularly, when designing cooling systems for eccentric annulus of oil-free friction assemblies.

Peer-review under responsibility of the Omsk State Technical University

Keywords: turbogenerator oil-free bearing; crescent-shaped gap; computational investigation; cooling systems

1. Introduction

One of the important issues in developing oil-free friction assemblies, such as “dry” structures of metal-polymer friction bearing, is the design of the efficient cooling system, which is especially relevant to high-speed machines, such as turbine expanders. In such machines, where rotors have high rotation frequency (up to 200000 rev/min), and non-metallic bearing is installed in the machine body, the most advanced cooling system is the cooling of surfaces forming crescent-shaped gap of tribocoupling. The development of such systems requires the investigation of fluid flow in channels, heat exchange between the medium and the crescent gap walls at various geometric, mechanical, hydraulics and thermodynamic parameters of the cooled oil-free bearing operation.

* Corresponding author. Tel.: +7-950-959-1207.
E-mail address: n_raykovskiy@mail.ru

Peer-review under responsibility of the Omsk State Technical University
doi:10.1016/j.proeng.2015.07.270

Nowadays there are a lot of researches [1–8], concerning the investigation of Newtonian and non-Newtonian fluid flow [1, 2] between two concentric [3, 4] and eccentric cylinders [5, 6], with one or two movable walls or with non-movable walls [6, 7], for the cases of cylindrical Couette flow [8], forcing axial and spiral flows [6]. However, in all the papers mentioned, as a rule, axial flow is analyzed; annulus size is more than 1 mm, while the size of the eccentricity ratio is from 0 to 0.95. Furthermore, the object under study can have various inlet fluid directions (from the radial to the axial one) and axial outlets in two opposite directions, the gap size being less than 1 mm, and the eccentricity ratio considerably differs from the parameters of the analogous objects, studied by other researchers, including those mentioned above. That is why this paper, aimed at the investigation of fluid flow in crescent-shaped gap of turbogenerator oil-free bearing, is of current interest.

2. Study subject

2.1. Physical object

The study subject is the crescent-shaped gap of the cooled oil-free friction bearing of the turbogenerator friction assembly with the fluid flowing inside (fig. 1). By fluid denote Newtonian fluids and gases contained in the turbogenerator as a working medium, or supplementary medium, such as water, fuel, air, nitrogen, natural gas and so on. The cooled oil-free bearing friction assembly is a structure (fig. 1), consisting of a rotor and a bearing, including channels between coupling surfaces, forming a crescent-shaped gap, where cooling medium is supplied. The radial bearing is considered. Mechanical contact between the bearing and the rotor, followed by the dispersion of heat energy, occurs on the surface, limited by the contact angle ($2\varphi^o$) not exceeding 180^o. Other geometric parameters of tribocoupling change throughout the wide range are defined by the machine and bearing structural characteristics.

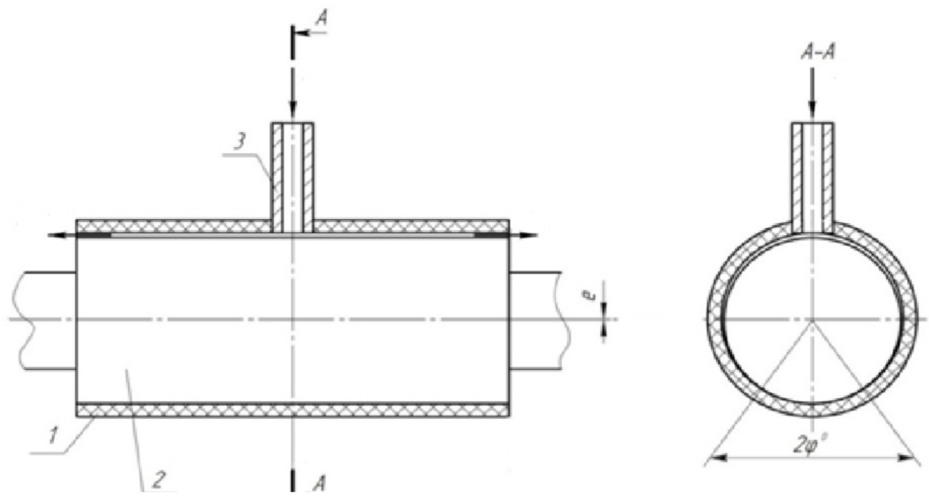


Fig. 1. Principle scheme of metal-polymer mounting friction assembly: 1 – non-metal bearing; 2 – rotor; 3 – fluid supply tube; $2\varphi^o$ – shaft and bearing contact angle

2.2. Mathematical modeling

Main assumptions, accepted in the paper, are the following: the process of gas and fluid flow is steady; heat exchange between a friction assembly, its cooling system and ambient environment is neglected; heat liberation in the friction couple is neglected; isothermal fluid flow is considered; single-phase fluid flow is considered; macro divergence of the bodies geometrical parameters are neglected; geometrical parameters, caused by the wearing process and the body temperature change, are neglected.

