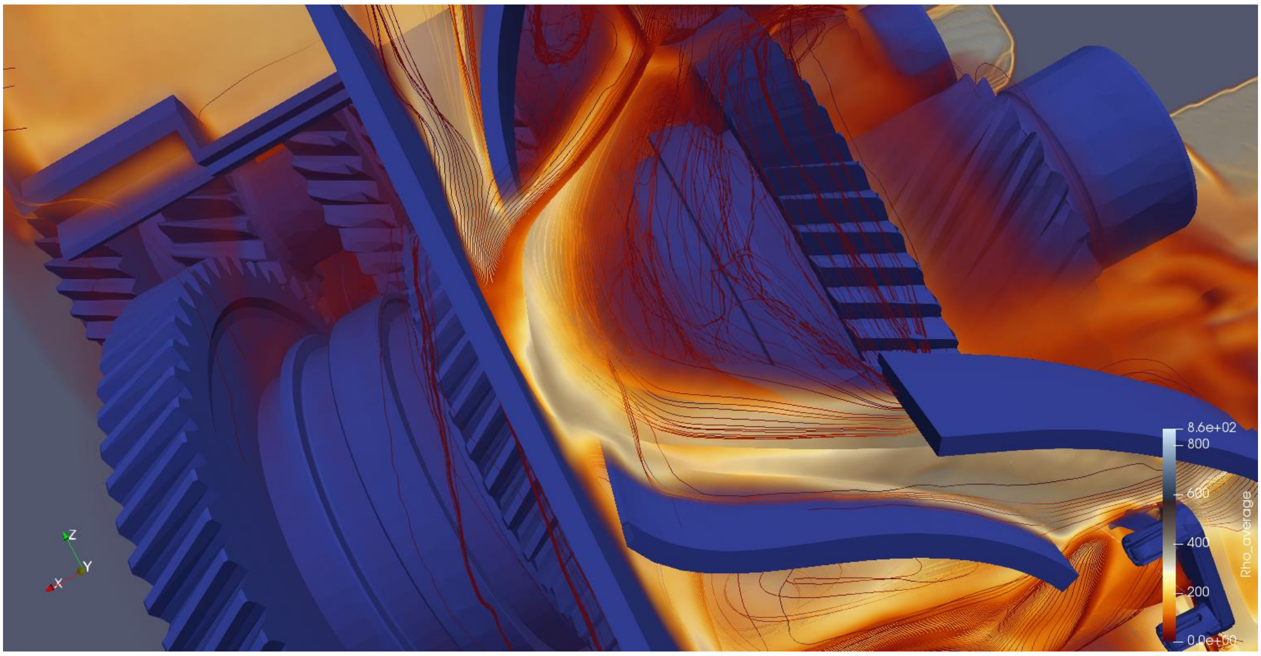


7DCT Transmission efficiency optimization

Design and simulation of oil traps that reduce the churning losses

Master's thesis in Automotive Engineering

Birk Forsström & Matus Minar

Department of Mechanics and Maritime Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2020

MASTER'S THESIS IN AUTOMOTIVE ENGENEERING

7DCT Transmission efficiency optimization

Design and simulation of oil traps that reduce the churning losses

Birk Forsström & Matus Minar

Department of Mechanics and Maritime Sciences
Division of Combustion and Propulsion Systems
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2020

7DCT Transmission efficiency optimization
Design and simulation of oil traps that reduce the churning losses
Birk Forsström & Matus Minar

Master's Thesis 2020:21
Department of Mechanics and Maritime Sciences
Division of Combustion and Propulsion Systems
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone: + 46 (0)31-772 1000

Cover:
Streamlines of oil through an early iteration of the new part

Department of Mechanics and Maritime Sciences
Gothenburg, Sweden

7DCT Transmission efficiency optimization
Design and simulation of oil traps that reduce the churning losses
Master's thesis in Combustion and Propulsion Systems
Birk Forsström & Matus Minar
Department of Mechanics and Maritime Sciences
Division of Combustion and Propulsion Systems
Chalmers University of Technology

Abstract

The automotive industry is undergoing a big change when switching from building combustion engine cars to electric powered cars. With the big limitation of range that comes with using battery to store energy there is an even greater need in finding ways to improve the efficiency. Together with CEVT (China Europe Vehicle Technology), this master's thesis purpose is to make an improvement of the gearbox that lower the torque that is required to rotate it. To verify the results a prototype is built with materials available in the company's workshop and investigated if it can withstand a test in a testing rig.

