

Two-Phase MRF Model for Wet Clutch Drag Simulation

Pengchuan Wang and Nikolaos Katopodes
University of Michigan

Yuji Fujii
Ford Motor Company

ABSTRACT

Wet clutch packs are widely used in today's automatic transmission systems for gear-ratio shifting. The frictional interfaces between the clutch plates are continuously lubricated with transmission fluid for both thermal and friction management. The open clutch packs shear transmission fluid across the rotating plates, contributing to measurable energy losses. A typical multi-speed transmission includes as many as 5 clutch packs. Of those, two to three clutches are open at any time during a typical drive cycle, presenting an opportunity for fuel economy gain. However, reducing open clutch drag is very challenging, while meeting cooling requirements and shift quality targets. In practice, clutch design adjustment is performed through trial-and-error evaluation of hardware on a test bench. The use of analytical methodologies is limited for optimizing clutch design features due to the complexity of fluid-structure interactions under rotating conditions. This article presents a two-phase Multiple Reference Frame (MRF) CFD model for simulating wet clutch behaviors, accounting for detailed design geometry. The model employs the Volume of Fluid (VOF) method to determine the air-fluid interface inside a computational domain. Model setup and simulation parameters, including initial conditions, boundary conditions, and relaxation factors are evaluated in terms of convergence behaviors. The model capabilities are validated against experimental data. Convergence of the CFD solver is demonstrated, capturing peak drag location as a function of rotating speed, until the phase fraction drops to a small value. The CFD model provides analytical insight into complex fluid interactions for grooved rotating plates, complementing hardware-based clutch design processes.

CITATION: Wang, P., Katopodes, N., and Fujii, Y., "Two-Phase MRF Model for Wet Clutch Drag Simulation," *SAE Int. J. Engines* 10(3):2017, doi:10.4271/2017-01-1127.

INTRODUCTION

Modern automobiles aim at providing transportation at higher efficiency with reduced environmental impact. Every component in a vehicle is carefully examined for an opportunity to improve fuel economy. Among them, an automatic transmission system undergoes continuous design refinements. The main function of the transmission system is to transfer drive torque from an engine to the wheels according to driver demands and operating conditions. In a typical design, multiple planetary gear sets are connected in a complex manner to achieve a desired number of gear ratios. Wet clutches [1] are commonly utilized to couple or de-couple gear elements to alter torque paths for automatic ratio-changing. When a clutch is actuated for engagement, its slip torque is directly transferred to a transmission output shaft, affecting both drivability of the vehicle and comfort of the occupants [2]. When the clutch is open, its viscous drag contributes to the measureable loss of fuel efficiency. Accordingly, the clutch remains one of the critical components in the transmission system where opportunities continues to exist for design improvements.

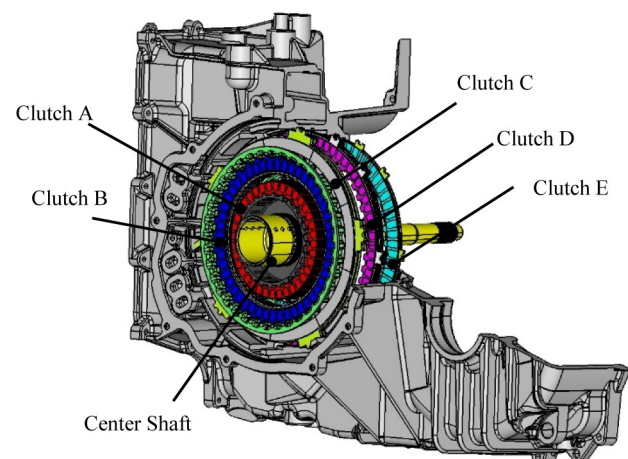


Figure 1. Illustration of clutch packs in an automatic transmission

Figure 1 illustrates a transmission system which includes 5 clutch packs in a nested structure. The Automatic Transmission Fluid (ATF) is supplied from the oil pump, travels through the center shaft, and

flows radially to lubricate all the internal components. ATF enters each clutch pack from its inner hub, travels through the frictional interface, and is discharged from the drain holes of the clutch housing. Under any driving conditions, two to three clutch packs are typically open, shearing ATF for viscous drag and thus affecting overall drivetrain efficiency [2]. When a clutch is commanded for engagement, heat is generated by the frictional force between the sliding plates. A sufficient amount of ATF must be channeled to the plate surfaces for cooling in order to avoid thermal damage and maintain stable frictional performance [3].