As a result, the system of equations [6], describing the laminar process of the fluid flow in the microgap consists of the mass-conservation equation and the impulse law:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho v) = 0, \ \frac{\partial \rho v}{\partial t} + \nabla(\rho v \cdot v) = -\nabla p + \nabla(\tau) + F,$$

where v is fluid velocity vector, τ is viscous stress tensor, F is volume force vector, p is static pressure, ρ is density.

Viscous stress tensor components $\tau_{i,j}$ are defined as follows:

$$\tau_{ij} = \mu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}),$$

where μ is dynamic viscosity; u_i is velocity vector components; δ_{ij} is Kronecker symbol.

For the investigation of turbulence flows semi-empirical turbulence models, which require the solving of averaged Navier-Stokes equations, are usually used.

Non-dimensional geometric characteristics, non-dimensional parameters and non-dimensional axial velocity are defined by the following equations [6]:

$$\theta = \frac{d}{D}, \ G = \frac{m}{g}, \ \delta = R - r, \ \varepsilon = \frac{e}{\delta}, \ \mathrm{Re}_{\omega} = \frac{\omega \cdot r \cdot \delta}{\nu}, \ \mathrm{Re}_{Q} = \frac{2\delta \cdot U}{\vartheta}, \ \overline{U} = \frac{u}{U},$$

where θ is diameter ratio, d is rotor diameter, D is diameter of the bearing internal surface, G is non-dimensional distance from the rotor wall, m is distance from the rotor surface, g is current gap height in various sections, δ is radial gap, R is radius of bearing internal surface, r is rotor radius, ε is eccentricity ratio, e is eccentricity, Re_{ω} is azimuth Reynolds, Re_Q is axial Reynolds, ν is kinematic viscosity, ω is angular rotor velocity, $\overline{U}$ is non-dimensional axial velocity, u s axial velocity component, U average axial velocity.

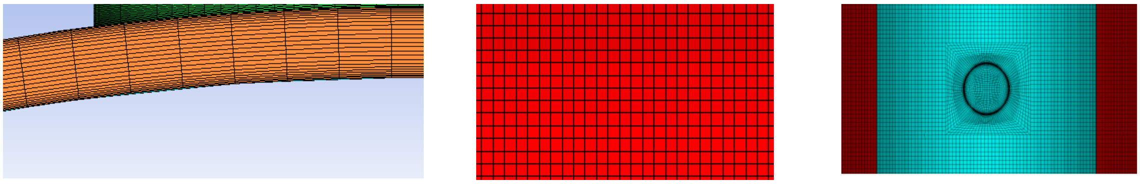
Single-valued conditions are the following: geometrical conditions are defined by diameter, axial and angular parameters of the friction pair and the tubes, as well as by the channel system, inside the bearing and on the bearing and rotor surfaces; physical characteristics of the medium (density, viscosity) were defined by using substances characteristics library ANSYS; initial conditions are neglected, because they are necessary while studying non-steady processes. The pre-set boundary conditions of the mathematical model are the following: the medium mass consumption at the inlet of the tube, the pressure at the outlet of the crescent-shaped gap ends; angle velocity, no-slip condition and roughness factor on the crescent-shaped gap wall, formed by the rotor surface; the slip condition on the wall of the crescent-shaped gap, formed by the bearing surface and on tube wall.

3. Methods

The computational model includes the following: geometrical model; grid model; fluid (gas) model; turbulence models; and boundary conditions. The development of the geometrical model as well as the fluid (gas) model is standard. Geometry development is made in SolidWorks package; while developing the fluid (gas) model physical characteristics of computational domain, analyzed in 2.2, are defined. The boundary conditions of the computational methodology are considered in 2.2.

In this paper, to complete the equation system RANS, Menter model SST was used. The main difficulty in SST model implementation is the necessity of using rather small grid in the neighborhood of the walls.

