

Spin Loss Computation for Open Clutch Using CFD

2011-01-1238

Published
04/12/2011

Anand K. Jammulamadaka and Prajal Gaokar
GM Technical Center India Ltd. - General Motors Company

Copyright © 2011 SAE International

doi:[10.4271/2011-01-1238](https://doi.org/10.4271/2011-01-1238)

ABSTRACT

Open clutch spin loss computation is of interest for new clutch designs. It is desirable to minimize open clutch spin loss. Spin loss in automatic transmission clutches is mainly due to the viscous shear of the transmission fluid. Depending on the relative rotational speed of the plates the spin loss varies. At low rotational speeds the gap between the plates is filled with ATF (Automatic Transmission Fluid) and spin loss increases linearly with the rotational speed. At higher speeds the ATF layer, which is held together primarily by surface tension, begins to breakdown due to higher centrifugal forces and air pockets form at outward radial locations of the clutch plates. This results in a decrease in spin loss for the open clutch. CFD (computational fluid dynamics) modeling is an attractive option in calculating the spin loss for an open clutch. Not only can a CFD model give an accurate estimation of the magnitude and trend of spin loss variation but it can also compare various groove designs. This is a significant advantage compared to some existing analytical methods for spin loss estimation of open clutches, as they are limited to either groove-less designs or designs with radial grooves. This work describes a 3D CFD modeling procedure that yields spin loss computations of sufficiently high accuracy by improvements made to some stated procedures in literature. Effect of variation of design parameters such as number of grooves, flow rates, etc, is studied.

INTRODUCTION

Automatic transmission fluid (ATF) that passes between the clutch plates performs the dual functions of lubrication as well as cooling. Heat generated during the clutch engagement, hold and release cycles due to friction between the clutch plates' surfaces is carried away by the ATF exiting the domain. The ATF also reduces wear of the clutch plate

surfaces, thereby increasing the durability of the clutch. For an open clutch the viscous shear forces acting on the rotating surfaces result in loss of power and a reduction in fuel economy. As a result, it is desirable to minimize spin loss for an open clutch. Currently open clutch spin loss is mainly obtained through testing. A computational approach for estimation of spin loss for an open clutch is complicated due to the existence of a two-phase flow regime consisting of oil and air. Some analytical tools are available that are based on empirical relations or simplified flow equations. These depend on calculation of an area factor to estimate the decrease in oil cover at higher rotational speeds. Even though these tools have been developed to good level of accuracy, their application is mainly to either groove-less designs or designs with radial grooves. In this aspect CFD offers a powerful alternative to get results of potentially higher accuracy for various groove profiles. Limitations exist in setting up the computational domain to obtain a good estimate of the spin loss. The clutch clearance is typically assumed as uniform while modeling. However, during tests it is noted that the clutch plates may not be uniformly spaced. Also, the nominal clutch clearance may vary with the rotational speeds. Tests show that the drainage of the ATF from the clutch has an impact on the clutch spin loss and depending on the type of drainage system in the clutch housing the clutch spin loss test data may vary. These effects are generally ignored during the development of a computational model.

Some analytical modeling efforts exist in literature that predict drag torque for an open automatic transmission clutch. An open clutch drag torque calculation model is available based on application of lubrication approximations for the navier-stokes equations [1]. The model, however, aims at showing aeration inception and is limited to the low rotational speed, single phase region. Mathematical models

for open clutch drag torque covering both low and high rotational speed regions are available. They are based on both laminar and turbulent flow assumptions and with and without inclusion of surface tension. They are obtained by solving the Navier-Stokes equations [2]. The models however are limited to grooveless clutch plate designs. Drag torque computation from a CFD model by considering a two phase flow regime has been performed [3]. However, the correlation with test data is achieved through arbitrary scaling of viscosity and surface tension. The current study aims to improve spin loss correlation by adding relevant physics to the model.

