POLITECNICO DI TORINO

Corso di Laurea Magistrale
in Ingegneria Aerospaziale e Astronautica

Tesi di Laurea Magistrale

Virtual Simulation for Clutch Thermal Behaviour Prediction

Relatore Accademico
prof. Roberto Marsilio

Candidato
Fabio Tosi

Supervisore Aziendale
Ing. Laura Maria Lorefice

Marzo 2018

If I have seen further
it is by standing on the shoulders of Giants
Isaac Newton

Contents

1 Introduction **5**
- 1.1 The Problem . . . 5
- 1.2 Behind the Problem . . . 5
- 1.3 How to Face the Problem . . . 6
- 1.4 The Strategy Followed . . . 6
- 1.5 Main results . . . 7

2 A Brief Technical Overview **8**
- 2.1 Main car components . . . 8
  - 2.1.1 The Engine . . . 8
  - 2.1.2 The Drivetrain . . . 11
  - 2.1.3 Cooling . . . 13
  - 2.1.4 Electrical system . . . 15
  - 2.1.5 Steering, Suspensions, Brake Systems . . . 15
- 2.2 The clutch . . . 15
  - 2.2.1 Clutch governing phenomena . . . 17
  - 2.2.2 Clutch Torque Capacity . . . 18
  - 2.2.3 A Real Clutch . . . 19
- 2.3 Heat transfer . . . 22
  - 2.3.1 Thermal energy . . . 22
  - 2.3.2 Conduction . . . 23
  - 2.3.3 Convection . . . 23
  - 2.3.4 Phase change . . . 24
  - 2.3.5 Radiation . . . 24
- 2.4 CFD methods . . . 25
  - 2.4.1 Governing equations . . . 25
  - 2.4.2 Turbulence modelling . . . 27
  - 2.4.3 Need for Discretization . . . 30
  - 2.4.4 Discretizing equations . . . 31
  - 2.4.5 Discretizing space . . . 32

3 Problem Presentation **33**
- 3.1 A Better Design . . . 33
  - 3.1.1 New complexity . . . 34
  - 3.1.2 Resources allocation . . . 34
  - 3.1.3 Improving Design Early Stages . . . 36
  - 3.1.4 Current Procedure . . . 36
- 3.2 The Other Side: Clutch Overheating . . . 37
- 3.3 Experimental test . . . 39
  - 3.3.1 Dangerous situations . . . 40
  - 3.3.2 The test . . . 40

3.4 Why CFD . . . 41
3.5 Air Pollution . . . 42

4 State of the Art Review **46**
4.1 Zero-dimensional Approach . . . 46
4.2 Uni-dimensional Approach . . . 46
4.3 Two-dimensional Approach . . . 46
4.4 Three-dimensional Approach . . . 47
4.5 Bibliography about Brakes . . . 49

5 Reaching the Solution **50**
5.1 Tools adopted . . . 50
5.2 Model improvement . . . 52
5.3 Complete simulation vs Reduced simulation . . . 53
5.3.1 The complete, though simplified, car simulation . . . 53
5.3.2 The reduced and more detailed subdomain . . . 56
5.4 Values monitored . . . 56
5.5 CHT Approach . . . 58
5.5.1 Components Description . . . 58
5.5.2 ANSA pre-processing . . . 58
5.5.3 Computational Domain . . . 60
5.5.4 Boundary Conditions Assignment . . . 60
5.5.5 Mesh and Physics setup . . . 66
5.5.6 Results . . . 66
5.5.7 Comments on sensitivity . . . 72
5.5.8 Cooling Improvements . . . 72
5.6 Thermal coupling Approach . . . 73
5.6.1 ANSA pre-processing . . . 73
5.6.2 Results . . . 73
5.6.3 Comparison with CHT Approach . . . 73
5.7 Complete and Detailed Car Simulation . . . 74
5.7.1 ANSA pre-processing . . . 74
5.7.2 Comparison with Reduced Simulations . . . 75

6 Conclusions **84**

Chapter 1

Introduction

The clutch is that mechanical part located in an internal combustion engine vehicle which allows the torque transmission from the shaft to the wheels, permitting at the same time gear shifting and supporting engine revolutions while the car is standing still. This simple but fundamental component exploits friction as working principle, therefore heat generation is in its own nature. The comprehension of all the critical issues related to thermal generation, and also of the principal physical parameters driving the phenomena, is a must in a design phase.

The subject of this Thesis is about the elaboration of an accurate, but also easy to use and easily replicable, methodology to simulate thermal behaviour of a clutch operating inside its usual environment. Using CFD, computational fluid dynamics, coupled with Thermal-BEM software to predict thermal behaviour, working limits can be foreseen, and strengths or design flaws can be highlighted. Definitely, it will allow us to design a better part with less efforts.

1.1 The Problem

One of the major dangers, for a clutch, is overheating. Two disks rubbing to transmit torque generates a certain amount of energy, which is dissipated mainly with absorption by disks acting as heat sinks. Distribution of heat generation over time determines a temperature profile inside the clutch which must remain under clear limits.

A clutch that have sustained overheating once is a damaged clutch. In fact, high temperature has irredeemably altered material properties and may have a major failure much more easily (a major failure occurs when the car driveline breaks down or people safety is threatened). CFD simulations and analysis may help to understand if the component risks overheating or it may point out possible oversize. The problem would be identified and corrected in advance, with less costs.

These are all really valid reasons to spend efforts in refining a design which (at least nowadays) lies too much on experience: it worked on previous types, why shall we change it? Resize a part in order to make more efficient all the rest involves non-negligible savings in terms of space and weight (and therefore operational costs).

1.2 Behind the Problem

In recent years, in automotive industry weight and space started to be key factors just as they are in aeronautics. There are many reasons to seek for the greatest performance out of each part. A lighter car allows less fuel consumption (or faster speed), while a smaller one would have less aerodynamic drag or, with the same chassis dimensions, more comfort for passengers.

Beside engineering reason there are also laws, luckily, that try to go more and more towards an ecofriendly transport industry. A way to reach less pollution, fact outstandingly important in this era, is to lighten the car.

Even performance must not be neglected, as customers want products always faster (still having the same spending power though...).

The aim of the Thesis is to provide a tool to facilitate the thermal target achievement for the clutch, avoiding in the meantime weight increase that would worsen exhaust gas emission. And possibly, even costs may be reduced.

1.3 How to Face the Problem

The design approach should base on means which are as faster and cheaper as earlier is the design phase in which they are adopted. It is not affordable to plan the production of a first prototype to put in a climatic wind tunnel with the only purpose than verifying that the thermal requirements are met. For these reasons all the analyses will be computational. Experimental analyses will be carried out but in an advanced phase, only to validate previously obtained data.

Factors are fundamentally imposed by time and money. In fact it results much cheaper to run a simulation on a server than build a vehicle and fill it with probes. It is anyway true that a correct simulation set up is time consuming and trial and error based (provided that a data correlation has not be studied yet). However, this is to be done *una tantum*. Furthermore, an eventual flaw found only in a prototyping phase would cause the company an unaffordable delay in production. Rather, additional costs for virtual simulations are really justified from this perspective.

1.4 The Strategy Followed

In order to reach a reliable, though not too heavy (computationally speaking), simulation model, several steps will be executed at an increasingly level of detail, in parallel with some simplification steps. With this we mean that we do not want to simulate phenomena or parts that consume computing power but whose behaviour do not influence the final result we have in mind.

Some data collected from an experimental test will provide a guideline in judging the accuracy of each simulation setup. This data is also used to validate the virtual clutch model. Last but not the least, many boundary conditions to set in the numerical simulation will come from this real test.

In a first plan, these may be some of the main stages:

1. Only the drivetrain will be simulated, neglecting the interaction with other underhood components. This part of the stage will add more and more details and realism, and it is not likely to remove other. Here, the aim is to understand how to set up a thermal simulation correctly, with much care spent for boundary conditions: where heat generation should be set, which temperature other components should have, and so on. Being the computational domain quite reduced, many more attempts can be run and a final model more easily found.

2. The drivetrain is then "bolted" to the whole car that is going to be simulated. There, radiative thermal path should also be considered. Thermal boundary conditions will be the same as in phase 1. There we are adding a great portion of realism, which had to be neglected in the previous phase. Now we would expect much accordance with experimental data. Little set up modifications may be required to bring the simulations closer to reality.

3. Environmental conditions like vehicle speed or external temperature will be changed in order to draw a working envelope for the clutch.

1.5 Main results

At then end of the internship, which has been a fundamental part of this Thesis, a new methodology for clutch thermal behaviour prevision will be written. Thanks to it, computational analyses will be set up with little efforts and their results will be interpreted in order to improve clutch design. Among the simulation output may be found temperature probes reports, air flow through the clutch case, influence of external speed and temperature and so on. By monitoring these values, engineers will be able to quickly state with precision if their choices are improving the design or they should be reviewed. The overall efficiency should then increase.

Chapter 2

A Brief Technical Overview

In this chapter main car components will be presented so to have a general sketch of the work environment. In the following, the main character, i.e. the clutch, will be described with further detail. Finally, in the last sections, some more mathematical concepts are outlined: heat transfer physics and formulation and a general description of CFD methods used to reach the objective.

2.1 Main car components

A car to move safely needs several hundreds components, usually grouped in a few assemblies which support a precise function in vehicle operation. In this section only reciprocating engines (i.e. internal combustion engines, or ICE) will be considered, as only this type of engine needs a clutch to operate. We will explain this fact with more detail in the following sections.

2.1.1 The Engine

The engine is the unit whose duty is to convert energy from chemical, stored in the fuel, into mechanical, in order to move the vehicle and its passenger. The fuel for ICEs is fossil fuel, thus hydrocarbons able to start a self-sustained oxidation reaction with air. In other words, they can ignite. This reaction is exothermic and, Consequently, great energy is delivered in form of heat beside exhaust gases like water vapour and carbon dioxide. Heat generated is used to warm a gas which will expand, executing work. This work is extracted by other components and addressed to the wheel.

The combustion reaction takes place inside a closed volume called cylinder, whose base, the piston, is free to move, and needs three actors to occur: fuel, the oxidiser, and the igniter. The first one is mostly represented by gasoline or diesel, sometimes methane and other natural gases; the second one is the oxygen contained in the air; and the last one may be either a spark or temperature. Being a chemical reaction, stoichiometric proportions must be respected and air mass vs fuel mass is carefully calibrated during gasoline engine operation (in the diesel engine this constraint does not exist). Once the fuel-air mixture is injected in the cylinder a spark plug or temperature itself ignites the mixture. This is the basic principle that lies under the ICE, now it will be explained *how* it can be actually exploited.

The most widely diffused engine in the automotive industry is the four-stroke engine. It is essentially made up of a cylinder, containing the gas; a piston representing the base of the mentioned cylinder whose duty is to collect the gas work; and, since nobody needs a linear movement, a piston rod that links the piston to a shaft, converting the movement into a rotational one. A sketch is provided in Figure 2.1. The shaft is then part of the drivetrain.

The four-stroke engine takes this name from its four-phases cycle. The piston moves up and down, and this corresponds to one shaft revolution. A cycle is composed by two shaft

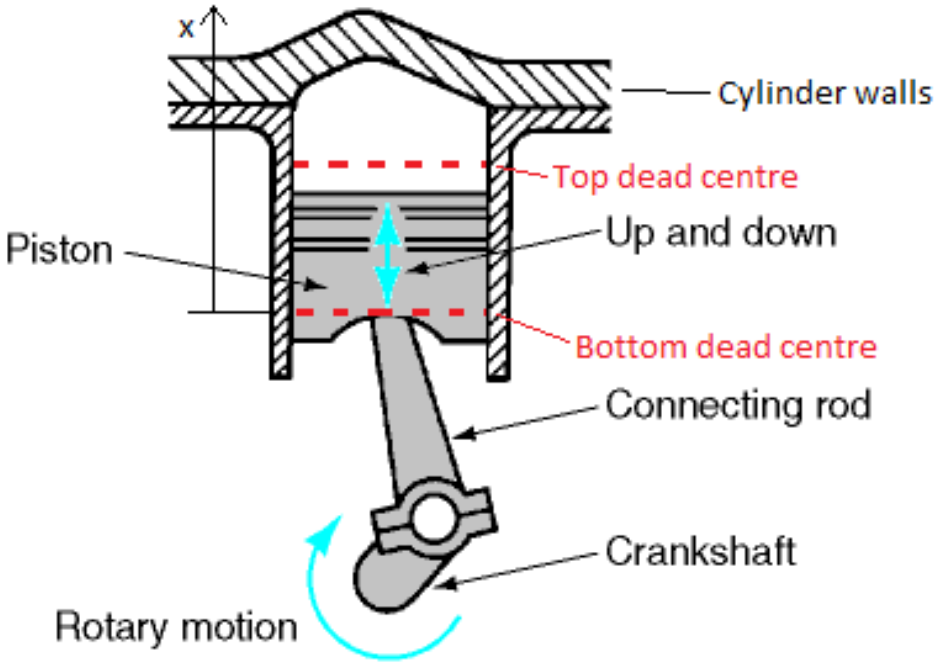

Figure 2.1: Sketch of Piston-rod mechanism. Piston is moving up and down from $x = 0$ (Bottom dead centre) to $x = x_{stroke}$ (Top dead centre).

revolutions, or four up/down piston movements: here it is, the four strokes (Figure 2.1). The thermodynamic cycle can be found with in-depth details in [10].

1. Intake: the piston runs from the top dead centre to the bottom dead centre while the intake valve is open. This allows the fresh fuel-air mixture to fill the cylinder;

2. Compression: the piston raises compressing the gas increasing its temperature. Pressure reached in the cylinder can be up to 12 times the atmospheric pressure for a gasoline engine and 30 times for a diesel engine;

3. Ignition and Expansion: in a gasoline engine the spark plug ignite the fuel-air mixture when the piston is in its highest position, while in a diesel engine the greater compression generates higher temperatures, causing the gas to self-ignite. The heat delivered in this exothermic reaction warms the gas that expands pushing the piston downward. Work is collected only during this phase, and it is spent to move the car and to complete the other three phases;

4. Exhaust: piston returns to the top dead centre with the exhaust valve open. Reacted gas leaves the chamber which is now ready for another cycle.

In real operation the ignition and valves opening do not exactly respect timing of strokes. Usually the ignition starts when the piston is still raising and valves opening and closing time are calculated to get maximum advantage from compression/expansion waves present in the engine. The aim is to enhance performance and to reach maximum thermodynamic efficiency. Opening and closing times, as well as spark times, are managed by an assembly of rocker arms, pushrods, camshaft etc generally called the valvetrain. Timing is synchronized with engine revolutions by means of chains or belts. Nowadays, a good gasoline ICE has an efficiency of about 0.28 with some hundredths more for a diesel one.

This reciprocation (with only one useful phase) causes a huge lack of continuity in power output. Indeed, vibrations and whole operations may become unacceptable. For this reason the

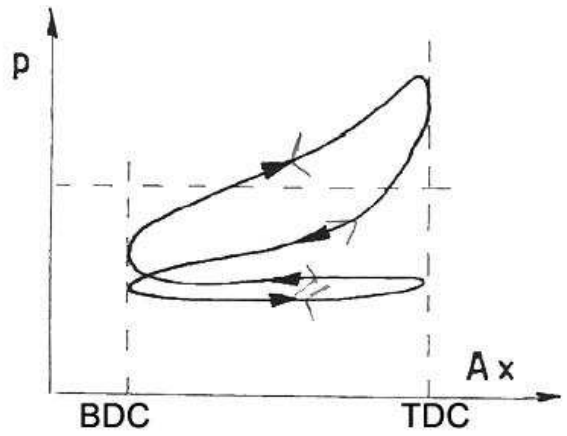

Figure 2.2: Qualitive diagram of gas pressure against piston stroke. Useful work is given by the upper closed area minus the lower closed area. Proportion are not respected: upper area is usually much larger.

transmission line is equipped with a flywheel. Its purpose is fundamentally to damp oscillations and smooth the power output outline. Being really large (low mass but high inertia moment is a must) its circumference provides also a great reduction gear for the engine starter. Improved designs may show also a double flywheel: two flywheels linked with a torsion spring. Another function that sometimes involves the flywheel is the valvetrain management.

With this description is clear that a four-strokes engine spends much work during exhaustion and intake (Figure 2.2). Furthermore, other work is necessary in order to maintain in operation many other components vital for the vehicle: cooling pumps, oil pumps for lubrication, alternators and so on. And we do not want to forget work lost for friction. Having these things in mind, we see that we need some fuel droplets just to keep the engine in rotation, with no power to the car wheels. This engine condition is known as engine idle. If we want to accelerate, we have to add fuel.

Power delivered by an engine is the result of several processes ongoing. An analytical formula, based on energy conservation principle, states that power delivered is equal to

$$W_{eng} = \eta_c \eta_{ac} \eta_{id} \eta_l \eta_{ip} \eta_p \eta_o \eta_t \eta_{vt} \frac{\rho_0 Z S H_i}{\alpha_t} \frac{u}{T} \tag{2.1}$$

as explained by Pignone and Vercelli in [6]. Lets explain all the factors. First of all, we have the engine architecture characteristics:

- T: is the number of stroke of the engine, 2 or 4 (4 for most vehicles);
- Z: number of cylinders, each with its own volume V and surface A;
- S: is the piston head area.

Then we find the fuel characteristics (keep in mind that the matter burnt is actually a mixture of air and fuel):

- H_i: is the fuel lower heating value. We consider the lower because exhaust products go away in form of gas and not liquid;
- α_t: is the ratio air/fuel introduced in cylinders $\frac{m_{air}}{m_{fuel}}$. The ratio may be stoichiometric, but further considerations lead to a rich (in fuel) mixture if we want high power or poor mixture if we are looking for maximum efficiency;

- ρ_0: is the reference density for air.

The three factors express the thermal power delivered by the fuel. Following parameters synthesize somehow how good is the engine in converting such thermal power in mechanical power:

- η_t, trap efficiency: not all the fuel introduced in cylinders actually *enters* it, as some may be refused because of valves timings and other factors;
- η_c, combustion efficiency: chemically, the reaction is not complete as the ratios are not stoichiometric;
- η_{ac}, adiabatic efficiency: some combustion heat may be lost in heating cylinders walls rather than gas;
- η_{id}, ideal efficiency: it states the unavoidable limit imposed by the ideal cycle (Otto or Diesel) that is less than 1 even in an ideal world;
- η_l, limit efficiency: the previous ideal efficiency is further decreased if we take into account parameters variability with temperature;
- η_{ip}, indicated pressure efficiency: considers the fact that, in a 4-strokes, part of the energy is spent during the intake and exhaust in thermodynamic terms;
- η_p: it states how good is, in fluid-dynamics terms, the exhaust/intake cycle;
- η_o, organic efficiency: accounts for all the power takings by engine pumps, generators, but also bearings friction and so on.

Finally, we find the quantities that set the engine:

- u: the piston speed, strictly related to engine rpm. As it can be easily seen, the more cycles per minute are made, the higher the power delivered. That is why Formula1 engines once revolved up to 18000rpm;
- η_{vt}, volumetric efficiency: it states how well the fuel-air mixture fills the cylinder. The filling process is not ideal (i.e. $\eta_{vt} \neq 1$ because of many factors (like valves timing, pressure waves, recirculating flow), but also because of pressure drops inside the intake line, which is a desired thing: the driver, acting on a valve mounted before the intake valve, actually varies η_{vt} in order to set the engine and get what she/he needs.

This summarizes the analysis of engine power output and power settings. It is still necessary, when equipping a vehicle with this engine, to provide a joint able to allow or deny torque transmission to wheels. Such component is the clutch and takes place inside the drivetrain.

2.1.2 The Drivetrain

Engine crankshaft is not directly linked to the wheels. A reduction gearbox is mandatory in order to reduce rpm and increase torque. It modifies the torque characteristic maintaining unaltered the power. A ICE characteristic curve is usually like the one shown in Figure 2.3. Beside it, load torque is represented by the dashed line. Indeed, the engine will not be able to overcome initial load torque without a reduction gearbox. A gearbox also keeps the engine rpm in the optimal range in a wide range of vehicle speed, thus having more performance or reduced fuel consumptions. A schematic of a very simple synchronous gearbox (with two gears) is reported in Figure 2.4. A further description of this assembly, along with many other gearbox configuration, can be found in [12].

Being far simpler, we start explaining gearbox operation with the non-synchronous one. Engine power comes from left on the green shaft (likely with high rpm and low torque). A

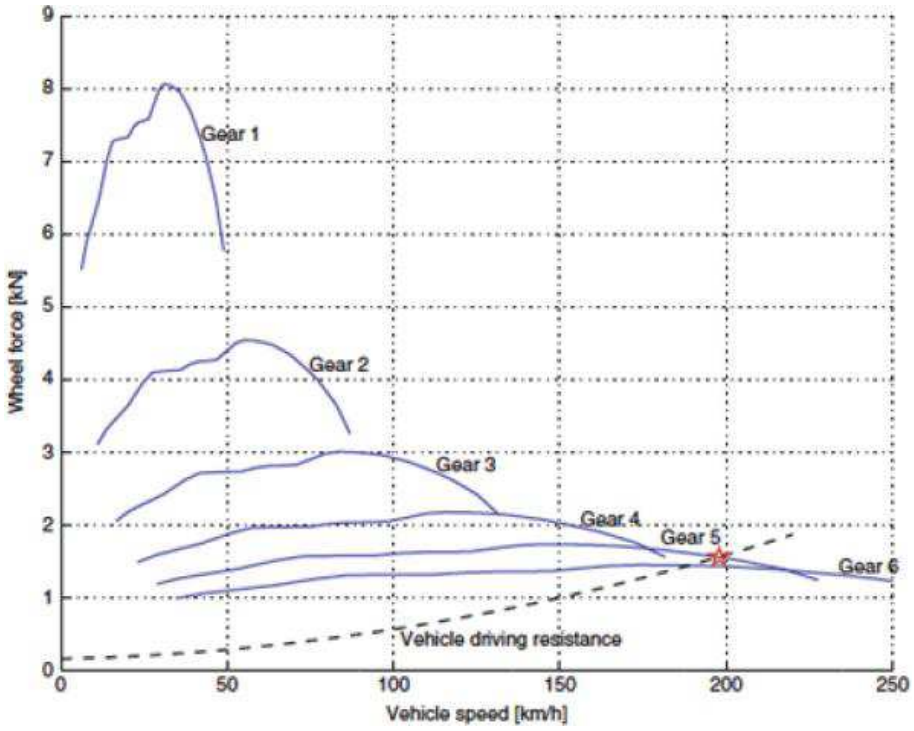

Figure 2.3: ICE haracteristcs. It shows how torque output varies with engine rpm, for different gear reduction. Acutally, to obtain C_ω, x-axis should be divided by wheel radius and y multiplied by wheel radius. In dashed line is reported the resisting force as sum of aerodynamic drag and rolling resistence.

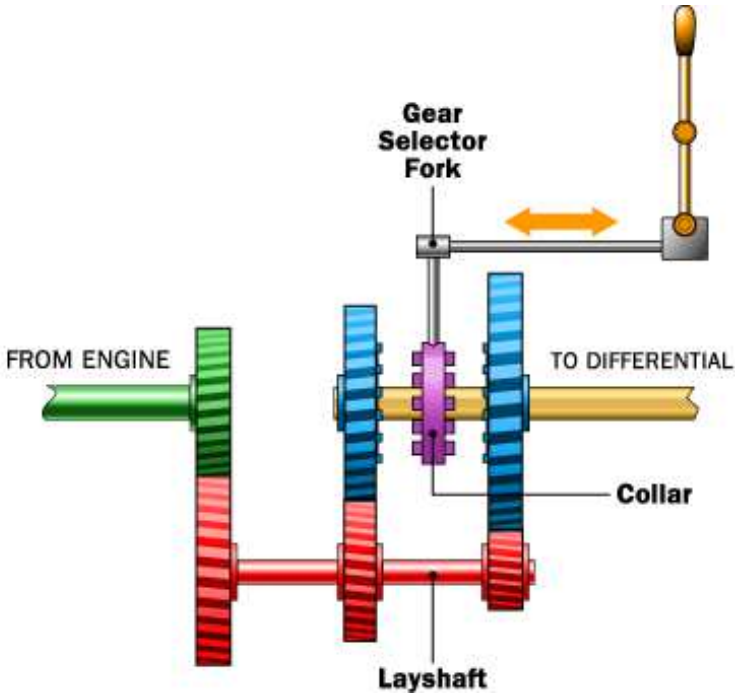

Figure 2.4: Representetion of a two-wheel gearbox, synchronous, with user selection lever.

first reduction occurs when meshing with the red shaft: the layshaft, which will mesh with the driven shaft providing a second rpm reduction (and torque increase, as conservation of power is the law). Here the layshaft meshes with the driven shaft with one gear at a time, thus the driven shaft must slide leftward and rightward to shift gear. Furthermore, for a correct meshing, driven shaft and layshaft must have the same angular speed (else gears will scratch), that never happens. This is the reason for the "non-synchronous" adjective used previously. Therefore, the old "double clutching" technique is required and the driver must have had some experience and sensibility in order to operate it properly.

Luckily, technical improvements made it possible to avoid double clutching with the introduction of a synchronizer. Some architectural modifications are required: first of all, driven shaft gears are not fixed on it: they are free to rotate, and they are constantly meshed with the layshaft gears (see again Figure 2.4 on the left). In addition, these blue gears are equipped with conical teeth on their sides so as to permit meshing with the purple collar. The purple collar is fixed on the driven shaft: when it matches with a blue gear, the shaft starts revolving. Now only the sliding of the collar is required to shift gear, saving space and structure weight. And mostly, the shafts can now have different angular speed as the clutch built inside the collar automatically adjusts speeds instantly. The price to pay is some more complexity and collar wear. This invention dates back to 1952, firstly introduced by Porsche only between second and third gear, but now every light-duty vehicle has got a completely synchronized gearbox that operates smoothly and unnoticed.

With more gears the basic principle remains completely unchanged. Only a bit complexity is added to the leverage system for the driver. All of these scratching and friction may lead to judge such a gearbox unreliable and short-lasting. However, being submerged in oil, wear is consistently reduced. A well designed gearbox may last at least as the engine without extraordinary maintenance.

It is worth saying also that the wear is strongly reduced if no torque is transmitted in gear shifting. This is the second reason for the clutch to exist. Being placed between the engine and the gearbox it is engaged by default, thus allowing torque transmission. When the driver presses the pedal, it disengages, and no torque is conducted to the gearbox any more. The driven shaft, being linked with the wheel axle, is still spinning, but the little inertia moment of the power shaft represents a very small load torque to overcome. When the gear shifting is complete, the clutch slightly slips to adjust the engine rpm with the (reduced) axle rpm.

Further details about clutches are provided in the following section.

The torque path carries on and the driven shaft leads into the driving axle, which must be provided with a differential. Its purpose is to allow differential wheel angular speed during cornering without slipping. Figure 2.5 shows a cad rendering differential in which the torque comes from a shaft (not visible) meshing with the greatest gear. Two semi-axles end with a homokinetic joint allowing wheel steering.

Finally, the global reduction ratio must take into account also tyre diameter.

2.1.3 Cooling

Each thermodynamic cycle, may it be a Diesel cycle, Joule cycle, Otto cycle, needs a phase in which temperature cools down. Heat has to be drained in some way. Beside this reason, there are several others. For example, thermodynamic efficiency raises as T_{min} in the cycle is decreased. Or even materials will not sustain too high temperature. Last but not least, an overheated fuel-air mixture will self-ignite too early causing engine damage.

It is of outstanding importance to equip a ICE with a well designed cooling system. Usually vehicle engine are liquid cooled, with a water circuit running inside the cases in channels built for the purpose. This is because water (or liquids in general) has a greater exchange coefficient rather than air. The water warmed up then flows into a radiator located in the front of the car,

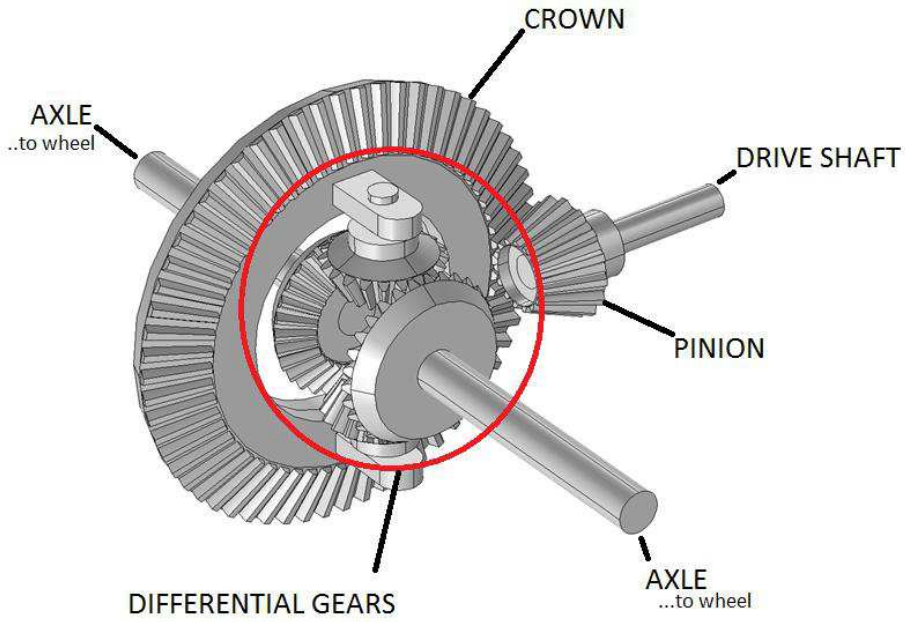

Figure 2.5: Rendering of a car differential. Engine torque comes from the driveshaft. Upper and lower differential gears are welded to the crown, so that thery rotate with the engine shaft. Side differential gears are welded to axles instead. While not cornering, axles have the same speed and side gears do not have any relative angular speed, and are equally pulled by the crown. During cornering, they have a relative angular speed, making the upper and lower gears rotate while still transmitting torque.

where it can be cooled down by the oncoming fresh air. A fan is bolted behind the radiator to provide forced convection even when the car is standing still.

However, cooling circuit does not contact only the engine. For excellent they are, gears and joints will always lose some energy in friction, thus heat is generated. It follows that gearboxes, differentials, crankshafts must be cooled like the engine (even if their thermal output is far less). Nowadays, the clutch is a part which has not a dedicated cooling circuit. It simply cools down with air flows around it, which is often very slow. Therefore, with severe external conditions or low car speed, the clutch becomes the critical car component.

2.1.4 Electrical system

This is a very complex assembly which does not know slowing in development. Many vehicle assemblies are managed by electronics with sensors, control units, algorithms and so on. Its purpose is to increase performance, reliability, safety, and to provide real time diagnostics and warnings. There is no doubt that such increase in complexity results in a much greater whole performance.

Beside these features, headlights, direction indicator, stops and so on rely on this system. Also the infotainment onboard system may be seen as part of the electronic system. We do not go deeper in the description of them, as our problem is quite independent of it.

2.1.5 Steering, Suspensions, Brake Systems

These assembly are fundamental too in vehicle operation, but have limited impact on the clutch heating, so only few words will be spent.

Steering system allows wheels orientation during cornering. The simplest type is the rack and pinion architecture, in which the steering wheel rotates a pinion meshing with a rack. The rack, sliding leftwards or rightward, steers the wheel with appropriate angles. As mentioned above, a constant-speed joint links the steering axle to the wheels permitting smooth torque transmission.

Suspensions are responsible of dumping oscillation induced by the road roughness (holes, cracks, rocks and so on) which may overload the structure (and for sure reduce passengers' comfort). The working principle of this system is represented by a mass-spring-damper system. Rods with careful studied geometries join the car chassis with the unsprung masses, i.e. uprights, wheels, brakes and so on.