The grid development is performed in the ANSYS ICEM CFD application by the block method with the use structured hexahedral grid. After the block structure development, the conversion into unstructured hexahedral grid is made. During the series of computation, the recommended values of $y+$, for the rotor surface (fig. 2a) are defined: $y+\leq 1.5$ velocity inaccuracy is less than 2%; for the bearing surface (fig. 2a) and tubes: $y+\leq 15$. Grid cell growth factor is assumed as equal to 1.2; cell growth law is linear. Finite elements grid is performed in azimuth (fig. 2a) and axial directions (fig. 2b); it is uniform, elements quantity in these directions is accepted is equal: 1-2 cells per 1 mm of the model. The exception is the crescent-shaped gap flow block neighboring to the tube block (fig. 2c), in this part the grid is made as non-uniform, the growth factor is assumed as 1.2. The tube grid is characterized by the following parameters: in the radial direction the value of $y+\leq 15$, the growth factor is 1.2; in the axial direction – 2 cells per 1 mm, in the tube area neighboring to the gap it is necessary to provide the grid clustering according to the linear growth law with the growth ratio equal to 1.2. The size of the last tube cell is assumed as equal to the size of the neighboring cell in the area of the gap.



a b c

Fig. 2. Grid flow model: (a) radial and azimuth direction; (b) axial direction; (c) flow inlet from the tube into the gap

4. Results and discussion

Taking into account the lack of experimental data for the system, considered in 2, computational methodology verification is realized for analogous objects, differing by the system part formed by the eccentric annulus. During the suggested computational methodology verification, the computations were compares with the experimental results obtained by other authors [6, 7 and others].