With the use of CFD-analysis (Computational Fluid Dynamics) the lubricant of the gears is investigated through a simulation of a benchmark. The aim was to find areas where reduction of churning losses can be made with the use of a new part that is created in CAD (Computer Aided Design). Inspiration for these improvements is found in previous studies made by employees at CEVT and at rival companies. The process of creating a new part is iterative and performed with rough simulations before a comprehensive simulation is made that can be compared with the benchmark.

Through the benchmark simulation it was concluded that the differential gear stands for most of the churning losses along with the output shaft above it. A new part was created that protects the top shaft from being hit by incoming oil transported by the differential, it also collects the oil in two containers for further redistribution. These changes resulted in a decrease in drag torque generated by the oil of 16.3% at 50km/h and a trend of larger reduction at higher speeds. For a prototype build, experimental tests were conducted on the ABS-plastic available at the company. It was concluded that the plastic would withstand the heat and oil inside of the gearbox and afterwards a prototype was built for future testing.

In the end of the study it was concluded that oil can be collected in the upper regions of the gearbox by taking care of the splash from gears. This along with protecting gears from splash that hit them in the opposite direction of speed can reduce the churning losses by around 16%. With this knowledge a future work would be to investigate how to distribute this oil from these regions to where it is needed. By doing so, future development of gearboxes can be made around how oil is being transported by gears to benefit from these possibilities.

Preface

This master's thesis project consisted out of designing an additional part to the existing lubrication system, simulating the performance of this part in Altair nanoFluidX and in the end, 3D printing of the new part and experimental tests. The main shareholder of this project is CEVT.

The first part of the project, design and simulations were carried out from January 2020 to March 2020. Afterwards, further simulations intended to finalize the design of the new part were performed from March 2020 until May 2020.

The 3D printing part of the project was carried out at CEVT, subsequently, preparations for experimental tests were made to be performed at CEVTs test facilities at Säve, Sweden.

Finally, it is worth noting, that the thesis could have never been performed without the tremendous amount of help and expertise of everybody involved.

Gothenburg 2020-06-02

Birk Forsström & Matus Minar

Acknowledgements

To do our master's thesis at CEVT have been an honour for us, all the support and help that we got made us feel included and like one in the team. We would first like to thank our supervisors Robert Johansson and Tommy Brandt at CEVT for creating this project. They truly made a great job with creating the project and helping us through it by sharing their expertise and introducing us to the right people.

We would also like to acknowledge and thank our examiner, Professor Sven B. Andersson at Chalmers University of Technology for taking on the responsibility of examine this master's thesis. He helped us to push through with deadlines and made sure that we did a correct academic report.

To all the people at the department of simulation engineers at CEVT, a very big thanks. This would never have gone as smoothly as it did without them. They helped us with all from computer problems to setting up the simulations, never that they hesitated when we asked them. For all the personal at the other departments, John the system engineer, Filip and Oscar the test engineers, Pedram that manages the 3D-printer and many others, we owe you a big thanks.

Nomenclature

CFD – Computational Fluid Dynamics
SPH – Smoothed Particle Hydrodynamics
ANSA – CEA pre-processing software
CAD – Computer Aided Design
CREO – CAD program
GPU – Graphic Processing Unit
CPU – Central Processing Unit
7DCT – 7 speed Dual Clutch Transmission
ANSYS Fluent – CFD simulation software
nanoFluidX – CFD simulation software
CFL – Convective Courant Number
FDM – Fused Deposition Modelling
ABS – Acrylonitrile Butadiene Styrene
7DCT – Transmission used in the project
PISO – Pressure-Implicit with Splitting of Operators
y^+ – Dimensionless wall distance
CEVT– China Euro Vehicle Technology
w – Kernel function
r – Radial distance
h – Height of the Kernel function
m – Weight of the particle
ρ – Density
α – Interaction between two phases
p – Pressure
μ – Dynamic viscosity
α – Volume fraction
VOF – Volume Of fluid
θ – Contact angle between fluid and a surface
u – Velocity
$F_{wall\_adhesion}$– Wall adhesion force

Contents

1 Introduction

1.1 Background

The seven-speed gearbox with the possibility of added hybrid drive, 7DCT (7 speed dual clutch transmission), developed by CEVT has a lubrication system that depends on splash from the gearwheels. This is called a wet sump lubrication system since there are no pumps that direct the oil to where it is needed. A master thesis project proposal was made, with an intention to investigate the 7DCT gearbox efficiency related to lubrication system and churning losses.

