Tribology International 92 (2015) 290–300

Contents lists available at ScienceDirect

Tribology International

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

Influence of spin flow on lubricating oil jet—Design method of oil spray parameters to high speed spur gears

Yanzhong Wang [a], Wentao Niu [a,*], Song Wei [b], Guanhua Song [a]

[a] School of Mechanical Engineering and Automation, Beihang University, Beijing, China
[b] School of Mechanical Engineering and Automation, Liaoning University of Technology, Liaoning, Jinzhou, China

ARTICLE INFO

Article history:
Received 26 March 2015
Received in revised form
12 June 2015
Accepted 7 July 2015
Available online 26 July 2015

Keywords:
Spur gear
Jet lubrication
Spin flow
CFD

ABSTRACT

As the gear speed becomes higher and higher in modern industry the influence of spin flow on lubricating oil jet must be taken into account in design of gearings. A multiphase flow model is established in computational fluid dynamics (CFD) to calculate the regimes of oil jet in spin flow. Schemes are developed to eliminate the effect of oil jet deviation in into mesh lubrication. The contra-flow in out of mesh condition is analyzed and found that the complete contra-flow arises when spray velocity is less than 1% (or higher for large module) of pitch velocity. The analysis results were verified by experimental results.

1. Introduction

The temperature is very important to the working performance of high speed and heavy load gears, in which the scoring, pitting and scuffing [1–3] happen frequently. In the jet lubrication which is commonly used in high speed gears, the oil spray parameters have a direct influence on the lubrication and cooling of the gear [4–8], hence determine the gear temperature. The design of the oil spray parameters has significant means to reduce the temperature of high speed and heavy load gears, as a result, considerably more power capacity can be provided by smaller gear drives.

The lubricating oil jet is deflected by the spin flow that could obstruct it from entering the meshing zone for lubrication or reaching to gear faces for cooling. The effect of spin flow on lubricating oil jet becomes more and more striking with the increasing of the gear rotation speed, however, it is still not considered adequately in the design of jet lubricated gears. The impingement and penetration depth of lubricant jet flow into a gear tooth space were analyzed in Refs. [9–11], their experiments indicate that the spin flow has little influence on the jet when the gear speed is slow and oil jet is thick. The fling-off cooling method is studied in Refs. [12,13] and different methods are discussed theoretically. The windage power loss is experimentally studied in Refs. [14–16] and numerical models are established in Refs. [17–19], their results show positive correlation between windage power loss and gear speed, and several methods to reduce the loss are discussed. The CFD method is applied to calculate the windage power loss of gear [20–23] and the spin flow around the gear is obtained.

The paper researches on the jet lubrication of high speed and heavy load spur gears. The direction deviation of the jet in into mesh lubrication as well as the contra-flow in out of mesh condition is studied in detail by an oil/air multiphase flow model established in CFD. The methods to eliminate the effect of deviation of oil jet and prevent the occurrence of contra-flow are developed and verified by experiments.

2. Model of oil jet and spin flow

2.1. Numerical model

The volume of fluid (VOF) method [24–25] can be used to model the oil/air multiphase flow which is formed when the lubricating oil issues in air. In order to arrive an appropriate efficiency and accuracy, the heat transfer and chemical reactions are ignored. The governing equations are given as follows.

a. Volume conservation equation

$$\sum_{\alpha=1}^{N} r_\alpha = 1 \tag{1}$$

* Corresponding author. Tel.: +86 15210980435.
E-mail address: w.niu@buaa.edu.cn (W. Niu).

http://dx.doi.org/10.1016/j.triboint.2015.07.017
0301-679X/

the lowercase letter α denote the fluid phase. r_α is the volume fraction of α phase. N is the number of fluid phases, here the fluid phases are oil and air, $N=2$.

b. Continuity equation

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha) + \nabla \cdot (r_\alpha \rho_\alpha U_\alpha) = S_{MS\alpha} + (\Gamma_{\alpha\beta} - \Gamma_{\beta\alpha}) \tag{2}$$

ρ is density, U is flow velocity, $S_{MS\alpha}$ is specified mass sources and here is 0, $\Gamma_{\alpha\beta}$ is the positive interphase mass flow rate per unit volume from phase β to α.

c. Momentum equation

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha U_\alpha) + \nabla \cdot (r_\alpha(\rho_\alpha U_\alpha U_\alpha)) = -r_\alpha \nabla p + \nabla \cdot (r_\alpha \mu_\alpha (\nabla U_\alpha + (\nabla U_\alpha)^T)) + (\Gamma_{\alpha\beta} U_\beta - \Gamma_{\beta\alpha} U_\alpha) + S_{M\alpha} + M_\alpha \tag{3}$$

where p is static pressure, μ is viscosity, $S_{M\alpha}$ describes momentum sources due to external body forces, M_α describes the interfacial forces acting on phase α due to the presence of phase β.