Figure 2 shows the detailed structure of Clutch C. Multiple friction plates and separator plates are alternatively placed around the inner hub. More specifically, friction plates are mounted to the inner hub through spline teeth, while separator plates are directly splined to the outer structure or the transmission case. A rotational motion of the inner hub generates a centrifugal force that pushes the ATF into the clutch pack through lubrication holes on the hub. When the clutch is open, the friction plates can rotate independently of the separator plates. However, a thin ATF layer between sliding plates generates parasitic viscous drag. When the clutch is engaged, the friction plates and separator plates become securely coupled to ground the gear components attached to the inner hub of the transmission case in this example. The separator plates also act as a sink that dissipates the heat generated at the frictional interfaces during engagement.

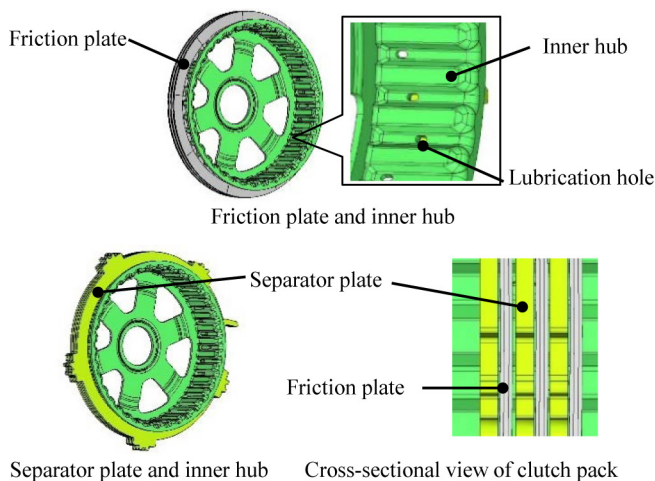


Figure 2. Detailed structure of Clutch C

The structure of a clutch pack can be complex with design considerations extending from fluid-structure interactions to friction material and ATF properties. Figure 3 illustrates four clutch plate designs. Grooves between segments of the friction paper provide conduits for the ATF to pass through the plates. A large groove area may be preferred for cooling purpose, but reduces the frictional contact area, affecting structural integrity of the friction material. A larger clearance between the plates reduces the velocity gradient of ATF between the plates and also promotes air entrainment. The large clearance reduces drag torque, but an increased stroke distance tends to degrade engagement control robustness. Some plates may be gently waved in the circumferential direction, aiming at uniform plate distribution for reduced drag torque. An overview of clutch design practice is found in [1].

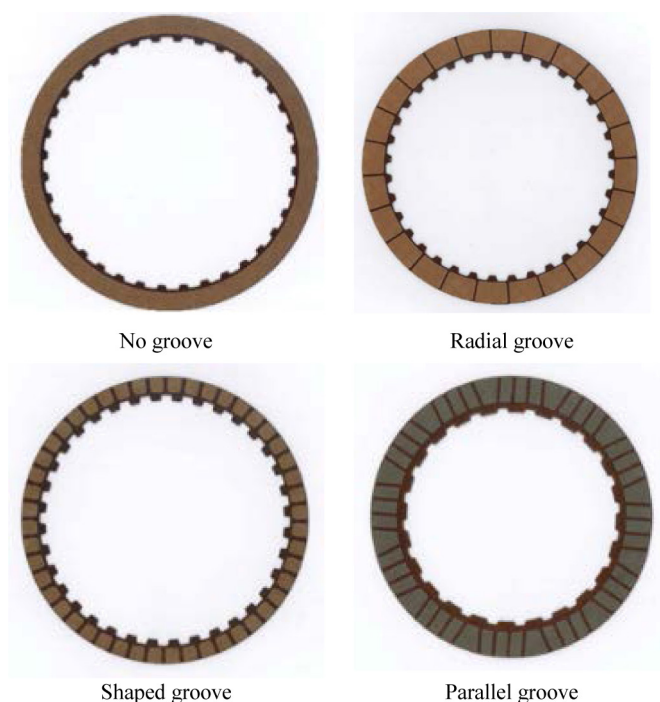


Figure 3. Friction plate groove designs

OPEN CLUTCH PHYSICS

The relative motion between the friction and separator plates in a disengaged clutch causes viscous drag torque as the ATF flows through the interface [1]. In Figure 4, the relationship between the drag torque and rotating speed is illustrated.

