Analysis of Lubricant Flow through Reynolds Equation

J.S. Crompton[1], L.T. Gritter[1], S.Y. Yushanov[1], K.C. Koppenhoefer[1], and R.O. Edwards[2*]
[1]AltaSim Technologies, LLC, [2] Cummins Fuel Systems
*Corresponding author: 130 East Wilson Bridge Road, Suite 140, Columbus, OH 43085, kyle@altasimtechnologies.com

Abstract: Reynolds equation is used to analyze fluid flow through small gaps. As such, the solution of Reynolds equation provides critical information for a wide range of tribological problems. In any case where a lubricant resides between two moving surfaces (e.g., thrust and journal bearings), the Reynolds equation can be used to solve for the flow. In the case considered in this paper, lubricant flows between a moving piston and the housing. A pressure decreases along the flow path cases the lubricant to heat and this heat is conducted through the piston and housing. The length scale difference between the lubricant thickness and the piston diameter represents a significant challenge to multiphysics software packages that do not have the ability to include weak form boundary conditions. In COMSOL Multiphysics v3.5a, Reynolds equation could only be fully implemented as a weak form boundary condition. This paper shows the implementation of Reynolds equation using the weak form.

Keywords: Reynolds equation, lubrication, weak-form boundary conditions.

1. Introduction

Many solutions in tribology use a thin layer of fluid between two surfaces to reduce friction and wear of the adjacent surfaces. These lubrication solutions can be found in industries that range from medical products to metal stamping. In each case, lubrication decreases the force necessary to move the adjacent surfaces, and increases the mean time between part replacement. Thus, solving this general class of problems provides a significant benefit to a broad range of industries.

2. Governing Equations

The equations that represent fluid flow in these hydrodynamic lubrication problems are often referred to as Reynolds equations due to the work of Osborne Reynolds in the late 1800s. These equations derive from the general Navier-Stokes equations based on the following assumptions: (1) the thickness of the fluid is small compared to the length and width, (2) the pressure gradients through the thickness of the fluid are small, (3) no external forces act on the fluid film, (4) no slip at the bearing surfaces, and (5) velocity gradients along the thickness dominate all other velocity gradients. Thus, the general fluid flow equations

$$\rho\ \mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p + \nabla\cdot\left(\eta\left(\nabla\mathbf{u} + (\nabla\mathbf{u})^T\right) - \frac{2}{3}\eta(\nabla\cdot\mathbf{u})\mathbf{I}\right) + \rho\ \mathbf{g}$$

can be simplified to

$$\begin{cases} \frac{\partial}{\partial x}\left(\frac{\rho h^3}{12\eta}\frac{\partial P}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{\rho h^3}{12\eta}\frac{\partial P}{\partial y}\right) = \frac{\partial}{\partial x}\left(\frac{\rho U_0 h}{2}\right) + \frac{\partial}{\partial y}\left(\frac{\rho V_0 h}{2}\right) + \rho W \\ P\big|_{y1} = P_1; \quad P\big|_{y2} = P_2 \end{cases}$$

where, P is pressure, h is gap, η is viscosity, ρ is density, and U_0, V_0, W_0 is velocity.

3. Use of COMSOL Multiphysics

The models described in this paper were constructed using COMSOL Multiphysics v3.5a running on a computer with Red Hat Enterprise Linux v5.4. The analyses consist of heat transfer by conduction (ht) with the Reynolds implemented as a weak-form boundary conditions (wb). The analyses also considered the thermal expansion using the stress-strain features in the structural mechanics module (smsld). This module provides the thermal expansion capabilities that are not native to the COMSOL Multiphysics base package.

Figure 1 shows a sketch of the axisymmetric model developed for this work. The lubricant flows between the plunger and the barrel. Pressure applied at the top of the barrel drives the lubricant flow. As the plunger moves vertically, it modifies the lubricant flow. The weak-form boundary condition is applied to the outer diameter of the plunger and the inner diameter of the barrel.

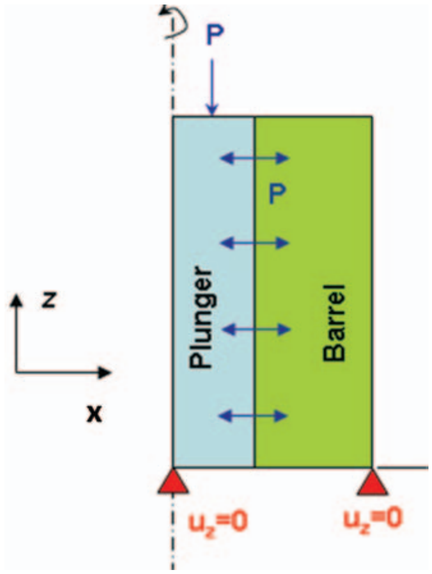

Figure 1. Axisymmetric model of lubrication path between a plunger and barrel subjected to large internal pressure.