1.2 Goal Statement

The goal of this project is to see if it's possible to improve the efficiency of the 7DCT gearbox and then see if a prototype can withstand the environment of a gearbox.

1.3 Limitations

The project will work with a gearbox that is being supplied by CEVT as a virtual model, this is then analysed in a CFD software that is operated on two GPUs. The biggest limitation is expected to be the time available on the GPUs due to that CFD calculations are notorious to demand a lot of computing power. Accuracy of the software and methods used can be altered with in order to reduce the time needed on the GPUs but this creates another limitation regarding how well the simulations correspond with data from a testing rig.

Regarding the confidentiality of the values from simulations, the analysis will be limited to comparing a percentage. 100% is a value picked from the benchmark simulation.

2 Method

This project can be classified into a few sub-systems as shown in Figure 1. Each of these sub-systems has a certain input and a certain output of deliverables to the rest of the project.

The process begins with the idea of optimizing the efficiency of the 7DCT gearbox, to do so, a literature study is made in order to identify the losses in a gearbox and ways how to reduce them. Afterwards, an approach is developed regarding how this goal should be achieved.

This approach starts with using a CAD-model of the gearbox as an input, this is investigated with a CFD simulation in nanoFluidX. With help from the literature study a new part is created and then simulated in order to be compared to the benchmark simulation. This process is repeated until satisfactory results in terms of churning losses reduction is achieved. Additionally, in the iterative process, simulations are made in ANSYS Fluent to analyse and dimension holes that are used to discharge oil from containers.

When satisfactory results in the simulations are achieved, the new part is put through a process of preparing it for an experimental test. This begins with designing it for 3D-printing where the new part is divided into pieces that can be printed, installed in a gearbox and survive the environment of heat and oil exposure in the gearbox. A prototype is then manufactured.

In the end, the results from investigating a possible experimental test along with the simulations will be analysed, compared and presented in the report.

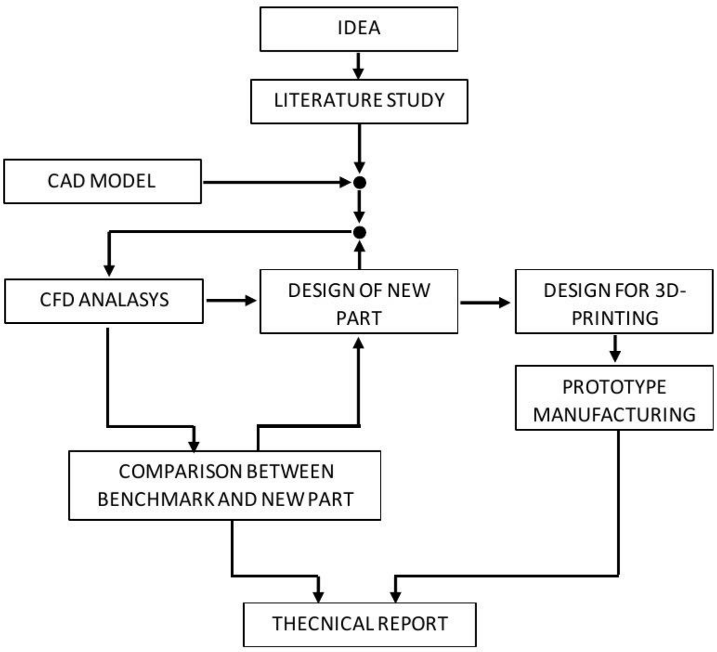


Figure 1 Flowchart of work

3 Literature Study

A literature study is made to support the approach and results of this project, here the studied papers are presented along with the findings.