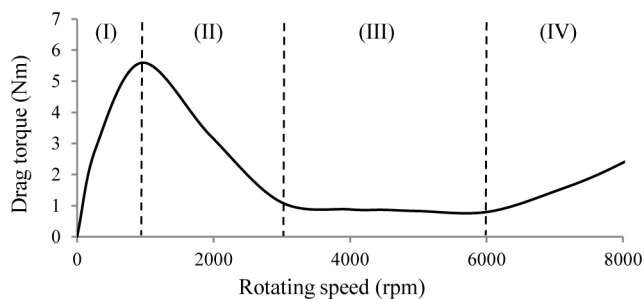


Figure 4. Typical open clutch drag torque characteristics

The drag torque curve can be divided into four regimes based on torque transfer mechanisms. In regime (I), a well-developed ATF film is present between the plates because centrifugal pumping at the interface is small due to the low rotating speed. Assuming a single phase laminar flow, an analytical solution can be derived for the motion of the fluid within the interface. For a flat plate with no grooves, the tangential velocity is expressed as [4]:

$$v_{\theta} = \frac{\omega}{h} r z \quad (1)$$

where ω is the rotational speed, h is the clearance between plates, r is the fluid film radius, z is the axial displacement, and v_{θ} is the velocity component in the circumferential direction as shown in Figure 5.

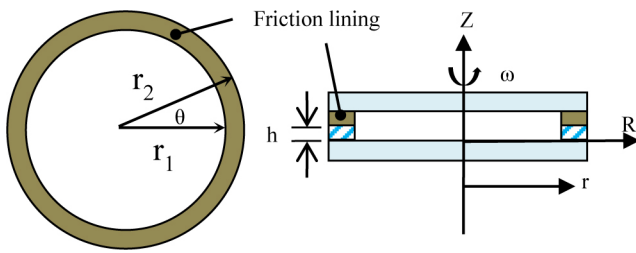


Figure 5. Clutch geometry for simple drag torque analysis

The drag torque is transmitted by viscous shear between the separator and friction plates. The shear stress $\tau_{\theta z}$ on the rotating plates is shown below:

$$\tau_{\theta z} = \mu \frac{\partial v_{\theta}}{\partial z} = \frac{\mu \omega r}{h} \quad (2)$$

where μ is the ATF's viscosity. The drag torque on each of the plates can be expressed as follows:

$$T = 2\pi \int_{r_1}^{r_2} r \tau_{\theta z} r dr = \frac{\pi \mu \omega}{2h} (r_2^4 - r_1^4) \quad (3)$$

where r_1 and r_2 are the inner and outer radii of the clearance between the plates. This model is often utilized to approximate open clutch behaviors in the regime (I) where drag torque monotonically increases with rotating speed.

In regime (II), additional air is entrained into the frictional interfaces as the pumping capacity of the clutch pack becomes larger. The average viscosity of the mixture becomes smaller and the drag torque decreases with sliding speed. In regime (III), the interface is mostly filled with air because of the large centrifugal pumping effect. Drag torque remains small and becomes insensitive to slip speed. In regime (IV), when the rotating speed is very high, the clutch plates start fluttering, resulting in increased friction due to mechanical contact between plates.

Many attempts have been made to predict the behavior of wet clutches. Stewartson [5] and Szeri [6] studied the flow between two co-rotating flat plates, providing explicit solutions for different speed cases. Szeri reported that the centrifugal force causes the fluid to increase its radial velocity near the rotating plate, while back flow is present near the stationary plate to compensate the centrifugal flow effect. Kitabayashi [7] empirically investigated the effects of frictional facing area, wave height, number of grooves and ATF flow rate. Lloyd [8] evaluated the groove patterns, fluid flow rate, friction material, as well as clearance and viscosity. The importance of plate waviness in reducing drag torque was also discussed. Yuan [9] proposed an improved hydrodynamic model with the inclusion of the effects of surface tension. To capture the effects of air entrainment, the concept of an equivalent radius is introduced to divide the fluid and air regions through a hypothetical boundary. The model formulation is only applicable to non-grooved flat plates. The ATF viscosity is treated as a tuning parameter to match experimental observations. Although this model provides a relatively simple analytical framework, the concept of equivalent radius has no

