

Dynamic Analysis of the Lubrication in a Wet Clutch of a Hydromechanical Variable Transmission

Andrea Bassi, Massimo Milani, Luca Montorsi, and Stefano Terzi
Universita di Modena e Reggio Emilia

ABSTRACT

The paper investigates the oil flow through a multi plate clutch for a hydro-mechanical variable transmission under actual operating conditions. The analysis focuses on the numerical approach for the accurate prediction of the transient behavior of the lubrication in the gear region: the trade-off between prediction capabilities of the numerical model and computational effort is addressed. The numerical simulation includes the full 3D geometry of the clutch and the VOF multi-phase approach is used to calculate the oil distribution in the clutch region under different relative rotating velocities. Furthermore, the lubrication of the friction disks is calculated for different clutch actuation conditions, i.e. not-engaged and engaged positions. The influence of different geometrical features of the clutch lubricating circuit on the oil distribution is also determined. The results show the areas where poor lubrication occurs and extend the experiments where measurements are difficult to carry out. The simulation highlight the regions where high thermal stresses are observed during tests.

CITATION: Bassi, A., Milani, M., Montorsi, L., and Terzi, S., "Dynamic Analysis of the Lubrication in a Wet Clutch of a Hydromechanical Variable Transmission," *SAE Int. J. Commer. Veh.* 9(2):2016, doi:10.4271/2016-01-8099.

INTRODUCTION

Numerical analysis is gaining an important role in the design of hydraulic components and systems. In particular, multidimensional simulation is increasingly applied to the investigation of the fluid dynamics behavior of hydraulic systems in order to broaden and complete the experimental campaigns. Moreover, the human and computational resources to be involved in the numerical analysis are now not only acceptable, but also advantageous, due to the continuous development of computational platforms and CFD tools. Numerical modeling is particularly useful in analyzing the fluid dynamic behavior in regions where experimental measurements are very difficult or even impossible to carry out.

In this paper, the ATF (automatic transmission fluid) flow through a multi plate clutch of a hydro-mechanical variable transmission is investigated with a multi-phase multi component numerical analysis. The CFD tool enables simulation of the fluid dynamic behavior of the ATF, accounting for the real complex geometry of the clutch under actual operating conditions and configurations. Thus, the lubrication process can be addressed and critical geometrical features or not efficient oil distribution outlined. Experimental investigation of simplified systems is possible with transparent components and modern acquiring techniques; for instance, in [1] the air velocity profile between rotating disks made of transparent polycarbonate has been measured using laser-Doppler anemometry. In [2] the application of the fast image processing was adopted for the characterization of a multiphase flow through transparent polymethyl-methacrylate prototypes. Indeed, the experimental approaches are very difficult to employ when analyzing the full physics of the

lubrication within a multi plate clutch, including not only a complex geometry, but also a multiphase flow, high friction and structural stresses and high rotating speeds.

In literature, there are many examples that prove the reliability of the numerical modeling for the simulation of multi-phase flows by comparing the calculations to experimental values. For instance, in [3] a CFD multiphase analysis was implemented in order to study the cavitation phenomena in hydraulic components and the results were validated against experimental measurements available in literature. Similar approach was adopted in [4] and [5] for evaluating the multi-phase flow in a directional control valve for load sensing application and in a closed center load-sensing proportional control valve respectively. Though, complex physics including multi-phase and multi-components phenomena can be addressed [6], only recently CFD modeling has been employed in the analysis of the ATF flow through the plates of a wet clutch. In [7] the multidimensional simulation was adopted for investigating the multiphase flow through an open grooved two-disc system, but a single phase and constant flow inlet boundary were assumed, while in the real phenomenon each couple of plates has a non-uniform air-oil mixture and pressure distribution at the boundaries. In this paper, the whole clutch geometry has been considered and the oil distribution to each plate has been predicted as a function of the component design and operating conditions. In fact, the flow distribution has a fundamental role for the heat dissipation and wear out of the friction plates. Different numerical methods have been proposed for the wear prediction considering thermal degradation as in [8] and [9]. Furthermore, in [10] the thermal aspect of the engagement process

in a wet clutch have been also investigated numerically. Since it is very difficult to assess the oil distribution between the plates, heavy assumptions have been made; as in [8] where the ATF is assumed to completely fill the plates' clearance, even though the same authors suggested that indeed the ATF film fills the clearance only partially at high rotating speeds, especially for the friction plates with grooves. Furthermore, the flow through the clutch plates is not only a multi-phase mixture, but it is also turbulent as determined in [7] and [11]. In this paper, a turbulent flow has been considered and the k- ϵ turbulence model was adopted due to its good tradeoff between accuracy and numerical stability as demonstrated in [12].