To apply these weak-form boundary conditions, the Reynolds equation must be derived in the weak form. The Reynolds equation can be recast as

$$\nabla \cdot (-\mathbf{q}) = \rho W_0$$

Where $\mathbf{q}$ is the mass flow rate

$$\mathbf{q} = \left\{ \rho \left(\frac{U_0 h}{2} - \frac{h^3}{12\eta} \frac{\partial P}{\partial x} \right), \rho \left(\frac{V_0 h}{2} - \frac{h^3}{12\eta} \frac{\partial P}{\partial y} \right) \right\}.$$

By multiplying each side of the equation by P_{test} and integrating over the domain Ω , yields the following equation

$$\int_{\Omega} P_{test} \nabla \cdot (-\mathbf{q}) d\Omega = \int_{\Omega} P_{test} \rho W_0 d\Omega$$

By using the identity,

$$\nabla \cdot (P_{test} \mathbf{q}) = P_{test} \nabla \cdot \mathbf{q} + \mathbf{q} \cdot \nabla P_{test}$$

along with Gauss' theorem, the equation may be written as

$$\int_{\partial\Omega} P_{test} \mathbf{q} \cdot d\mathbf{n} - \int_{\Omega} \mathbf{q} \cdot \nabla P_{test} d\Omega = \int_{\Omega} P_{test} \rho W_0 d\Omega$$

where, **n** is a unit normal to the surface. By introducing a Lagrange multiplier, μ, and the Neumann boundary condition, the weak-form of the Reynolds equation may be written as

$$\begin{cases} 0 = \int_{\Omega} \left(\nabla P_{test} \cdot \mathbf{q} + P_{test} \rho W_0 \right) d\Omega + \int_{\partial\Omega} P_{test} (G + \mu H) dn \\ 0 = \left(P_0 - P \right)_{\partial\Omega_1} \end{cases}$$

These equations can then be implemented in PDE mode as a weak-form boundary application mode for a stationary analysis.

The critical parameters for this model appear in Table 1. The parameters *a* and *b* define the relationship between temperature and pressure for the lubricant. As pressure decreases, the temperature of the lubricant increases. The pressure along the lubrication path decreases form an initial pressure of P_2

Table 1: Model Parameters

Name	Expression	Description
g_0	0.0057e-3[m]/2	initial gap
P_2	2600[bar]	inlet pressure
P_1	0[Pa]	outlet pressure
ρ	1e3[kg/m^3]	fluid density
V_0	1[m/s]	plunger velocity
η_0	6e-3[Pa*s]	reference viscosity
a	-24.7713	P=a*T+b, [P]=bar; [T]=C
b	4600.03	P=a*T+b, [P]=bar; [T]=C
E	2e11[Pa]	elastic modulus
ν	0.33	Poisson ratio
dens	7850[kg/m^3]	density of solid
α	1.2e-5	CTE
K_{th}	50[W/(m*K)]	solid thermal conductivity
T_0	80	inlet temperature, [C]
T_{ref}	25	TE reference temperature, [C]

4. Results

The results developed from this model were compared against the exact solution of the Reynolds equation. As shown in Figure 2, the COMSOL solution shows excellent agreement with the exact solution. The pressure varies nonlinearly along the lubrication path from the atmospheric pressure at the bottom of the plunger to the high pressure in the reservoir located at arc length of one. Results were also developed from the COMSOL model at plunger velocities of 10 m/s upward and downward. These results show that the pressure variation does not depend on the velocity of the plunger.

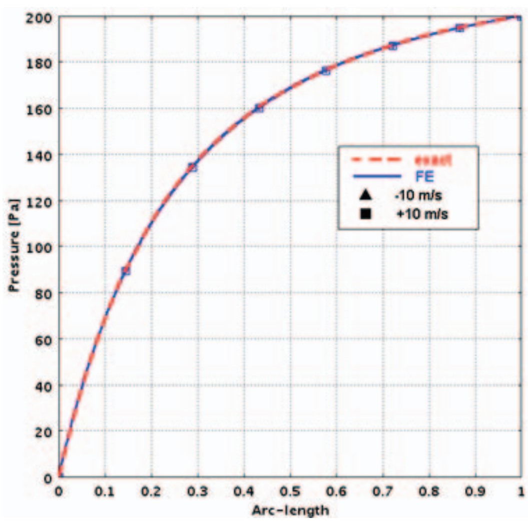

Figure 2. Variation of pressure along lubrication path for plunger velocities of 0, -10 and +10 m/s. The arc length is defined from both to top of plunger.

Although the pressure along the path length remains constant with different velocities, the flow rate of lubricate varies with plunger velocity. Figure 3 shows the effect of plunger velocity on flow rate. The model defines the direction of flow (down) as negative. Thus, a negative velocity moves the plunger in the direction of flow, and increases the flow rate. A positive plunger velocity works against the flow and decreases flow rate. In each case, COMSOL accurately calculates the flow rate as compared to the exact solution.