Brakes obviously are used to slow or stop the car. Steel disk brakes are the most widely used type as they provide good braking power together with easy construction and maintainability. Brake heating shall be taken into account during design and heat management analysis.

2.2 The clutch

The clutch is a mechanical joint whose function is to transfer torque between two coaxial shafts by friction, with the ability to stop this transfer with an appropriate command. These shafts may have different angular speed, in that case torque transfer is essentially determined by sliding friction, or may have the same speed, with torque being transferred by adherence. In both cases, friction torque depends on several factors, among which the most important are pressing force, friction coefficient, area involved. Therefore, it exists an upper limit on the maximum torque to transmit. If exceeded, the clutch starts slipping (with all the consequent troubles). Usually, when the shafts have different speed, friction torque is greater than the resisting torque, causing the driven shaft to accelerate until they have the same speed.

In automotive applications, the clutch has to perform two major duties:

1. Keeping the engine running even when the vehicle is standing still;

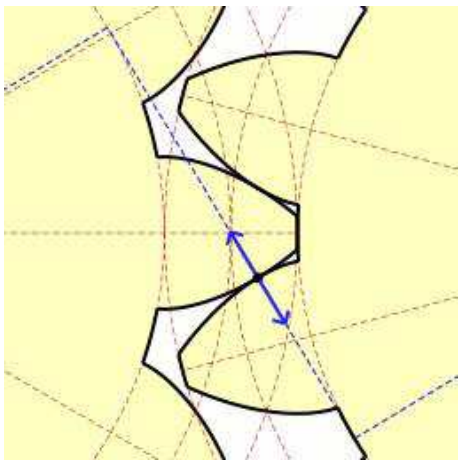

Figure 2.6: Sketch of a perfect contact between meshed gears. If gears were slightly closer or farther teeth would likely fail.

2. Strongly reducing gear shifting stresses for the gearbox.

Let us better explain why a clutch should carry out those tasks. First of all we have to keep in mind that a ICE has only one useful phase out of four. Three phases spend part of the energy delivered by the one remaining. The engine idle condition occurs when the power output is that strictly necessary to cover that part spent by engine itself (plus a part lost for friction, and that necessary to keep in rotation pumps and so on). The vehicle must have the chance to stop with its engine still running, which has therefore to stay above a minimum rpm (usually less than 1000). Therefore, a reversible disengagement between wheels and engine is to be provided. The clutch performs this disengagement.

Furthermore, the clutch allows a better gear shifting. We remind that during a shift the collar (rotating with the wheels) meshes with the layshaft (with no angular speed). The only torque the teeth are facing is that required by the layshaft to reach the driven shaft rpm. The layshaft does not have a huge inertia moment compared to the engine's one, thus teeth are quite unstressed. Later, when gears are meshed, the clutch re-engages slipping a little to adjust engine rpm with the wheels' ones. If we would not disengage the clutch the shift is possible without stress only when engine rpm equals wheels (reduced) rpm. In addition, if we force the shift, teeth would face a huge force even during meshing, when contact points are not those the teeth were designed for (see Figure 2.6 for more clarity). In some car competition a particular gearbox has been developed (the so-called dog-box transmission) to permit gear shifting without clutching, exploiting greater collar teeth (called dogs), but this can occur a limited number of times. More complex transmissions which allow shifting with no power interruptions do exist but are based on a completely different design (and where first introduced by Ferrari in 2007).

The clutch thus represent a reversible mechanical joint transmitting torque by friction. In principle, it is composed by two disk, fixed one to the power shaft and the other to the driven shaft. When the clutch is engaged, the disks are pressed together and the torque is transmitted by the friction between them. When the clutch disengages, the pressing force is removed and the friction reduces to zero, as the torque output. During re-engagement it is very likely that shafts have different rotating speeds: disks will slip until the driven shaft speed will match the power shaft's. In this situation the torque accelerating the driven shaft is provided by skin friction, thus heat is generated. After engagement, torque is transmitted by adherence, and no power is wasted. Maximum torque managed by a clutch relies on this adherence condition, depending on applied normal force, friction coefficient, involved area, and many others.

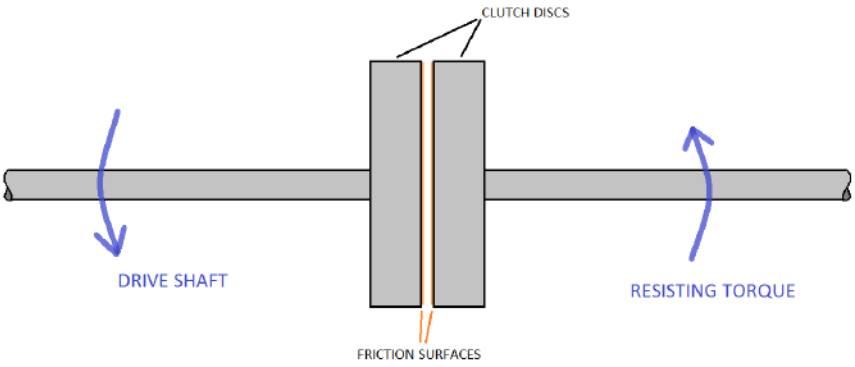

Figure 2.7: Sketch of the simplest kind of clutch between coaxial shafts.

2.2.1 Clutch governing phenomena

In this section we outline the phenomena driving clutch operations, with simple associate equations, to provide a minimum of mathematical support and to give some logics to what will be shown later about the simulations environment.

The simplest clutch architecture is the one sketched in Figure 2.7, with a driving shaft (on the left), a driven shaft (on the right) and the clutch between them. This clutch is disengaged, meaning that it is not transferring torque as the discs are not touching. The driving shaft is providing a torque C_m with a speed ω_1 while the driven shaft is resisting with a torque C_r (thus opposite in sign) and a speed ω_2, generally different from ω_1. They have their own moment of inertia. When disc 1 and 2 are set to contact, a friction torque start acting and, for its own nature, develops negative work (dissipated as heat).

If we call θ_1 and θ_2 the generic angle ranged by the shafts during the engaging operation, we are able to write down a first energy balance equation for that system (as exposed by [8]:

$$C_m d\theta_1 - C_r d\theta_2 - dL_{fr} = dE_{kin} \tag{2.2}$$

where dL_{fr} is energy lost for friction and dE_{kin} is the global increment of kinetic energy (which may also be negative if C_r is really strong). We can integrate this relation on the total operation duration τ to obtain the energy loss:

$$L_{fr} = \frac{1}{2}[I_1\omega_{1_0}^2 + I_2\omega_{2_0}^2 - (I_1 + I_2)\omega_f^2] + \int_0^\tau C_m\omega_1 dt - \int_0^\tau C_r\omega_2 dt \tag{2.3}$$

where subscripts $_0$ stands for "initial condition" and $_f$ for "final condition". ω_1 and ω_2 are angular speeds during the generic time instant, and vary accordingly to their respective dynamic equilibrium equations:

$$\begin{cases} C_m - C_r - I_1\dfrac{d\omega_1}{dt} = 0 \\ \\ C_{fr} - C_r - I_2\dfrac{d\omega_2}{dt} = 0 \end{cases} \tag{2.4}$$

When $\omega_1 = \omega_2$ the system behaves as a rigid body and the equilibrium equations get a little simpler. A typical engagement diagram is provided in Figure 2.8, where it is clearly seen the difference between clutch slipping (early instants) and clutch fully engaged (later instants).

If engagement period is really fast, the whole operation may be assimilated to a collision, with the equations getting far simpler, but without much interest in this work (as no heat would

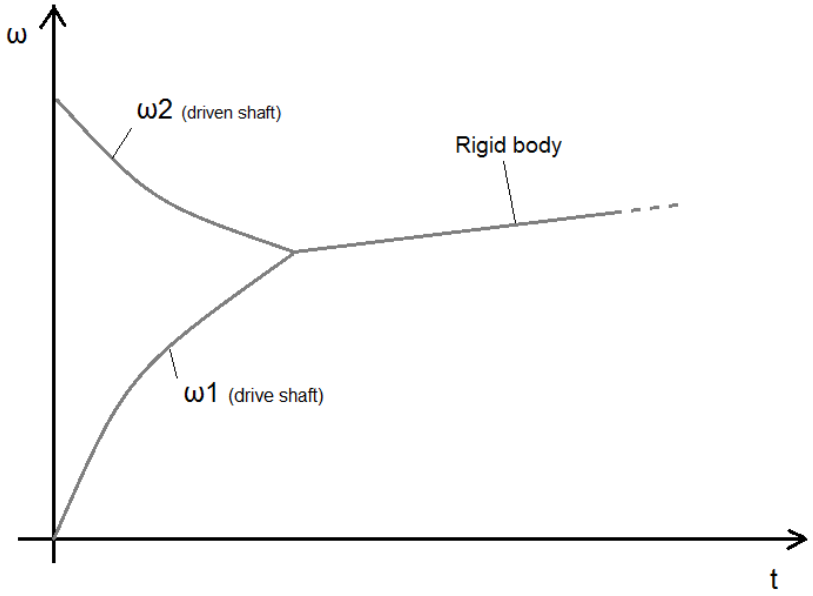

Figure 2.8: Angular speeds of shafts during clutch engagement. Trends are given by respective dynamic equilibrium relations.

be generated). In contrast, the more the engagement lasts, the more heat is generated, with temperature approaching critical.

In our simulations, we are not interested in system dynamics, but only in heat production. However, a brief description of clutch torque capacity is provided in the following Section.

2.2.2 Clutch Torque Capacity

Here we briefly report a mathematical expression for the torque a clutch is able to transmit. The clutch model is very similar to ours: a single disc clutch. As we would expect, maximum torque transferred by clutch depends on disc area A, normal force F_N acting on it and the dynamic friction coefficient μ. Normal force, provided by the diaphragm spring action, generates a friction force on the pressure plate and flywheel that translates into a torque.

Referring to Figure 2.9, we say that pressure acting on clutch disc is as usual

$$p = \frac{F_N}{A} = \frac{F_N}{\pi(r_2^2 - r_1^2)} \tag{2.5}$$

since the disc area is $A = \pi(r_2^2 - r_1^2)$. However, in order to express a torque, we must divide the disc into several strips of width dx at a certain distance r from the disc centre, which can be considered constant. Thus,

$$dA = 2\pi x dx \tag{2.6}$$

Consequently, the differential normal force dF_N acting on the differential area dA becomes

$$dF_N = pdA = \frac{2F_N x dx}{r_2^2 - r_1^2}. \tag{2.7}$$

Furthermore, we know that the friction force generated by a normal force is given by $F_a = \mu F_N$. Then,

$$dF_a = \frac{2\mu F_a x dx}{r_2^2 - r_1^2}. \tag{2.8}$$

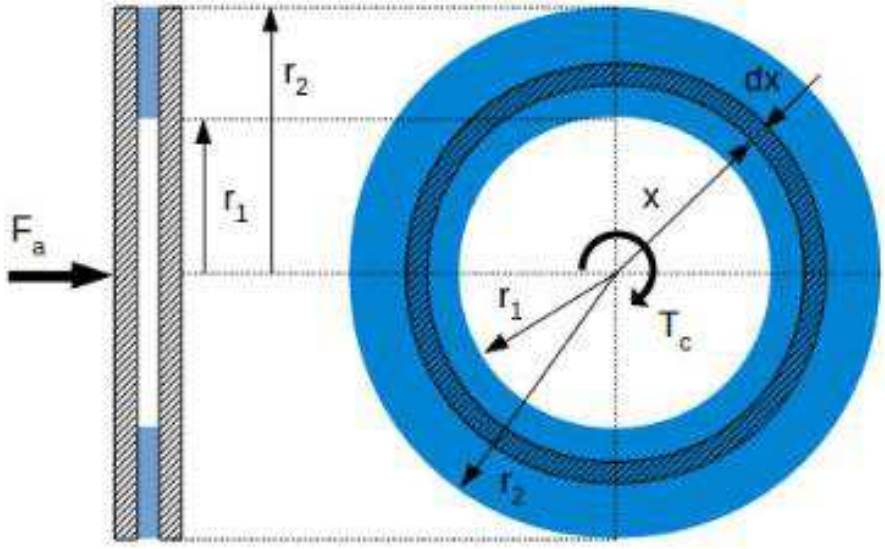

Figure 2.9: Clutch disc scheme for torque calculation.

Finally, expressing the torque as usual with $dT = xdF$, we integrate from r_1 to r_2 to obtain the total torque generated by friction force F_a:

$$T = \int_{r_1}^{r_2} \frac{2\mu F_a x^2 dx}{r_2^2 - r_1^2} = \frac{2}{3}\mu \frac{r_2^3 - r_1^3}{r_2^2 - r_1^2} F_N \tag{2.9}$$

that is the torque transmitted by an engaged clutch. As we have expected, it depends on normal force, geometric dimensions and friction coefficient. The higher μ, the higher the torque a clutch can handle. We cannot increase too much F_N by equipping the clutch with a stiff spring, as the same F_N has to be exceeded by the driver's foot in order to disengage it. A bigger disc, on the other hand, may cause packaging difficulties. Research is therefore directed to find materials with the highest friction coefficients.

2.2.3 A Real Clutch

In this section we describe how phenomena shown in the last section are implemented in a working object. We will focus on the simplest clutch architecture, the dry single-disc for manual transmission, which is by the way the most used in European common applications.

As stated earlier, a clutch connect and disconnect the engine and the wheels. It is therefore located between the engine and the gearbox. In a commercial car, its external dimensions are the same as the flywheel, then its diameter is similar to engine height. Being used during usual vehicle operations, an easy-to-use command must be provided to the pilot for clutch disengagement (engaged clutch, thus transmitting torque, is its default state). The release mechanism comprehends the pedal in the cockpit, its mechanical linkage (or even hydraulic) to the clutch fork. However, we are not much interested about that mechanical linkage, so we will assume that the pilot in some way transmit a force to the clutch, and then we will focus on how this force is effectively exploited in disengagement. A real clutch section is provided in Figure 2.10, where mechanical linkage to the cockpit is not shown.

There, the power (i.e. the engine) comes from the left and the gearbox is located on the right. The flywheel is bolted to the engine crankshaft to operate properly, and its right side acts also as a friction plane. It is easy to suppose that this side of the flywheel has sustained

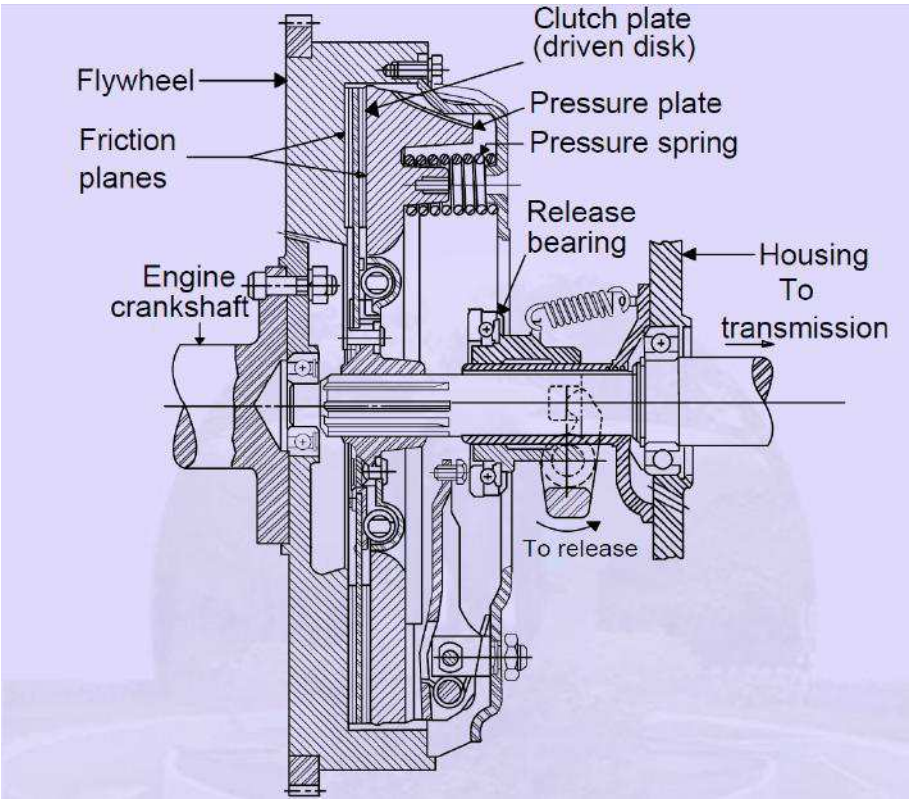

Figure 2.10: Technical drawing of a clutch section.

some superficial treatment to better face wear. Then we find the pressure plate, bolted to the flywheel and thus fixed to the crankshaft. A little gap exists between them, namely $7 - 8mm$, and there the actual clutch plate (a steel disc) is placed. This plate is then bolted to the driven shaft. While the clutch is engaged, the pressure plate presses the disc against the flywheel. In an adherence condition, the friction between them is enough to transmit torque from the crankshaft to the gearbox. A simpler way to build this assembly would see no disc and the pressure plate bolted not to the flywheel, but to the driven shaft instead (this scheme is very similar to that presented in the previous section). However, by using a disc between flywheel and pressure plate, we double the friction area with little costs. Having a great friction area is important to manage high torques.

Beside these three fundamentals elements (i.e. the flywheel, the pressure plate, and the clutch plate) there are others whose function is to make the clutch a reversible joint. Therefore, the pressure plate is linked to the flywheel by means of springs. These springs ensures a proper pushing force to maintain the clutch engaged. When the driver wants to disengage it, he/she applies a force which goes against the springs action, thus lowering, and cancelling, their pushing force. At this point, no normal force acts between the clutch plate and the pressure plate/flywheel, no friction exists and engine torque does not reach the gearbox any more. As the driver releases the pedal, his/her contrasting force is removed and the springs crush again the clutch disc, allowing torque transmission.

Those springs are represented in Figure 2.10 by usual coil springs. There, a release bearing (i.e. a ball bearing with a collar packed in a lubricated case) slides on a hub sleeve responding to driver's commands of engagement/disengagement. It can also be seen a little arm linked to the release bearing as the last part of the mechanical leverage. Anyway, coil springs are rarely adopted for this application. They are better represented by a diaphragm type spring (shown in Figure 2.11). This spring is secured to the pressure plate on its outer rim, while on its inner rim the driver force is applied. When this force is applied, an opposite force generates on the

Figure 2.11: A diaphragm spring.

outer rim, causing the pressure plate to get away from the flywheel. The main strength of this spring is its highly non-linear characteristics, as it is nearly sinusoidal (a good force-displacement law has been provided by Almen and Laszlo). If cleverly exploited, the driver effort is strongly reduced. Other advantages in using this kind of spring comprehend more uniform pressure force, less mechanical leverage and easier assembly, while the release bearing acts in the same fashion.

Spending two words about the linkage between the driver's pedal and the release bearing, usually a not-in-pressure hydraulic circuit links them; the circuit is pressurized in a semi-automatic transmission and it is electrically actuated in a fully automatic transmission.

Finally, the clutch housing (not represented in Figure 2.10), also called the bell housing due to its shape, keeps the assembly together. It provide support for bearings, springs, leverage, and so on. It is bolted directly to the engine case. The gearbox is in turn bolted to the bell housing.

Other parts add to the assembly, with purposes that do not influence directly clutch operation, but whose presence facilitate driving or improve comfort. An example is the damped flywheel, containing some springs between its structure and the crankshaft joint, in order to provide increased damping for engine oscillation. We may find joints between flywheel and pressure plate that are more than a simple nut and bolt, to further improve damping. We will find for sure supports for the clutch plate and flywheel enclosure, plus other component to fit the flywheel to pressure plate at proper distance, providing also means for regulation, as well as engine-flywheel joints. Finally, we expect to find the engine starter very close to the clutch, as it meshes with the teeth on the flywheel outer diameter.

What we described so far applies both to dry either wet clutch. The only difference between them concerns the medium surrounding the clutch: the first is air, the latter is lubricant oil (and of course the bell housing must be sealed). Reasons to use a more complex wet clutch involve the torque to be transferred, the maintainability issue, how much performance is required and so on. In fact, a wet clutch is service-free, meaning that it is designed to last the entire machine life-cycle. Furthermore, it is able to manage much higher torques than a dry clutch (when several discs are required) or, if rapid gear shifting is a must, it allows to build a double-clutch architecture. It brings also some disadvantages beside an augmented complexity: there is viscous drag, the friction coefficient is reduced (thus a larger area must be provided), but the heat dissipation is enhanced. Summing up these elements, often the choice falls in favour of the dry clutch, leaving the wet for heavy duty trucks or racing cars.

As just introduced, heat dissipation is an issue to take into account during design. A dry clutch sometimes shows holes or grooves on its external perimeter to improve cooling, while the bell housing is left open to allow some fresh air to come in and provide some heat exchange. However, heat sources are many (the engine, the gearbox, friction) and heat sinks few. The disc itself has a low mass, being relatively thin, whilst the flywheel, being much more massive,

allows heat dissipation for some time. Troubles arise during continuous clutch operation: in this case some heat dissipation devices are required. An air flow, coming from a fan or ram air, is a starting point, even if gaps between part slow it down a lot. In conclusion, determining how much this air flow is effective is the subject of this Thesis.

2.3 Heat transfer

Heat means thermal energy. Thermal energy can be identified as the kinetic energy owned by single particles in their microscopical chaotic agitation. The physical quantity used to quantify the amount of such kinetic energy is *Temperature*. Heat (or kinetic energy, which is the same) can be transferred across a well defined boundary, in a thermodynamic system, by means of:

- conduction;
- convection;
- phase change;
- radiation.

Such energy transfer always occurs between two sides of a boundary at different temperature, exploiting usually a combination of the three mechanisms stated above. The process stops when the sides are at the same temperature, reaching locally thermal equilibrium. We refer to this principle as the *zeroth law of thermodynamics*, as reported in [**?**]. The total amount of energy transferred depends on the path followed, and not only on initial and final states (making it a path function, opposed to a functions of states). Furthermore, the energy flow is always from the warmer side to the colder side, as stated by the *second law of thermodynamics*. For completeness, we report also that the *first law of thermodynamics* states that, in an isolated system, energy cannot be created nor destroyed.

2.3.1 Thermal energy

All particles, whatever the phase, have got an intrinsic kinetic energy due to microscopical motion. It is possible to give a classical statistical mechanics description of such behaviour, rather than quantum statistical mechanics. Briefly, each particle (atoms, molecules and so on) has its own velocity in space. This velocity (magnitude, direction) depends on the kind of binding of the owner particle and collisions between different particles. The magnitude of the velocity is temperature-dependent, and collisions determine energy transfers. Therefore, conduction can be seen as energy flow via collisions, thus kinetic energy transfer from a particle to its neighbours.

With statistical mechanics it is possible to evolve from a submicroscopic point of view to a macroscopic one, relating particle-size physical quantities (such as kinetic energy) to intensive ones (like temperature and pressure). This theory is called *kinetic theory of gases*. One of the most important results from this theory states the relation between temperature and motion:

$$T = \frac{2}{3} \cdot \frac{\frac{1}{2} m \overline{v^2}}{k_b} \tag{2.10}$$

where $\overline{v^2}$ is the mean quadratic velocity and k_b is the Boltzmann constant.

Another important result says that gas pressure is given by particles colliding the system boundary (where pressure is measured), thanks to their momentum change.

There are many ways, however, to quantify "how much hot" a body is, or in more scientific words, "how much energy it has got". One of these is, as stated, a temperature measurement. Beside it there are at least two other physical quantities that are more strictly related to energy. Their names are *internal energy* and *enthalpy*.

The first one, e_{int}, takes into account only contributions from molecular motion. It is the minimum quantity of energy needed to "create" a thermodynamic system, excluding its whole motion or interactions with other systems or forces. The last one, h, considers also the "existence" of that system with its surroundings in space, thus with its own pressure and volume. In other words, enthalpy is a thermodynamic potential. Mathematically, their expression (per unit mass) are

$$e_{int} = c_v \cdot T \tag{2.11}$$

$$h = e_{int} + pV = c_p \cdot T \tag{2.12}$$

which are both functions of states.

2.3.2 Conduction

Heat transfer via conduction occurs between two bodies when they are put in contact. Physical contact is needed to allow molecular collisions. Energy flow continues until the bodies reach the same temperature, or they are physically separated. Conduction occurs also inside a single body if, in its initial condition, its temperature field is not uniform (it is indeed the prevailing heat transfer mechanism inside a solid).

A substance that allows great energy flows via conduction is said to be conductive, while one that prevents such great flows is called insulating (or non-conductive). The best conductive materials are the metallic ones, while gas are very insulating. *Thermal conductivity* λ quantify how much heat a material is able to transfer via conduction.

As already stated, heat transfer are always driven by temperature difference (excluding phase change). Fourier's law relates heat flux with this temperature difference in a steady-state condition:

$$\vec{q} = -\lambda \nabla T. \tag{2.13}$$

However, it often happens to have an unsteady situation to analyse. Temperature field will depend on time too, and the Fourier's law modifies to take into account this dependence. The resulting expression is the heat equation:

$$\rho c_p \frac{\partial T}{\partial t} - \nabla \cdot (\lambda \nabla T) = \dot{q} \tag{2.14}$$

which takes into account also a source term $\dot{q}$. This equation is a diffusion equation, and can therefore be useful in many fields, even far from the thermodynamics one. Furthermore, we can re-write it as (neglecting source therms)

$$\frac{\partial T}{\partial t} - \nabla \cdot (\alpha \nabla T) = 0 \tag{2.15}$$

where $\alpha = \frac{\lambda}{\rho c_p}$ is highlighted, called *thermal diffusivity*. It is clearly a quantity that assumes importance in a transient analysis, and it expresses how good a material diffuses heat inside itself (or, in other words, how much uniform its temperature field will be).

2.3.3 Convection

Convection (sometimes called advection) is a transfer of heat by mass motion, i.e. by physically moving particles at a certain temperature towards areas at different temperature. For it to be possible, there must be fluid movement inside the system, thus this mechanism only applies to gases, liquids, or plasmas. During this transport, anyway, conduction takes place as usual, so that it is nearly impossible to have strictly convection without conduction or vice versa.

Fluid convective motion may be self generated either imposed by external conditions. In the first case, temperature field interacting with gravity field generates buoyancy forces which impress a motion to the fluid. This kind of convection is said to be natural, because no external

phenomena play any role. Temperature and motion fields are strictly coupled. In contrast, forced convection involves external momentum source, like a fan, a pump, or ram air too. In this case, heat transfer is greater than in natural convection. For sake of completeness, natural convection always occurs when a ΔT exists inside a gravity field, but it is completely obscured by forced convection when present. In forced convection, in fact, motion field have a huge influence on temperature field, but the contrary rarely happens. As a rule of thumb, when average speeds are less than $1\frac{m}{s}$ natural convection is predominant.

The Rayleigh number provides a qualitative tool to determine who, between conduction or convection, is playing the major part:

$$Ra = \frac{g\beta}{\nu\alpha}(T_w - T_\infty)x^3 \tag{2.16}$$

where β is the thermal expansion coefficient and x the characteristic length. When $Ra > 2000$, convection is predominant. However, this method only applies to natural convection, as forced convection is for sure higher than natural.

Laws to express convective heat flux clearly reminds Fourier's Law for conduction, with the only difference relying in convective coefficient h:

$$\vec{q} = -h\nabla T. \tag{2.17}$$

This law applies only when there is moving fluid contacting a fixed solid; fluids at different temperature mixing is by far a more difficult problem to solve, involving first of all Navier-Stokes equations. Anyway, even in cases when the law holds, determination of h coefficient may result difficult as it depends on fluid, fluid speed, turbulence intensity, boundary layer etc. Actually convective heat transfer coefficient is a simplified mathematical abstraction of a more complex physical phenomena.

2.3.4 Phase change

Actually this is not usually thought of as a heat transfer mechanism, but it still offers an interesting way to manage heat. In a few words, each phase change needs an amount of heat (positive - required, or negative - released) to occur. Such amount per unit mass is called *specific latent heat*, which is different for each material and phase change.

For example, during a constant-temperature phase change of boiling water from liquid to vapour it is required $2.26\frac{MJ}{kg}$, that is a considerable quantity of energy (five times the heat required to bring liquid water from $0°C$ to $100°C$).

2.3.5 Radiation

This heat transfer mechanism is the only one to occur without physical contact, as it exploits electromagnetic waves. Charged particles randomly moving due to thermal agitation irradiates energy because they undergo accelerated motion. Each charged particle in acceleration produces electromagnetic waves. The energy so emitted is collected by surrounding bodies, which are themselves emitting radiation for the same reason.

The Stefan-Boltzmann equation states that the amount of heat exchange between two bodies infinitely large of the same material is

$$Q = \epsilon\sigma(T_a^4 - T_b^4) \tag{2.18}$$

where ϵ is the surface emissivity (unity for a black body, i.e. the best radiator) and σ the Stefan-Boltzmann constant ($\sigma = 5.67 \cdot 10^{-8}\frac{W}{m^2K^4}$). This constant is very low, thus radiation is visible only when temperatures (or ΔT) are quite high. However, Q is related to T^4, so it raises very rapidly. For bodies of different material and finite dimensions the equation becomes more complex, involving different ϵ and view factors F, but the basic rule remains the same.

Radiative heat transfer occurs among bodies provided that they are in line of sight, even if they are in vacuum or there is a transparent medium between them. This is the reason why it is usually implemented in satellites thermal control systems. There, other surface properties assume importance, like reflectivity, absorptivity and transmissivity.

Radiation is not considered inside the same body because as one of its particle emits photons they are near-simultaneously received and re-emitted by neighbours of this particle.

2.4 CFD methods

Computational fluid dynamics (CFD) is the investigative approach adopted for this Thesis. It is therefore an approach based on numerical simulation. CFD is that branch of fluid dynamics that applies results from numerical methods to solve fluid-related equations and problems. Its objective is to discretize physics in time and space, along with the proper governing equations, setting boundary conditions and solve the equations (i.e. finding a solution). These equations are the Navier-Stokes ones, which are far from being analytically solved, and therefore CFD studies are strictly coupled with advancing in mathematics and numerical analysis.

Before deepening in CFD methods, we open a section to describe equations which lies beneath fluid motion and driving phenomena.

2.4.1 Governing equations

Many scientists and engineers from the past made lot of efforts in developing mathematical tools in order to understand how nature works. These tools are equations, concepts and schemes that are able to describe fluid dynamics.

To describe problems there are unknown and known quantities and the relations between them. The aim is to find enough relations to pair the number of unknowns. Relations must be sought in those physical principles that are globally assumed valid without hesitation. In fluid dynamics, conservation laws are widely used: mass, momentum and energy. We analyse their integral formulation and meaning in the following, as brilliantly explained in [13]. We will consider a fixed and finite control volume of fluid, and we will apply conservation laws to this volume (large enough to contain several fluid particles).

- Conservation of mass: single scalar equation.

$$\frac{\partial}{\partial t}\int_V \rho dV = -\int_S \rho(\vec{v}\cdot\vec{n})dS \tag{2.19}$$

  In words: the time-rate change of mass inside the volume equals the net of the mass flow across its boundary.