Multi plate wet clutches are widely employed in powertrains where high-energy transmission and durability of the friction material are required. The lubrication process is crucial to prevent the degradation of the friction material, reducing the wear behavior. For this reason, the design of the clutch lubrication layout is important in order to distribute adequately the ATF to the plates. A simulation of the complete geometry has been implemented in order to study the fluid dynamic behavior of the ATF in the entire geometry and in particular on the plates' surfaces. The results obtained from the entire geometry have been compared to the values that can be achieved by a simplified geometry that provides similar accuracy but it is characterized by a much lower computational effort and calculation time. Using the simplified model, it is possible to study the oil flow field for different operating conditions and configurations of the clutch, i.e. disengaged and engaged clutch. The calculation of oil distribution for each plate identifies the regions where a poor lubrication may cause failures or fast wear out of the disks. The numerical results are compared to the plates' conditions at the end of a real clutch test.

CASE TEST

The clutch object of this study is shown in Figure 1 and represents a classic architecture of a wet clutch. The analyzed geometry of the clutch includes the following elements: the shaft, the return spring, the gear, the piston, the steel plates, the friction plates and the clutch housing. The friction plates are connected to the gear, while the other components to the shaft. The void volume in this region represents the fluid domain where the oil flows for lubricating and cooling the different parts. Even though the geometry is essentially cyclic, due to the different rotation speeds of the gear and the shaft before engagement the fluid dynamics problem is fully three dimensional and no symmetry boundaries can be adopted to simplify the domain.

The lubrication oil reaches the clutch through the duct realized in the shaft from the main lubricating circuit. A single inlet duct supplies the ATF to the gear internal chamber, which is the main responsibility of the ATF distribution to the plates region. In the gear chamber, a multi-phase flow develops since the ATF mixes with the air inside the clutch. The holes realized on the gear (GHs) distribute the ATF to the plates' regions. The ATF can also reach the plates' regions through the lateral ring (LR), since its area is relevant. The LR is the annular volume on the left of the Figure 2b. The oil distribution determines

the inlet boundary conditions for the gaps between each steel plate and grooved plate. Therefore, the velocity field in the plates' clearance depends on the inlet conditions from the GHs as well as on the rotation speed of each plate. The ATF reaches the tank through the shaft holes (SHs) and the lateral outlet (LO). The tank pressure is assumed at atmospheric pressure.

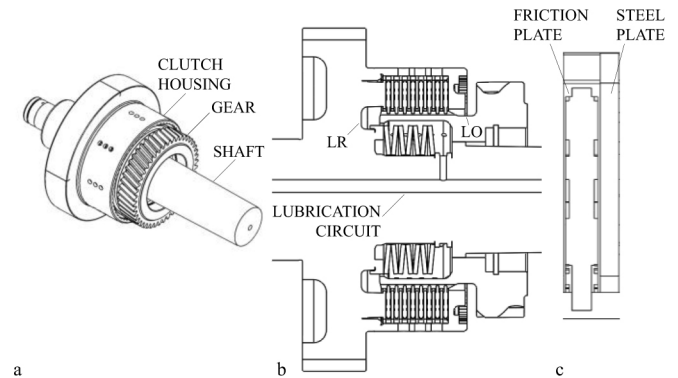


Figure 1. Schematic of (a) the wet clutch and (b) the lubrication circuit. (c) Zoom of a couple of steel and friction plates.