FLOW PHYSICS

The flow in an automatic transmission clutch is a two phase flow regime. It is a well known fact that automatic transmission fluids contain dissolved air, in some cases up to 14 % [4]. Air, in addition, can enter the gap between the plates from outside through the oil discharge holes. At low rotational speeds the forces acting on the oil film are dominated by the surface tension. Hence, the oil film covers nearly the entire surface of the clutch plates. At higher rotational speeds the centrifugal forces dominate and cause the film to break up. There is sufficient pressure drop for the dissolved air to be released in the form of bubbles. Significant quantity of dissolved air is released to form air cavities on the outer radial locations of the clutch plates. The force acting on the clutch plates of a open clutch is primarily the viscous shear force. Thus the rotation of the clutch plate results in generation of heat, heating up the oil as it passes through the gap between the friction and reaction plate. The heat thus generated is dissipated either by the exit of the heated fluid from the domain or through the clutch plates and housing to the atmosphere. In past modeling efforts both laminar [1,3,5] and turbulent [2,6] flow regimes have been considered to model the flow in the narrow gap between the plates. However, most approaches are based on the laminar assumption as wall effects are expected to be dominant for a narrow gap flow making the laminar flow assumption reasonable.

CFD MODEL

In the current effort, FLUENT (12.r.1) is used to model and solve the flow field. The two phase flow regime is modeled using the Volume of Fluid (VOF) scheme. An alternative multiphase scheme, Eulerian scheme, which solves separate momentum and continuity equations for each phase was tested as this would give results of higher accuracy as compared to the VOF scheme. However, no perceptible improvement is seen over the results obtained with VOF scheme. Hence, VOF scheme is chosen to model the two-phase flow regime due to the lower computational effort required. The implicit formulation of VOF scheme is chosen for the model as a steady state flow field is being computed. The flow is assumed to be laminar and incompressible.

Surface tension and wall adhesion effects are included. For all the results in this effort the oil is assumed to contain 10 % dissolved air. Effect of gravity is neglected.

A small section of the gap between the plates is chosen as the computational domain. This is based on the assumption that the clearance between the clutch plates is uniform and flow is periodic in the rotational direction. In the current effort only radial grooved friction plates have been considered for CFD analysis. The considered computational domain consisted of a circular section with 2 radial grooves.

The inlet boundary is modeled as a constant velocity inlet. The outlet boundary is assumed to be a natural pressure boundary. The grooved face boundary is modeled as a wall with no-slip shear condition. This wall is provided a rotational motion to implement the clutch rotational motion. The flat face boundary is modeled as a stationary wall with no-slip shear condition. Rotational periodic boundary condition is applied to the faces along the radial edges. [Figure 1](#) shows the schematic of the flow domain as well as the boundary conditions applied. The flow domain consists of a pair of adjacent reaction and friction plate faces and the clearance volume in between. The dashed line is an imaginary line that encloses the computational domain.

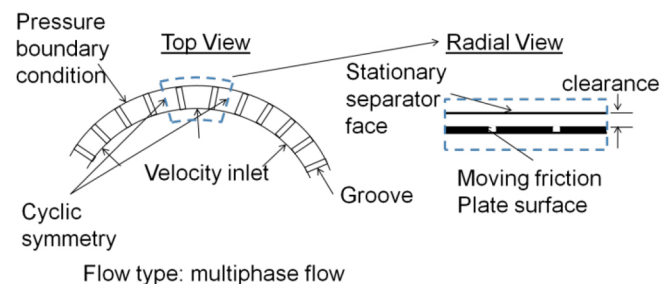


Figure 1. Schematic of the flow domain and the boundary conditions applied in CFD model

Mesh consists of approximately 50,000 hexahedral cells. 10 layers of cells are used to discretize the axial clearance. To assess variance due to mesh, finer meshes were considered with up to 150,000 hexahedral cells. However, variation of peak value drag torque value of less than 3 % is observed. [Figure 2](#) shows the hexahedral mesh used in CFD model.

