

# Numerical Computation of the Air Flow and the Thermal Behavior of a Double Dry Clutch Automotive Transmission

Anthony Levillain and Pascale Brassart  
Valeo Powertrain

Béatrice Patte-Rouland  
Université de Rouen

## ABSTRACT

New directives and increasing competition push OEMs to get better performances (engine power increase), along with mass and size reduction (consumption). These evolutions lead to an increase of the thermal solicitations undergone by the clutches whereas their weight must be decreased, as it is one of the main influent factor on CO<sub>2</sub> emissions. As the compactness is even more reduced for a Double Dry Clutch (DDC), this issue is critical.

In order to improve the thermal behavior of the DDC, a CFD study has been performed. The present paper will introduce the first results obtained with a validated CFD model. The computations are performed on a DDC where the air flow around the rotating parts, along with the convection and the conduction of all solid parts are modeled. Then four different cases are used to evaluate the impact of holes on the air flow path and on the thermal behavior of the different parts of the system. The global and local air flow results are then determined in addition with the heat transfer coefficients. Results show that the heat transfer coefficients are mainly piloted by the local air flow. Moreover, it shows that large temperature differences can occur between the different cases, and that a design improvement can be proposed regarding pressure plates cooling.

**CITATION:** Levillain, A., Brassart, P., and Patte-Rouland, B., "Numerical Computation of the Air Flow and the Thermal Behavior of a Double Dry Clutch Automotive Transmission," *SAE Int. J. Engines* 8(4):2015, doi:10.4271/2015-01-1661.

## INTRODUCTION

Automotive clutches are rotating systems subject to high heat loads. These heat loads are generated by the slipping of Cast Iron discs (called Pressure Plates), rotatable with the engine, and the Facing disc, rotatable with the gearbox shaft. Both parts are rotating at different speeds, dependant on the engine speed and the engaged gear ratio. While engaging the clutch, the two parts slip on each other before being fully engaged, and heat is generated. These heat loads may be significant during traffic jam or hill starts. During such events, the temperature cyclically rises. While the temperature reaches a high value, close to 400°C, the friction coefficient, which is related to the temperature, can drop suddenly, thus making the vehicle unusable, as engine cannot transmit its torque to the gearbox. Furthermore, this issue is critical for a Double Dry Clutch (DDC), as its compactness is increased compared to manual transmission clutches.

The cooling of the clutch parts is directed mainly by the convection as it has been shown on previous study [1]. The convective heat exchange is done with the air enclosed in the clutch housing. Previous studies focused on the thermal behavior of manual transmission clutches. Beitler H. [2] made an experimental study of a clutch which combined both air flow rates and temperature measurements. Wittig S. and al. [3] were the first, to the knowledge

of the authors, to apply CFD on clutches. They abstracted the clutch to a simplified model and showed that the heat transfer was dependent on the local flow field. Hauschild J. [4] performed a CFD study on clutches in their environment, and evaluated the air flow rates and the heat transfer of different clutch systems.

In order to improve the thermal behavior of the DDC, a CFD study has been performed. The present paper will introduce the first results obtained with a validated CFD model [5]. The computations are performed on a DDC where the air flow around the rotating parts, along with the convection and the conduction of all solid parts are modeled. As the thermal behavior of the system is mainly dependant on the air flow field, improvements of the air flow path could increase the convective heat transfer. Then, the aim of this study is to evaluate the impact of holes on the air flow rates and on the thermal behavior of a DDC. Four different cases are used to evaluate this impact.

First of all, an overview of the DDC system is made, considering the operating principle and the heat load generation. Then, the numerical procedure used in this study is described. As results, global and local air flow rates are determined, giving information on the air flow paths

in the system. In addition, global and local convective heat transfer coefficients and temperatures of the different parts allow the comparison of the thermal behavior of the different cases.

## SYSTEM OVERVIEW

The Double Dry Clutch (DDC) is a clutch system combining two clutches. Each independent clutch consists of two cast iron pressure plates and a facing disc (Cf. Fig. 1). The clutch 1, on the engine side, consists of the Pressure plate 1 and the Central plate, rotatable with the engine. This clutch is used for start-up and odd gear engagements with the use of the Disc 1. On the gearbox side, the clutch 2 consists of the Pressure plate 2 and the Central plate, shared with the clutch 1. It is used for even gear engagements with the use of the Disc 2. Each clutch transmits the engine torque to its assigned input gearbox shaft thanks to the facing discs. The Covers assembly, rotatable with the engine, enables the engagement/disengagement of each clutch. The actuation system activates/deactivates the clutch 1 moving the Diaphragm 1 and the Pulling-Cover, while clutch 2 is actuated by the Diaphragm 2 and the fixed Cover. The full DDC system is located in a housing between the engine compartment and the gearbox.