3.1 Different types of losses in transmissions

According to a study made by Zhou [1], there are five types of losses in a DCT; gear related meshing, windage and churning losses, bearings and oil seal related losses, concentric shaft related viscous shear losses and disengaged wet clutch caused drag torque losses. It would however be too much for this project to look at them all due to that the model would be too complicated for the timespan of this thesis. When narrowing down the losses to the ones that are considered in this project the ones depending on mechanical friction, thus dependent on the load, were quickly eliminated. This due to that the program used cannot simulate this and that it's not a part of the question answered in this project. Therefore, gear related meshing, bearings and oil seal related losses are eliminated in the simulations. The concentric shaft related viscous shear losses are depending on the oil film between input shaft one and two because of that one goes inside of the other. The size of the particles in the simulations cannot capture this small oil film and are therefore eliminated. Last one eliminated is the losses in disengaged clutches so the model becomes simpler and have more focus on the gears.

In a study made in corporation with BMW [9] the concept of how oil hits a moving gear is explained. According to this study a wet sump lubrication generates more resistance then a spray lubrication, a conclusion is drawn from this that lower oil level corresponds to lower drag torque.

Spray lubrication is further divided into four different types [9] illustrated in Figure 2. Spray that hits the approaching contact areas generates more losses then when hitting regressing contact areas, i.e. BO and BU generate more losses then AO and AU. In the regressing case, AO and AU, the centrifugal force makes the oil film very thin compared to the approaching case in BO and BU. Therefore, less oil must be squeezed out from in between the gear teethes in AO and AU.

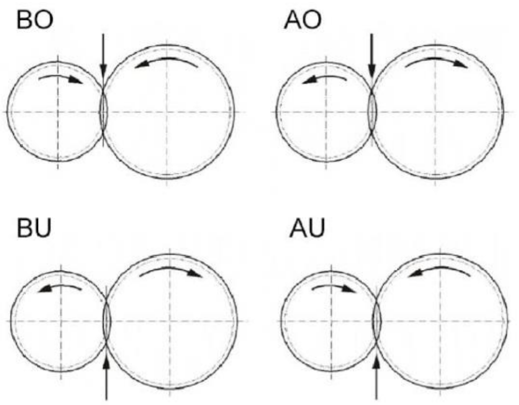


Figure 2 Different cases of spray lubrication [9]

The study [9] also explains that more losses are generated when the spray hits the gears from above due to the added force of gravity, AO compared to BU. Less acceleration of oil corresponds directly to lower losses and therefore controlling the direction of impact of oil on a rotating gear can be beneficial. Therefore, shielding the teeth on the gearwheels from oil that hits them in the opposite direction of movement will reduce the acceleration of the oil and reduce the losses.

4 Computational simulation theory

In the following chapter, an explanation about the numerical methods used in simulations will be given. It's limitations and advantages will be also discussed. Furthermore, a validation of the numerical method will be presented as well.

4.1 SPH-Smoothed particle hydrodynamics

The theory behind Smoothed particle hydrodynamics (SPH) was first formulated in 1977 by Lucy [3] and in 1977 by Gingold and Monaghan [2]. Nowadays, it is being widely applied and has many applications in an engineering industry.

In general, SPH is a Lagrangian method, where the single particle is being tracked unlike Euler method, where the control volume is of importance. The fundamentals of this method revolve around computation of the density from an arbitrary distribution of point mass particles. Where the point mass particles refer to such particles, that have non-zero mass although, their other properties are not being defined. In addition to that, the advantage of using this framework is that the advection term is not present in Navier-Stokes equation as pointed out in 4.1.

$$\rho \frac{\partial u}{\partial t} + u * \cancel{\nabla \rho u} = -\nabla p + \mu \nabla^2 u + g \qquad 4.1$$

There are three different approaches when it comes to computing the density as shown in Figure 3. In the first one, one constructs a mesh around the particles and computes density by dividing the mass in each cell by the volume. Next approach is the construction of the local volume around sampling point and dividing the total mass by the sampling volume. The third approach, which is also primarily used in SPH, computes the density from weighted summation over nearby particles. The equation describing this approach can be seen at 4.2.