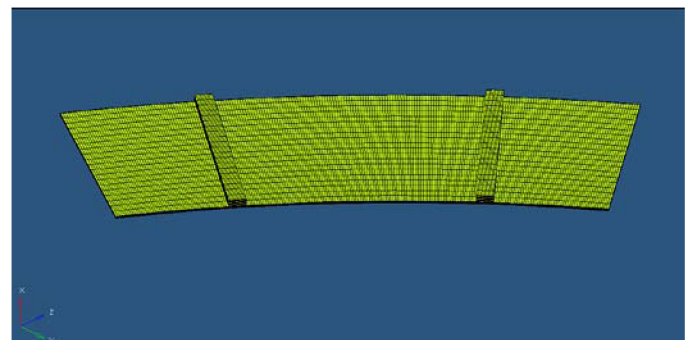


Figure 2. Mesh consisting of approximately 50,000 hexahedral cells

The flow domain is initialized with zero values for velocity, oil volume fraction and gauge pressure. The solution is run with default under-relaxation factor solution controls. Under-relaxation factors slow down the rate at which solution changes thereby providing solution stability. In FLUENT these values are set to their optimum and don't need to be varied for most flow problems. To assess convergence criteria, in addition to monitoring the residuals the net mass flux from the domain is monitored and ensured that it is at least two orders of magnitude less than the mass flux through the inlet and outlet boundaries.

CFD MODEL VALIDATION

In order to validate the CFD model the simulation data is compared with test data. The correlation is performed assuming that the test data is accurate and that the assumptions made in the CFD model hold with respect to the test apparatus and procedure. [Table 1](#) lists the test data parameters.

Table 1. Parameters for the test data.

Number of friction surfaces	10
Number of grooves on each friction surface	32
Groove type	radial
Inner diameter	170.12 mm
Outer diameter	200.3 mm
Clutch clearance	1.5 mm
Oil inlet temperature	40 C
Oil flow rate	1.0 liter/min
ATF	Dexron VI
Groove width	1.2 mm
Groove depth	0.495 mm
Hub rotational speed	400-3000 RPM

The oil inlet temperature is specified as 40 C in the simulation model. Steady state solutions are obtained at various rotational speeds across the range of the rotational speeds used during test. [Figure 3](#) shows the oil layer extent over a domain mid-plane at various rotational speeds.

The drag torque calculated for the two-grooved section is scaled by value of 160 to obtain the drag torque for the entire clutch pack. [Figure 4](#) shows the comparison of the drag torque predicted by CFD model and the test data.

The slope of CFD model predicted drag torque at lower speeds is close to that seen in the test data. The drag torque peaks at a certain rotational speed and follows a decreasing trend with increasing rotational speed. However, the model over predicts both the peak value as well as the torque values at higher rotational speeds. The location of the peak is also wrongly predicted. In order to improve the correlation, the assumption that both phases are incompressible is relaxed and air is treated as compressible fluid. This consideration is more appropriate as with compressible air the size of air cavities will become dependent on the local pressure value. And since drag torque is primarily dependent on the extent of oil cover on the friction plate surface it will have a direct affect on the drag torque value at higher rotational speeds. [Figure 5](#) shows the comparison with air treated as a compressible fluid.

Improvement is seen in the predicted peak location and a reduction is seen in the higher prediction of drag torque values at higher rotational speeds. In order to further improve correlation a sensitivity study is performed with model parameters such as, operating pressure, volume fraction of dissolved air, outlet pressure and oil temperature. Oil temperature is found to have most influence on the calculated drag torque value primarily due to the sensitivity of oil viscosity with temperature. It is well known that the viscous shear experienced by the oil film due to the rotating friction plate results in heat generation. In order to capture this effect, the energy equation is added to the model. In FLUENT, the viscous heating option under the viscous model is turned on to include the viscous dissipation terms in the energy equation. Temperature dependent density, viscosity, surface tension, thermal conductivity and specific heat for the oil are considered. The generated heat is either carried away by the fluid exiting the domain or dissipated through the clutch plates and casing to the atmosphere. The heat dissipation capacity of friction surfaces is very low compared to the steel separator plates [7]. Hence, heat is assumed to be dissipated through the steel separator plate surface. Conduction of the heat from the steel stationary plates to the housing through the spline joint is considered to be an effective mode of heat transfer for automatic transmission clutches [8]. The transfer of the heat from the fluid to the separator wall is implemented through the wall convective heat transfer model in FLUENT. This requires specification of a convective heat transfer coefficient (HTC) as well as a sink temperature. The convective HTC for open clutches is dependent on various factors such as wall velocity, radial location, clutch plate and housing design, etc. In order to keep modeling simple a single lumped value for the HTC has been used in the current analysis for the entire stationary plate face. Such an averaged value for HTC, viz. 1500 W/m²K, is available from a separate analysis based on clutch temperature test data for a