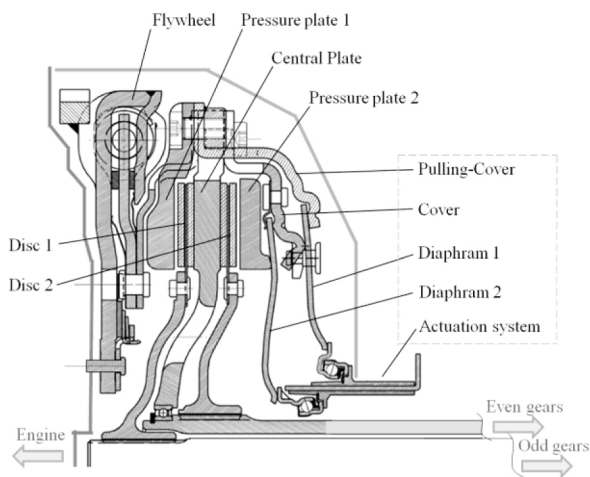


Figure 1. Double Dry Clutch (DDC).

During clutch engagement, heat is generated at the slipping interfaces. For clutch 1 engagement, for example, heat flux is generated between the Pressure Plate 1 and the Disc 1 and between the Central Plate and the Disc 1. Assuming all the energy is transferred into heat, a uniform pressure distribution and an equal friction coefficient on both sides of the disc, the total heat flux can be computed by equation (1).

$$Q = T_o \cdot \Omega_s \quad (1)$$

Where  $\Omega_s$  is the slipping speed and  $T_o$  is the transmitted torque computed by equation (2).

$$T_o = \frac{4}{3} \cdot \mu \cdot F \cdot \frac{(R_o^3 - R_i^3)}{(R_o^2 - R_i^2)} \quad (2)$$

Here,  $\mu$  is the friction coefficient,  $F$  is the clamp load,  $R_o$  and  $R_i$  are the facing disc outer and inner radius, respectively.

## NUMERICAL PROCEDURE

The numerical model used in this study is a commercial CHT code (Conjugate Heat Transfer). This type of model can determine both the conduction within the solids and the convection from the solid walls to the fluid. To model the rotation of the system, the computation is performed using a Single Reference Frame at a rotating speed  $\Omega$ . Wittig et al. [6] has showed that the method could be apply on rotating systems with holes. To simplify the model, the Discs rotate at the same speed as the Pressure plates and the Covers assembly, assuming that the rotating speed difference between these parts does not affect the air flow field. Then, a steady-state computation can be achieved, assuming the flow field within the rotating frame is stationary. The continuity equation (3) and the momentum equation (4), used to determine the air flow field, can be written in an absolute velocity formulation as follows [7] :

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}_r) = 0 \quad (3)$$

$$\frac{\partial}{\partial t} \rho \vec{v} + \nabla \cdot (\rho \vec{v}_r \vec{v}) + \rho (\vec{\omega} \times \vec{v}) = -\nabla p + \nabla \cdot \vec{\tau} + \vec{F} \quad (4)$$

The thermal field is computed using the energy equation (5) within the fluid and the solid parts :

$$\frac{\partial}{\partial t} \rho E + \nabla \cdot (\rho \vec{v}_r H + \rho \vec{u}_r) = -\nabla \cdot (k \nabla T) + \vec{\tau} \cdot \vec{v} + S_h \quad (5)$$

A RANS model (Reynolds Averaged Navier-Stokes) is introduced in order to take into account the turbulence effects.

The equations are spatially discretised using a conformal tetrahedral mesh. The thermal gradients from the solid walls to the fluid are of critical importance, as they will be used in the calculation of the convective heat transfer. Then, in order to get well discretised thermal gradients in this area, prisms layers are added. A mesh criterion is used such that the first mesh point respects the relationship in equation (6) [8].

$$y^+ = \frac{u_{\tau} y}{\nu} \leq 4 \quad (6)$$

For the mesh cells number limitation and thanks to the symmetry of the system, only 120° of the DDC is meshed. The total mesh count is approximately 15 million cells. Periodic boundary conditions are set on the symmetric parts. At the slipping points, between the Disc faces and the Pressure plates, heat flux is used as boundary condition. As a steady-state computation is performed, the total heat flux is constant and is distributed between the Disc and the Pressure plates. Assuming an equivalent torque on both sides of the facing Disc, the heat flux distribution between two slipping parts can be computed using the thermal effusivity ratio [9] :

$$\alpha_i = \frac{E_i}{E_i + E_j} \quad (7)$$

Where  $i$  and  $j$  refer to the slipping parts and the effusivity is defined such as :

$$E_i = \sqrt{\lambda_i \rho_i C_{p_i}} \quad (8)$$

Where  $\lambda$  is the thermal conductivity (-),  $\rho$  is the density ( $kg.m^3$ ) and  $C_p$  is the specific heat ( $J.kg^{-1}.K^{-1}$ ). In this study, only the Clutch 1 is under investigation. Then, half of the total heat flux  $Q$  is distributed between the Pressure plate 1 and the Disc 1, and another half between the Central plate and the Disc 1. Thus, according to material characteristics, the heat distribution at the slipping face computed at  $100^\circ C$  is of approximately 6% for the Disc 1 and 94% for the Cast Iron Discs (Pressure plate 1 or Central plate) respectively. The scheme in Figure 2 summarizes the distribution steps.