A typical engagement cycle for a wet clutch can be divided into four stages: *engagement*, *soaking*, *dwell* and *stabilization*. The engagement stage starts with the pressurization of the actuation cylinder chamber and ends when the relative speed between the steel and the friction plates becomes null. During this period, the clutch goes from the open configuration, where the clearance between the plates is maximum, to the closed configuration, where the zero gap is reached. The lubrication of the steel plates is extremely important during the engagement phase, since the maximum heat generation occurs and the high conductance of the steel plates allows the dissipation of most of the friction heat generated. Furthermore, during the engagement the oil on the steel surface permeates into the lining due to its porous structure. In addition, the ATF flow into the grooves of friction material also cools the interface temperature. The soaking stage initial condition is the end of the engagement period and it lasts until the pressure in the actuation chamber is released. During this period, the relative speed between the plates is null since the plates are locked. The soaking stage final condition overlaps the dwell stage initial condition, which ends when the gap between the plates is restored to the initial value. In the stabilization stage, the ATF flows through the clearance of the plates and cools down the clutch.

The most critical stage, in terms of heat exchange and temperature development, is the engagement period. For this reason, in this paper, the lubrication distribution is studied for two configurations and operating conditions close to the engagement stage. The numerical analysis addresses two relative rotation speeds for the gear and the shaft that are considered the most critical ones. In particular, in the first operation the gear rotates faster than the shaft, while in the second one the shaft rotation speed is only slightly larger than the angular velocity of the gear.

METHODOLOGY

The CFD analysis of the clutch is carried out by means of the computational fluid dynamics code Star-CCM+ [13], licensed by CD-Adapco.

The Volume Of Fluid (VOF) approach is adopted for the solution of the mass conservation and the momentum equations of the multiphase isothermal flow. Eulerian multiphase physics for the air and oil phases is defined. For the oil incompressible flow is assumed and the density and viscous properties at 70°C are used in the calculation. The interface between the gaseous and liquid phase is accounted for by means of the interaction due to the liquid surface tension. In this analysis, the cavitation and aeration phenomenon are not included. The turbulent flow is solved by means a two equation approach and the k-epsilon turbulent model is adopted due to its good tradeoff between accuracy and numerical stability. Two-layer wall treatment is considered to address the wall flow behavior. Bounded central differencing is used for the discretization of the momentum, second-order scheme for subgrid kinetic energy, and the mixture fraction. The conservation equation for mass, momentum is solved simultaneously using a pseudo-time-marching approach. The second-order implicit method is used for time integration scheme. Finally, the gravity contribution is included in the simulation and its effect on the flow field is considered.

The [Figure 2a](#) shows the lateral section of the domain used in the simulation for the open configuration. The section shows the number of domains in which the complete geometry is subdivided. The partition takes the different rotational speeds of the regions into account; thus, the proper angular velocity is set up to the regions rigidly connected to the gear and the shaft respectively. The partition of the entire domain into sub-domains enables to account effectively for modifications to the initial geometry, since only the sub-domain where the modification takes place has to be remeshed. The spring region is neglected, since it is assumed that the flow through its plates do not obstruct significantly the ATF flow.

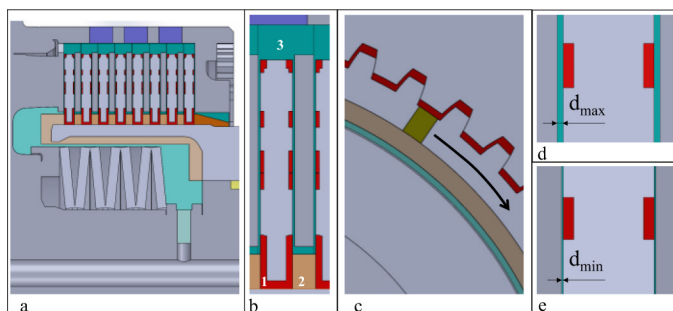


Figure 2. (a) Multicomponent domain for the simulation of the open configuration; (b) Modular approach, single reference unit; (c) tooth leakage due to gear and friction plate engagement; (d) open configuration; (e) closed configuration

The [Figure 2b](#) shows the modular approach for the computational domain of the plates' pack and the three repeated regions are highlighted with different colors: 1 - the groove region, 2 - the inner region and 3 - the steel plate region. In order to reconstruct the entire domain, the three regions are repeated 7 times. The last plate for the clutch housing is meshed separately, since it is slightly different from the other plates' regions. Further advantage of this meshing approach is the possibility of tailoring the mesh characteristics for each sub-domain thus improving the overall quality of the entire mesh.