- Conservation of momentum: three component vector equation.

$$\frac{\partial}{\partial t}\int_V \rho v_x dV + \int_S \rho v_x(\vec{v}\cdot\vec{n})dS = -\int_S p n_x dS + \int_S(\vec{\tau_x}\cdot\vec{n})dS + \int_V \rho f_x dV \tag{2.20}$$

$$\frac{\partial}{\partial t}\int_V \rho v_y dV + \int_S \rho v_y(\vec{v}\cdot\vec{n})dS = -\int_S p n_y dS + \int_S(\vec{\tau_y}\cdot\vec{n})dS + \int_V \rho f_y dV \tag{2.21}$$

$$\frac{\partial}{\partial t}\int_V \rho v_z dV + \int_S \rho v_z(\vec{v}\cdot\vec{n})dS = -\int_S p n_z dS + \int_S(\vec{\tau_z}\cdot\vec{n})dS + \int_V \rho f_z dV \tag{2.22}$$

  where $\vec{\tau_x}$, $\vec{\tau_y}$, $\vec{\tau_z}$ are the three component of the shear stress tensor $[\tau]$, explicitly written as

$$[\tau] = \begin{bmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_{yy} & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_{zz} \end{bmatrix} \qquad \rightarrow \qquad \begin{bmatrix} \vec{\tau_x} \\ \vec{\tau_y} \\ \vec{\tau_z} \end{bmatrix} = \begin{bmatrix} \sigma_{xx}\vec{i} + \tau_{xy}\vec{j} + \tau_{xz}\vec{k} \\ \tau_{yx}\vec{i} + \sigma_{yy}\vec{j} + \tau_{yz}\vec{k} \\ \tau_{zx}\vec{i} + \tau_{zy}\vec{j} + \sigma_{zz}\vec{k} \end{bmatrix}$$

while pressure p is the thermodynamic pressure, and shall not be confused with normal stresses σ_{ii} (even if in case of incompressible fluid the average σ_{ii} equals thermodynamic pressure p). For a newtonian fluid, shear stresses τ_{ij} are written as

$$\tau_{ij} = \delta_{ij}\lambda_{bulk}(\vec{\nabla} \cdot \vec{v}) + \mu\Big(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\Big) \tag{2.23}$$

and $\lambda_{bulk} = -\frac{2}{3}\mu$ for our aerodynamic purposes.

In words the equation says that the global variation of momentum inside the control volume must equal the forces acting on it, as the second Newton law wants ($F = \frac{d(mv)}{dt}$).

- Conservation of energy: single scalar equation.

$$\frac{\partial}{\partial t}\int_V (\rho e^0)dV + \int_S \rho(e^0 + \frac{p}{\rho})(\vec{v} \cdot \vec{n})dS = \int_S [([\tau] \cdot \vec{v}) \cdot \vec{n}]dS - \int_S [-\lambda\vec{\nabla}(T) \cdot \vec{n}]dS + \int_V \rho\dot{\zeta}dV + \int_V \rho(\vec{f} \cdot \vec{v})dV \tag{2.24}$$

where e^0 is the total energy (thus internal energy plus kinetic energy) per unit mass, and $\dot{\zeta}$ represents heat sources. This equation is saying that variations in energy inside the control volume are equal to energy supply by forces acting on the volume. In fact it is easy to see that most of terms here are strictly related to those in momentum equation, but written as force instead of power. It is worth noting also that the second term $e^0 + \frac{p}{\rho}$ is nothing else than total enthalpy h^0. Also, the second term on the right-hand side represents conductive heat transfers, with explicit Fourier's law.

We wrote integral form of conservation law for sake of clarity, but it is possible to write them directly in differential form, or derive it from integral form using Gauss theorem. Integral form has anyway an advantage: it admits discontinuities in fluid domain without breaking down the solution process. It is therefore "more fundamental" than the differential form. This last one does not admit discontinuities, like shock waves, being therefore inappropriate for some problems. For this reason differential form is said to be a *weak* form.

We put into formulas immovable physic principles. These are globally known as the *Navier-Stokes equations* (even if only the momentum equation comes from the work of Navier and Stokes). They are exact equations and do not contain any assumption or simplification. However, mathematics contained inside is so heavy that it is impossible to have an analytic solution for most problems, and it is neither sure that this solution exists and is unique (it doesn't exist a closed form solution). Non-linear terms like $u_i \cdot \frac{\partial u_i}{\partial x_i}$ are really stiff to manage, and these equations have plenty of them. This is the reason why CFD studies often are strictly connected with numerical analysis.

However, the problem is not closed yet. By now we only have 5 equations and 7 unknowns: ρ, v_x, v_y, v_z, p, μ, T. Two more relations have to be found.

- Ideal gas law.

$$\frac{p}{\rho} = \frac{R}{M_m}T \tag{2.25}$$

where $R = 8314\frac{J}{K \cdot mol}$ and M_m is the molar mass of the gas, a known quantity. Thus we add one relation without introducing unknowns.

- Sutherland's law.

$$\mu_{(T)} = \mu_0\frac{T_0 + C}{T + C}\Big(\frac{T}{T_0}\Big)^{\frac{3}{2}} \tag{2.26}$$

where C, μ_0 and T_0 are reference values for that gas, found in laboratory tests, then known. In this case too we added one relation and no unknowns. The Sutherland's law is only one

relation for $\mu_{(T)}$, one can use the one which better fits his aims. Same for the ideal gas law.

Finally, the problem is closed and we can look for a solution. This means that we would like to find time-dependent field functions for each unknown in the problem (for example, we would like to have an explicit expression $\rho = \rho_{(x,y,z,t)}$). At actual state-of-the-art, this is impossible. Now we are therefore going to see how it is anyway possible to get the fluid field description.

2.4.2 Turbulence modelling

A model is a mathematical description of a physical phenomenon, usually based on some hypotheses or assumptions, aimed to greatly simplify this description without invalidate too much reality. This reduces efforts in studying or foreseeing phenomena behaviour, with negligible accuracy losses. An example of model is the Eulero-Bernoulli beam theory: nobody can know the exact three-dimensional stress distribution in a I beam during bending, but nobody cares: if (assumption) the beam is slim and homogeneous, with one simple equation (mathematical description) we obtain a modelled stress distribution along the beam which is really close to reality, with modelling errors completely negligible. Assumptions and descending equations set up the model.

Modelling may also be the only way forward, specially when phenomena are too complex. Navier-Stokes equations previously written do not model fluid dynamics: they are exact equations and account for every fluid motion behaviour. However, turbulence studies report that motion structures exist at such microscopical scales (e.g. Kolmogorov scale) that discretization would become too small (too expensive as computational costs). Usually too refined mesh are not affordable, thus some models are introduce to still take into account turbulence effects (which remain unresolved) but with much coarser grids. By doing so, turbulence is no longer simulated, but modelled. Mathematical details can be found in [11].

DNS

Direct numerical simulations make no use of modelling. Turbulence, being already accounted for in Navier-Stokes equations, is brutally simulated. This requires extremely fine computation grid, in order to get even the smallest eddy structures, with incipient dissipation. Such simulations can be run in limited fluid domain, with extremely simple geometries, as computational costs are extremely high. They do not find any practical industrial use, but they have major importance in research sectors.

LES

Large eddies simulations provide higher accuracy than RANS simulation (described in detail in the following paragraphs). The underlying idea is to reduce computational efforts of a DNS by resolving only bigger and more powerful eddies. Computational costs are anyway times higher than RANS, so that they will not be used in this Thesis. They will only be described quickly.

As the name suggests, this approach resolves larger eddies on an appropriate mesh, without any assumption, but models they smaller ones (to keep computational costs reasonable). The choice between what is going to be simulated (or resolved) and what is going to be modelled is done through a low-pass spatial filtering of the Navier-Stokes equations, which effectively removes small vortices effects from solution. These little vortices, however, play an important role in the flow field, and must be accounted for. The are called sub-grid stresses (sub-grid because actual eddy size is smaller than the finest cell). Models to add these effects to the actual flow is the central problem in LES approach, and it is still matter of research.

DES

Detached eddies simulations use a hybrid approach for unsteady flows that stands between RANS and LES. The aim is to reduce computational costs of a LES approach. The idea is to actually simulate turbulence when the grid is fine enough (LES approach) but then, when either the grid is coarser or the turbulent length scale get smaller, a RANS approach jumps in instead of using sub-grid models. The result is that one gets the best of both worlds: RANS simulation in boundary layer and LES in separated regions.

However, DES set up is quite tricky, as it is highly sensible to mesh parameters. Drawing a grid that properly sustain this approach is really difficult and something that is more an art than a science, thus much experience is required.

RANS

RANS stands for Reynolds averaged Navier-Stokes. In his work, Reynolds took fluid motion unknowns and expressed their instant values as a mean contribute plus a fluctuation: for example, for the pressure p, he wrote $p_{(x,y,z,t)} = \overline{p}_{(x,y,z)} + p'_{(x,y,z,t)}$. Then, after substitution in Navier-Stokes equations, he took the time average of the results. After some manipulation, equations are now far simpler thanks to the "dampening action" of the average, but still accounting for the turbulence in fluctuation terms.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \qquad\qquad \frac{\partial u'_i}{\partial x_i} = 0 \tag{2.27}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u_j}\frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu\frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j} - \frac{\partial}{\partial x_j}(\overline{u'_i u'_j}) \tag{2.28}$$

These are the Navier-Stokes equations after Reynolds manipulation. Turbulence effects are considered in the last term $\overline{u'_i u'_j}$. This term closely approaches a stress quantity, thus the term $\rho\overline{u'_i u'_j}$ is called Reynolds' stress tensor R_{ij}. Formally, $\frac{\partial}{\partial x_j}(\overline{u'_i u'_j})$ represent a force (though fictitious), the average momentum flux due to turbulent velocity fluctuations. In this way, turbulence still interacts with the mean flow, even if it is not resolved. This is the strength of the model.

It may be misleading thinking that, having taken the time average, RANS simulations cannot handle unsteady problems. In fact, the averaging period has to be large enough to damp turbulent fluctuations, but not as large as macroscopic fluid motion changes. Therefore, the mean contribute to the instant value may still be long-period time-dependent, and it will not "disappear" after short-period Reynolds averaging. RANS equations for unsteady problems are commonly used and they are referred to as URANS, and they resolve only the varying mean flow.

Anyway, Reynolds' set of equations, shows the so-called closure problem: how do we calculate all the $u'_i u'_j$, considering only mean flow quantities? The answer is: by using turbulence models. In RANS, some widely used models are:

- Spalart-Allmaras model;
- $k - \epsilon$ model;
- $k - \omega$ model;
- Reynolds stress model.

The first three models use the Boussinesq hypothesis, which introduces a new quantity: the turbulent viscosity. This quantity enhances natural molecular viscosity, and this is how turbulent effect are considered. Turbulence is not resolved, but its effect are calculated by solving transport

equation for turbulent related quantities. In contrast, the last one directly solve transport equations for each Reynolds stress (with appropriate models), and it represents the most accurate turbulence model for RANS simulations (models based on eddy viscosity assumption hold for mild flows, while their behaviour in complex flows is unsatisfactory).

In the following we briefly report mathematical description for each model, along with pros and cons.

- Boussinesq's eddy viscosity: it provide a model to estimate Reynolds stress tensor components using only mean flow values. The mathematical description is:

$$-\overline{u_i' u_j'} = \nu_t\Big(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3}\frac{\partial \bar{u}_k}{\partial x_k}\delta_{ij}\Big) - \frac{2}{3}k\delta_{ij} \tag{2.29}$$

  where ν_t is the turbulence eddy viscosity and k the turbulent kinetic energy. We do not have none of them. Following models exist to provide, in different ways, k and μ_t values, though exploiting other constants. However, those constants will hold for several different flows, so it is for sure worth the effort.

- Spalart-Allmaras model: in a one equation model, it solves a single transport equation for eddy turbulent viscosity. The model was originally developed for aeronautic-only purposes, thus wall bounded flows with mild pressure gradients, though adverse, but no separated flow. It describes well the boundary layer at all y^+ distances (provided the mesh is properly tuned). Unfortunately, its application outside attached external flow lacks of accuracy.

- $k-\epsilon$ model: is a two equations model, meaning that two turbulent quantities are considered and their respective transport equations solved. These quantities are turbulent kinetic energy k and rate of dissipation of turbulence energy ϵ. The fundamental assumption on which it is based, beside Boussinesq hypothesis, is that this eddy viscosity is isotropic. During the model elaboration, authors used best known processes, focusing on most relevant ones. This minimized unknown constants, but they are still present in number of 5. On the other hand, these constants provide a fine mean for model tuning. Transport equations to be solved therefore are:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\Big[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\Big] + 2\mu_t E_{ij}E_{ij} - \rho\epsilon \tag{2.30}$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho \epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\Big[\frac{\mu_t}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial x_j}\Big] + C_{1\epsilon}\frac{\epsilon}{k}2\mu_t E_{ij}E_{ij} - C_{2\epsilon}\rho\frac{\epsilon^2}{k} \tag{2.31}$$

  where E_{ij} represents rate of deformation, μ_t the eddy viscosity. Constants have been obtained by data fitting of several turbulent flows, giving the following: $C_\mu = 0.09$, $\sigma_k = 1.00$, $\sigma_\epsilon = 1.30$, $C_{1\epsilon} = 1.44$, $C_{2\epsilon} = 1.92$. In words: Rate of change of k or ϵ + Transport of k or ϵ by convection = Transport of k or ϵ by diffusion + Rate of production of k or ϵ - Rate of destruction of k or ϵ.

  It is the most widespread turbulence model because of its adaptative quality for many different turbulent flows. Computational costs are low compared to the more complete Reynolds stresses. However, it works well for mild pressure gradient, though adverse, and recirculating flows, but if fails when strong pressure gradients jump in.

- $k-\omega$: is a two equations model and it closely related to the $k-\epsilon$ one. In fact, ω is the dissipation rate per unit energy, thus $\omega = \frac{\epsilon}{k}$. Assumptions and derivation are the same, so that equations are quite similar:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \rho\tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta^*\rho\omega k + \frac{\partial}{\partial x_j}\Big[\Big(\mu + \sigma_k\frac{\rho k}{\omega}\Big)\frac{\partial k}{\partial x_j}\Big] \tag{2.32}$$

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho\omega u_i)}{\partial x_i} = \frac{\gamma\omega}{k}\tau_{ij}\frac{\partial u_i}{\partial x_j} - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\Big[\Big(\mu + \sigma_\omega\frac{\rho k}{\omega}\Big)\frac{\partial\omega}{\partial x_j}\Big] + \frac{\rho\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \quad (2.33)$$

with constants values obtainable in [9].

It improves the $k-\epsilon$ model as it is able to manage stronger pressure gradients, but finds its weakness in huge sensitivity in ω value in the free stream. Boundary layer calculations depend too much on inlet ω value, making the model unsuitable for internal flows.

- Reynolds stress model: it solves several exact transport equations, but makes no use of eddy viscosity - and this is its great strength. Reynolds stresses are directly evaluated using the knowledge about interactions in turbulent flows. Without eddy viscosity, accuracy is greater than any other turbulence models for RANS, but at a heavier cost. This cost is anyway less than a LES, and much less than DNS.

Many variations do exist, specially for two equations models. Each of them applies better to a problem rather than another, so that a little time should be spent in analysing the situation to face. Literature is continuously developing and the closure problem should not be considered closed yet.

2.4.3 Need for Discretization

Since no analytical approach is permitted (so far), a numerical one must be performed. That means to use a calculator, which deals with bits (binary entities), thus discrete quantities. A computer can manage only discrete information, 0 or 1, with no intermediate states. Therefore, every entity that is seamless in the real world has to be discretized when fed to a calculator. This holds for physical quantities like space, time, but also for equations and algebra. Calculus cannot exist for a binary calculator.

Luckily, scientists from the past did manage to transform calculus operator (derivatives, integrals), which use infinitesimal quantities, into schemes and rules using finite quantities to get the same result, though approximated. This discretization error is unavoidable, as each signal born continuous and read discrete loses information.

Functions are therefore not evaluated continuously but only in certain points disposed in a spatial lattice. We can enumerate components that make up a numerical method:

- Mathematical model: set of partial differential equations driving the physics;
- Discretization method: how to discretize those *PDE*, i.e. finite differences, finite volumes, finite elements (described in detail in the following section);
- Coordinate system;
- Numerical grid (i.e. the points lattice, also called mesh);
- Solution method: how to solve large algebraic set of equations coming from PDE discretization. Iterative methods are a must.

Choices for each of the items above will determine if the following requirements of the numerical method are satisfied or not:

- Consistency: the discretization method should return a more and more exact solution as the grid spacing become closer and closer to zero;
- Stability: unavoidable discretization errors and truncation errors must not amplify during iterations;
- Convergence: solution of discretized equations must tend to exact equations solution as the grid spacing tends to zero;

- Conservation: conservation laws must be verified in their discretized form too;
- Boundedness: numerical solution may oscillate, but within proper bounds;
- Realizability: complex phenomena that must be modelled to be included in simulation must base on models that guarantee realistic solutions.

The whole accuracy, which is affected by modelling errors, discretization errors, truncation errors and iteration errors, must be kept as high as possible.

2.4.4 Discretizing equations

Equations, regardless of the subject, can be discretized by using one of these method: Finite elements method (FEM), Finite volume method (FVM), Finite difference method. Their purpose is to convert a set of partial differential equations into a linear set of much simpler algebraic equations. This set is then solved with one well known sparse matrix solver. Each method is based on a discretization of the space into a grid of points (called nodes) or cells (enclosed by nodes), and the evaluation of some kind of functions at nodes or cell centres. The nature of these functions determines the difference between the three methods, that are going to be briefly described.

Finite Differences Method

This method starts from conservation equations in their differential form. Really straightforwardly, each derivative is substituted by a finite difference, i.e. $\epsilon \to 0$ from the difference quotient is no more infinitesimal but finite, though little. How much little depends on nodes spacing. In this way the solver has only to evaluate conserved quantities at nodes and then take their difference between adjacent node pairs. The obtained set of algebraic equations, along with boundary conditions, results in conserved quantities values in each node.

Advantages of this method are the really simple programming and accuracy tuning capability: derivatives may be expressed as a first order Taylor expansion (like the one described previously), but also higher order - though this increases dramatically costs. However, the method is suitable only for simple geometries. Accounting for sharp edges, thin skins etc. is difficult with finite differences and modifications in equations are required.

Finite Volumes Method

The spatial grid entities here considered are not nodes but cells. At each cell centre a value for conserved quantities is set (initial condition), and its trend into the cell is given by a piecewise linear polynomial. The set of linear equations is obtained by balancing conserved quantities fluxes across cell boundaries. Therefore. equations must be in integral form. Fluxes must then be evaluated for each boundary. When mesh is structured, or globally regular, problems will not arise. In an irregularly shaped mesh (like a tetrahedral 3D mesh) faces are not so wisely oriented and flux calculations may result in non effective values, causing slow convergence (if not divergence).

Finite volumes method is commonly used in CFD software, because of its natural formulation in conservation laws. With some shrewdness in meshing it allows for complex geometry modelling. Since only fluxes are evaluated, this method is more robust than others, but it is very difficult to increase accuracy with higher order piecewise polynomials.

Finite Elements Method

In the finite element method, Galerkin's method of weighted residuals is generally used. In this method, the governing partial differential equations are integrated over an element or volume

after having been multiplied by a weight function. The dependent variables are represented on the element by a shape function, which has the same form as the weight function. The shape function may take any of several forms. Algebraic equations set comes directly from these manipulations.

It is commonly used in structural analysis softwares, but it is a very general method. Potentially, each geometry and physics (complex or not) can be modelled with the same effort. Furthermore, like finite differences, it is easy to increase accuracy thanks to higher order elements. There is no physics involved in deriving equations, but this has the shortcoming of difficult mathematics to handle, which is its greater disadvantage. Programming a FEM software is more difficult than any other one.

2.4.5 Discretizing space

The remaining task to develop a working simulation software is to create the computation grid. This process is called meshing (mesh is another name for grid). The algorithm performing the task is industrial property of each software house, so mesh parameters and terminology vary from one software to another. However, the workflow remains more or less the same in all situations, and is described below.

First of all, the geometry must be created. This is done with a separate CAD/CAE software by the enterprise's design office. Then the CAD model is sent to simulation office where detailed geometry is cleaned and set suitable for good quality meshing using other *ad hoc* pre-processor softwares. Now the geometry is ready to be meshed by the simulation software mesher.

Cleaned CAD geometry, usually 3D, must be discretized into a number of finite-dimension cells. We start from describing surfaces with triangles for CFD applications or quadrangles for FEM applications. The resulting surface mesh acts as the basis for the following volume mesh (being a 3D problem, all the volume must be discretized with 3D elements). The mesher is coded so that it automatically shrinks triangle dimensions where geometries are sharper or wherever more elements are required to reach better convergence. The user should then check if this automation worked properly and if node spacing is regular enough. When a good quality surface mesh is reached, the mesher starts inserting nodes inside the fluid volume, thus creating 3D cells: it is creating the volume mesh. Cells are created with the user specified shape, which may be tetrahedral, exahedral, or polyhedral. Each one has its own pros and cons. The software stores node positions and connectivity matrix, assigns initial conditions to cell centres and starts iterative calculations.

Chapter 3

Problem Presentation

New global directives for automotive industry are pushing further and further towards increased design efficiency and lower CO2 emissions, while overall performance are getting better. Automotive technology has been evolving since the nineteenth century, making it very difficult to have a new major breakthrough in design concepts. As research carries on, the main path followed in order to reduce CO2 emissions is weight reduction. In the past the powertrain was essentially made of steel, but now many drivetrain components can be cast from Aluminium. Smarter designs are required in order to have the same structural reliability at a lower weight.

In a harmonic design, all components must undergo the same lightening process. Efforts in that way are as big as the performance required, making racecars the cutting edge of technology. However, even cheaper everyday vehicles have got many engineers behind it who struggle to keep weights low, even if price and reliability constraints set rigid limits in minimum weight reachable.

However, clutch design, apart from extreme performance, does not involve many efforts, usually, in weight reduction. This occurs because heat generated during its operation has to be disposed somehow, and the bulk metal which it is made of really accomplish this task, acting as heat sink. It comes that if metal is cooled in another way, then the bulk may be reduced (and weights with it).

In this chapter it will be shown why clutch design should be improved, how and when it may be achieved. In Section 3.1 it is carefully described why an efficient design shall start from the very beginning, and why current clutch design strategy is improvable; in Section 3.2 is also explained why a poor design is dangerous, and how FCA manage to stay at a proper distance from such risks; then it is described how we may improve clutch design. In the last section, it is shown the global necessity to improve designs, regardless of the component involved.

3.1 A Better Design

Our goal is to improve clutch design in order to reduce CO2 emissions, first of all, but also to ensure greater component reliability, safety, and reduced costs (both operative either purchasing ones). Better design means, in most cases, smarter material distribution to have lighter weight with the same structural strength, wiser material selection and assignment, proper element sizing, and so on. The factors required to perform these choices right are essentially represented by the most complete knowledge of phenomena acting on the system.

In this section, we will describe when these choices have to be made, and why, with eyes wide open to product success, economically speaking. In the following, a description of what we mean for "complete knowledge" is provided (Section 3.2).

3.1.1 New complexity

Technology is improving as researchers publish papers and new devices are invented. In few years, these new technology can be implemented in everyday objects at reduced costs and high reliability. Theoretically, this should lead to better user experience, or higher safety, or improved efficiency, or whatever better aspect one may find. However, on the other hand, these improvements require harder efforts by designers who have to handle that new technology, which often introduces more complexity into the system.

The higher complexity engineers have to face may also be added by external regulations that are trying to make products better because the older ones were too polluting, too energy consuming, or too unsafe. The new product may be, for example, more expensive, less user-friendly, even less performing, but much more healthier. Think about asbestos covering used in the past: asbestos is a phenomenal insulating material, is light, cheap, easy to produce and manage. Until you do not consider long-term effects on health. In fact, after years of investigations and research, they found out that it is the unique cause of a particular form of lungs cancer. Obviously, asbestos use cannot be tolerated any more. As a consequence, buildings that have it in their walls have to be remediated (that costs), new insulation devices, likely more complex, have to be invented (adding other costs) and newer houses must use those devices (more and more costs). But people is happy to spend increased money because they know they are safer now.

Such complexity, in any case, involves a re-thinking of older design processes. In most cases, if a final product is more developed and uses newer technology, its design has been more difficult. In fact, older technology is better understood and more documented, experience gained during years is a lot, leading to more effective design and less possibility of error. When this experience and documentation lack, brains must start running again. At least to keep final costs comparable (or less) than previously.

3.1.2 Resources allocation

An industry product, whatever it is the sector concerned, it is not limited to the object itself. Its success does not depend only on goodness of design, but also on the research behind it and customers assistance. In addition, production too should be accounted for, having to be as fast and easy as possible (easy and fast are both synonyms of cost-effective). An effective design should find the best trade-off among these crucial aspects, which form the global product system. We describe this system summarily as: design - production - logistics. A sketch of costs allocation during the product lifecycle (along with a description of such lifecycle) is provided in Figure 3.1. The most straightforward strategy to set up an excellent system, from design to aftersale product assistance, would see a first guess design, immediately sent to production, it follows a construction of an assistance network based on such design, and then iterate after careful review of system analysis.

Times and costs required by such architecture are by far impossible to sustain. Setting up a network assistance or assembly line require a lot of financial investments and physical time for building it. Deep changes there mean pure loss of previously spent money and time. Design strategies used by most industries try to reach from the earliest stages the best compromise between an efficient product, an easy production, great maintainability, and effective network assistance. This is what is called an integrate design - and what we are aiming for.

The reason to spend lots of efforts in design early phases is that it is here that most of the future money spent is allocated. Small variations at this scale do involve in deep changes later. However, at the same time, the money and effort spent in these first stages is negligible compared to the total budget amount. Later, when production starts, a huge amount of money flows out and affordable adjustments would not result in major decreases. The concept is well explained by Figure 3.2.

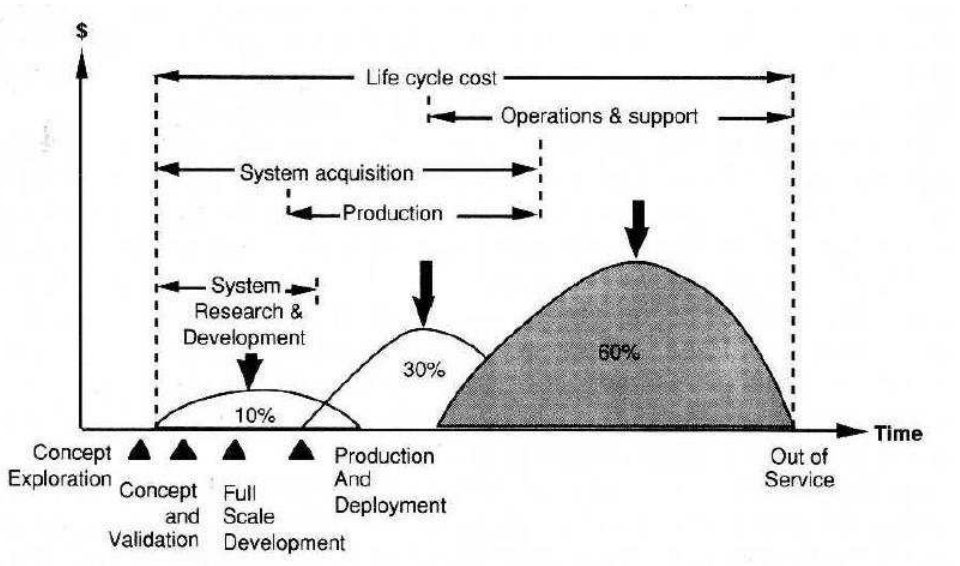

Figure 3.1: The birth of the product coincides with early conceptual choices following the identification of a precise requirement. Then it goes through actual design and production, but its life comprises also its operation and support. Costs are allocated over each one of these phases, for sure they do not end with production. Indeed, operations and support represent the greater slice! An excellent design should aim for easy production, distribution and maintenance, in order to decrease costs of major weight.

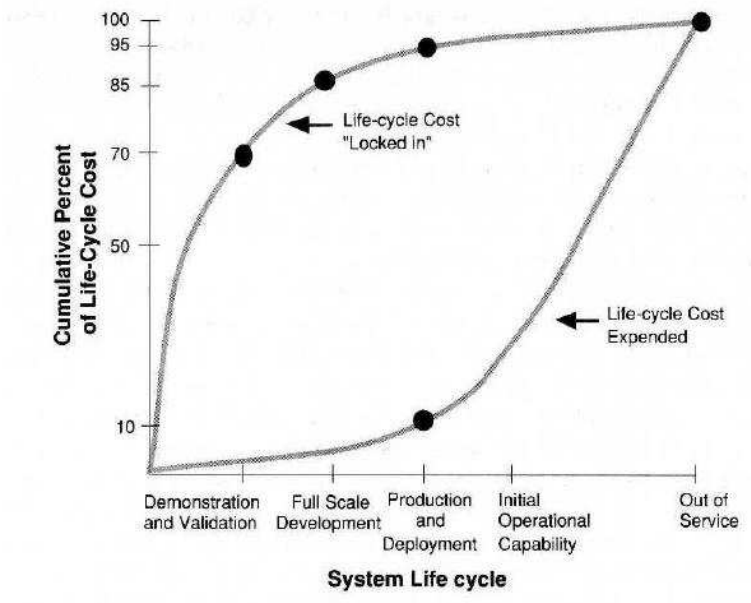

Figure 3.2: Life cycle cost allocation. Money spent during production and operations are the result of choices made during design early phases. It is there that most of the money is allocated - with more or less awareness - as it is a direct consequence of design architecture.

3.1.3 Improving Design Early Stages

needs to predict behaviour in design early phases logistics must be considered with the same importance given to performance nowadays aftersales costs are equally important to purchase cost in product success lifecycle cost must be as low as possible

With this premise, it follows that most intellectual efforts should be made during conceptual design choices. This will avoid, during last stages, late adjustments which will impact heavily on system effectiveness. The system is not "stiff" at the design very beginning. Variations made there do not involve delays nor money loss, and they do not compromise anything in product final success, as it is still completely evolving. However, the same variations, when should design be "frozen", will determine the success or the failing of the product.

The impact a choice may have on final design must be foreseen in some way. Its influence on other aspects, along with side effects, have to be handled with great care. The actual analysis should not stop at the mere component performance. The product is more than that. Production and logistics should be considered with equal importance as performance.

It is evident that the number of factors to be considered is huge. A human being cannot handle all of them without the aid of a computing machine. Nowadays, the most efficient design methodology in this preliminary phase is trying a strong engineering solution, analyse effects with software algorithms, understand strengths and flaws, and iterate with other as much strong changes. When satisfying convergence is reached (i.e. further big variations only worsen the results), a detailed design may commence.

3.1.4 Current Procedure

iterative design would require too much resources (time and money), therefore integrated design is required. planning - analysis - detailed design - prototyping, verifications and tests - mass production - distribution - aftersale support (rather than analysis after prototyping)

In FCA, clutches are bought from a third party supplier. That means that clutches provided which do not fit perfectly in a new vehicle may not be so uncommon. A tailored design should bring more satisfactory results.

In any case, once the clutch is bought and bolted on vehicle, tests have to be made to ensure proper working (see Section 3.3). An effective design process would follow these steps:

1. planning;
2. first design attempt;
3. design analysis;
4. design adjustments;
5. iterate 2 to 4;
6. detailed design;
7. prototyping and validations.

Once the first prototypes meet requirements, mass production shall start along with distribution and aftersale support. Clutch design procedure, currently, switches step 7 with 2, because it is not actually designed by FCA, making it impossible to iterate and provide an efficient product.

If a problem is found during validation, it is very likely that it will remain unsolved until a downgrade, somewhere and somehow, is accepted. Such downgrade should be kept as discrete as possible. Its repercussions on the system must remain minimum. As long as the clutch is concerned, when an overheating issue arises the assembly to modify in order to bring the least damage has been identified as the gearbox ratio. Adding or removing a tooth in a gear actually modifies engine torque, giving some relief to the clutch.