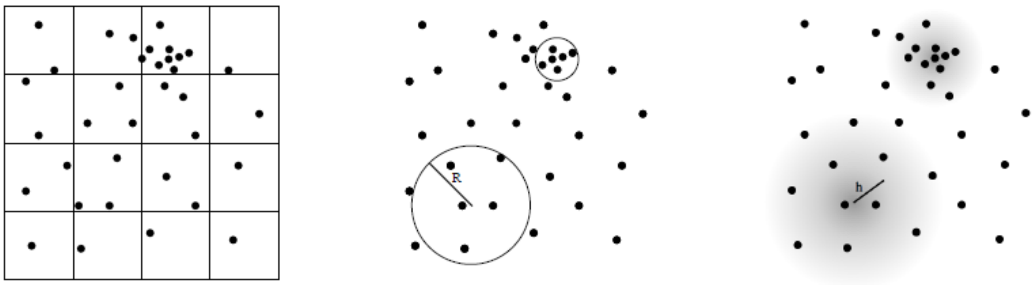


Figure 3 Three different approaches to compute the density [8]

$$\rho(r_i) = \sum_{j=1}^{N_{neighbours}} m_j * W(r) * \left(|r_i - r_j| * h\right) \qquad 4.2$$

The $W(r)$ refers to a weight function. Its dimensions are dependent on the radial distance r, between a particle of interest and neighboring particle and h, which is the height of the $W(r)$ at the point of considered particle. The accuracy of the final density is mostly dependable on the Kernel function itself, which can be represented for example with 3D Gaussian distribution as projected in Figure 4.

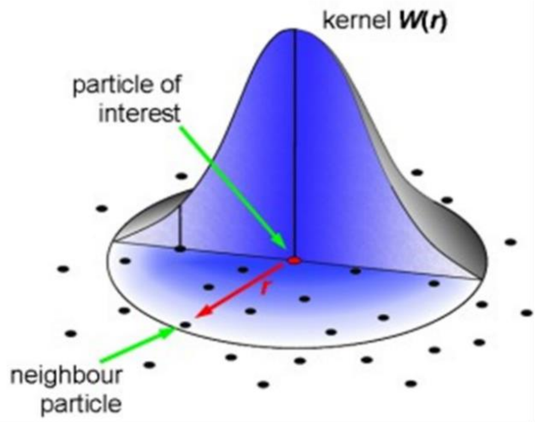


Figure 4 Gaussian Illustration of the Kernel function [6]

There are also some requirements, which Kernel function must meet, such as:

- It must be positive with smooth monotonic decrease as a function of distance
- Symmetry around the particle of the interest
- $\int W * \left(r^{\rightarrow} - r^{\rightarrow\prime}, h\right) \, dr^{\rightarrow\prime} = 1$

Afterwards, once the density is determined all other properties such as velocity, internal energy, acceleration etc. can be calculated. From equation 4.3 one can conclude that it is a slightly more complicated Newton's second law equation, where the acceleration of particles is a result of combination of the forces acting upon them.

$$a_i = \frac{\partial u_i}{\partial t} = -\frac{1}{m_i}\left(\sum_{j=1}^{Nnbs} \widetilde{p_{ij}}\left(V_i^2 + V_j^2\right)\nabla W_{ij} + \sum_{j=1}^{Nnbs} \widetilde{\mu_{ij}}\left(V_i^2 + V_j^2\right)\frac{u_{ij}}{r_{ij}}\nabla W_{ij} + g + \alpha\kappa\nabla C\right) \qquad 4.3$$

The first term on the right side of the equation 4.3 above represents a pressure gradient, the second one is a viscous term and the last two terms represent body forces and surface tension respectively. The term α represents an interaction between two phases and is only valid in multi-phase simulations, otherwise it is equal to zero. Furthermore, in equation 4.4 pressure and viscosity require a modification when it comes to the multiphase simulations.

$$\widetilde{p_{ij}} = \frac{1}{2}\left(p_i + p_j\right) \Rightarrow \widetilde{p_{ij}} = \frac{\rho_j p_i + \rho_i p_j}{\rho_i + \rho_j} \qquad 4.4$$

And instead of specifying a viscosity for single phase, there needs to be a combined viscosity accounting for both phases as expressed in following equation 4.5.

$$\widetilde{\mu_{ij}} = \frac{2\eta_i \eta_j}{\eta_j + \eta_i} \qquad 4.5$$

Regarding the boundary conditions at walls in SPH, it is required to have no-slip condition at the walls and Impermeability. When it comes to particle generation at walls, it is important to use at least three particles in order to avoid problems with unphysical results, such as squeezing of particles through the walls.