d. Turbulence model and flow near the wall

Homogeneous model is used to the calculation of turbulence, in which the turbulence fields is shared by the two fluids. The k-epsilon Model is employed for its broad applicability and high precision [26]. The turbulence viscosity is defined as:

$$\mu_t = C_\mu \rho k^2 / \varepsilon \tag{4}$$

where C_μ is the model constant equal to 0.09, k and ε are the turbulence kinetic energy per unit mass and the turbulence dissipation rate, which obtained by the differential transport equations for the turbulence kinetic energy and turbulence dissipation rate.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon + P_{kb} + S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho U_j \varepsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}(C_{\varepsilon 1} P_k - C_{\varepsilon 2}\rho\varepsilon + C_{\varepsilon 1} P_{\varepsilon b}) + S_\varepsilon \tag{6}$$

where $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ is the model constant equal to 1.44 and 1.92, P_k is the turbulence kinetic energy due to the gradient of average velocity, P_{kb} is the turbulence kinetic energy due to the buoyancy, here is zero; σ_k and σ_ε is Prandtl number; S_k and S_ε is external source.

The wall function method [27] is employed to calculate the flow near the no-slip wall, in which the fluid shear stress is a function of the tangential velocity. The empirical formulas are used to avoid the need to resolve the boundary layer and account for viscous effects in the turbulence model. The near wall velocity is given by:

$$u^+ = U_t / u_\tau = \ln(y^+)/\kappa + C \tag{7}$$

where U_t is the velocity tangent to the wall at a distance of Δy from the wall, κ is the von Karman constant and C is a log-layer constant depending on wall roughness (here a smooth wall is

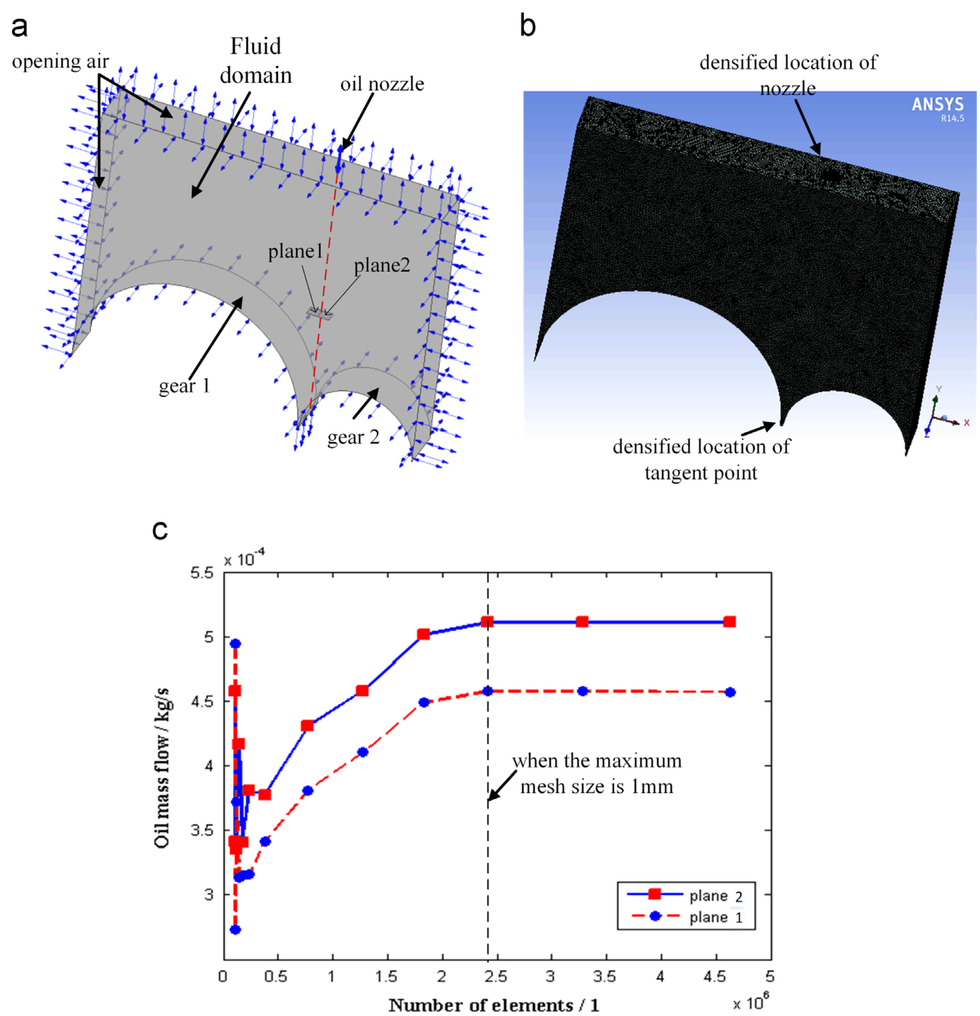

Fig. 1. CFD model.