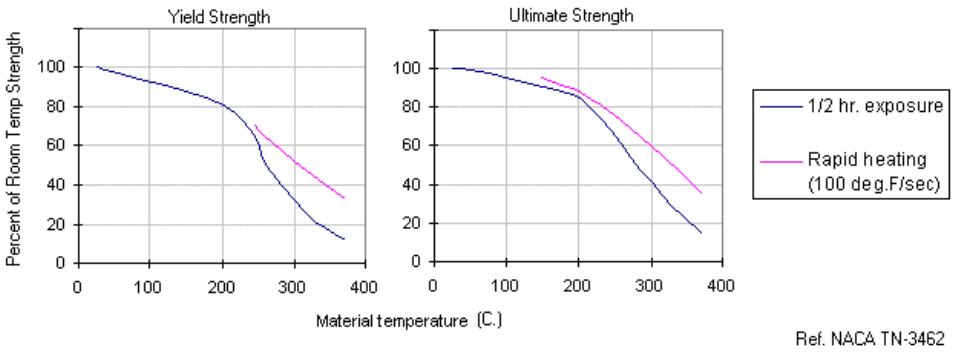

Figure 3.3: In this diagram, yield and ultimate strength vs temperature are reported. In fuchsia are plotted trend for a rapid heating, while in blue the temperature is uniform inside the specimen.

However, powetrain engineers spent a lot of time in trying all of those ratios possibilities to find the best one. Gearbox ratios are fundamental to keep engine revolutions in their optimal range, so that efficiency is maximum and CO2 emissions are minimum. If they are modified later, it is unaffordable to iterate the procedure until the same level of efficiency is reached. CO2 emissions increase is accepted, we will do better next time.

What if FCA will not buy clutches from external suppliers any longer and it will design its own? With the work of this Thesis, new tools are becoming available and this goal is getting closer.

design philosophy: safe life, fail safe, fault tolerant, functional redundancy, fault avoidance. fault avoidance - design with proper margins, high quality components and materials, design and production monitoring, tests.

3.2 The Other Side: Clutch Overheating

Concepts described in previous section are accomplished as more successfully as more complete is the knowledge about phenomena acting on the component (i.e. the clutch). We refer to "phenomena" both as component physical working principles either its mission profile. While we already talked about working principles in Section 2.2.1, where we discovered that heating is the clutch major enemy, there we only gave a brief description of what its mission profile is. In this section we will infiltrate deeper in the problem. However, only a qualitative description will be provided and it will focus on most dangerous situations (traffic jams, hill starts, etc.), as the remaining mission (most of it really) is by far negligible for our purposes.

During a hill start or a traffic jam, clutch spends a lot of time slipping and generating heat. Temperatures may become dangerously high in these phases, even up to 400°. If a proper heat management is not provided, it may cause large damage to a clutch.

Materials properties are usually temperature-dependent, and they might vary a lot even in ranges far from melting point (where of course properties sustain a deep change). For example, a usual Aluminium alloy (Dural) has a yield strength that is halved already at 250°, while its melting point is well above 500°. Figure 3.3 displays σ_y and σ_r trends with temperature.

Materials should be assigned not only with strength and weight criteria, but also considering which temperature they are going to face, even if this temperature is below the melting point. This is the reason why jet engines are made of super-alloys: Nickel or Cobalt are mixed with several other chemical species in order to have good mechanical strength at extremely high temperature, and outstanding resistance to thermal creep. Titanium would have better elasticity modulus and higher specific yield strength, but this happens at room temperature. Above 1000°, as in a jet engine, steel behaves as "rubber", and Titanium even worse (as it can be seen in

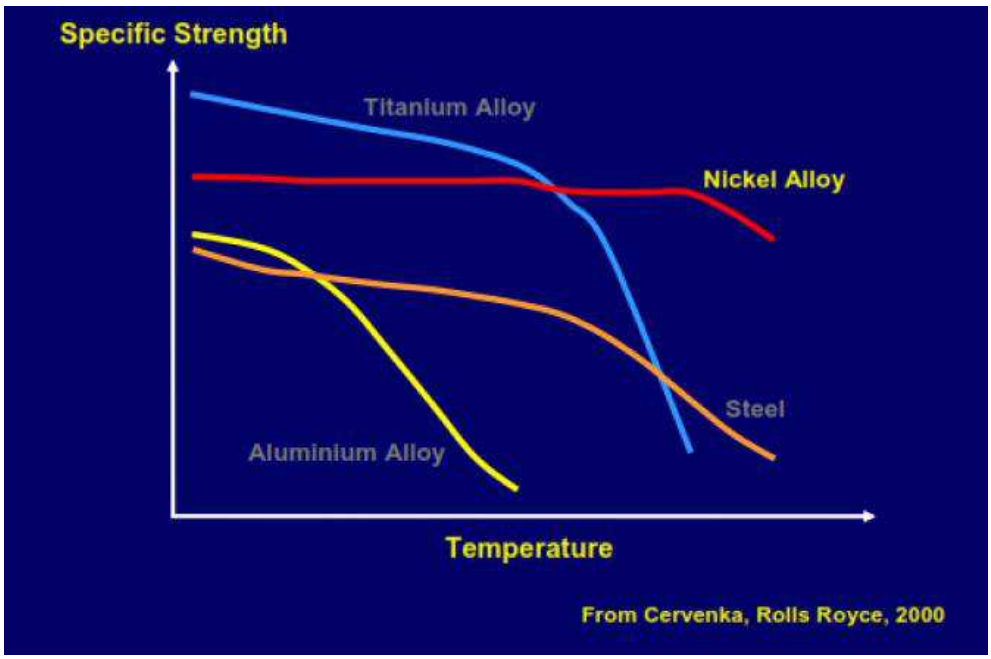

Figure 3.4: Comparison among specific strength (i.e. $\frac{\sigma_y}{\rho}$) of different materials against temperature.

Figure 3.4). Many factors are responsible of such properties decaying with temperature, like crystal lattice disposition, precipitate percentage, alloying elements solubility in matrix and so on. Furthermore, also friction coefficient is temperature-dependent, a really important factor in clutch performance.

If a clutch temperature raises above a safe limit, material starts degrading, and wear increases. This occurs particularly in outer regions where tangential speed (and friction work) is higher. Clutch disc lining gets thinner there, and pressure plate action is no more uniform on the disc: it will only touch the inner parts of the disc as they are not been consumed yet. This results in less total normal force and less torque transmitted. In addition, wear may fall on the disc with no escape from there, likely worsening friction coefficient. Such coefficient must be as high as possible, in order to transmit more torque with the same pressing force (related to driver's effort). If clutch disc surface is ruined, or physically altered by high temperature, friction coefficient may drop below a value for which diaphragm spring action is no more sufficient in providing the required pressing force. Recollecting Equation 2.9, expressing F_N as a generic spring coefficient k with its displacement d we quickly obtain $F_N = k \cdot d$. Then, the torque expression becomes

$$T = \frac{2}{3}\mu\frac{r_2^3 - r_1^3}{r_2^2 - r_1^2}k \cdot d. \tag{3.1}$$

If μ drops too low, the displacement d provided on design may not be enough to compensate for friction loss. Then, normal force does not generates enough friction force and T decreases. As a consequence, engine torque exceeds friction torque, and clutch starts slipping. The driveline has broken down, and the vehicle is hardly drivable.

The clutch is made up of three main components: the flywheel, the pressure plate and the disc (other parts are the springs and mechanical linkage between them). The formers are made of cast iron since there high wear resistance is needed, while the latter is a composite material of bitumen matrix with copper particles to improve thermal conductivity. The reason of bitumen to exist finds its answer in its high friction coefficient. However, it is not as hard as cast iron, thus it is the first to wear out. Clutches are designed to sustain wear for all the vehicle life cycle, and when it fails there are not fail-safe or safe-life features to protect the car. The disc has to

Figure 3.5: In the photography are shown a pressure plate and its disc after failure. Discoloration areas can be seen on pressure plate, while lining particles blacken clutch disc.

be substituted or proper running conditions will not exist. A high quality product must avoid overheating in each possible situation the vehicle may come to face.

A failure analysis is provided in [7]. In this accident, a clutch broke down after only $40.000km$ of service, very far from expected life, and required the entire assembly replacement. The pressure plate was damaged too (see "discoloured" areas in Figure 3.5). Investigations found out that some weeks earlier, about $2.000km$ before, and incident took place as the clutch was overloaded for a too long time. Smoke was noticed coming out of the under hood, so it was very likely that it was sustaining overheating. As a consequence, a failure occurred very soon even if, after that incident, nothing unusual was noticed.

In conclusion, care has to be paid during heat management design, and in later phases it may be a good practice to experimentally test the assembly in real conditions.

3.3 Experimental test

After several computer-aided designs and analysis, one cannot help for testing its products in real conditions in order to be sure that they will work properly and as expected. Usually, real testing is a must for those feature that are vital to the car operations or concerns people safety in some way. In addition, real testing is required when numerical analysis are too inaccurate to grant precise and reliable data.

In this case, specially in the past, virtual clutch simulation were really difficult to implement because of computational costs and efforts in setting up the simulation (i.e. preprocessing). The clutch is also a component that, for large productions, does not involve high technology and its developing is essentially based on "what it was done before". In other words, if a clutch assembly worked on previous model, the same clutch will equip the new incoming prototype. Later, just to make sure that this really works well on this new model too, a track test is made. A procedure like that simplifies early design phases, since an assembly is ready, but what if it

does not fit properly or does not meet requirements? In this chapter it will be described how a clutch may be tested and usual corrective actions to undertake.

3.3.1 Dangerous situations

Usually, during ordinary march, a driver would not perform many starts and stops. Even during driving through the streets of a city the clutch has the time to breathe between a traffic light and the next one. Czel et alii in [2] found that, in experimental tests, temperature increase is not dangerous if dis- and engagement occurs a few time in a short period. In their experiment they were able to measure directly disc temperature (a fact that it was not possible during FCA testing as pointed out in Section 3.3.2), which increased of 100° and over, but it quickly dropped.

Dangerous situations exist when engagement occurs many times one after another. In such a situation, the bulk of metal still cools the disc acting as heat sink, but it gets hotter and hotter with no easy way to cool down (by comparison, the flywheel is thicker and its cooling is much slower). As a result, being clutch temperature higher and higher, heat exchange with clutch disc is more and more difficult. Average temperature dramatically approaches critical values, until during one more engagement the disc overheats since no more heat may be disposed of: smoke is produced and surface deteriorates. The next engagement may not work as planned.

Examples of repeated starts and stops are provided during whatever traffic jam. Here, the situation is even more critical as car speed is usually low (thus no air flow cooling clutch case) and the little air still present is overheated by surrounding cars engines. Traffic jams are not so unusual, specially in highways. A customer would not be happy if she/he is aware that the car would break down if the traffic jam lasts more than half an hour. Vehicles must withstand situations much worse than this one.

Another really tricky situation is represented by a hill start. There, only one engagement may kill the clutch. The gravity force the engine has to face may be significant (it adds a $mg \cdot sin(\alpha)$ term to the total force to overcome), so its power output has to be remarkable. The driver has to raise rpm to make the car climb. The manoeuvre to succeed has to be performed with an extended use of clutch pedal. Engine power is directly proportional to engine revolutions (recollect Equation 2.1):

$$W_{eng} \propto \eta_u \eta_{vol} \cdot n_{eng} \tag{3.2}$$

The driver has to deliver this power as much as possible to the ground, but it cannot be 100% since wheels are at $0rpm$ while the engine is at, say, $4000rpm$. Different angular speeds must be agreed by clutch slipping where friction wastes part of engine power output, as it is during each start, with the uncomfortable fact that now speeds are very different. The time required by the clutch to adjust speeds is consequently longer. The result is that disc temperature raises not only of 100°, as would be in a flat start, but it may be up to 300°, burning the disc. Heat conduction is not as fast, so that the disc has to rely only on its own heat capacitance (which is not so high). On the other hand, the driver should not release the clutch too quickly or the engine will stall.

Clutches must be therefore tested to be sure that they react positively to stressful situations like those just described.

3.3.2 The test

While there is not much a producer could do to protect the clutch against hill starts badly performed, as it is up to driver's ability making a good one, it can anyway provide the clutch with proper cooling and heat management to sustain traffic jams without damage. The test, therefore, is a traffic jam simulation. (For sake of completeness, we report that a hill start is made easier by the producer by equipping the vehicle with an assisted driving software: the hill sustain).

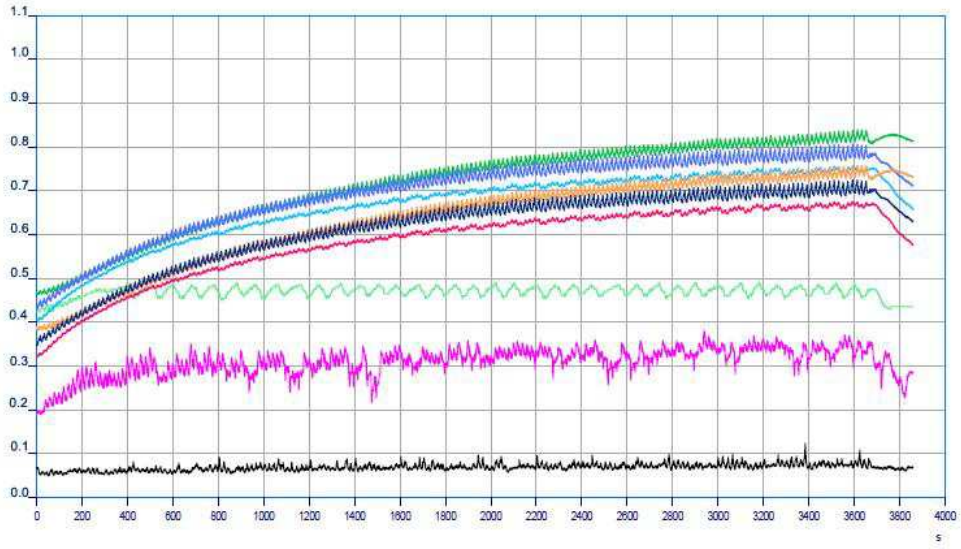

Figure 3.6: This plot shows the trend of several thermocouples placed close to the clutch, along with other useful positions. Temperature scale is normalized. Start and stop repetitions are clearly visible in sawtooth shape.

There will not be many cars during the test: one is enough. But, really like it was stuck in a clogged highway, the test driver shall perform several time a start from standing still, accelerate to moderate speed, and then come to a complete stop. Thermocouples wisely placed monitor gearbox and clutch temperatures, while other sensors keep track of usual quantities like speed, rpm, torque and so on. They are necessary in post processing, to compute derived quantities, and to provide a comparison term with virtual simulations.

The test is carried on for about one hour, just to make sure that the clutch is undergoing a really heavy duty. Internal procedures and regulations are adopted to state if the test is passed or not. We can anyway report that, if the clutch starts slipping with pedal fully released or smoking, the test is failed.

In Figure 3.6 are reported temperature values against time during the test. These values tend to an upper limit, though not reached during the test, but what is clear is that they are bounded. This is a fundamental result for the research of our methodology, as it allow us to set steady simulations in order to evaluate virtually temperatures at the end of the test.

The test, as stated above, is also useful in determining the clutch thermal power output, which is a boundary condition (a source term) in the virtual simulation. Sensors detecting engine torque and rpm, gearbox rpm, wheel speed, vehicle acceleration and other are exploited to achieve a concrete value. Details are provide in Chapter 5.

3.4 Why CFD

As stated in Section 3.1.3, it is very hard to make conceptual decisions at the very beginning of the design process, since the number of factors to take into account is enormous (some of which are final product performance, costs, ease of production, logistics...), and they are all interrelated. Some of these relations are not so easily seen by the engineer. Ways to facilitate the job are provided by experience, models, benchmarking, statistical analysis, testing. Each of these ways has its own flaws and strength:

- Experience is fundamental because it states if previously done assumptions were correct, thus they can be still used, or they were not (so they should be changed). It shows

unexpected phenomena or failure modes. However, it fails when new technologies and processes are implemented, right because they are *new*, while experience is past-based;

- Models based on experience and theoretical considerations are able to foresee system behaviour, becoming extremely useful during early stages. Such models, because of several factors involved, are implemented on powerful calculators. Their primary flaw is that no one is sure if the current model is describing properly real unseen phenomena and their interrelations, and (as a consequence) a trade-off between accuracy and time-effective analysis is to be found;

- Testing is important both because phenomena are really occurring and reality is *never* wrong, either because it provides data to be compared with model results (which can be used to improve those models). However, to test a component, it must previously be built, spending time and money, and further adjustments testing require another production (with other costs and delays);

- Benchmarking and statistical data relates our internal product with the one from our competitor, declaring who is winning the race. Obviously, no one is happy when being spied, so that everyone carefully protects its own data. Benchmarking and statistical analysis must be done to keep the pace with others, but it may not be so easy.

Our work has to improve design early stages, when components production would be too expensive and quite useless (apart from precise specimens), therefore we should exclude direct testing. Benchmark actually is quite difficult because underhood heat management is strongly influenced by engine, gearbox and packaging. Exploiting a solution from a foreign company may not work at all in FCA vehicles, and viceversa. The only feasible way is to set up a new model, actually not present in FCA procedures as long as clutch is concerned, to simulate its thermal behaviour. Knowledge behind such model is provided by experience and theoretical analysis, described respectively in Section 2.2.1 and Section 3.2. In Chapter 4 a state of the art review tells that models different than CFD numerical analysis lead to inaccurate results. Clutch thermal behaviour and its interaction with other components are complex and there are not many assumption we can do to simplify it. Only a powerful calculator is able to model it with sufficient precision, both in terms of physics either geometry discretization.

In addition, testing is anyway quite important to have comparison terms with numerical results. The model is therefore developed using an existing prototype, both for clutch and vehicle which it is mounted on. Its accuracy is determined with a comparison between its results and those provided by testing the existing component. When satisfying agreement is reached, the model is ready to be used in real design for prediction purposes.

The subject of this Thesis is focused on setting up the model, then only comparison with existing vehicles are performed. A data collection campaign to provide correlation between numerical and test data will be object of further research.

3.5 Air Pollution

To conclude the Chapter, we would like to spread some light into the issue that first claims for general efficiency increase - air pollution. We previously talked about dangers of clutch overheating, its reliability, its cost and so on, but the main reason to make it better is to reduce the emission of polluting elements into the air.

Too often is heard that road transportation is not the cause of greenhouse effect, sometimes that the greenhouse effect does not exist at all. Unfortunately, it does exist (see Figure 3.7 for NASA temperature data) and CO2 is part of the problem. The latter fact is simply demonstrable with a lab test: CO2 is transparent to visible light (sun light crosses the atmosphere) but not

to infrared waves (radiation emitted from earth is blocked by CO2). Thus, the higher the CO2, the more heat retained.

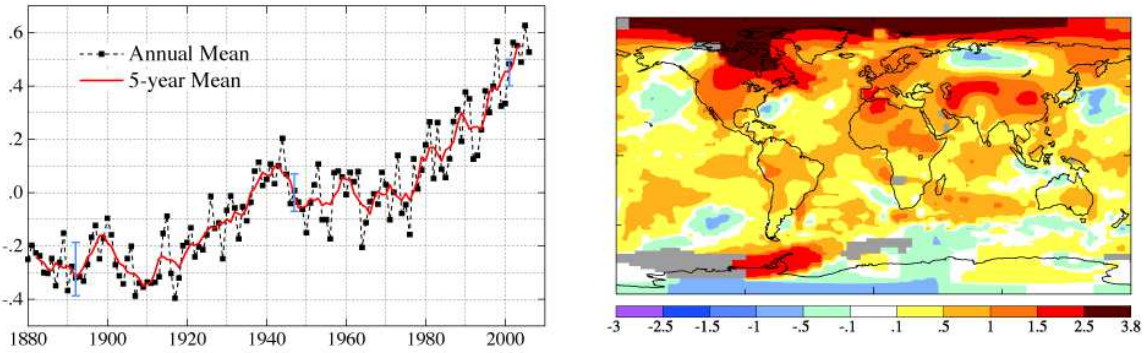

Figure 3.7: On the left, a plot of past 130 years mean temperatures is reported, along with a 5-years period average. On the right, surface mean temperature increase in 2006 is reported compared to a reference temperature, which is considered as the one preceeding industrial revolution. Credits to NASA https://www.giss.nasa.gov/research/news/20070208/

Data collected from 2010 to 2015 in Italy (by ISPRA, Superior institute for environment research and protection) clearly show that road transportation is one of the primary CO2 source in the country, along with energy production (see Figure 3.8). Road transportation (both commercial either private) represents the 29% of the total (energy production processes are the 31%) and little decrease in a single vehicle, multiplied by a huge number of cars that everyday travel across Italy, may have appreciable air quality improvement.

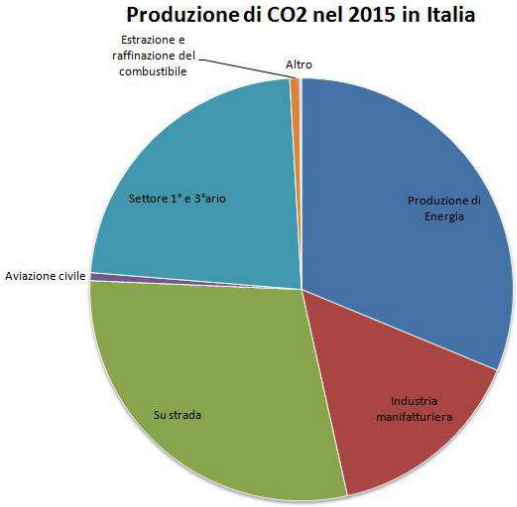

Figure 3.8: Allocation of total CO2 emissions between 2010 and 2015, in Italy.

For sake of completeness, we report also another greenhouse gas: methane (CH4). Its impact on greenhouse effect is far more pronounced than CO2 (nearly 30 times as high), but its concentration in atmosphere is (in average) $2ppm$. On the other hand, CO2 concentration has risen up to $400ppm$ (0.04%). In any case, methane is not a combustion product and it is not emitted into the atmosphere by cars or power plants. Rather, it is produced by usual animal

metabolism, or it leaks from gas pipelines during extraction and distribution. See Figure 3.9 for a diagram.

Water vapour (H2O) is another greenhouse gas, the more abundant in the atmosphere to be honest. It is considered the responsible of about 80% of total greenhouse effect. However, in contrast to CH4 and CO2, it has always been present, and its concentration is not influenced by human activities. It has got, unfortunately, a positive feedback mechanism: the higher the air temperature, the more water vapour it could be present, and thus the more temperature is increased by increased greenhouse effect. Little variations in other greenhouse gases may be amplified by this positive feedback mechanism.

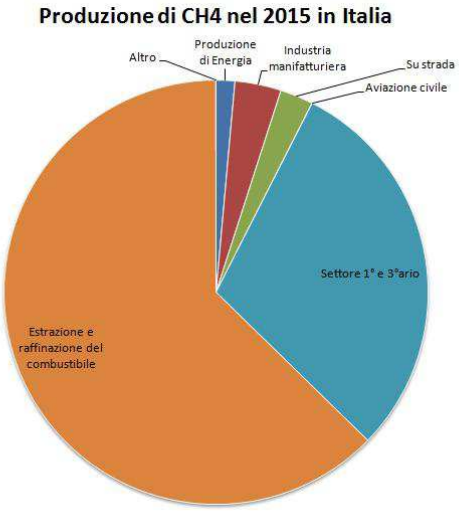

Figure 3.9: Allocation of total CH4 emissions between 2010 and 2015, in Italy.

Other harmful gases usually associated with air pollution are nitrogen oxides NOx and sulfur dioxide SO2. They are both strictly linked with energy management and production, but only NOx find their direct cause in road transportation. NOx is not a greenhouse gas (though in small contribution), but it is plenty of other harmful effects: it irritates respiratory tracts, eyes and mucosa, it cause acid rains, and it may also enhance cases of cardiac illnesses or asthma. SO2 has the same effects on humans and it is thousands time more harmful. A few part per billion are enough to cause damages to living beings. Luckily, its emissions has already been strongly reduced. In Figure 3.10 are reported main NOx and SO2 producers in Italy.

The conclusion is that, as long as CO2 and NOx are concerned, improvements in automotive industry would have significant impact on total emissions of such gases.

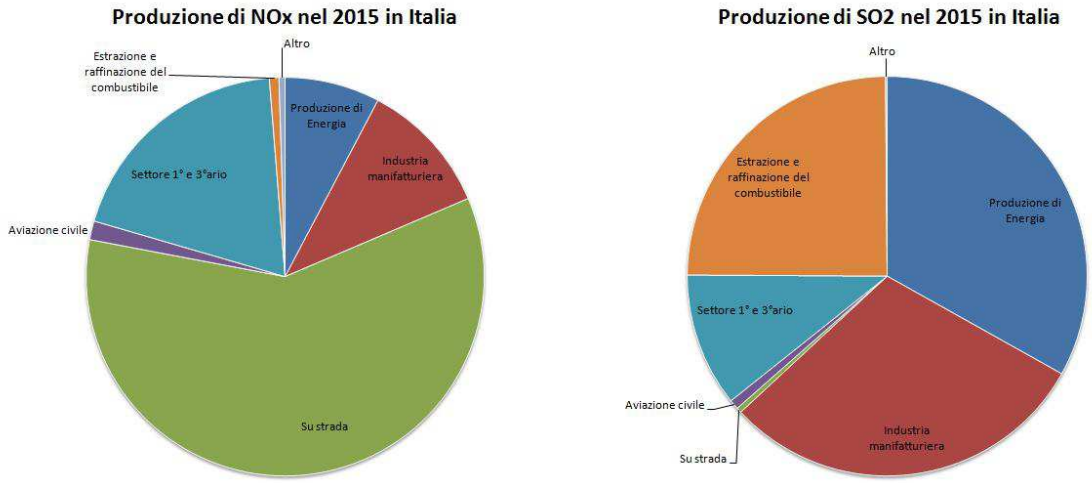

Figure 3.10: Allocation of total NOx and SO2 emissions between 2010 and 2015, in Italy.

Chapter 4

State of the Art Review

As long as clutch is considered, some attempts have been made in years in order to foresee clutch thermal behaviour. Researchers put their efforts especially in studying how the torque transmission is affected by heat generation and temperature rising. This reason closely approaches the subject of the Thesis: a better knowledge of clutch working limits may lead to better design and improved efficiency, and in this case to lower CO2 emissions. Literature found focuses on modelling the clutch alone, detached from its usual surroundings (e.g. a gearbox, a crankcase...). This coincides with the first stage of the workflow described in section 1.4: simulating a reduced subdomain, runtime are strongly reduced and setup simplified. This allows to run many more attempts and to more easily tune the initial model.

In this short chapter, literature found is presented along with strength and flaws of each work. Unfortunately, a clutch is a real component with its own market, definitely limiting the freedom to publish data and research results because of intellectual property. Papers are not so easily accessible and they are quite few. Those found by the author are presented in the following sections, categorized by space-dimension approach. In the last part of the chapter it will be shown how current Thesis is adding originality in what previous researchers produced.

4.1 Zero-dimensional Approach

No articles were sought using this approach. A real clutch is a complex assembly, with many components linked differently to each other and made of different materials. They take also place in a motion field that is the opposite of uniform. Therefore, lumped parameters modelling cannot take into account all of these particularities. Temperature distribution plays a fundamental role in this problem. A space discretization is strongly required.

4.2 Uni-dimensional Approach

A first attempt for space discretization may model the clutch as an axisymmetric component. This 1D model would neglect material variations along clutch axis, which is far from accurate (think about the three main clutch components: flywheel, disc, pressure plate, all coaxially mounted). A correction may divide the problem in a few slices along this axis, each with proper material parameters and boundary conditions. However, this is a more 2D approach and is outlined in the following section.

4.3 Two-dimensional Approach

Now it is worth spending two words about articles found. Awrejcewicz and Grzelczyk [1] carried out a numerical analysis along with a simple experimental test to validate the mathematical

model. In their numerical model, they considered only the clutch disc, composed by different materials along its axis, and they discretized it radially with m nodes. This allowed the researchers to assign as boundary condition a heat flux radius-dependent. Such r dependence comes from an elementary theoretical analysis of work done by friction on the disc: considering a pressure p acting equally on a disc surface, the normal force is given by

$$F_N = \int_{r_{int}}^{r_{ext}} \int_{-\alpha/2}^{+\alpha/2} p r dr d\theta \tag{4.1}$$

and, being the work done by friction $L = F_N \cdot l$ and ω the disc angular speed, the work produced by the clutch (i.e. the heat generated) in a certain period Δt is

$$L_f = F_N \cdot \omega r \cdot \Delta t \tag{4.2}$$

which is directly proportional to the distance of disc portion from its rotation axis (assuming a constant pressure distribution). This brings with it some consequences, the most important of which is a non uniform wear on the disc.

Anyway, going back to Awrejcewicz studies, they assigned a heat flux generation $\dot{q}_{(r)}$, resulting in a temperature distribution $T_{(r)}$. Furthermore, with their axial discretization they managed to divide the clutch in two slices with proper material properties: thermal conductivity $\lambda_{1,2}$, heat specific capacity $c_{1,2}$, and also heat transfer coefficient between air and material 1, material 1 and material 2, and material 2 and air.

Calculations were made with a home made code. They analysed also an unsteady behaviour. Numerical results are only qualitatively confirmed by simple experiments, whose purpose was to show qualitative temperature distribution on clutch disc. For a quantitative agreement too, the approach is likely to simple.

This paper show the need to consider different material composition of the component due to different thermal behaviour under the same boundary conditions. However, quantitative data agreement is not reached most likely because of the assumption of fixed convective heat transfer coefficient. The subject of this paper is somehow quite far from the one of this Thesis, as it examines the disc alone, not considering the bulk of many other components.

From this paper, it is anyway clear that a numerical analysis of a real clutch must take into account other clutch components, and convective heat transfer coefficient h must be evaluated precisely, thus opening the way to CFD.

4.4 Three-dimensional Approach

With three dimensions no geometry assumption is going to be made and much better accuracy can be reached. Some authors only exploited FE method, while others even CFD analysis. Their results are provided in the following.

FEM Analysis

Czel et alii [2] built a FEM model of a ceramic dry clutch to investigate temperature distribution calculated with a commercial software and validated this model with careful experiments in a dedicated testing facility. Model setup could be much more precise and according to reality than [1], thanks to FE discretization. In details, Czel considered the clutch disc engaged to the steel plate fixed to the engine shaft. He only discretized 1/6 of the system as it is clearly axisymmetric. He also assigned a heat generation flux using physics as basic as he could:

$$\dot{q} = \mu \cdot v \cdot p \tag{4.3}$$

where μ is the local friction coefficient, v the local disc speed, and p the local pressure applied. These parameters values come from experimental tests. Simulations (and tests) were unsteady,

so that variations with time of these parameters should be accounted for. Once materials were set, the challenge was to assign a meaningful convective parameter h that, without CFD, had to be fixed and previously evaluated. Czel arrived at the conclusion, after several attempts, with different value of h, that convection does not have a major role in heat transfer as long as this case is concerned. Thus he neglected completely convection relying only on conduction to spread out heat. This conclusion is in total disagreement with the work presented by Levillain et alii [3], showed in detail in the following section, and also with the result of this Thesis. In any case, Czel work shows the fundamental need for a precise CFD approach.

Abdullah and Schlattmann [4] used a FEM approach to evaluate the influence of grooves dug inside clutch disc to improve cooling (grooves may actually be useful in collecting wear from disc surfaces and to get rid of it). It also considers the thermoelastic problem. Modelling approach was as fine as the one by Czel [2], as well as the way to evaluate heat generation. An improvement is made by taking into account convection, but radiation is still neglected. The paper shows that the larger the grooves, the higher the convective heat transfer. A trade-off between contact area and grooves size had to be done during design. Unfortunately, a comparison with real data is not provided.