physical justification in the clutch system where fluid is dispersed over the clutch surface. Aphale [10, 11] investigated the effects of clearance and number of grooves. A CFD model was developed to optimize the design of grooves. In his model, only the friction material area is included, excluding the detailed design features, which cannot present the real environment of the fluid within the interface. Takagi [12, 13] developed a three-dimensional, single-phase CFD model for clutch plates. Effects of clearance, flow rate and groove numbers were studied at a low slip condition. The single phase assumption limits the model's capability to predict the peak torque location with respect to slip speed. Yuan [14] developed a two-phase flow model to study the aeration process for a flat clutch plate. The simulation indicated that the aeration process is initiated near the plate where the rotational speed is small. Yuan [15] attempted to improve model performance by treating the surface tension coefficient and the ATF's viscosity as adjustable parameters. Although the simulation appears to match the experimental data, the model does not account for realistic boundary conditions such as the inner hub and outer structure.

It remains a challenge to capture drag torque behaviors, including peak location, accounting for key design features and operating conditions. This article presents a two-phase Multiple Reference Frame (MRF) CFD model for simulating wet clutch behaviors, including detailed design geometry. The model employs the Volume of Fluid (VOF) method to determine the air-fluid interface inside the computational domain. Model setup and simulation parameters, such as initial conditions, boundary conditions, and relaxation factor are evaluated in terms of convergence behaviors. The model capabilities are validated against experimental data for the regime (I) and (II). Convergence of the CFD solver is demonstrated, capturing peak drag location as a function of rotating speed, until the phase fraction drops to a small value. The model provides analytical insight into complex fluid interactions between grooved rotating plates, complementing hardware-based clutch design improvement processes.

MODEL FORMULATION

The computational domain is shown in Figure 6, representing a single fluid interface of a clutch pack between the inner hub and outer structure. The domain includes the friction plate, the frictional material area, the spline teeth geometry, the space between the friction plate and outer structure, and the drainage holes on the housing. The inner boundary of the domain corresponds to the spline teeth area of the friction plate. There are six evenly spaced ATF inlet holes at the spline teeth area, where a fixed flow rate boundary condition is applied. The outer boundary of the model corresponds to the clutch housing with six drainage holes where wall type and outlet boundary conditions are used. The top drainage hole is designated as the air inlet, where air enters the domain. It is assumed that this hole does not allow ATF fluid to leave the domain. The ATF and air mixture can exit or enter the domain through any of the remaining holes. This assumption is found to improve the velocity distribution in the simulation. A structured mesh of hexahedral cells is selected and generated in order to improve the numerical accuracy. The complete mesh contains more than four million hexahedral cells. A part of the mesh and a typical hexahedral grid is shown in Figure 6.