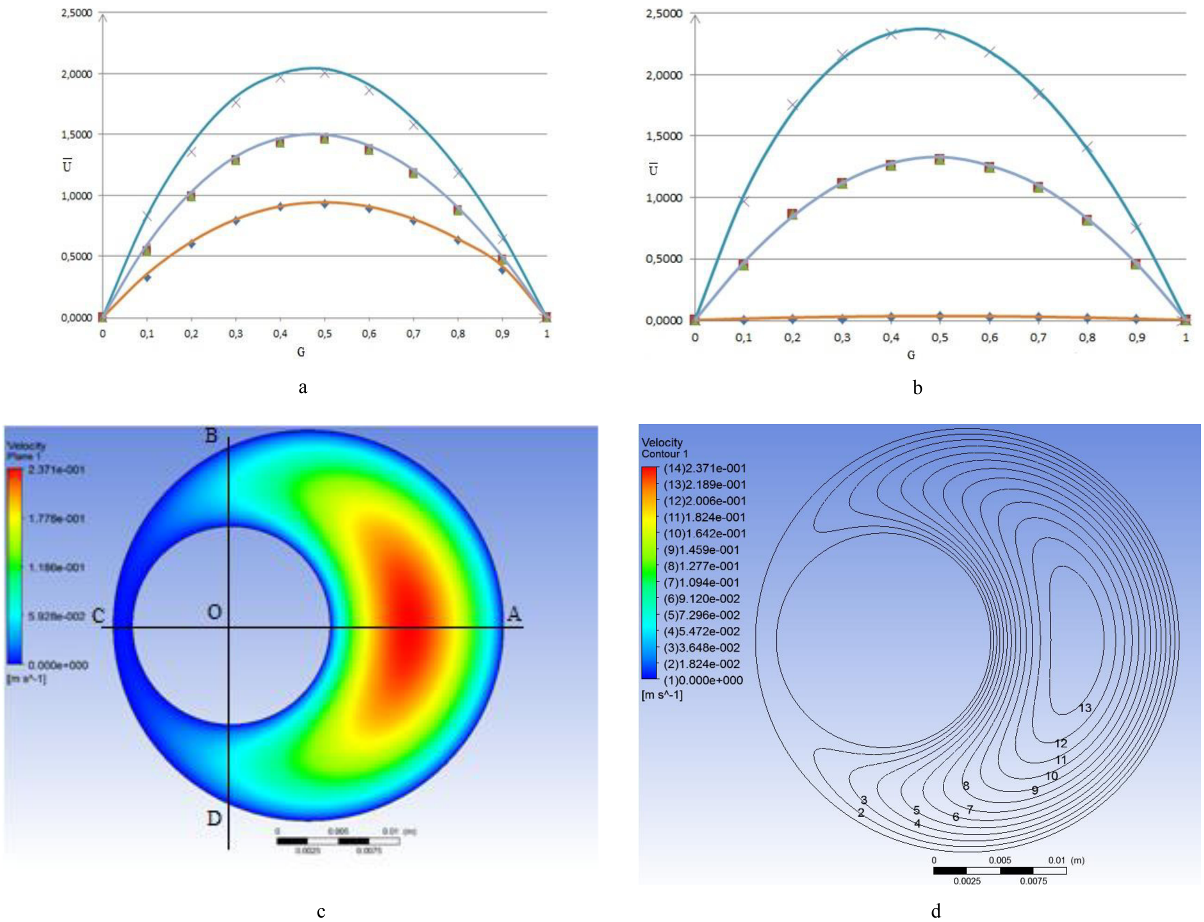


Fig. 3. Comparison of the distribution of the dimensionless axial velocity of the air flow for the computational Laminar model with experimental data [6] at Θ=0.506, ReΩ=105, Reω=0: a – ε=0.2; b – ε=0,8; c, d – velocity parametres in radial section at ε=0,8 (c – colored; d – contoured); × - section OA, ■ - section OB, ◆ - section OC, ▲ - section OD

For example, fig. 3 presents verification results for the fully developed axial laminar air flow through the eccentric cylinders. The computation made proved the adequacy of the suggested computational methodology for the study of the cooling system of the eccentric annulus, formed by the bearing and the rotor surfaces. It was established, that, concerning the laminar flow, fine precision of the experimental and computational results is provided by the computational model Laminar ANSYS CFX; concerning the transition and the turbulence flows – by the computational model SST CFX. At the same time, for the case of circular fluid flow maximum divergence values for the velocity do not exceed 1%, for eccentric channel with medium axial flow (fig. 3) – 2.5%, for spiral flow – 5%. In general, the results obtained can be regarded as satisfactory, and the computational methodology – as applicable to the tasks of this type.

5. Conclusion

In conclusion, the methodology of computational investigation of the gas flow in the crescent-shaped gap of the turbogenerator oil-free bearing on the basis of ANSYS CFX package was developed. Taking into account the lack of experimental results for the object, considered in 2, computational methodology verification was carried out for analogues objects. The comparison with the experimental research results proved the adequacy of the suggested computational methodology and the possibility of its implementation, at the first stage, in the limited area of engineering applications for investigating and developing the cooling system of crescent-shaped gap in the turbogenerator oil-free bearing. In general, average divergence of the fluid flow parameters in various coaxial and eccentric annuli, in the area of laminar and turbulence flows do not exceed 5%. It is estimated, that, in the area of the laminar fluid flow it is necessary to use computational model Laminar ANSYS CFX, in the area of the turbulence fluid flow the satisfactory divergence of the results is provided by computational model SST CFX. As a result of computation series, the recommended values for $y+$ are found: for the rotor surface $y+\leq 1.5$; for the bearing and tubes surfaces $y+\leq 15$. Further, with the aim to enlarge the sphere of the suggested methodology implementation, experimental study and verification of the computational methodology for the object, design scheme of which corresponds to the one considered in 2, are necessary.

References

[1] M. Rudman, H.M. Blackburn, Direct numerical simulation of turbulent non-Newtonian flow using a spectral element method, Applied Mathematical Modelling 30 (2006) 1229-1248.

[2] D.O.A. Cruz, F.T. Pinho, Turbulent flow predictions with a low- Reynolds number k-e model for drag reducing fluids, J. Non-Newt. Fluid Mech., 114, 109-148.

[3] S.Y. Chung, H.J. Sung Large-Eddy, Simulation of Turbulent Flow in a Concentric Annulus, Int. J. Heat and Fluid Flow. 26 (2005) 191-203.

[4] J.A. Naser, Prediction of Newtonian and non-Newtonian Flow through Concentric Annulus with Centerbody Rotation, Int. Conf. on CFD in Mineral and Metal Processing and Power Generatio, CSIRO, 1997.

[5] G. Ooms, B.E. Kampman-Reinhartz, Influence of drill pipe rotation and eccentricity on pressure drop over borehole during drilling, SPE Drill and Completion. Vol. 15, 4 (2000) 249-253.

[6] M.P. Escudier, I.W. Gouldson, P.J. Oliveira, F.T. Pinho, Effects of inner cylinder rotation on laminar flow of a Newtonian fluid through an eccentric annulus, Internat. J. Heat and Fluid Flow. 21 (2000) 92-103.

[7] L.N. Tao, W.F. Donawan, Through-Flow in Concentric and Eccentric Annuli of Fine Clearance with and without Relative Motion of the Boundaries, Transactions of the ASME, 1955, pp. 1291-1301.

[8] J.M. Nouri, J.H. Whitelaw, Flow of Newtonian and non-Newtonian Fluids in an Eccentric Annulus with Rotation of the Inner Cylinder, Int. J. Heat and Fluid Flow. V. 18, 2. (1997) 156-162.