CFD Analysis

Wittig et alii [5] were the first to implement a CFD model in order to investigate effects of holes in clutch disc, to improve cooling. Being in 1998, their best improvement was given by the writing of Navier-Stokes equations in a moving local reference frame, in order to take into account clutch angular speed. Therefore, they pointed out how convection is dominated by this spinning and cannot be neglected. Furthermore, they show that a precise geometry description is required to properly resolve turbulent structure that further increase heat exchange.

However, neither in this case surrounding components were accounted for even if they could somehow affect the motion field. There was not much to do, since computational resources were limited (and only 1/8 of the region have been simulated). They have put anyway focus on the importance of a clever orifice design in the whole clutch assembly to grant a proper heat exchange with the external flow field.

Levillain et alii [3] improved Wittig model thanks to 20 years of advancing technology. They focused once more on the importance of studying air flow crossing clutch assembly during design to achieve the desired thermal behaviour. The clutch under investigation is a more compact double clutch, and four different holes layouts are numerically simulated. They paid further attention on heat flux distribution between different clutch discs sides, as well as on evaluating the thermal conductivity of a composite material (which the clutch disc is made of).

The balance between conductive and convective heat transfers was evaluated by Levillain in about 40% the former and 60% the latter. They reported as results relative deviation of the present layout from the baseline, without providing comparisons with real test data. Furthermore, no hint was given about radiation or surrounding components. They found out that the rotating disc (and plates) act as a pump ingesting air, increasing forced convection.

Improvements Needed

From the review of papers dealing with virtual simulation of clutches, it emerges that some aspects have to be studied deeper:

- being the convective heat transfer of primary importance, each element that may potentially affect the flow field must be considered;
- for the same reason, flow surrounding clutch assembly is determined also by vehicle speed and underhood layout, which has to be accounted for somehow;

- conductive heat transfer strongly determines temperature distribution: different components made of different material must be modelled properly;
- radiation may play some role, though marginal;
- an experimental validation is required.

With this Thesis the aim is to solve all of the items above. If the ultimate purpose is to clearly foresee clutch thermal behaviour, in order to improve clutch design and reduce CO2 emissions, the virtual model must be accurate, easy to use and flexible. The only assumptions affordable are those involving computational costs (i.e. runtime). These issues are faced as reported below.

- The clutch analysed is a real clutch and its geometry is carefully meshed. Components surrounding clutch are taken into account into numerical domain. CFD approach is used;
- free field boundary conditions for a reduced numerical subdomain are mapped directly from a detailed car with underhood geometry running at a certain speed (more details are given in Chapter 5);
- an investigation on real clutch materials is carried out, with special care put on clutch disc. As long as materials are concerned, this part is the most challenging one, because real clutches are protected by industrial property or patents, and so their exact material composition. Uncertainty is kept as low as possible. To properly model heat conduction, even solid components are volume meshed. A BEM thermal software is also used;
- the software used can handle radiation modelling;
- a real car test is performed in order to validate the model (the clutch equipping the car is the same as the one virtually analysed), as described in Section 3.3.

Many more details can be found in Chapter 5.

4.5 Bibliography about Brakes

We may expect that, since the working principle for brakes and clutches is the same (exploiting friction to transform energy), we could find literature about brake thermal behaviour simulation, and then adapt it to our case. However, a few differences do exist, and their presence suggested us to consider only clutch literature. Examples of these differences are the great contribution given by conduction in clutch temperature field, being far more massive than a brake; the impulsive engagement typical of a clutch; clutch surroundings.

Those issues were not considered in simulating brakes, because they actually did not exist. Therefore, the little literature found did not help much in our research.

Chapter 5

Reaching the Solution

The aim of this Thesis is to find a methodology to analyse the clutch thermal behaviour in earlier design phases, so that modifications can be accepted without major negative impacts on other assemblies and on performance. A series of instruction easily replicable to obtain an accurate clutch thermal simulation are considered. Instruction must take into account initial geometry description, cleaning, meshing, boundary conditions setting and post processing description.

The approach is typically bottom-up. Since external flow impact on clutch is not currently numerically simulated at FCA, but it is directly tested on first prototypes, and no models are available on literature, the research must start from scratch. Using also a trial-and error approach, we supposed at the beginning that we would have run many attempts before finding a satisfactory model setup. Concerns regard first of all proper assignment of thermal boundary conditions, also some software features, usually neglected, should be investigated too, to improve numerical analysis.

Considered that, it would be unsuitable to perform many complete system CFD thermal analysis. It is simply too heavy to run, and small adjustment would require days of calculations to see if they were worth the effort. We adopt therefore a reduced computational domain, involving the clutch and the neighbours may affect the temperature field. Details behind this choice are provided in Section 5.3.

This Chapter starts with a brief description of software used (Section 5.1), followed by a completion of the Thesis aim already presented in this introduction. After a comparison between complete and reduced models (Section 5.3) we report the issues faced during the setup in Section 5.5.4, as they may provide a starting point for further works. Numerical results are found in Section 5.5, Section 5.6 and Section 5.7, based on the different approaches.

5.1 Tools adopted

In Section 3.4 we stated the need for computational fluid dynamic analysis as the only accurate enough. Component geometry should be discretized completely, since the motion field is three dimensional and using an axisymmetry domain is a too heavy assumption. A correct description of the motion field is fundamental for convective heat exchange, which transfers at least as much heat as conduction.

CAD geometry needs cleaning in order to be correctly imported in a CFD simulation software. First of all, there cannot be any ambiguity about what is fluid and what is solid (i.e. what is *inside* and what is *outside*). Finite volumes-based software (FV from now forth) use normal vectors from object surfaces to understand if that face looks at the fluid or not. Therefore, normal vectors must be uniquely defined and surface geometry must be closed. Unfortunately, CAD software deal with CFD is not as good, so an intermediary should be present. In our case, the intermediary is represented by the pre-processor software $ANSA^{®}$ by Beta-Cae.

$ANSA$ duty is to handle both CAD geometries either FE description (Figure 5.1), with

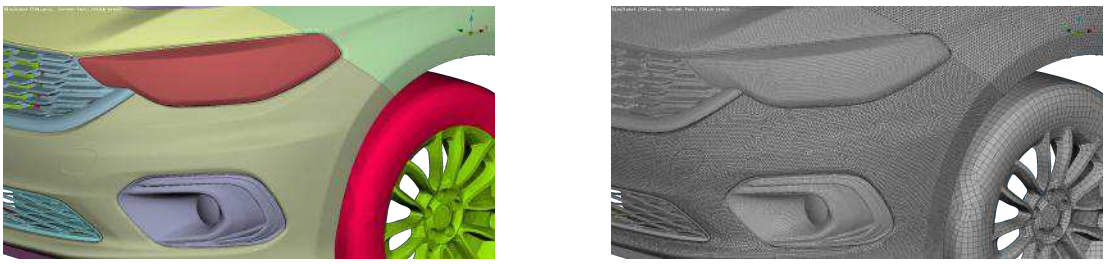

Figure 5.1: On the left, Geometry description of our prototype. On the right, its surface discretization (surface cells are visible specially on the wheel, where their size is wide).

several tools for their manipulation, from surface repair to surface reconstruction and meshing. Actually, we are describing ANSA for CFD. Meshing is anyway outstandingly important in each of these applications, and *ANSA* performs surface meshing, volume meshing, mesh quality evaluation and fixing, and many other tasks. Meshing methods and parameters relies on user choices and experience, while fixing tools are quite automated. Also geometry defeaturing has to be done in order to launch a CFD calculation: no one is interested on what it might happen if that nut faces that side to incoming flow. The depth of defeaturing is up to user experience and needs. Finally, when geometry has been cleaned and meshed, it can be exported with the proper file format.

As CFD software, $StarCCM+^{®}$ from Siemens is used. *StarCCM+* is a finite volume method-based software. It is able to solve most cases of flows with several models of turbulence and physics. Furthermore, it also handles heat exchange and its transfer mechanisms, making it suitable for our purpose. Many different kinds of boundary conditions can be set to domain boundaries, from flow motion to thermal. Both fluid and solid regions can be implemented. However, also for heat transfer it uses finite volume discretization, while it would be more appropriate to solve thermal problems with a finite elements approach. Part of the Thesis purpose is also to test *StarCCM+* capabilities to see if it can be used for thermal simulation purposes too. A scene of such thermal simulation is shown in Figure 5.2.

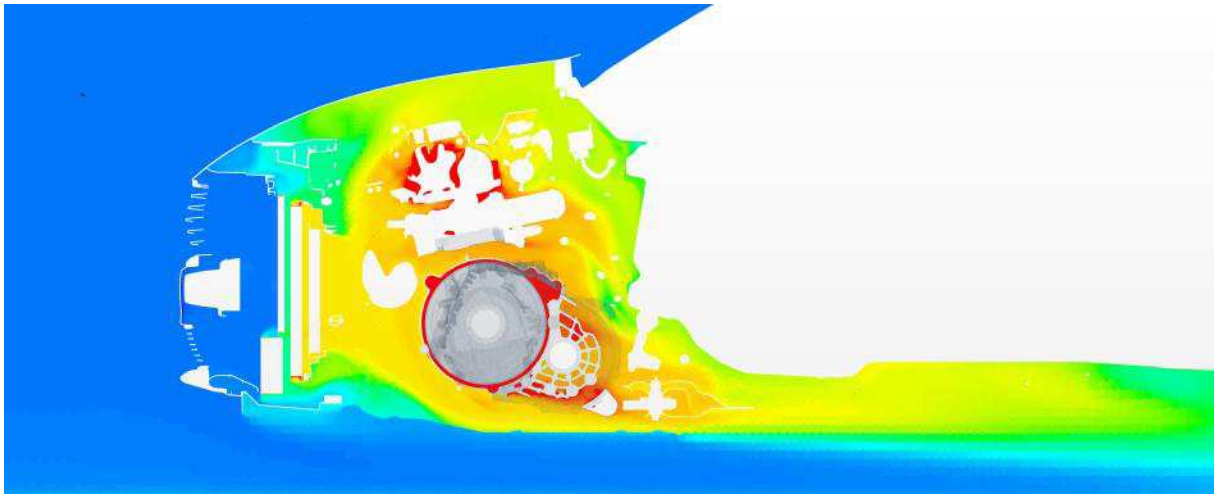

Figure 5.2: A thermal simulation for underhood components. The scalar quantity displayed is temperature. Solid components are unmeshed (they appear white in the scene) but their temperature is fixed on their boundaries.

Usually, thermal simulations can be calculated also with a finite elements method approach (*FE* from now forth), rather than a finite volumes one, obtaining very accurate results.

TaiTherm® from ThermoAnalytics) is a thermal BEM (boundary elements method) software. It discretizes surfaces with shell meshes and solves conduction and radiation equations locally between elements. Convection may be implemented with a fixed value. CFD data may be imported to provide a distribution of h coefficient, but a CFD simulation is previously required.

Shell mesh concept

We stated earlier in this section that for a finite volume software surfaces must be closed and non manifold. A FE software is not that stiff instead. Volumes are modelled using a skin (a two dimensional entity) with a constant value assigned as thickness. In most cases components thickness is well below other dimensions, and a single skin is accurate enough. However, bulk components are poorly discretized using this method and corrective actions should be performed. Meshing of skins gives a shell mesh.

StarCCM+ - TaiTherm Coupling

It is useful to exploit software features in order to get the best of them and put it together. In this case, StarCCM+ is optimized for flow calculations, while TaiTherm it is for heat transfer calculations. Motion field and temperature field are interrelated because of convection. Therefore, it is clever to import the same geometry model in both software and assign proper boundary condition to perform a coupled calculation. The procedure is graphically sketched in Figure 5.3. Iterative steps are listed below:

- StarCCM+ side: geometric representation of the model, which should be closed and non manifold, is used to extract the fluid volume. No solids are modelled: only fluid. Where there should be a solid part, it remains simply unmeshed with a "wall" boundary condition specification. Motion field is calculated along with fluid temperature and h coefficient. These spatial data is mapped and transferred to TaiTherm;
- TaiTherm side: the model geometry is made up of shell meshes with assigned thickness and materials. Conduction and radiation path are set by default, while it is up to user import CFD data to simulate the interaction with the fluid. Here, wall temperature is mapped and passed to StarCCM+, where it is assigned as "temperature" thermal specification in boundary conditions;
- Iterate: Surface temperature is determined also by fluid cooling (or warming). However, motion field is also influenced by surface temperature because of buoyancy forces. The data mapping from StarCCM+ to TaiTherm and viceversa should be iterative and the process has to be carried on as long as temperatures (or whatever we are interested in) globally reach a constant value.

5.2 Model improvement

In the ending of Chapter 4 we identified some aspects to be studied with more care in order to set up a methodology for clutch thermal analysis. This added care should improve models found on literature, making them suitable for industrial product development. We report those aspects here briefly:

- Convection as much important as conduction;
- Accounting for other underhood components;
- Accounting for radiation;

- Precise material assessment.

Furthermore, we also stated in this Chapter opening that we adopt a reduced subdomain, as a consequence of our choice to use a bottom-up approach. With this approach we are able to run several simulation attempts within the time allowed. These results may be inaccurate in absolute terms, but they should agree with reality when trends or deltas are considered. When post processing of reduced simulations show meaningful results, then we can implement the methodology in a complete car simulation. Here, more accuracy is expected. Details of this distinction between complete and reduced domains are provided in the following Section.

However, the choice for a reduced subdomain, while keeping in mind that also neighbours components play a major role, involves some issues that had to be faced at the very beginning of the work. In a few words, clutch thermal behaviour is determined not only by its heat generation and conduction within its parts, but also by heat exchange between these parts and the surrounding environment, which is furthermore determined by components packaging and temperatures. How can we take into account their influence, when we have just stated our will to exclude them from our reduced subdomain? Moreover: the clutch heat generation, never simulated before, will not strongly impact on global underhood motion field?

We try to answer these question in Section 5.5.4. But first, a description of what we mean with "complete" and "reduced" model is required.

5.3 Complete simulation vs Reduced simulation

Equations governing fluids motion are non-linear, making it very hard to formulate predictions using the superposition principle. A fluid-dynamic system may be remarkably influenced by a factor that, in a first analysis, was not so critical. Moreover, it is a risk stating that subsystems $A + B$ together will behave as subsystem A + subsystem B alone. Therefore, the whole system should be simulated globally during design early phases. However, this theoretical consideration clashes with any enterprise cost and time requirements. A trade-off between simulation accuracy and computational costs has to be found.

In FCA procedures, clutch components were set as adiabatic surfaces. They did not participate in heat management. With this set up, simulations could run faster.

When the issue of clutch overheating came along, some thermal simulations were modified to account for clutch too. However, in order to keep simulations runtime increase in a reasonable range, the clutch assembly were meshed as case-only. All those gaps filled with air were sealed and excluded from numerical domain. Results showed remarkable disagreement with test data. Further investigations suspected that the clutch internal architecture and packaging strongly influence its thermal behaviour. To prove this, a reduced and accurate subdomain is simulated (and this corresponds to Section 5.5 of the Thesis).

5.3.1 The complete, though simplified, car simulation

We describe here how the complete car simulation were set, why it is improvable, and how it interfaces with the reduced one.

In Figure 5.4 is reported a volume mesh representation of the complete car thermal simulation, where the clutch were geometrically described as case-only. Thus, details were quite poor. The simulation, despite its lack of accuracy, was anyway fundamental to draw the path for a better methodology study. In Figure 5.5 is reported the mesh as it should be instead. The Thesis work should end with the implementation of such geometry in a complete car, including clutch, thermal simulation.

At the time the problem of clutch overheating was faced, the simulation baseline was a complete thermal underhood simulation: it accounted for external aerodynamics, engine and gearbox heat production, radiator heat exchange with airflow (engine radiator, intercooler, and air

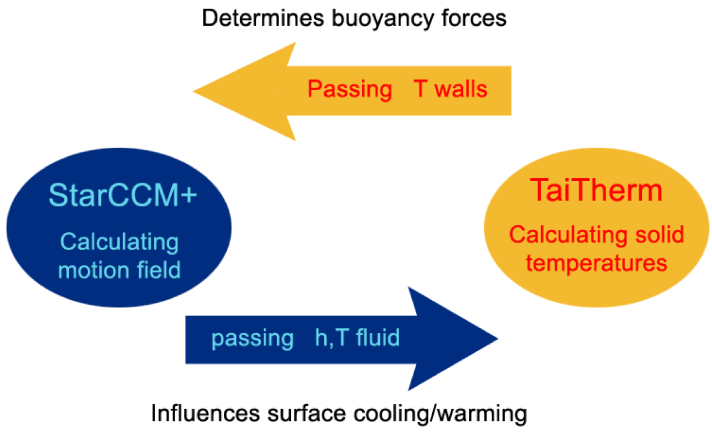

Figure 5.3: A scheme of the thermal coupling approach. It allows us to get the best from both worlds.

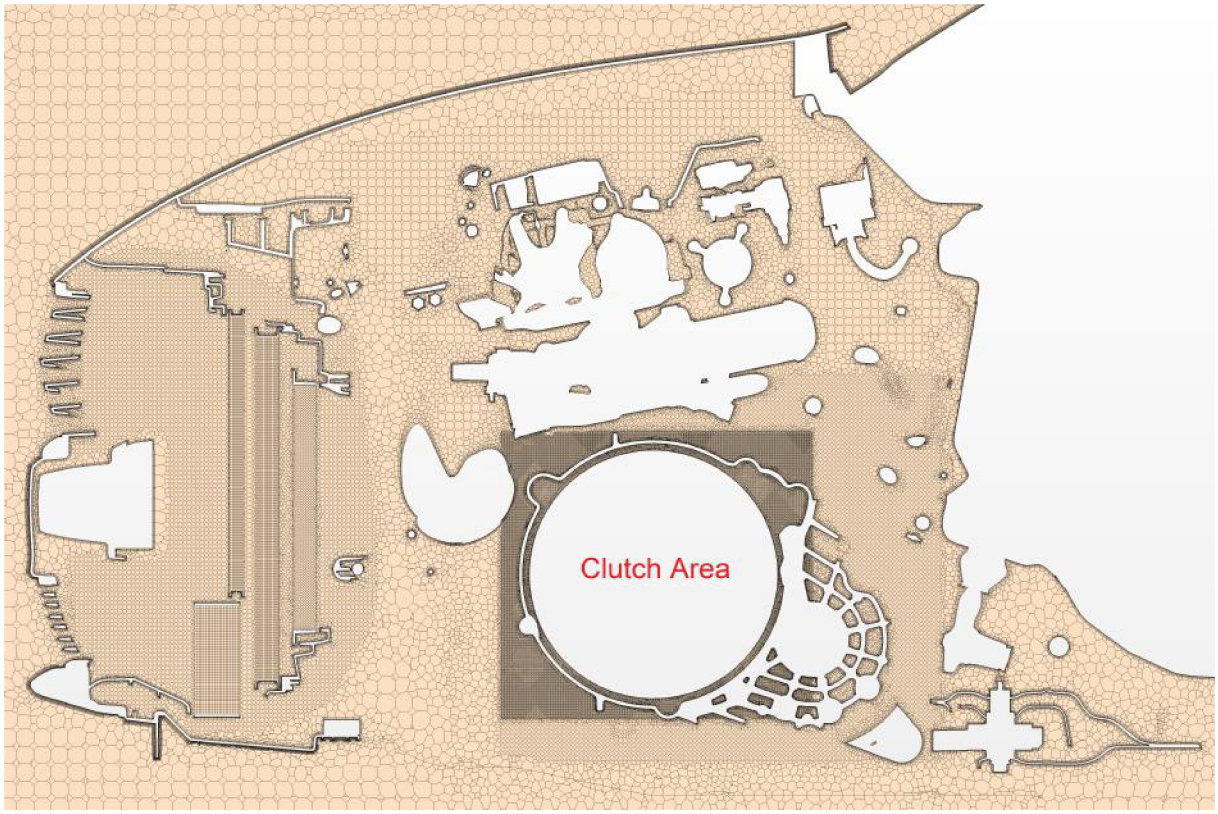

Figure 5.4: A volume mesh section of car simulation with clutch poorly detailed.

conditioner radiator were modelled), exhaust system, electric power system (battery, electronic control units), and ground heating. All of three heat exchange mechanisms were considered, and the coupled approach described in Section 5.1 was used. In this baseline set up clutch already existed but it did not participate in heat exchange: its boundaries were adiabatic. Then, in a first attempt to simulate clutch thermal behaviour, the less detailed clutch case is left free to develop thermally.

Since data obtained was to be compared with a real test, described in Section 3.3.2, boundary conditions were set as close to that test as possible. The test is inherently unsteady: it has plenty of starts and stops. However, temperatures reached an upper limit quite soon (Figure 3.6), so a steady simulation could be set. Without losing too much accuracy, average values were easily computed from telemetry data. Precisely, it was taken the average of:

- External temperature;
- Car running speed;
- Engine temperature;
- Gearbox oil temperature;
- Gearbox rpm.

Those averaged values are assigned then to the car steady simulation. One, fundamental, parameter had to be found yet: clutch thermal power output. This is the most delicate parameter in this simulation. Clutch output is nearly impulsive, rather than unsteady. The idea to evaluate it was to find a way to calculate total engine energy wasted by clutch, then divide it by the duration of the test, to obtain an average thermal power output. It was assumed that:

$$\dot{q}_{CL} = f_{(C_{eng},\omega_{eng},\omega_{GB})} \tag{5.1}$$

where suffixes $\dot{q}_{CL}$ stands for clutch thermal power generation, C for torque, ω for angular speeds, suffixes $_{eng}$ and $_{GB}$ for engine and gearbox respectively. All these values were monitored during the test.

Then, we take the integral time average of $\dot{q}_{CL}$:

$$\bar{\dot{q}}_{CL} = \frac{1}{t_f - t_0} \int_{t_0}^{t_f} \dot{q}_{CL} dt. \tag{5.2}$$

This provided a value of about $600W$ generated in average (as it was produced continuously during the test) by the clutch, to be set inside the simulation. The definition of the function above is not the only one possible. We may also use the theoretical relation based on energy conservation principle, expressed by Equation 2.3. There, we neglect gearbox inertia I_1 and resisting torque, since they are much smaller than car inertia I_2 and inertial forces. Reportin it here, and substituting values from telemetry, we have

$$\bar{\dot{q}}_{CL} = \frac{1}{t_f - t_0} \cdot \left(\Delta K + \int_{t_0}^{t_f} C_{eng}\omega_{eng} dt \right) = 680W \tag{5.3}$$

Values obtained with different models are quite similar. In a first guess, we adopted the round value of $600W$. This value was set as heat source, inside StarCCM+ simulation, on the clutch case boundary.

So far, the model is improvable considering two aspects:

- the clutch is not properly detailed. This forced us to set heat generation on its case, while it should be set on the clutch disc. Heat source set on the case corresponds to an enormous area lapped by air, thus providing a heat exchange much bigger than what may happen in the real world. Furthermore, the heat capacitance acting as heat sink was there completely neglected;

- since there is no solid in StarCCM+ nor in TaiTherm simulations, there was no way to control cooling by material property tuning.

We try to solve these issues (introducing newer ones though) with the reduced subdomain simulation: with less car to simulate, we can detail better the clutch without worsening runtime.

5.3.2 The reduced and more detailed subdomain

We consider the clutch CAD file without neglecting any part. The clutch case is bolted to the gearbox on one side and to the crankcase on the other side, both quite hot during vehicle operation. Reduced subdomain includes then the clutch assembly of course, but also the lower gearbox and the crankcase. In Figure 5.6 is provided a sketch of simulated components. If we remove the case (Figure 5.7) we finally find the added detail, with flywheel, pressure plate, clutch disc and many other smaller parts represented and meshed.

We are therefore able to assign materials properly to each clutch component. Furthermore, the heat generation can be set exactly where it should be: on the disc surface, which faces the flywheel on one side and the pressure plate on the other. Actually, the present model is that of an engaged clutch: there is no air gap between the disc and flywheel or pressure plate.

Other components influence on that subdomain is taken into account by mapping velocity and temperature data in clutch area from the complete car simulation, where such components do exist. Their influence on clutch can effectively be evaluated with the motion and temperature fields they produce. However, even radiation is important, and excluding them from the reduced domain means to exclude their radiative contribute. But temperatures are relatively low, say under $100°C$, thus radiation may be neglected in a first attempt. A better description of how mapped data from complete simulation is useful in the reduced one is described in Section 5.5.4.

Later on, anyway, radiation has been added as a test case.

5.4 Values monitored

Before getting deep into the problem, it is useful to spend some words about what we are monitoring. We keep in mind the aim of the Thesis: to provide a methodology to properly simulate clutch thermal behaviour. Consequently, temperatures are among the quantities monitored. During the real test some thermocouples where placed on the clutch case and clutch actuator. Temperature values found in those region in the simulation are compared with real values. If values show some accordance, we may assume that the methodology direction is the right one.

Another value monitored is the maximum temperature reached on the clutch disc. The value is easily measurable in a numerical simulation, but much less in a real vehicle. There, an hypothesis is needed: since during test no smoke or smell were noticed, we must agree that clutch temperature remained in their safe range ($< 270°C$). Above this limit, smoke and smell is expected. Thus, if simulated temperatures raised much above $270°C$, something is going wrong.

In addition, it could worth the effort the creation of an "equilibrium" index, in order to know when the steady simulation has really reached convergence. Therefore, we monitored how much energy entered the system and how much left it, then we computed the difference: when that delta was 0 we are sure that solid bodies are not warming or cooling anymore, and temperature field is stable.

Summing up, we use the following criteria to state if simulations are improving or not:

- A simulation is ok only if temperatures reach stable values, and energy entering the system is equal to energy leaving it;
- A simulation is ok if maximum temperature on the disc does not exceed $270°C$;

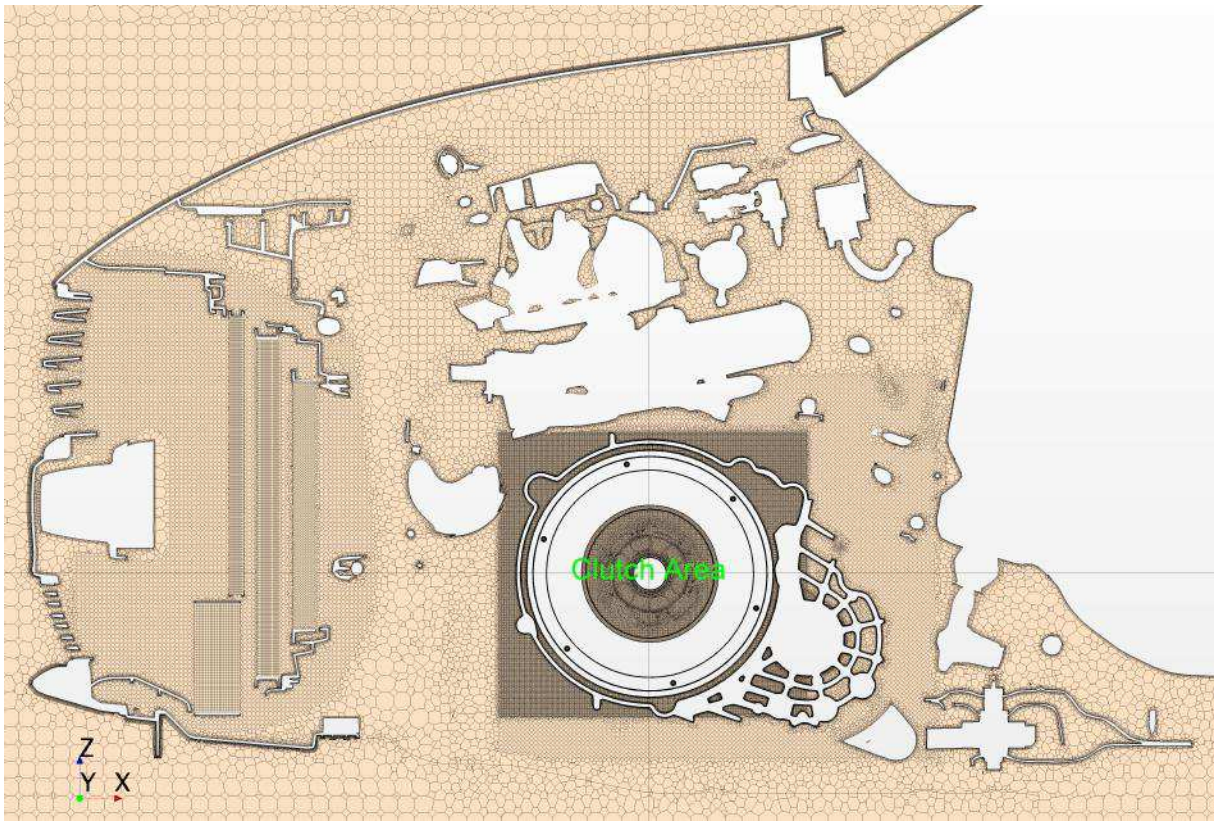

Figure 5.5: A volume mesh section of car simulation with clutch properly detailed.

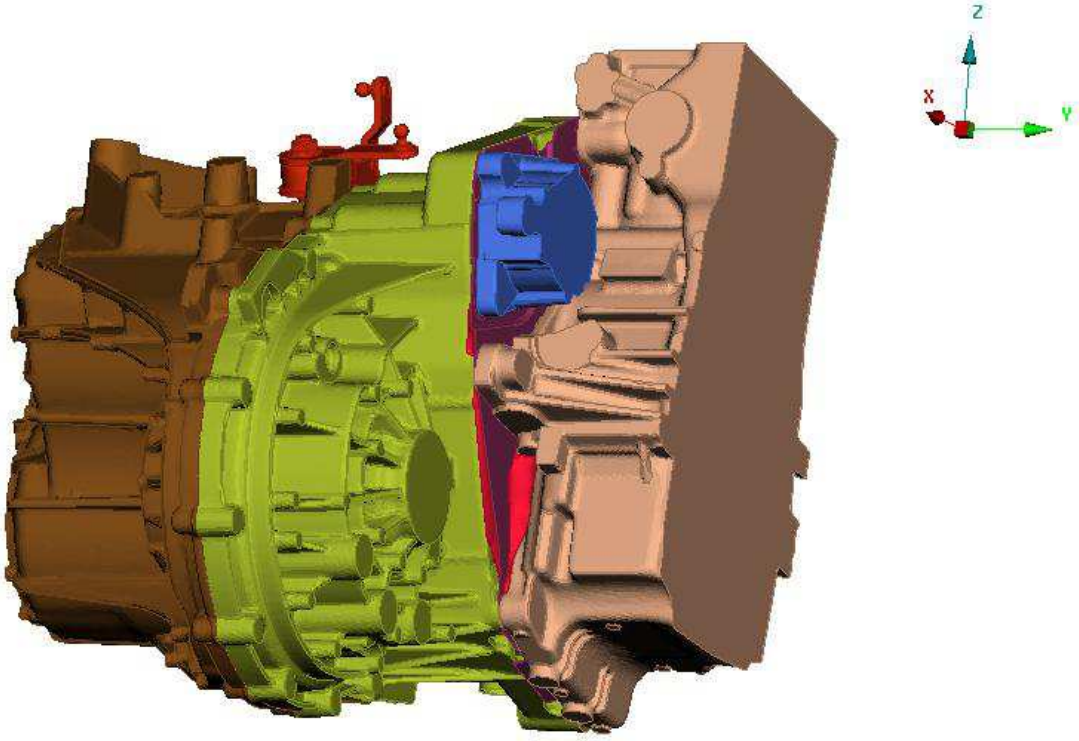

Figure 5.6: Components included in the reduced subdomain. Pink is the crankase, yellow the upper gearbox (also called clutch case), brown is the lower gearbox.