The engagement region between the friction disk and the gear is accurately discretized in order to reproduce the leakage and the contact area due to the spin direction between the teeth, see [Figure 2c](#). In fact, the leakage is an important feature to account for, since some GHs' outlet is located in front of the plate teeth and thus the oil would flow only through the leakage. Therefore, if no leakage is included those GHs would be obstructed.

[Figure 2d](#) and [Figure 2e](#) show the different gaps between two plates in the open and closed configurations, i.e. d_{max} and d_{min} respectively. In the closed configuration the considered clearance of the plates is assumed as the 25% of the gap in the open position ($d_{min} = 0.25 d_{max}$).

In order to simulate the fluid dynamic behavior of the lubrication process in the complete geometry and in different operating conditions, two simulation strategies are implemented. The first one accounts for the entire domain and requires a remarkable computational effort; the second strategy adopts a simplified domain that is characterized by a lower computational effort and therefore it can be used for the simulation of different configurations.

The [Figure 3](#) depicts the mesh of both the complete and the simplified geometry. In order to approximate the fluid dynamic behavior of the plates, the simplified geometry is cut where the grooves start. Since the grooves are the minimum cross section area and the pressure in these region is very close to the ambient value, as it will be shown in the results section, the geometry approximation does not affect remarkably the pressure distribution in the GHs' region. The meshing approach adopts a trimmed hexahedral discretization with a characteristic cell size in the order of 1 mm. Local cell refinements are employed for a better definition of the geometry features in particular regions, such as the plates' regions and the teeth leakages. A preliminary sensitivity analysis with respect to the mesh size was carried out in order to determine the best tradeoff between the accuracy of the results and the computational effort. The zoomed views of [Figure 3b](#) and [c](#) show the refinements in the plates' regions in the complete geometry; while the [Figure 4b](#) and [c](#) report the zooms in the plate regions in the open and closed configurations respectively. The number of cells of the complete geometry results 36.7 Million, instead the simplified geometry mesh amounts only to 14.1 Million cells. The number of cells for the open and closed configurations is similar since the hexahedral mesh can be resized non-uniformly along the three axes, thus preserving a minimum number of cells in the gap direction.

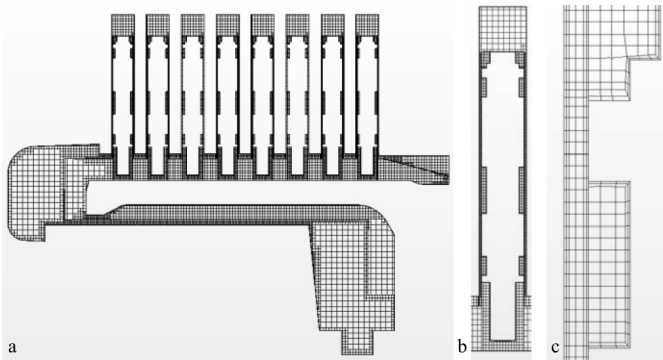


Figure 3. (a) Hexahedral mesh of the complete geometry. (b) and (c) zoomed views of the mesh in the plate regions.