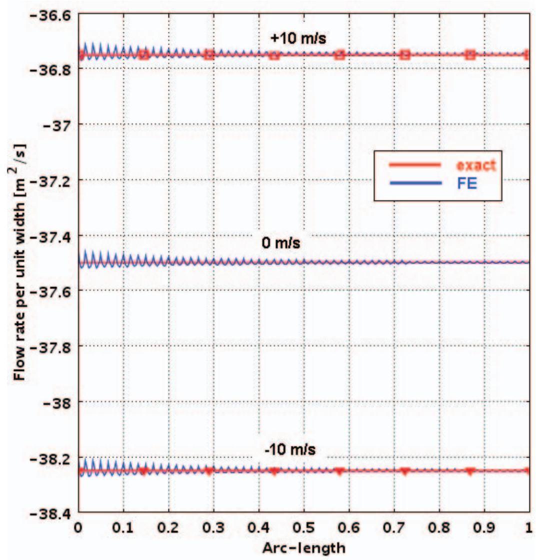

Figure 3. Variation of flow rate for plunger velocities of 0, -10 and +10 m/s. The arc length is defined from both to top of plunger.

The pressure drop in this problem causes the temperature of the lubricant to increase. Thus, the flow acts as a heat source to the solids. This temperature increase produces deformation and stress in the structure.

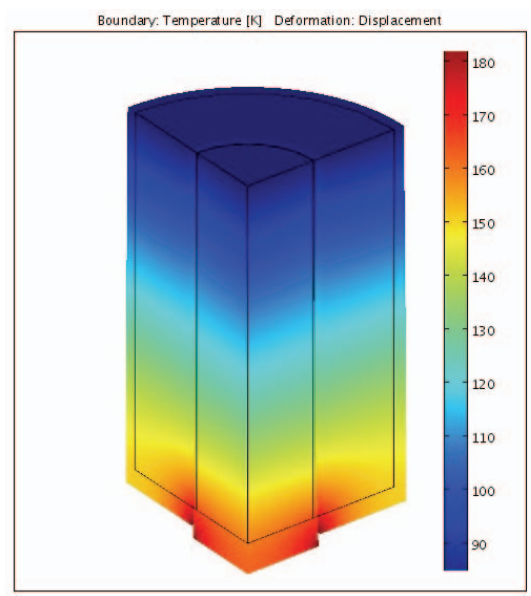

Figure 4. Variation of temperature in the plunger and barrel.

Figure 4 shows the variation of temperature in the plunger and barrel. As lubricant flows from the top to the bottom, it increases in temperature. The plunger and barrel conduct the heat from this source and produce the observed temperature gradient.

The temperature gradient combined with the thermal expansion of the material and restrains shown in Figure 1 produce a stress in the plunger and housing.

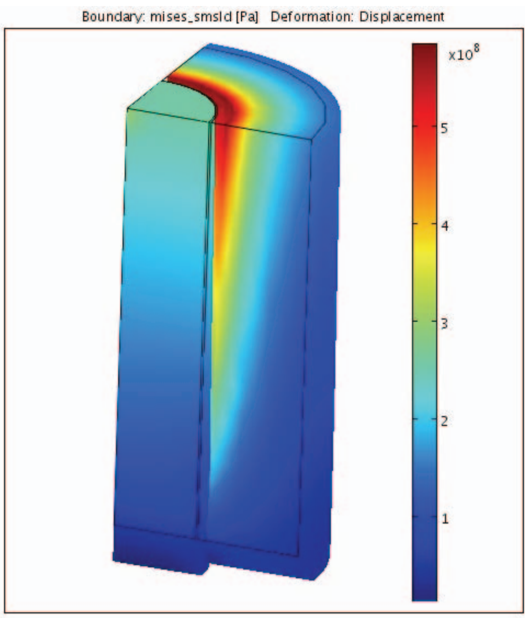

Figure 5. Variation of Mises stress in the plunger and barrel.

The material at the top with the lower temperature experiences less thermal expansion than the hotter material at the bottom of the plunger and barrel. Thus, top section restrains the bottom section from expanding. This restraint leads to high stresses near the top.

Although not presented in this paper, additional information can be calculated for this case. The distance between the plunger and barrel can be calculated to determine if any location along the flow path might choke off the flow of lubricant. In addition, the temperature dependant material properties of the lubricant (e.g., viscosity) could be calculated to determine if the lubricant breaks down with temperature.

5. Conclusions

This work demonstrates the capabilities of COMSOL Multiphysics to solve the complex problem of lubricant flow in a narrow gap between two solid parts. The strength of COMSOL for this class of problems is the ability to define the Reynolds equation as a boundary condition in the model, and eliminate the need to mesh this region. Other multiphysics codes attempt to solve this problem by meshing this region and solving the Navier-Stokes equations along the leak path. This method produces meshing difficulties due to the difference in length scales between the flow region and the rest of the model.

The work conducted here was implemented using COMSOL Multiphysics v3.5a. In v4.0a, this capability has been implemented for the user in the CFD module. Thus, the weak form boundary conditions identified here are not required in v4.0a. For users without access to the CFD module, this method remains valid, and it can be implemented with only the base package COMSOL Multiphysics.