5.5 CHT Approach

We start the methodology description with the CHT approach. Here, only ANSA pre-processing and StarCCM+ simulations are used.

5.5.1 Components Description

We report in Table 5.1 a list of parts modelled, each one with its own region and physics inside StarCCM+. Material is also reported, along with if it is in rotation or not.

5.5.2 ANSA pre-processing

Parts reported in the previous Section are wrapped in ANSA environment to provide StarCCM+ with a cleaned and closed geometry. Only very few details are removed: most of them remain. The biggest geometric manipulation is done on the diaphragm spring, as its radial grooves are sealed.

It is worth reporting that, while each component has a surface that actually represents its shape and volume, crankcase and lower gearbox have not. They are still represented with a closed shell mesh, with zero thickness, so that they can be seen as predetermined boundary conditions, from a CFD point of view. We leave them without proper description because we are not interested on what it happens inside those components: their temperature is fixed and imposed as boundary condition, there is no need to model their actual physical material distribution.

It is fundamental, for solution convergence reasons, that interface surfaces between two different parts are meshed exactly in the same way (Figure 5.9). This will allow StarCCM+ volume mesher to create a conformal mesh between different regions, that is outstandingly helpful for residuals convergence. The result is reported in Figure 5.10.

Conformal mesh

Conformal mesh is important in calculating quantities fluxes conservation, represented in CFD problems by mass, momentum, energy, turbulent viscosity and turbulent kinetic energy. Best practices require that the line connecting the centroids of neighbouring cells must ideally pass as closely as possible to the centroid of the face separating the cells (this is one of several guideline to judge mesh quality). A two dimensional example is shown in Figure 5.12. Cell aspect ratio

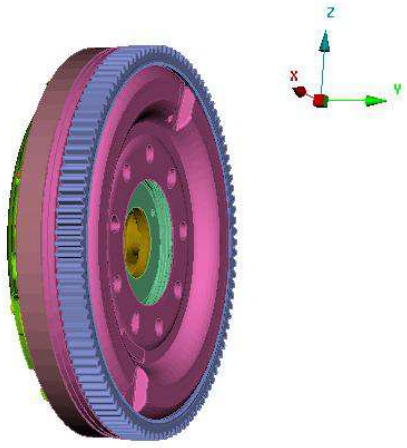

Figure 5.7: Inside the clutch case we found the flywheel, the pressure plate, the clutch disc and several other smaller components

Table 5.1: Components modelled, with their material and rotation

	Component	**Material**	**Rotation**
1	Disc	Bitumen Copper	Yes
2	Disc pin	Steel	Yes
3	Disc plate	Cast iron	Yes
4	Disc support	Cast iron	Yes
5	Pressure plate	Cast iron	Yes
6	Pressure plate attachments	Steel	Yes
7	Pressure plate inner disc	Cast iron	Yes
8	Pressure plate outer disc	Cast iron	Yes
9	Spacers	Aluminium	Yes
10	Lower gearbox	Aluminium	No
11	Clutch actuator	Steel	No
12	Actuator bellow	Rubber	No
13	Actuator tube	PVC	No
14	Gearbox control	Aluminium	No
15	Upper gearbox	Aluminium	No
16	Crankshaft	Steel	Yes
17	Crankshaft plate	Steel	No
18	Crankcase	Steel	No
19	Crankcase plate	Aluminium	No
20	Starter	Steel	No
21	Flywheel attachments	Steel	Yes
22	Flywheel support	Cast iron	Yes
23	Flywheel engine attachments	Cast iron	Yes
24	Flywheel inner disc	Cast iron	Yes
25	Flywheel gear	Steel	Yes
26	Flywheel separator	Cast iron	Yes

is related to such problem: usually, with an aspect ratio close to unity quality is high and fluxes are well determined.

If we consider a non conformal mesh, as the one in Figure **??**, in addition to "gap of void" that take place between different regions we also note that the line connecting two centroids crosses unwanted cells. When such gaps are many the software struggle to reconstruct proper fluxes, often without success. The result is a huge slowdown of convergence rate. It specially happens when thermal calculations are concerned, i.e. when energy fluxes have to be evaluated between different materials.

It is worth spending hours in ANSA pre-processing in order to obtain a high quality surface mesh globally as long as finite volumes software are used.

5.5.3 Computational Domain

Once we determined which components we want to simulate (i.e. clutch and its closest neighbours), we have to determine also the surrounding volume of fluid to be simulated. As we said in Section 5.3.2, we want to map complete car data of $\vec{v}$, T, p in order to set it as boundary conditions in the reduced simulation. This data is fundamental if we want to account for the presence of other components, even thought we are not simulating them explicitly. Where shall we map such data to achieve that?

The idea is to map data as close as possible to clutch assembly. If mapping surface is too far from clutch, other components in between (yet not accounted) influence the flow but we would not be aware of them; if mapping surface is too close, boundary layer may invalidate the external flow field. Seen that, in the complete simulation, boundary layer thickness is $\delta \sim 2mm$, if we map data $15mm$ far from surfaces we expect that this data is free from boundary layer effect and other components influence is kept at minimum. Flow evolution in an area so close to clutch is mostly dominated by the presence of the clutch itself, rather than other components. Furthermore, by mapping data on a tight surface we avoid intersecting too much other parts, where mapping would be impossible.

Therefore, we draw an envelope of clutch assembly along with crankcase and lower gearbox, to obtain Figure refinv1. The purple shadow surface is both the computational domain (i.e. the bulk of fluid) and the surface where complete car data is mapped on.

To better identify the uncomfortable situation of mapping data on such surface, the same surface with components highlighted are represented on board in Figure 5.14. There, it can be noticed how tight the surface is to other components, and how close they are to each other. The trade-off we made keeps at minimum intersections with other components, but at the same time accounts for their influence in terms of flow field and temperature field. However, it is not safe from issues. We analyse these issue in the following Section.

5.5.4 Boundary Conditions Assignment

We need to assign boundary conditions which determine the far field and boundary conditions which determine the thermal problem. Obviously, both represent a challenge. Let us start with far field BC.

Since it is fairly sequential, we report the procedure as a list of operations to do.

1. Run the complete, though simplified in clutch zone, complete car thermal simulation;
2. Import in StarCCM+ complete simulation the envelope surface discussed in the previous Section, as arbitrary section;
3. Map $\vec{v}$, T, p data on such surface and export as .csv table;
4. Pre-process the reduced simulation:

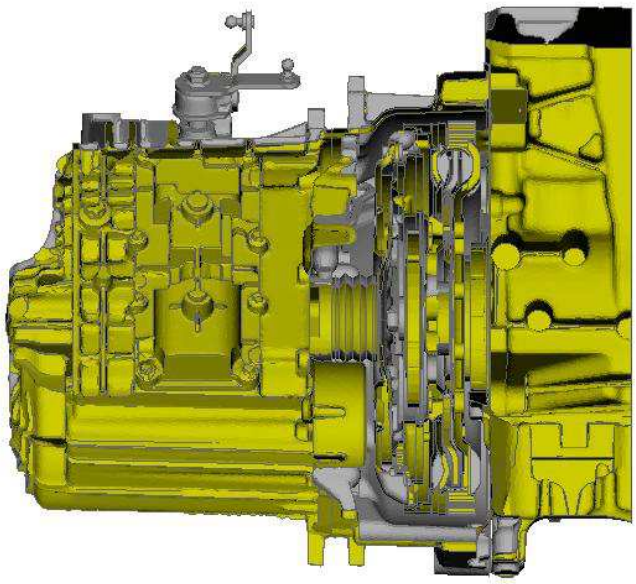

Figure 5.8: In this picture it can be seen the several air gaps (in gray) distributed all over the clutch assembly. Yellow stands for metal (solid) regions. Crankcase on the right and lower gearbox on the left will remain unmeshed in StarCCM+.

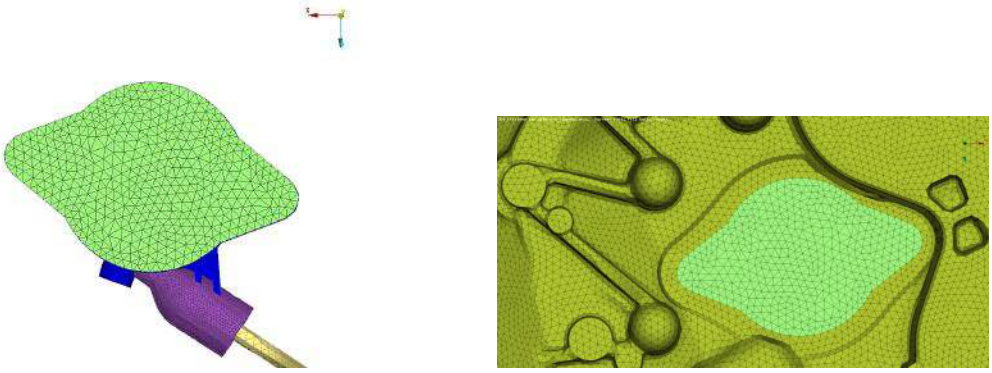

Figure 5.9: On the left, clutch actuator is shown with its surface mesh discretization. On the right, the same surface representation but caught on the clutch case side. Note the specularity of the axes (in the top right of the pictures) and the perfect nodes coincidence at the interface.

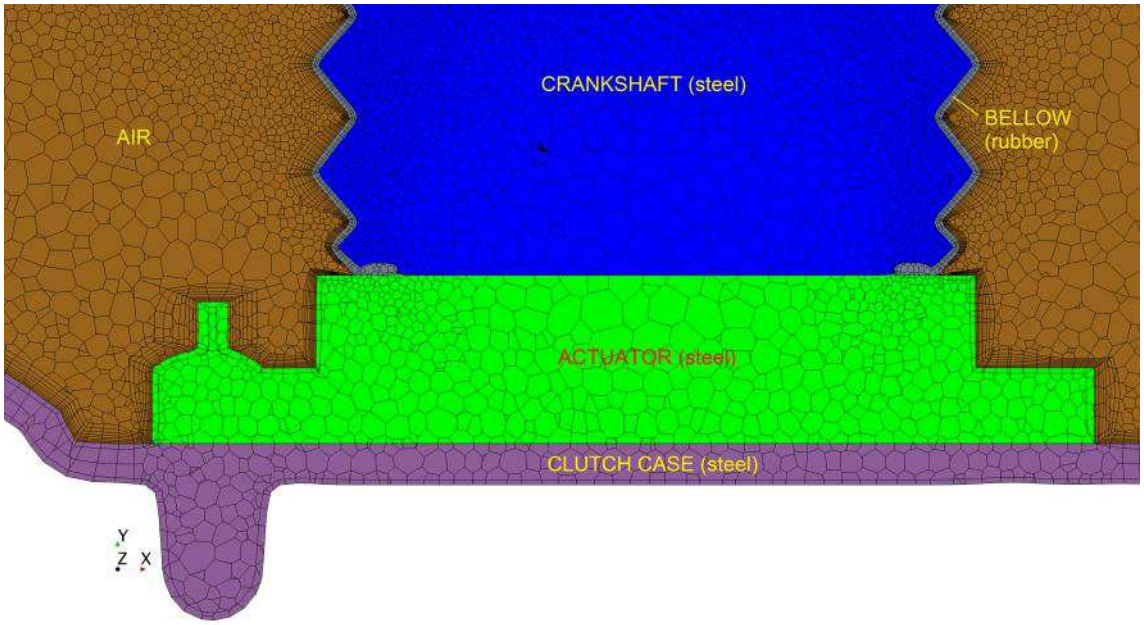

Figure 5.10: Conformal mesh among different regions.

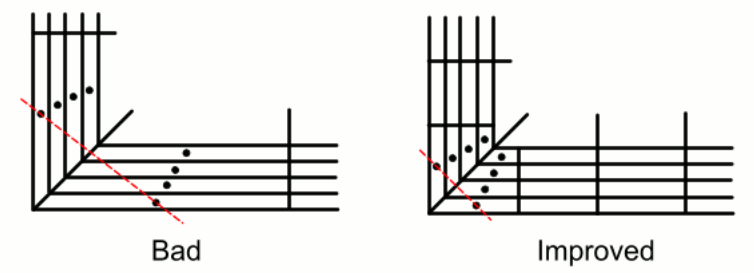

Figure 5.11: Two dimensional mesh in the boundary layer region of a flow. As cells shrinks, the line connecting centroids is closer and closer to separation face centroid.

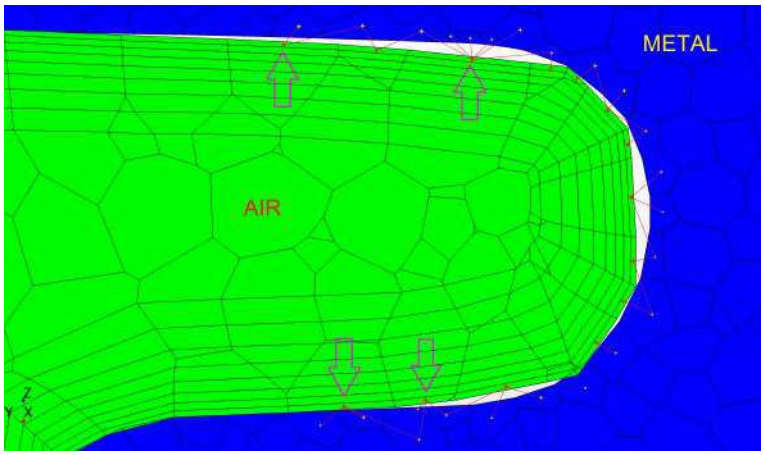

Figure 5.12: Centroids and lines linking them are highlighted. Arrows point to cells particularly involved with this issue.

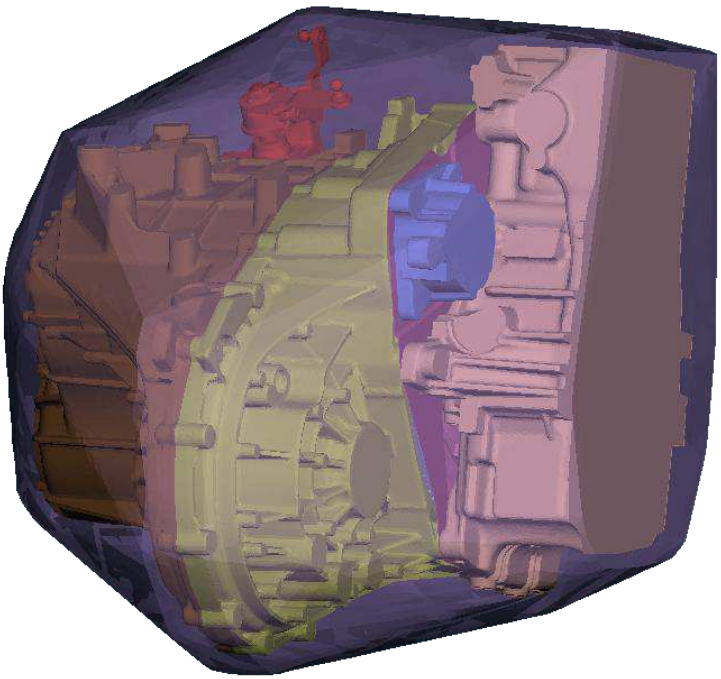

Figure 5.13: Figure 5.6 is here reported with the envelope enclosing components. The light purple envelope is the computational domain.

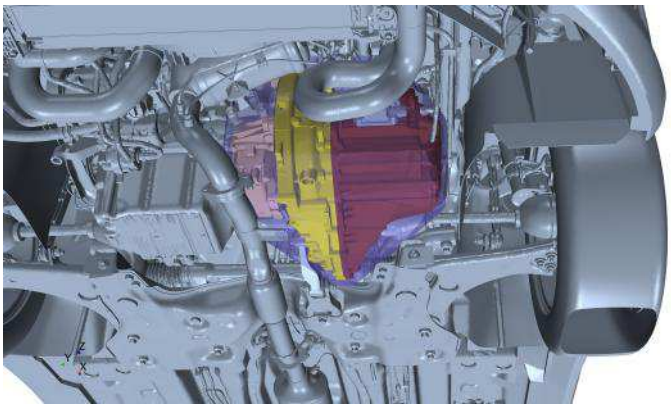

Figure 5.14: The computational domain as it appears once bolted on the car. Several components are very close to it, while others pierce it. Other components have been hidden for clarity reasons.

- Import in StarCCM+ components after ANSA pre-processing;
- Import the envelope surface;
- Extract fluid volume using the proper StarCCM+ operation;
- Assign proper physics to each region and set the boundary corresponding to envelope as "velocity inlet";

5. Import in StarCCM+ the table of item n. 3;
6. At the inlet boundary, specify the entering of $\vec{v}$ as table;
7. Set also T at the inlet with the same procedure.

We immediately notice some issue to solve. We examine them one by one in the following, then we provide the solution found.

Velocity Inlet

Under these circumstances, there is no outlet. And this is a big issue. Furthermore, mapped $\vec{v}$ may point both inward either outward the computational domain, while the whole boundary is still set as "velocity inlet". Even if mapping is precise, which means that the net mass flow equals 0, CFD theory states that, with a subsonic flow, two condition must be set at the inlet, while a third is calculated. On the other hand, at an outlet, one condition must be imposed and two other are calculated. Therefore, if we impose $\vec{v}$ and T at the inlet, only pressure p may be calculated. T is not free to evolve, even where flow leaves the domain. Therefore, the domain behaves as an isolated system: it cannot exchange mass ($\dot{m}_{net} = 0$) nor energy. Having an energy source inside it, temperatures will not stabilize at all during the computation.

We choose to split the single boundary inlet into two separated ones, using the velocity direction (inward/outward) as splitting function:

$$\vec{n} = \frac{\vec{A}}{|\vec{A}|} \qquad\qquad i == \left[(\vec{v} \cdot \vec{n})\right] \leq 0 \tag{5.4}$$

The normal vector $\vec{n}$ always points outward the domain. i is a boolean operator. When $\vec{v}$ points outward, it has the same sign as $\vec{n}$, thus $i = 0$. When $\vec{v}$ points inward, $i = 1$. We can therefore say that i answers the question: *"is that cell an inlet?"*.

If we display the i distribution on the domain, we obtain Figure 5.15 (where we zoomed on a portion of its). Velocity vectors actually point out the domain where cells are blue, corresponding to $i = 0 \rightarrow$ outlet. Where $\vec{v} = 0$ there should be a solid component, and actually flow cannot exist in that region. Components intersected are represented by part of the crankcase and both semi-axles. We try to take account of them by setting those cells as inlet, with null velocity imposed. Despite of this counter-measure, flow still evolves in cells between walls and inlet with $\vec{v} = 0$. But, having set the envelope surface really close to the walls, this evolution is kept at minimum and air, there, does not move. There should still be solid body there anyway, we will remove this further issue with the complete simulation. Another possible solution is to make the domain end with a solid region, rather than a fluid region.

So steps to be performed to solve this first issue are:

1. After having entered $\vec{v}$ as table, initialize the solution;
2. Create the field functions $\vec{n}$ and i;
3. Run 200-300 iterations to stabilize the flow;
4. Split the inlet boundary by function i;

5. Set the obtained boundaries as "velocity inlet" and "pressure outlet" accordingly, and set p as table on the outlet;

6. Iterate 3 to 5 twice for better stabilization, otherwise reverse flow on outlet will be detected.

Far Field Really Close to Walls

StarCCM+ best practices (but also common sense) would say that if the boundary condition name is "far field", it should be set *far* from other boundaries, where they do not feel the influence of other conditions. However, that is not really the case in our reduced subdomain. This may cause convergence slowdown, when no convergence at all.

Luckily, StarCCM+ manages to keep the solution process stable and the simulation converges anyway. And this issue is out of the way.

Mutual Influence Between Inside and Far Field BC

The situation is the following: we mapped flow field around a region poorly detailed, and then set them around another region, similar to the previous one, but much more detailed. The question that we should ask is: are we sure that what *now* happens inside the subdomain does not affect the flow field where we mapped it before? If this happens, and there is no reason to claim the opposite, we are not setting proper far field boundary conditions.

We are therefore assuming that the flow field around the clutch assembly is not affected by what happens inside the clutch case. With steps describe two paragraphs ago, the simulation actually self-adjusts those boundary issues, but we are not sure that this adjustment will agree with reality. However, we could not find a way that did not involve this strong assumption.

The only way to be sure that this assumption is not so strong to invalidate simulation result is to perform a complete car thermal simulation, now including the whole clutch assembly, and then compare both temperature either velocity fields on the envelope surface with those coming from the reduced simulation. If fields are similar, then the clutch thermal behaviour can be analysed with a reduced simulation, saving much time in pre-processing and runtime (complete car thermal simulation without clutch, that provide far field data, would be run anyway). If, on the other hand, they differ a lot, we have probably no choice than to analyse clutch assembly along with the rest of the car.

Other Boundary Condtions: Thermal BC

Once we set up the simulation with steps described so far, only thermal boundary conditions remain to be imposed. Briefly, they are:

- Crankcase temperature: being cooled by a dedicated circuit, its temperature remain fixed throughout the test, thus its thermal specification is "constant temperature";

- Lower and Upper gearbox temperature: this is more tricky. Telemetry show a constant oil temperature, therefore we may set that temperature on gearbox boundaries. However, with this approach, we obtain results that quite disagree with thermocouples data. We therefore set thermal specification as "convection" with oil temperature as ambient temperature along with an usual force convection h coefficient. A more accurate model would see the gearbox as a heat source, because it actually wastes part of engine power into thermal power because of friction. An investigation on gearbox global efficiency should be done;

- Clutch thermal power output: we set an heat source (whose value is calculated in Section 5.3.1) on clutch disc sides, set as contact interfaces: flywheel-disc and disc-pressure plate;

- Material properties taken into account are: thermal conductivity λ, density ρ, and specific heat c. The most challenging values to set are those concerning clutch disc: its composition is description is property of the supplier, who is not so happy to share it with us. Therefore, we assumed it is made of a bitumen matrix to provide a high friction coefficient with copper particles dispersed in it, to enhance thermal conductivity. Composite λ have been evaluated thanks to Maxwell relation:

$$\lambda_{comp} = \lambda_1 \cdot \frac{2\lambda_1 + \lambda_2 - 2\phi_2(\lambda_1 - \lambda_2)}{2\lambda_1 + \lambda_2 + \phi_2(\lambda_1 - \lambda_2} \qquad \phi_2 = \frac{V_2}{V_{tot}} \tag{5.5}$$

  where subscripts $_1$ and $_2$ stands for respectively Bitumen and Copper.

5.5.5 Mesh and Physics setup

In FCA thermal simulations are carried out on a polyhedral mesh. We choose a minimum size that allow to model properly sharp surface angles and thin plates. Furthermore, we use a thin mesher model to increase mesh efficiency in these plates. Prism layers are set in order to reach a wall $y^+ < 2$. Referring to Figure 5.16, we want to remain safely enough in the linear region in order to simulate, rather than to model, the boundary layer. A well simulated boundary layer provides more accurate convective heat exchange. Prism layers are also set inside solid regions: physically is an error, but computationally it helps in detecting steep temperature gradients. We already talk about the importance of a conformal mesh: to be sure that it is obtained, an imprint operation is made before surface remeshing. A representative cut plane is shown in Figure 5.17.

As long as physics is concerned, we set up a standard RANS steady simulation. We also choose a segregated flow. The turbulence model is the well-known $k - \epsilon$, with all y^+ wall treatment. We also have an ideal gas with gravity, to account also for natural convection.

As convergence criteria, we use the steadiness of temperatures along with the net energy entering the system equalling zero. Usually, with 2000 iterations convergence is ensured. Runtime are about 3-4 hours with 256 processors at FCA high power computers.

5.5.6 Results

As a premise, we would like to say that absolute results are not so meaningful. It is really hard, in fact, to run a virtual simulation and obtain values that perfectly agree with reality. These agreement may happen only if we run a definitely too heavy calculation for any reasonable purpose (apart from research ones). It is common, anyway, to perform several simulations with little variation between one another, compare them with the same situations in reality, and try to extrapolate correlations and data fitting. But this is an expensive task.

What our methodology should do to declare it suitable is to correctly foresee trends. At the moment qualitative results can be considered sufficient: if one physical quantity decreases by a delta in a test, the same quantity must decrease by the same relative delta (more or less) in the virtual simulation. Trends prediction is very helpful during preliminary design phase, since it states if an adjustment is correct or it is better leave things as they were.

In the first part of this Section we provide a comparison between allowable thermocouples output and corresponding simulation probes. Then we report a few scalar scenes ((Figure 5.18 to Figure 5.26), along with some plots that describe the problem and its findings. Descriptions and comments are attached to images as captions for better clarity. Values are normalized because of non disclosure reasons (Table 5.2). In the following Section we add some fundamental comments about solution sensitivity to setup parameters.

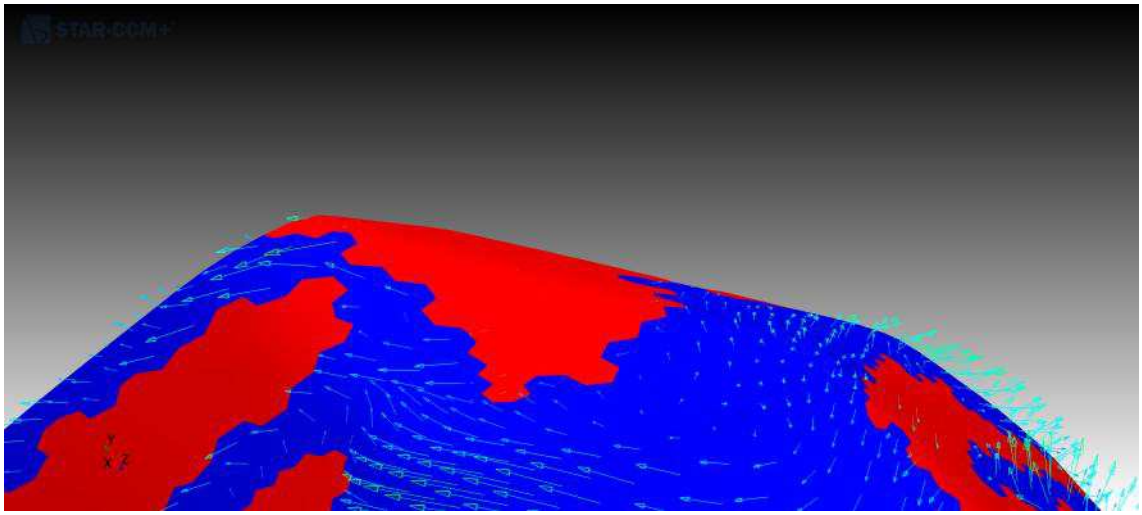

Figure 5.15: A portion of the computational domain with the i field displayed, along with velocity vectors.

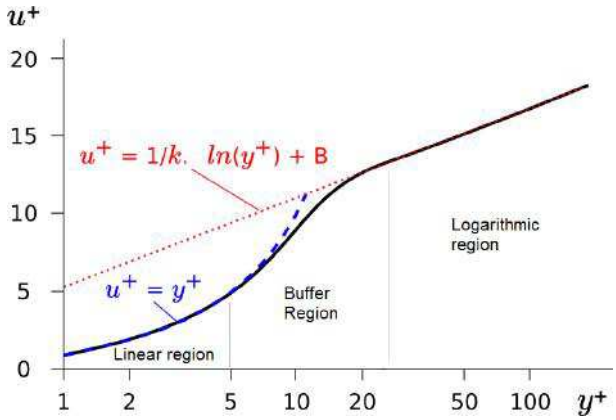

Figure 5.16: A classical $y^+ - u^+$ diagram.

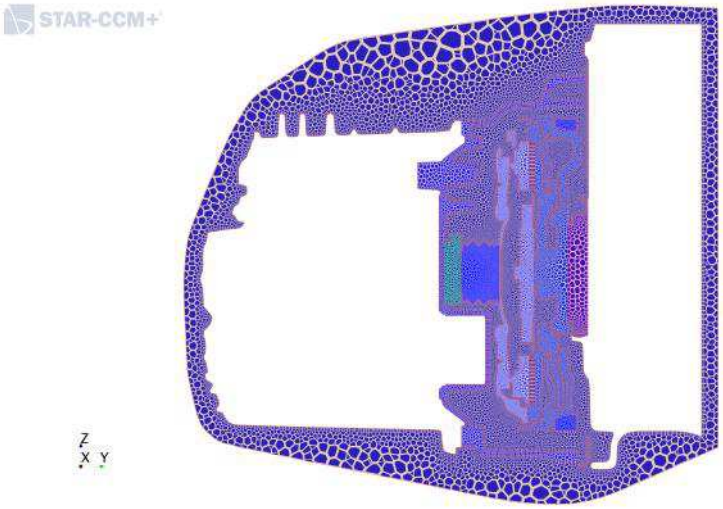

Figure 5.17: A cut plane of volume mesh, with components boundaries highlighed.

Table 5.2: Comparison between real and simulated thermocouples output. Results are normalized with real test values because of secrecy reason. Relative variation is provided as $\Delta T_{rel} = \frac{T_{num} - T_{exp}}{T_{exp}}$

	ΔT_{exp}	ΔT_{CHT}
T clutch case 2	T_1	+34%
T clutch case 3	T_2	+22%
T clutch actuator	T_3	−1%

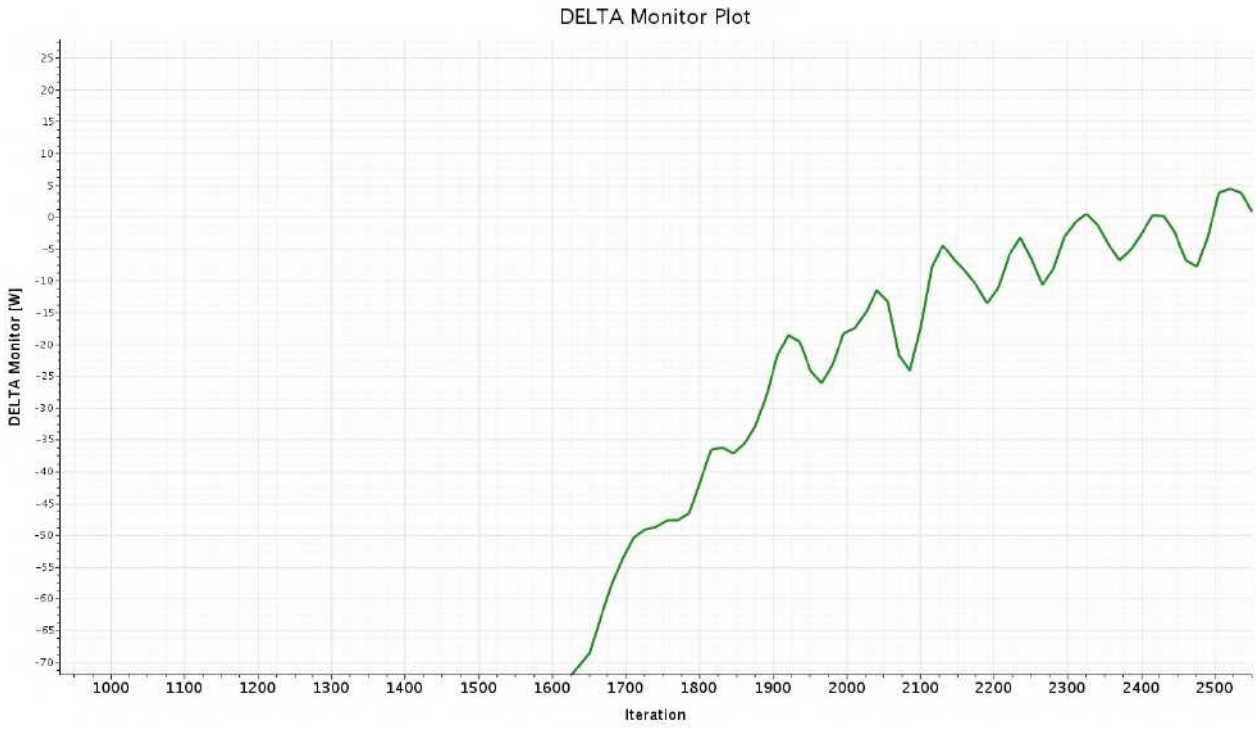

Figure 5.18: **CHT Approach.** Plot for Δq_{net}: the balance of energy entering and leaving the system. Source terms are represented by clutch thermal power output and high temperature on surfaces. Heat is removed by air flow that enters the system and leaves it at warmer temperature. When the balance between q_{in} and q_{out} equals 0 then the simulation is in thermodynamic equilibrium and components temperature is stable. Order of magnitude of q_{in} is about $2000W$, thus we can well accept oscilations of $10W$. The trend asimptotically approaches 0 in 2500 iterations.