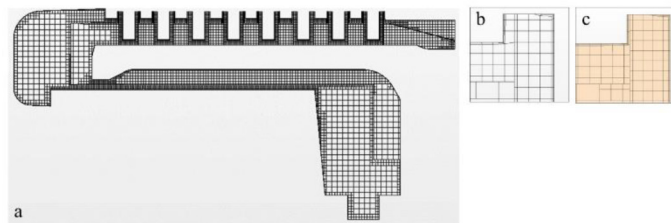


Figure 4. (a) Hexahedral mesh of the simplified geometry. (b) Zoomed view of the mesh in the plate region for the open configuration. (c) Zoomed view of the mesh in the plate region for the closed configuration

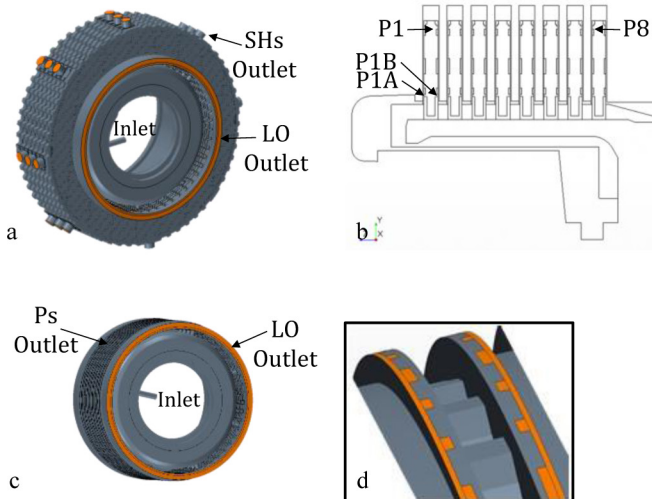


Figure 5. Boundary conditions (a) in the complete and (c) in the simplified geometries; (b) section of the complete geometry and (d) zoomed view of the clearance outlet boundary in the simplified geometry

Figure 5 outline the boundary conditions adopted in the simulations. The multiphase flow includes the incompressible flow phases for air and oil. At the inlet boundary from the lubricating circuit, a constant mass flow rate of pure oil is set. The relating mass flow rate is determined by a preliminary CFD simulation of the entire lubrication circuit for the considered operating conditions. In particular, the

measured values of the inlet mass flow rate and the pressure in several sections of the circuit were compared to the calculated ones. The comparison was carried out for different rotational speeds, since the inlet mass flow rates are different for the simulated operations due to the effect of the centrifugal forces. The outlet boundaries are different in the complete and in the simplified geometries. In the complete geometry, the outlets are real surfaces and correspond to the SHs and the LO outlet areas. These boundaries are set at ambient pressure since they are connected to the tank volume. The simplified geometry maintains the same pressure outlet condition in the LO and atmospheric pressure is set for the plates.

The rotation speed rates are hold constant during each simulation and different values are set for the gear and the shaft regions. In the complete geometry, a particular wall boundary velocity is used in the plate region to simulate the different rotation speed between the friction disk and the plate. Sliding interfaces are employed between gear and shaft regions in order to allow the fluid exchange in adjacent regions characterized by different rotations.

Table 1 summarizes the main characteristics of the simulations accounted for in this analysis. In particular, the boundary condition, the fluid properties and the angular velocity are highlighted. Table 2 details the shaft and the gear rotational speeds in the two operating conditions. Only the case #0 simulates the complete geometry (Comp.), while for the other cases the simplified geometry (Simpl.) is adopted. A fully transient approach is employed to simulate the filling of the clutch with the lubricating oil. Different simulation durations are adopted for the different cases in order to reach a stable condition of the flow. The time step for the implicit unsteady solver is selected accordingly to the simulated set of angular velocities in order to be shorter than the time required to sweep one degree for the fastest region (e.g. a time step equal to 6.43E-05s is adopted for the first velocity set and equal to 1.04E-04s for the second one).

Table 1. Main characteristics of the simulations.

Case #	Geom.	Set of velocities	Clutch Status	Q_{in} [l/min]	Clearance [mm]	T [°C]
0	Comp.	1	Open	Q_{ref}	d_{max}	70
1	Simpl.	1	Open	Q_{ref}	d_{max}	70
2	Simpl.	1	Closed	Q_{ref}	d_{min}	70
3	Simpl.	2	Open	$0.9318 Q_{ref}$	d_{max}	70
4	Simpl.	2	Closed	$0.9328 Q_{ref}$	d_{min}	70

Table 2. Speed rotation of the shaft and the gear regions.

Set of velocities	n_{shaft} [rpm]	n_{gear} [rpm]
1	1240	2590
2	1600	1500