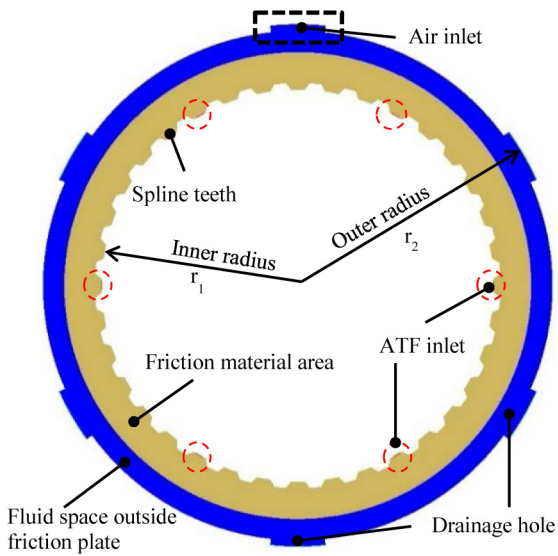


Figure 6. The computational domain

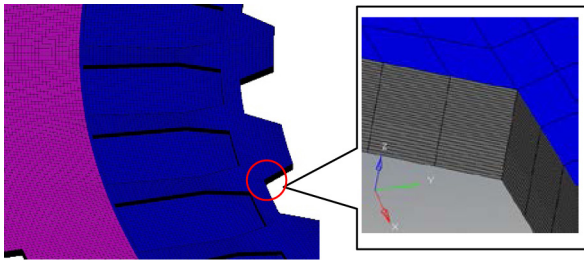


Figure 7. Mesh view and zoomed grid cell

The flow is modeled as a laminar, two-phase flow at room temperature (40°C). A segregated solver is used to solve the governing equations sequentially. The VOF method is used to determine the interface between the ATF and air. Each cell is described by a phase fraction α , defined as the percentage of the local fluid volume each phase occupies. The equations for conservation of mass and linear momentum are written as:

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{u}_k) = 0 \quad (4)$$

$$\alpha_k \rho_k \left(\frac{\partial \mathbf{u}_k}{\partial t} + \mathbf{u}_k \cdot \nabla \mathbf{u}_k \right) = -\alpha_k \nabla p_k + \nabla \cdot \alpha_k \mathbf{T}_k + \alpha_k \mathbf{f}_k \quad (5)$$

where ρ is the density of phase k , $\mathbf{u}$ is the corresponding velocity, p is the pressure, $\mathbf{T}$ is the stress tensor and $\mathbf{f}$ is the sum of external forces. The phase fraction α is conserved and bounded. A sharp interface cannot be maintained if $\nabla \cdot \mathbf{u} \alpha_k$ is diffusive.

The discretization of the governing equations is shown below. It represents the steady-state conservation equation for a quantity ϕ , such as pressure, phase fraction and velocity components, in the integral form for an arbitrary control volume V .

$$\oint \rho \phi \mathbf{u} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \oint_V S_\phi dV \quad (6)$$

where $\vec{A}$ is the surface area vector, Γ_ϕ is the diffusion coefficient for ϕ , and S_ϕ is the source term. At each iteration, the variable ϕ is calculated for each cell using an adjustable parameter λ , which is commonly referred to as a relaxation factor, i.e.

$$\phi^{new,used} = \phi^{old} + \lambda(\phi^{new,predicted} - \phi^{old}) \quad (7)$$

where λ is the relaxation factor for pressure, which can also be applied to VOF method and velocity. The relaxation factor is a constraint for adjusting a dependent or auxiliary variables from one solution to the next during the iteration process. The relaxation factor maintains the stability of the coupled, non-linear system of equations. If $\lambda < 1$, the process represents under-relaxation. The speed of convergence may be slowed but this increases the stability of the calculation. In contrast $\lambda > 1$, results in over-relaxation. This may accelerate the convergence but tends to decrease the stability of the calculation. For transient simulations, the governing equations must be discretized in both space and time. The spatial discretization for the time-dependent equations is identical to the steady-state case. Temporal discretization involves the integration of the differential equations using a time step Δt . The time step is determined by the Courant number

$$C_r = \frac{u \Delta t}{\Delta x} \quad (8)$$

where u is the magnitude of the fluid velocity, Δt is the time step and Δx is the cell size. When an explicit solver is used, the time step Δt is selected to keep the Courant number smaller than 1. A detailed discussion about selecting the appropriate time step is discussed in the next section.

The governing equations are discretized implicitly with respect to the velocity and pressure variables. A second order central differences discretization scheme is used to approximate the momentum equations:

$$\left[\frac{\partial \phi}{\partial x} \right]_i^n = \frac{\phi_{i+1}^n - \phi_{i-1}^n}{2\Delta x} + O(\Delta x^2) \quad (9)$$

The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used to introduce the pressure field into the continuity equation to enforce mass conservation and to determine the pressure field. Instead of using linearly an averaged value, the face value of velocity is calculated through momentum-weighted averaging by introducing weighting factors. Then, the flux through each face of the cell, J_f , can be written as follows

$$J_f = \hat{J}_f + d_f(p_{c0} - p_{c1}) \quad (10)$$

where p_{c0} and p_{c1} are the pressures within the two cells on either side of the mutual face, and $\hat{J}_f$ contains the influence of the velocities in these cells. The term d_f is a function of the average of the momentum equation coefficients for the cells on either side of face. Once a solution is obtained, the cell pressure and the face flux are corrected using