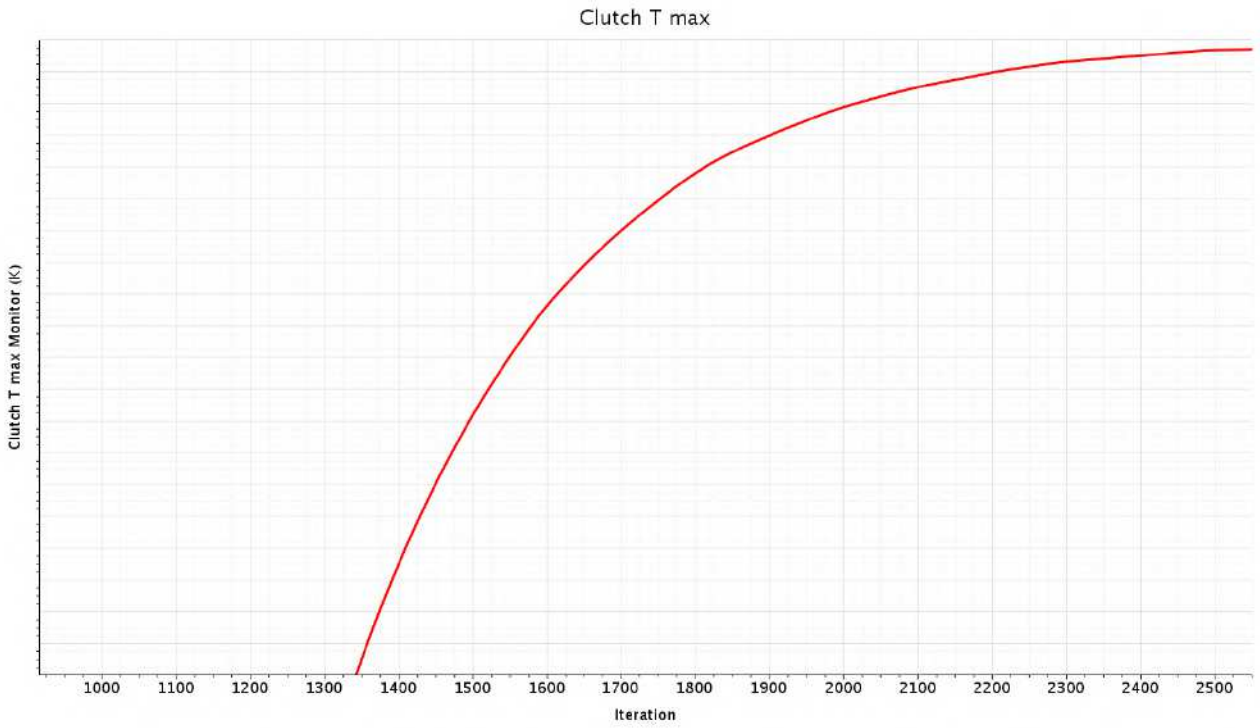

Figure 5.19: **CHT Approach.** Plot for maximum temperature reached on clutch disc. As in Figure 5.18, temperature stabilizes approaching 2500 iterations, starting from an initial condition of $300K$. T_{max} always remains under the critial value of $270°C$ we set at the beginning.

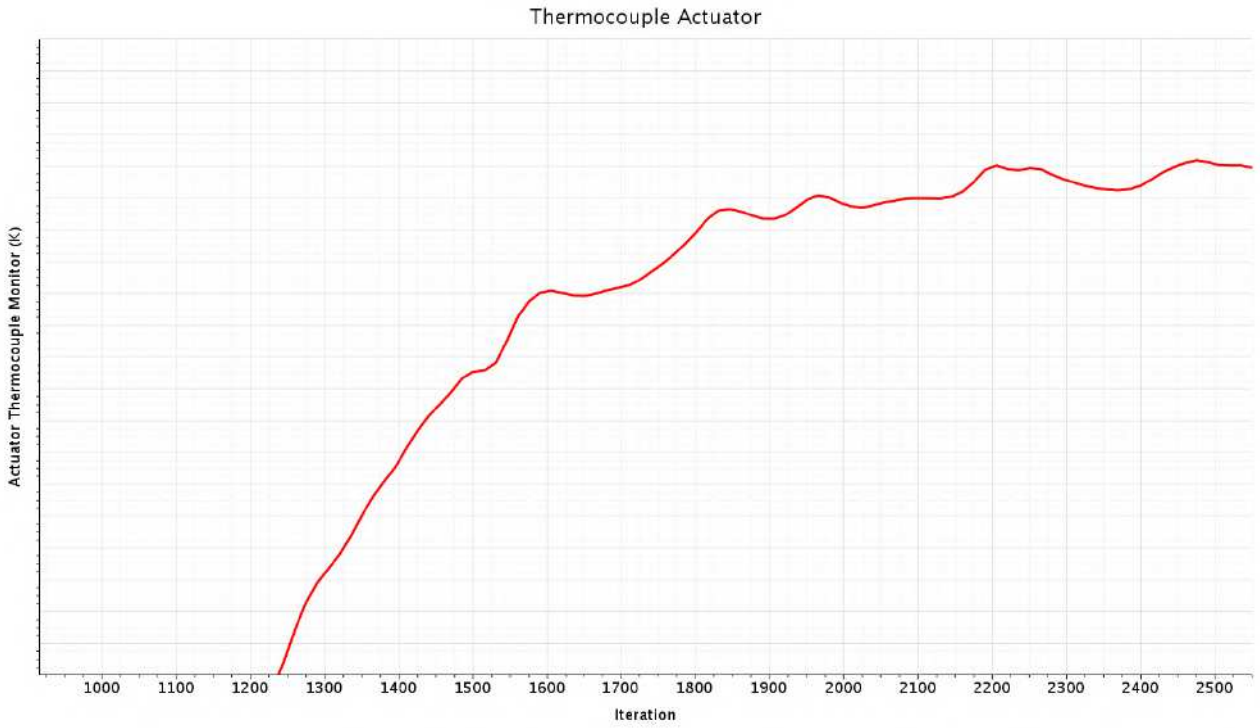

Figure 5.20: **CHT Approach.** Plot for temperature probe set on the clutch actuator. This value really approaches the experimental one.

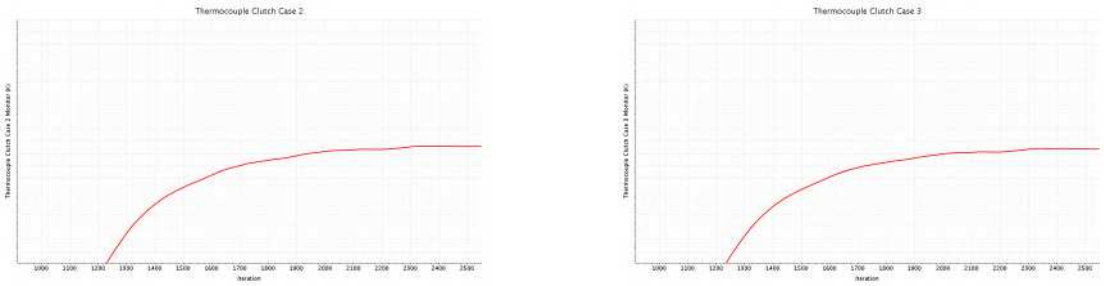

Figure 5.21: **CHT Approach.** Plots for temperature probes set on the clutch case. Their values are very similar to each other, but they remain significantly lower than the experimental ones. However, they show that the simulation is proceeding in the right way: corresponding thermocouples show very similar values to each other and probes actually reports equal temperatures.

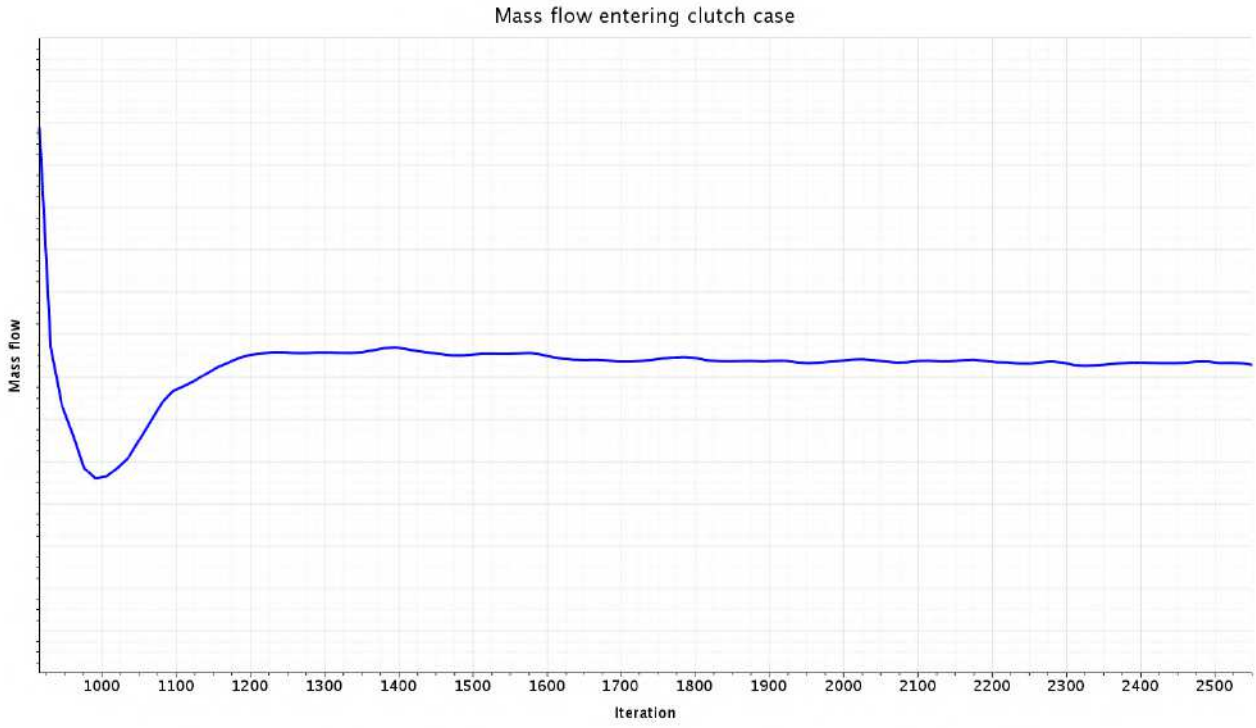

Figure 5.22: **CHT Approach.** Plot for mass flow entering (or leaving) the clutch case. Mass flow has been evaluated as $\Sigma(\Delta\dot{m})$, where $d\dot{m} = \rho|\vec{v}|dS$. Each cell contribution is taken despite of its sign (i.e. inward/outward). Since there are not mass sources inside the clutch case, the obtained result is halved (half enters, half exits). This value is important in monitoring if convective heat exchange may improve or not.

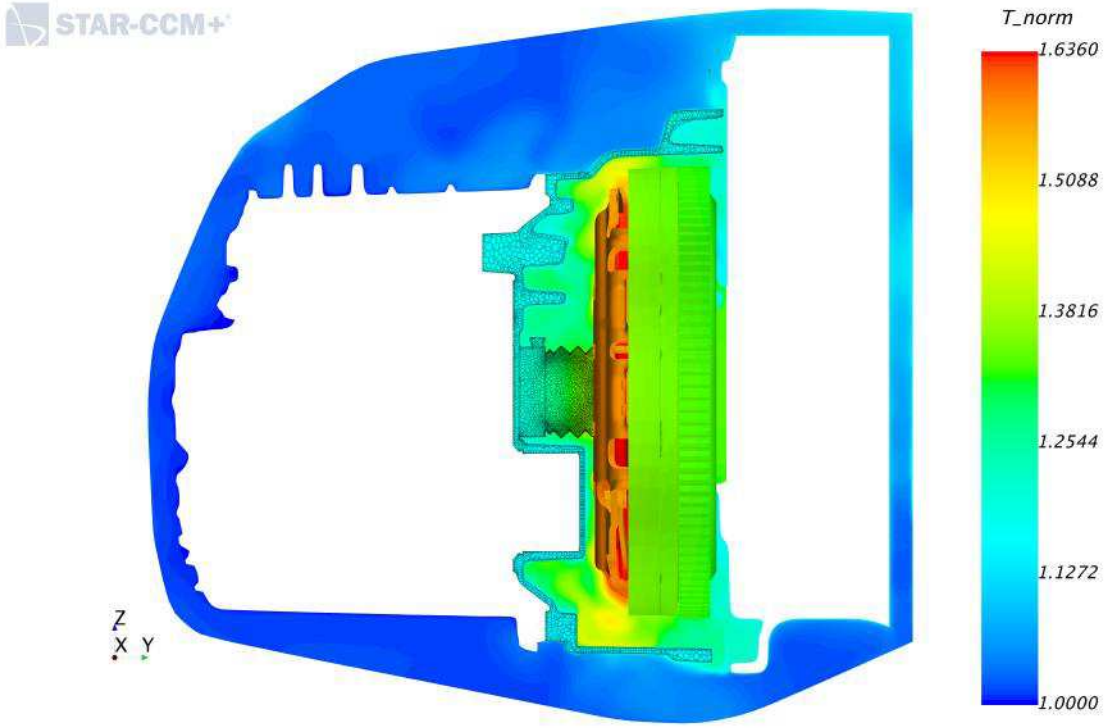

Figure 5.23: **CHT Approach.** In this scalar scene normalized temperature is displayed. It can be seen how the pressure plate is the more critical element, while the flywheel (with higher heat capacitance and area) manages to stay cooler. The clutch bell, as the crankshaft and the actuator, are highlighted thanks to the mesh representation on them.

5.5.7 Comments on sensitivity

Careful parameters selection allows to obtain numerical data that is tending to agreement with experiments. Such parameters include materials thermal properties, clutch thermal power output, and "far field" boundary conditions.

About "far field" boundary conditions, we remind that they are extracted from a complete car simulation running with the same average speed and external temperature evaluated during the test (Section 5.3.1). A sensitivity investigation on these values is carried out. The complete simulation described in Section 5.3.1 has been repeated with different far field conditions: the car runs from $6\frac{km}{h}$ to $50\frac{km}{h}$ at a temperature from warm to hot summer day. Six different combinations are therefore tried. The most severe one, as long as maximum clutch temperature is concerned, is the run at $6\frac{km}{h}$ with external temperature of $40°C$. It is noticed that the motion field in the purple area of Figure 5.14 is completely dominated by the radiator fan exit speed, and the temperature field is dominated by engine and clutch heat generation. In Figure **??** a qualitative comparison is provided. The conclusion is that the reduced simulation is not very much affected by car running conditions, therefore speed changes during experimental tests should not be considered an issue.

Clutch thermal power output is the greatest source term in the reduced simulation. Only two different values are tried: a telemetry value of $600W$, and a theoretical value of $680W$ (+13% added to telemetry value). Evidence is that special care has to be given when setting this value, since the simulation is mostly driven by that. However, virtual thermocouples show, with $680W$, an increase of only 5%, while T_{max} on clutch disc increases by 10%.

Oil temperature inside upper gearbox, set as temperature of fluid boundary, greatly influences clutch case temperature, thus probing results. In details, "Thermocouple Clutch Case 2" and "Thermocouple Clutch Case 3" mostly depend on such oil temperature, which has been evaluated with standard procedure, and modelled without proper details, likely causing inappropriate results. Also, convective coefficients (h) set in simulations should be properly evaluated. Further work on such modelling is required.

Radiation may be implemented with StarCCM+ model, its impact remains anyway really negligible (but it has the drawback of heavy slowdown in computation process).

Finally, thermal conductivity is the most important material parameter, and should be set with scientific precision. Probing is influenced by material parameters as much as it is by clutch power output. Unfortunately, clutch discs supplier does not state exact disc composition, forcing us to set a first guess value estimating material composition and final overall thermal conductivity, as explained in Section 5.5.4.

5.5.8 Cooling Improvements

Due to time matter, we will not be able to test improvements in clutch case geometry or materials to better manage heat. However, we try here to spread some light on the problem basing on what we have seen so far. We found out specially that air is overheating inside the case and few vents are provided. We expect that if flow speed increases in this area, then Re increases too and Nu with them (Nu is ratio between convection and conduction, and it is proportional to Re). As a consequence, convection coefficient increases with speed. Furthermore, with more ventilation, more fresh air is provided and ΔT is higher, thus enhancing heat exchange both with a higher ΔT *and* a higher h. Air flow may also be amplified equipping the pressure plate with blades that behave like a pump. Adding mass to improve heat capacitance is excluded, as we are aiming to reduce drivetrain mass.

Table 5.3: Comparison between real and simulated thermocouples output. Results are normalized with real test values because of secrecy reason. Relative variation is provided as $\Delta T_{rel} = \frac{T_{num} - T_{exp}}{T_{exp}}$

	ΔT_{exp}	ΔT_{coupl}	ΔT_{CHT}
T clutch case 2	1	+22%	+34%
T clutch case 3	1	+22%	+20%
T therm actuator	1	+8%	−1%

5.6 Thermal coupling Approach

Now we analyse the thermal coupling approach. The computational domain remains the same, but now software involved are ANSA, StarCCM+ and TaiTherm. StarCCM+ performs $(h, T)_{fluid}$ calculations while TaiTherm computes T_{walls} in solid regions. Details are explained in Section 5.1.

5.6.1 ANSA pre-processing

ANSA pre-processing is essentially identical to the one done for the CHT approach. The only difference is that components belonging to solid regions are not modelled inside StarCCM+, thus they must not be closed: TaiTherm handles only shell meshes. However, TaiTherm, as each FEM software, is more comfortable with quad elements, while StarCCM+ prefers tria. The geometry has been remeshed to be tessellated with quad before being fed to TaiTherm. Furthermore, being the clutch disc completely surrounded by solid regions, does not exist any more in StarCCM+ fluid region: the heat source is present only during TaiTherm iterations. This slows convergence, but it should not affect accuracy.

Where components should be piled up, contact is missing in TaiTherm: a thermal link is added in order to take into account conduction between them.

5.6.2 Results

Since volume mesh in StarCCM+ is now quite reduced (the total cell count is about $\frac{2}{3}$ of the CHT one), calculation is faster: TaiTherm iterations are very fast compared to StarCCM+ ones. In order to reach global convergence, 40 cycles between StarCCM+ and TaiTherm are required. Results are reported in the following, along with a comparison with CHT approach (Table 5.3). Comments can be found in captions (Figure 5.27 to Figure 5.33). The following Section provides also a critical comparison between the two approach presented so far.

T_{max} on clutch disc is not reported because it cannot be probed in StarCCM+: solid regions do not exist any more. However, from TaiTherm, it is found that this time T_{max} is about 10% lower than the temperature found with CHT approach.

5.6.3 Comparison with CHT Approach

Some comparison comments have already been written in captions. We report them here anyway.

Radiation seems to have a much stronger impact on wall temperatures in the thermal coupling case, due to TaiTherm radiation models. However, since components out of the reduced subdomain where not accounted for as long as radiation is concerned, gearbox and clutch assembly only see a heat flow departing from their outer surfaces, and no heat flow incoming. Radiation has a cooling contribution, and outer surface temperatures are smaller in the thermal coupling case.

Dependence on "far field" boundary conditions, as well as clutch thermal power output, results to be the same both for CHT approach and thermal coupling approach.

Shell meshes have been given a constant thickness. Components thickness was measured in some locations and taken the average. The complex geometry makes that a strong approximation: ribs and bolts (for example) have a very different thickness value from the average one, but they were not accounted for (no easy way to perform the calculation was found). Therefore, their heat capacitance contribution has been neglected. In a first approximation, we thought we could neglect their presence without many drawbacks. Actually, a cause for higher temperature value in inner components (compared to the CHT approach) may be really the lack of heat sink effect, as mass is numerically approximated.

We should also report that the test thermocouple has been placed next to the surface, but not drowned in it: it always sense the air temperature. Therefore, the probe in virtual simulations is set to record only air temperature (and its position is about half a millimetre over the walls). However, thermal boundary layer shows steep temperature gradients, so even a tenth of millimetre of position displacement may result in unexpected results (Figure 5.34). Due to thermocouple dimensions (about $1mm$ at its edge), it is reasonable suppose that it did not resolve boundary layer temperature, which has a (simulated) thickness of $0.3mm$. Therefore, we expect the probe is better placed immediately outside the thermal boundary layer, where temperature gradients stabilizes. Another consequence is that post-processing should be done in StarCCM+ environment, where the air temperature is accounted for. If we, in TaiTherm, record the temperature on the same element which the probe is set above, we would very likely get a completely different value because of thermal boundary layer presence. Air is a very insulating material, thus thermal boundary layer is thin. Wall temperature may be very different than the air temperature just $1mm$ above it. In conclusion, special care should be spent during thermocouples placement too, in order to be sure of what we are measuring (inside/outside the boundary layer).

5.7 Complete and Detailed Car Simulation

As the reduced simulations got clear enough, we noticed that outer components temperature (i.e. gearbox and clutch case) still have a remarkable disagreement with experimental data (see Table 5.2). Furthermore, we have no way to state surely enough if far field boundary conditions set on that reduced subdomain are realistic or not: at the time we assumed that what happened inside the properly detailed subdomain did not affect external flow field. This assumption may be too strong, as flow field modifications may origin from new simulated phenomena, like flywheel and pressure plate rotation, as well as in buoyancy forces arising from high disc temperature.

We also tried to set different far field conditions, emerging from different flow conditions in the complete car simulation (Section 5.3.1), and there we did not notice appreciable differences (Section 5.5.7). However, we do not want to lay on such assumption too much and we seek further confirmation by carrying out a complete and detailed car simulation. Now that the methodology has been routed, it is relatively fast to pre-process such simulation. We decide to perform the computation with the already seen thermal coupling approach, because simpler in pre-processing (and the standard in FCA procedures).

We report comparison with the reduced simulation at the end of this Section. That will teach us if we can trust reduced simulation output (thus saving much time and computational effort) or we will have to run heavy complete simulations in order to investigate clutch thermal behaviour.

5.7.1 ANSA pre-processing

Pre-processing in ANSA environment concerns only some hours in merging the detailed clutch assembly with the rest of the car. Conceptually, there is nothing new. As we did for the reduced simulation with thermal coupling approach, the finite elements car representation is provided by

Table 5.4: Comparison between real and simulated thermocouples output. Results are normalized with real test values because of secrecy reason. Relative variation is provided as $\Delta T_{rel} = \frac{T_{num} - T_{exp}}{T_{exp}}$

T	ΔT_{CHT}	ΔT_{coupl}	$\Delta T_{complete\ \ simple}$	$\Delta T_{complete\ \ detailed}$	Test
T clutch case 2	+34%	+22%	+73%	−29%	T_1
T clutch case 3	+20%	+22%	+70%	−32%	T_2
T clutch actuator	−1%	+8%	+50%	−40%	T_3

shell meshes. Though the starting geometry is the same, we meshed with tria vehicle components involved in CFD computation, while components involved in thermal calculations are meshed with quad. Since most of the geometry is present in both StarCCM+ either TaiTherm, we did perform twice the meshing procedure.

As we are not looking for car aerodynamic coefficients, cells can have a relatively large size. Just to give an order of magnitude, the total cells count in the CFD-side of this simulation is the same as the one of an usual aerodynamic simulation with FCA standards, despite the fact that we introduced, here, a highly refined volume around drivetrain assembly. The important quantity to collect is the convective coefficient h. Wakes and eddies do not affect underhood thermal behaviour. It would result in no advantage simulating them in such a simulation.

5.7.2 Comparison with Reduced Simulations

The comparison is made between the reduced computation and the complete (and detailed) car simulation. The aim is to show if the reduced simulation may be used to investigate clutch thermal behaviour, or our assumption on its far field BCs was so strong that a complete simulation is unavoidable.

We already reported our concerns about the issue: we did not consider the effects of clutch presence on surrounding motion field and temperature field. Their data was mapped once (from a clutch disc-missing simulation) and assigned as (constant) far field boundary conditions on the reduced subdomain envelope. However, if new phenomena are strong enough to modify those fields, reduced simulation results may be less accurate and we would have no way to notice it. The only way is, as said, a direct comparison between reduced and complete simulation.

Quantities involved in comparison are those assigned as far field boundary conditions in reduced simulations: $\vec{v}$, p, T fields collected over the subdomain envelope. The comparison is carried out only qualitatively: scalar scenes of such data are visually inspected to seek mismatches between *colours* on the reduced simulation envelope and the same envelope in the complete simulation. If remarkable differences are pointed out, then we should look at comparisons between other quantities (thermocouples data, mass flow entering the clutch case, and others) in order to declare the reduced simulation suitable or not for clutch thermal behaviour investigation. On the other case, if fields are similar, then a long runtime complete simulation may be avoided.

As long as quantitative data is concerned, it is clear that further development is required in order to give some utility to the complete and detailed simulation (see Table 5.4). As it is the longest simulation to run, we would expect better results before inserting it in FCA procedures. We have also to report that special care is needed when measuring temperature among clutch case. There, as it can be seen in Figure 5.36, the thermal boundary layer is about $2mm$ thick, while the probe has an edge thickness of $1mm$. It is likely that it detected an average temperature across the boundary layer. We tried to take account of it by averaging ourselves probing along it. We may think that in future tests thermocouples shall be simulated in the virtual environment too, with a proper geometric and probing model, in order to be sure that what we are detecting numerically and experimentally is the same.

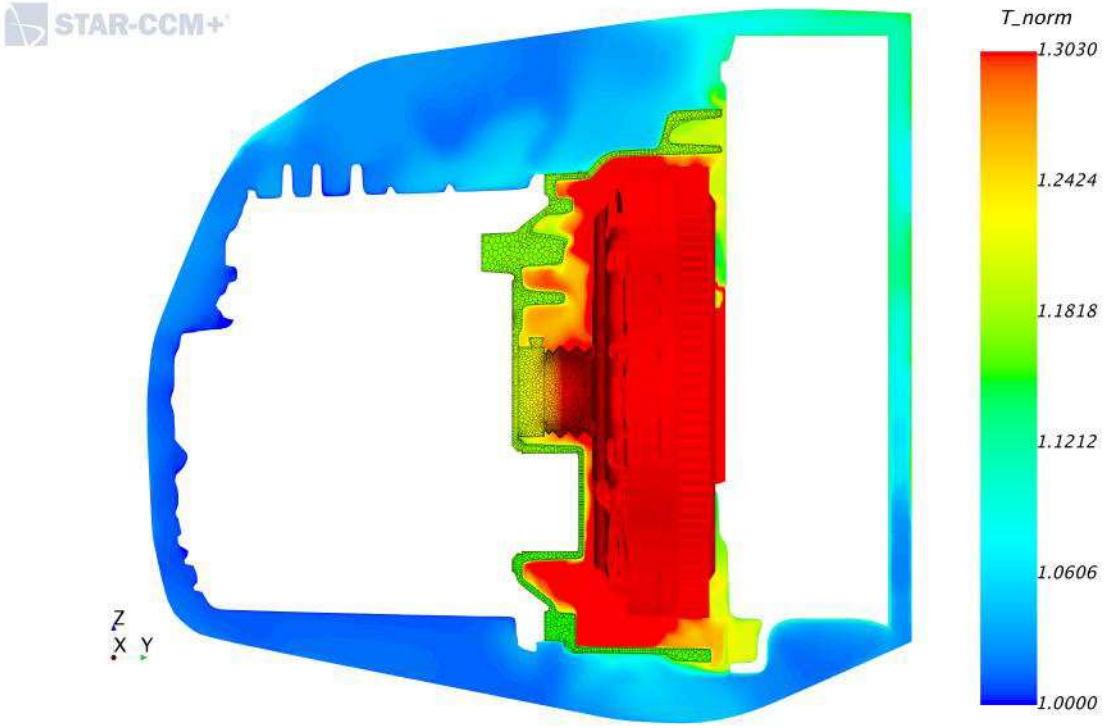

Figure 5.24: **CHT Approach.** Here a focus on air temperature is displayed. With hot air, as it is close to the pressure plate, heat exchange with metal is reduced. There are few airflow vents, namely at the ends of the case, where hot air mixes with surrounding cooler air. Air speed is reported in Figure 5.25.

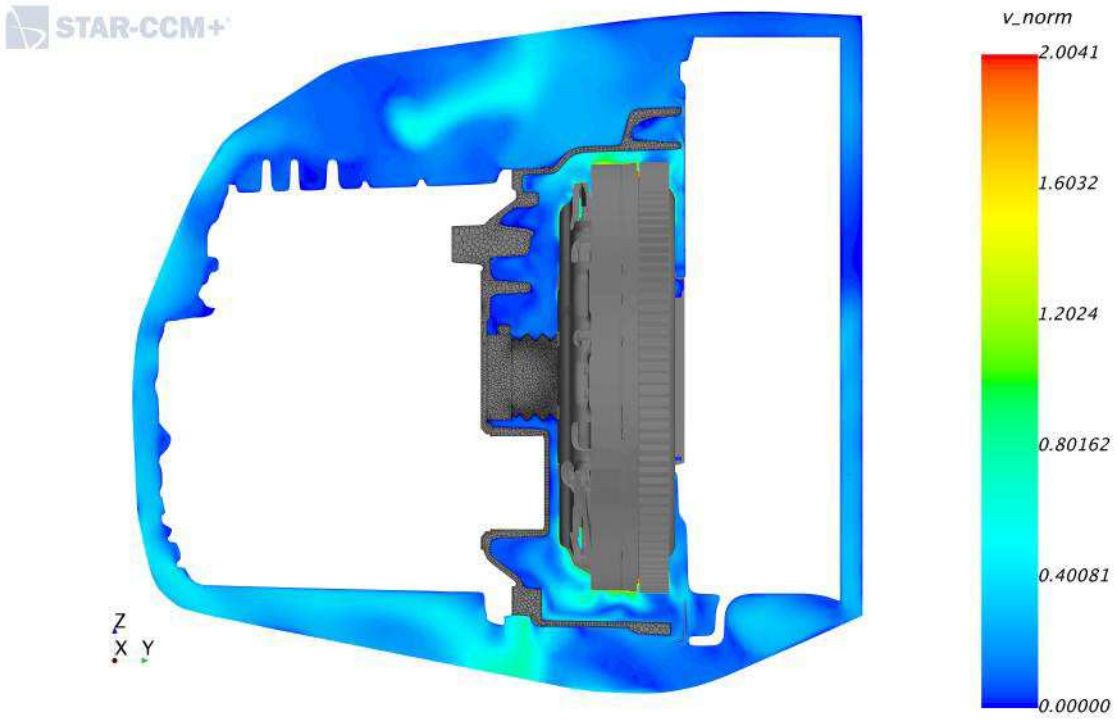

Figure 5.25: **CHT Approach.** Air velocity magnitude (normalized). The flow is generally slow, and convection is natural. Speeds are much higher inside the boundary layer around the rotating components: they are the only responsible of forced convection. Air-metal natural convection heat exchange on other surfaces provide a really small contribute if compared with metal-metal conduction. On the other hand, air-metal forced convection exchange is similar to metal-metal conduction (this result is explained in Section 5.6.