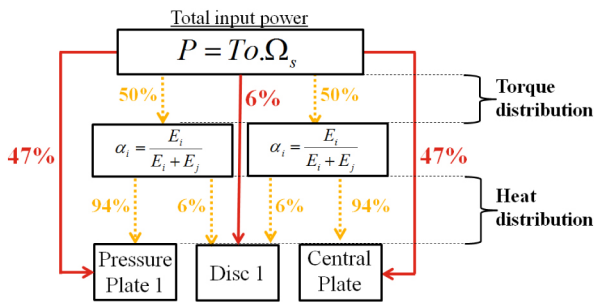


Figure 2. Heat distribution scheme.

Then, the heat is dissipated by conduction within the solid parts, and by convection with the air. Minereau H. [1] has evaluated the heat phenomena distribution on manual transmission clutches, and has estimated that 60% of heat is dissipated by forced convection at 1000 rpm. As convection represents the main part of the clutch cooling, no radiation is taken into account. The total heat flux is chosen as small as possible in order to get low temperatures and thus to reduce the radiation cooling impact. Moreover, only forced convection is modeled. Natural convection is assumed as negligible considering the rotating speed. It also simplifies the model, as a total 3D geometry ( $360^\circ$ ) would have been necessary.

## RESULTS

The model described in the part *Numerical procedure* has already been used in a validation procedure [5]. Measurement of local air flow velocity within the clutch housing has been performed with a technical laser technique called Laser Doppler Velocimetry (LDV) [10]. Furthermore, measurements of temperatures during constant heat flux tests (i.e. constant slipping and torque) has been done on test bench. These measurements results have been compared with the CFD model. It showed the capability to predict an air flow field and temperature trends in accordance with the measurements. Temperatures can be predicted within a range of  $10^\circ C$ .

The purpose of this paper is to present the results obtained with this model on a parametric variation. As the convective heat transfer is driven by the air flow field, creation of holes in the system could improve the pressure plates thermal cooling. Two hole sets have been defined and are used as parameters. The first holes set is located in the Flywheel (H3 on Figure 3), at small radius and consists of nine holes along  $360^\circ$ . The other set is located in the Central Plate (H2 on Figure 3) also with nine holes. Then, four cases can be defined in Table 1. Case-1 correspond to the reference case, where both holes sets are closed. In Case-2 and Case-3, holes sets are opened independently. Finally, in the Case-4, both holes sets H2 and H3 are opened. The four cases are computed and results can be compared.

Table 1. Holes opening.

|                    | Case 1 | Case 2 | Case 3 | Case 4 |
|--------------------|--------|--------|--------|--------|
| Central Plate (H2) | Close  | Open   | Close  | Open   |
| FlyWheel (H3)      | Close  | Close  | Open   | Open   |

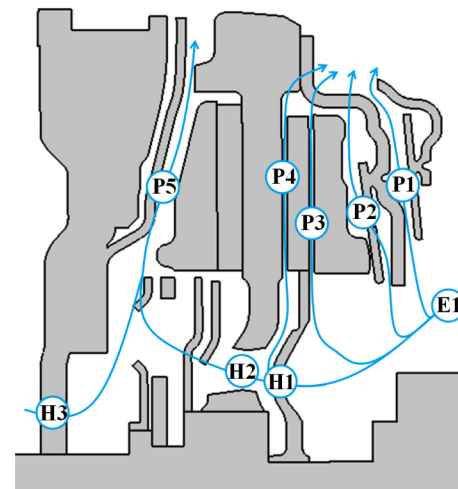


Figure 3. Air flow paths

## Flow Field

The rotation of the system causes a fluid rotation. As the fluid rotates, it is centrifuged and a radial velocity is generated while a depression occurs close to the axis. Then, in a DDC, the fluid is aspirated through the diaphragms, and is centrifuged by the different rotating parts of the system (Disc, Pressure plates, etc...). It can be said that the different rotating parts of the system act as a pump. Flow rates can then be calculated for the different paths of the system. The possible flow paths are indicated in Figure 3. The flow rates results obtained for each case are compared at a same rotating speed  $\Omega$ . The flow rates are calculated such as described in equation (9), where  $S$  is a path surface,  $\vec{V}$  the velocity vector and  $\vec{n}$  the vector normal to the path surface :

$$\dot{V} = \int_S \vec{V} \cdot \vec{n} \cdot dS \quad (9)$$

Indices of each flow rates  $\dot{V}$  refer to its corresponding path. The entrance flow rate  $\dot{V}_{E1}$ , calculated at the entrance of the DDC, is plotted in Figure 4.