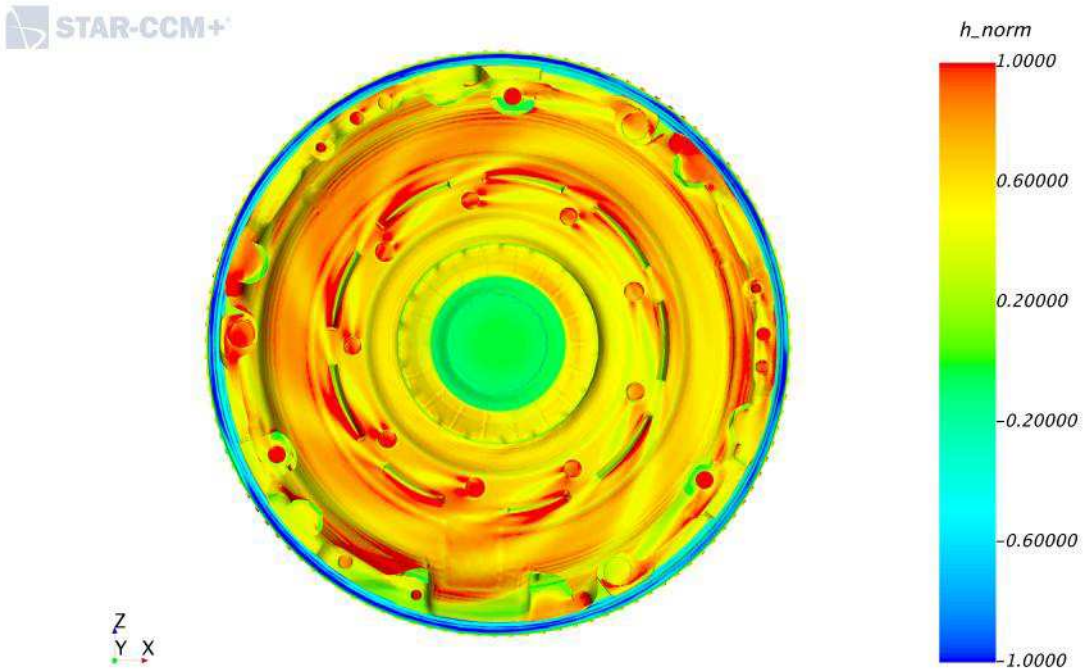

Figure 5.26: **CHT Approach.** Scene displaying convective heat transfer coefficient h (normalized) on the pressure plate. It can be noticed the wake behind bolts and gaps that reduces heat exchange because of recirculating and slower flow. The heat propagation direction is such that where the figure is red air is warming and viceversa. The flywheel, with its mass, is heating slowly: hot air actually warms it, acting as heat sink throughout the test.

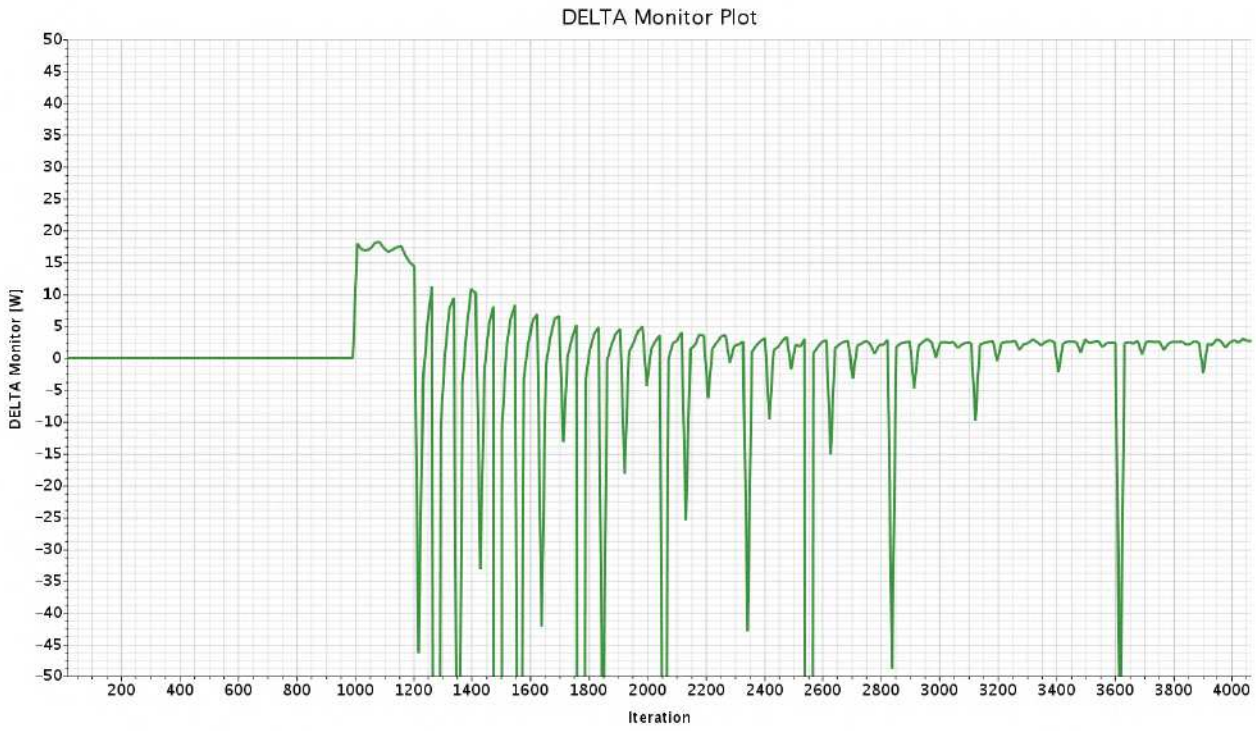

Figure 5.27: **Thermal coupling Approach.** This time the plot for Δq show an unusual trend: peaks are caused by coupled iterations between StarCCM+ and TaiTherm. When new temperature values are imported from TaiTherm the solution undergoes a brief periodo of consolidation. In any case, it asimptotically tends to 0, as it should.

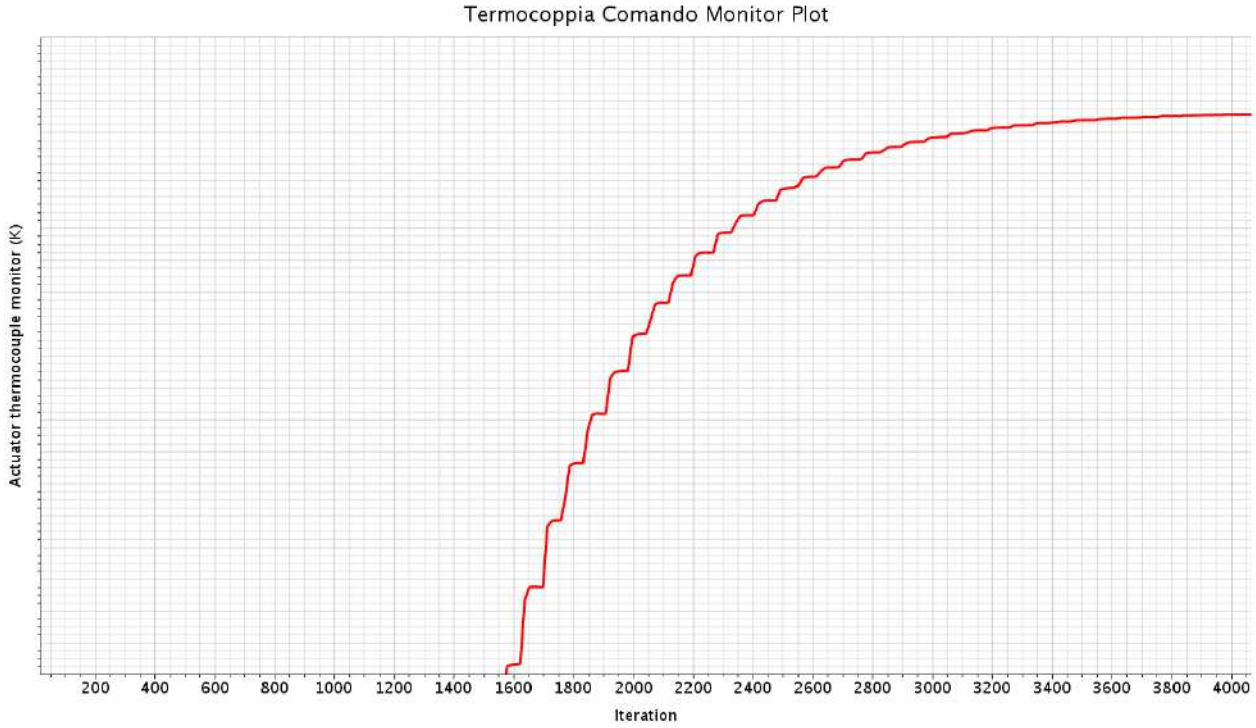

Figure 5.28: **Thermal coupling Approach.** The "stepping" trend is seen also in this plot for actuator temperature. The final value is slightly higher than the experimental one (8%), but there is not much concern. It can be seen that, even if temperature stabilizes inside each StarCCM+ run, it does not globally. Many coupled iterations are launched in order to reach proper equilibrium.

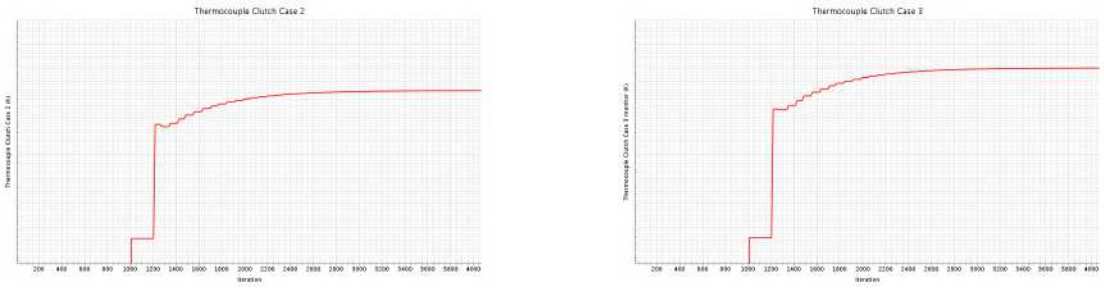

Figure 5.29:
small **Thermal coupling Approach.** Plots for temperature probes set on the clutch case. Their values are very similar to each other, but they remain even lower than the experimental ones. We suppose that radiation gives a cooling effect to gearbox case.

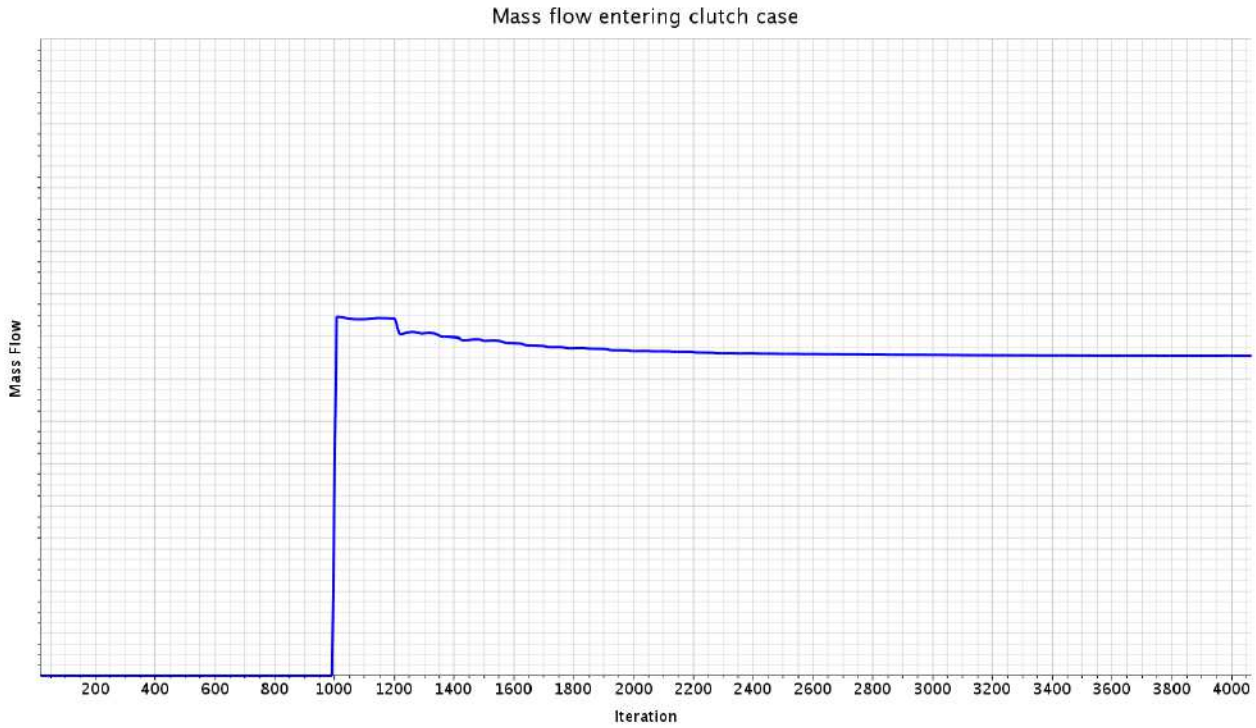

Figure 5.30: **Thermal coupling Approach.** Plot for mass flow entering (or leaving) the clutch case. The scale, though normalized, is the same of Figure 5.22. Both results are quite similar (they are computed in the same way), meaning that the flow field is quasi-identical with both approaches.

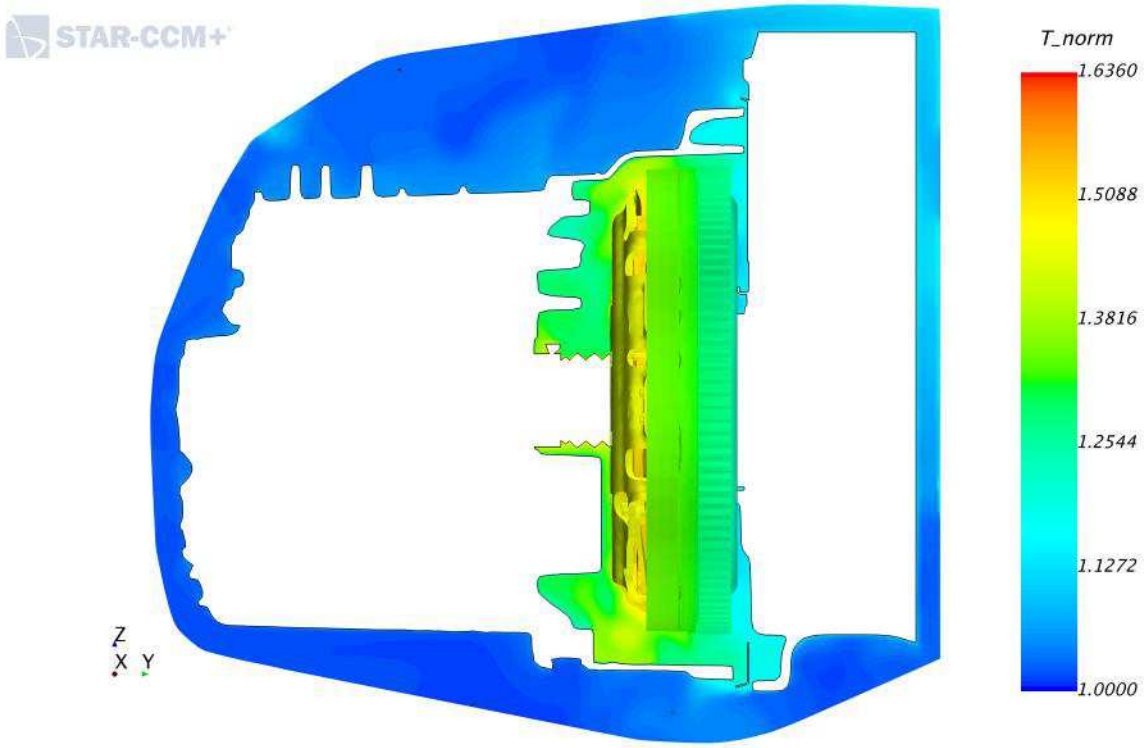

Figure 5.31: **Thermal coupling Approach.** In this scalar scene normalized temperature is displayed. The pressure plate, if compared with Figure 5.23, is quite cooler. We interpret this fact with the cooling contribution from radiation.

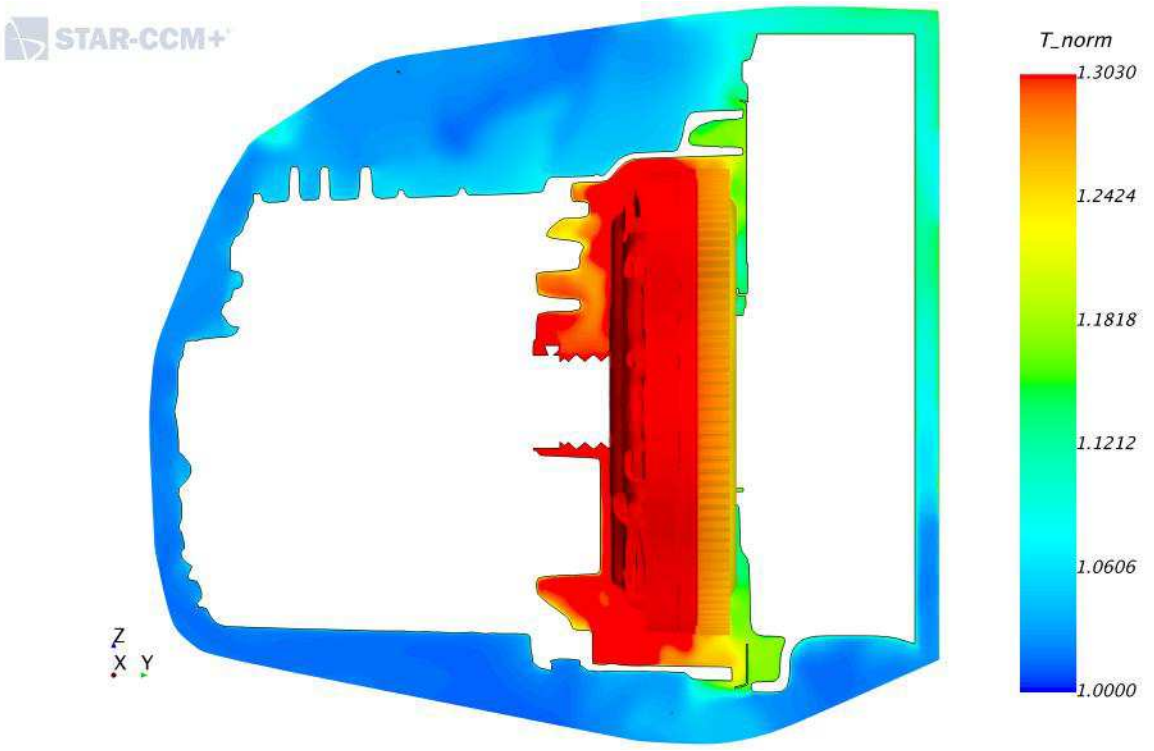

Figure 5.32: **Thermal coupling Approach.** Here a focus on air temperature is displayed. Another proof that radiation actually cools components is found on the flywheel: it is cooler with this approach, while the air (which does not participate in radiation path) has the same temperature distribution if compared with Figure 5.24.

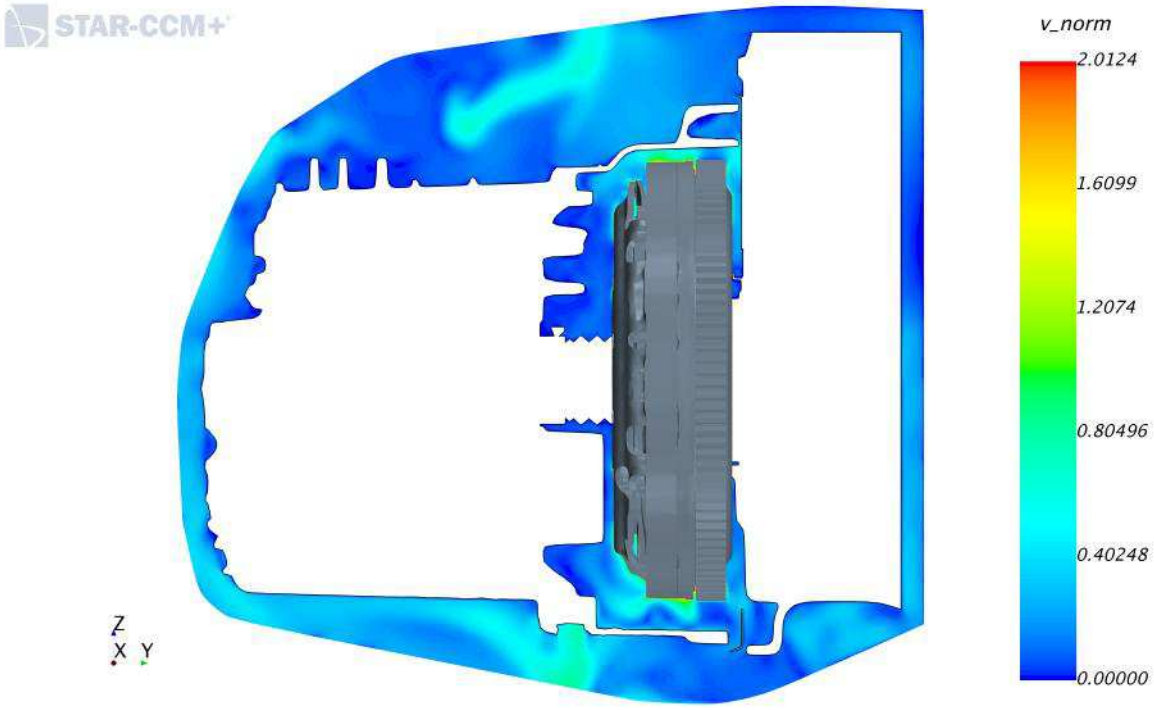

Figure 5.33: **Thermal coupling Approach.** Air velocity magnitude (normalized). Flow field is very similar with both approaches.

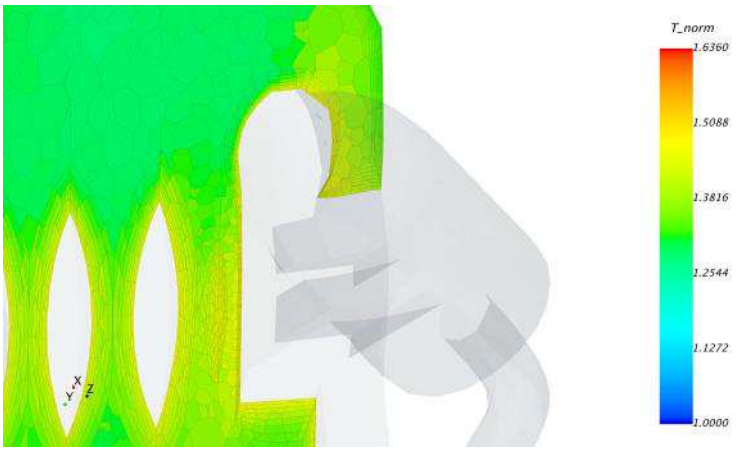

Figure 5.34: **thermal coupling Approach.** A focus on thermal boundary layer. Prism layers total thickness next to the probe position (highlighted in red) is $2mm$, while the boundary layer itself is about $0.3mm$.

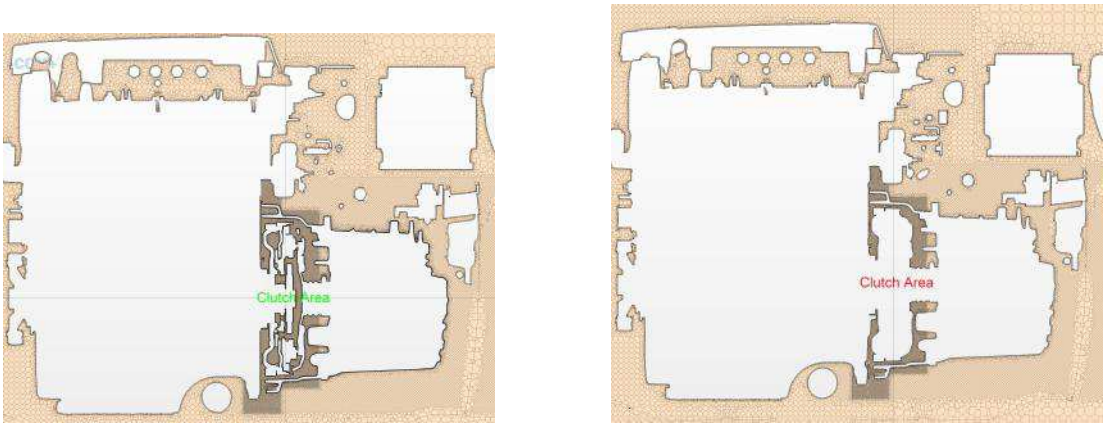

Figure 5.35: Fluid region volume mesh representation. On the left, the clutch area properly detailed is pointed out. Many components are missing on the right hand picture, which is a mesh scene from the simplified car simulation described in Section 5.3.1.

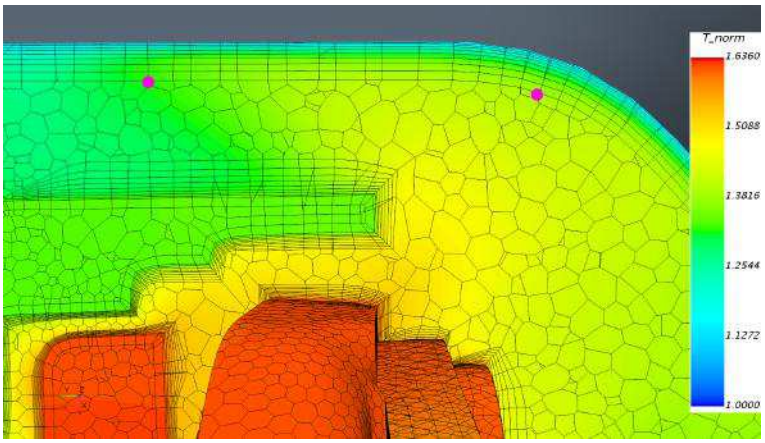

Figure 5.36: **CHT Approach.** A focus on thermal boundary layer. Thermocouples are placed on the external edge of the thermal boundary layer, whose thickness is about $2mm$.

Chapter 6

Conclusions

A new methodology to predict clutch thermal behaviour using virtual simulations in a reduced domain has been found. This will allow engineers to foresee in early design phases if clutches will perform properly in all environments, even under heavy duty cycles. Direct consequences of that will be improved clutch design and drivetrain efficiency (and reduced CO2 emissions).

We believe that the reduced subdomain with both CHT either thermal coupling approach is to be preferred. Methodology critical issues have been found in correctly evaluating clutch thermal power output and setting exact material properties, specially clutch disc properties, whose values are not so easily accessible. Using meaningful values, a satisfactory agreement between numerical and experimental data is found. Radiation seems not to have strong influence on results (at least on reduced model). The thermal coupled model is to be preferred when shorter runtimes and easier setup are required, keeping in mind that about 10% error is introduced on best matching probe (see Table 6.1). Even though underhood components surrounding the reduced domain are taken into account, their radiation contribute is neglected.

Being the clutch assembly almost completely surrounded by its case, what happens inside the case is not affected by what happens outside, so that environmental conditions (T_{inf}, v_{inf}) have a marginal impact on temperature field. Convection is completely dominated by clutch spinning, and it offers a 50% contribution to heat exchange. The other half is provided by conduction; radiation play a much smaller role in cases studied.

Complete vehicle simulations, detailed or not, are still requiring further development to reach reliable results as far as clutch thermal analyses are required. The complete though simplified simulation is a must since it provides mapped data to set as far field BC on the reduced subdomain simulations. We expect that the best procedure is really to run one single complete and simplified simulation (which is usually already available thanks to other underhood thermal analyses), collect and store data, and then carry out several reduced simulations to analyse different engineering solutions. These reduced simulations, in fact, provide simple means to switch clutch materials either substitute different geometries for clutch housing.

We also found out during this Thesis work that StarCCM+ is suitable for CHT simulations. Involving only one, finite volumes software we are sure that we are modelling components geometries as they are, but drawbacks are found in difficult and long preprocessing and increased

Table 6.1: Cost comparison between different approaches.

Mesh	**Complete simplified**	**Complete detailed**	**Reduced subdomain with thermal coupling**
StarCCM+ side	67 milion	84 milion	20 milion
TaiTherm side	2 milion	2.4 milion	400.000
Runtime	18h	28h	4h

runtime. The thermal coupling should be chosen if time is a strict requirement.

Further works will try to solve presented issues: experimental tests on clutch discs may be scheduled to find thermal conductivity. Analysis of telemetry data, from test described in Section 3.3, may give more precise thermal power output. Phenomena inside gearbox, which strongly affect clutch case temperature, are going to be studied deeper.

Bibliography

[1] Awrejcewicz J., Grzelczyk D., 2013, *Modeling and Analysis of Thermal Processes in Mechanical Friction Clutch - Numerical and Experimental Investigations*, International Journal of Structural Stability and Dynamics, Vol. 13, No. 7. DOI: 10.1142/S021945541340004X

[2] Czel B. et alii, 2009, *FE Thermal Analysis of a Ceramic Clutch*, Tribology International, Vol. 42, pp 714-723

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

[4] Abdullah O., Schlattmann J., 2014, *An Investigation Into the Thermal Behavior of the Grooved Dry Friction Clutch*, Journal of Tribology, Vol. 136

[5] Wittig S. et alii, 1998, *Numerical Study for Optimizing Heat Transfer in High Speed Rotating Components*, International Journal of Rotating Machinery, Vol. 4, No. 3, pp 151-161

[6] Vercelli, Pignone, 2010, *Motori ad alta potenza specifica*, Seconda edizione, Nada Editore, Italia

[7] Psylaki P., Pantasopoulos G., Karaiskos P., 2012, *Failure Mechanisms of an Automobile Clutch Assembly Cast Iron Pressure Plate*, Journal of failure analysis and prevention, Vol. 12, pp 16-23, DOI: 10.1007/s11668-011-9523-3

[8] Jacazio G., Pastorelli P., 2001, *Meccanica delle Macchine*, Prima edizione, Levrotto E Bella Editore, Italia

[9] Wilcox D., 2008, *Formulation of the $k-\omega$ Turbulence Model Revisited*, AIAA Journal, Vol. 46, No. 11, DOI: 10.2514/1.36541

[10] Cengel Y., Cimbala J., Turner R, 2012, *Fundamentals of Thermal-fluid Sciences*, Fourth Edition, McGraw-Hill, Singapore

[11] Pope S., 2015, *Turbulent Flows*, First Edition, Cambridge University Press, United Kingdom

[12] Bosch, 2014, *Automotive Handbook*, Sixth Edition, SAE, USA

[13] Anderson J. jr, 2010, *Fundamentals of Aerodynamics*, Fifth Edition, McGraw-Hill, USA

[14] Siemens, *StarCCM+ Documentation*, 12^{th} Edition