When it comes to validation of the NanoFluidX, a Particle Image Velocimetry (PIV) study by Hardno, Golubev and Chernoray [4] of fluid flow inside of gearbox with two gears was performed, which results are displayed in Figure 5. One can notice here the comparison of the two measurements, one performed in the beginning and one after some time. The results from experiment and simulation align quite well especially in the beginning of the simulations, however as the time progresses the speed of the flow in the vicinity of the gears differs.

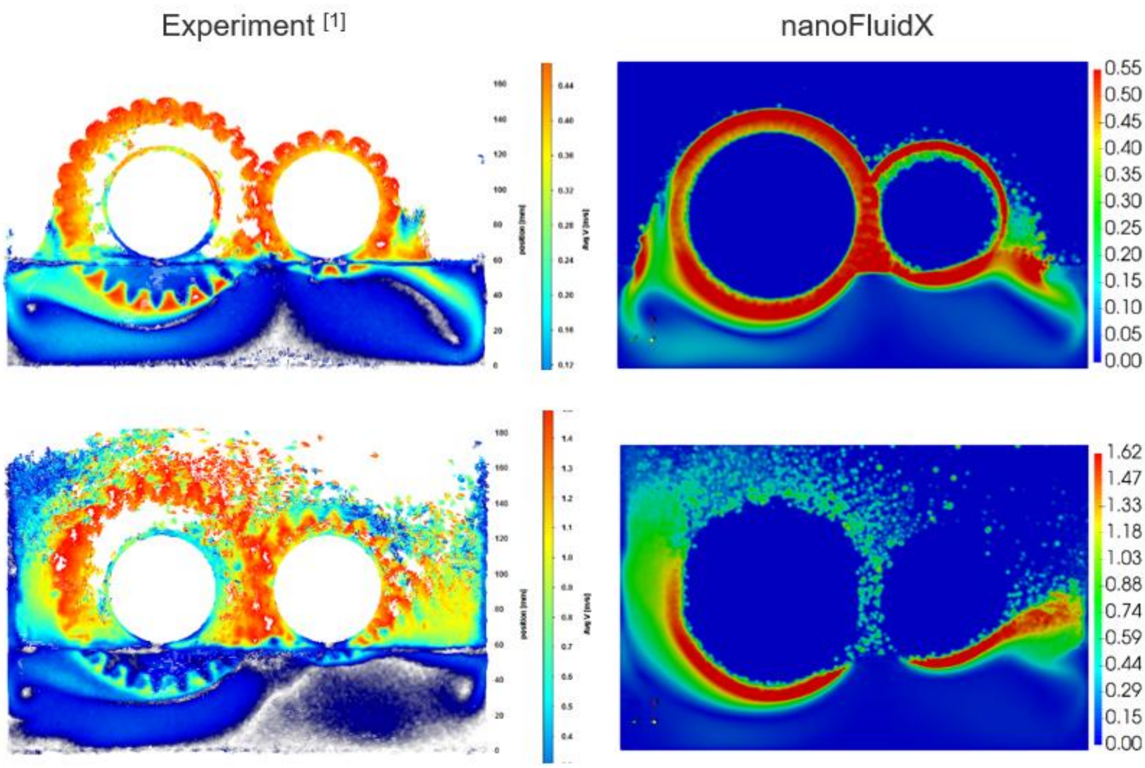


Figure 5 Particle image velocimetry study of two gears rotating in oil [4]

In the same study [4], another comparison of the experimental and simulated studies shows excellent alignment of the results, as shown in Figure 6. In this case, the primary focus was the velocity profile beneath the pinion gear. The conclusion from these studies is that the nanoFluidX displays realistic results in comparison to the experimental tests. However naturally, there are some limitations such as stability issues and choice of the Kernel function W_r representation, which is crucial for accuracy of the results.

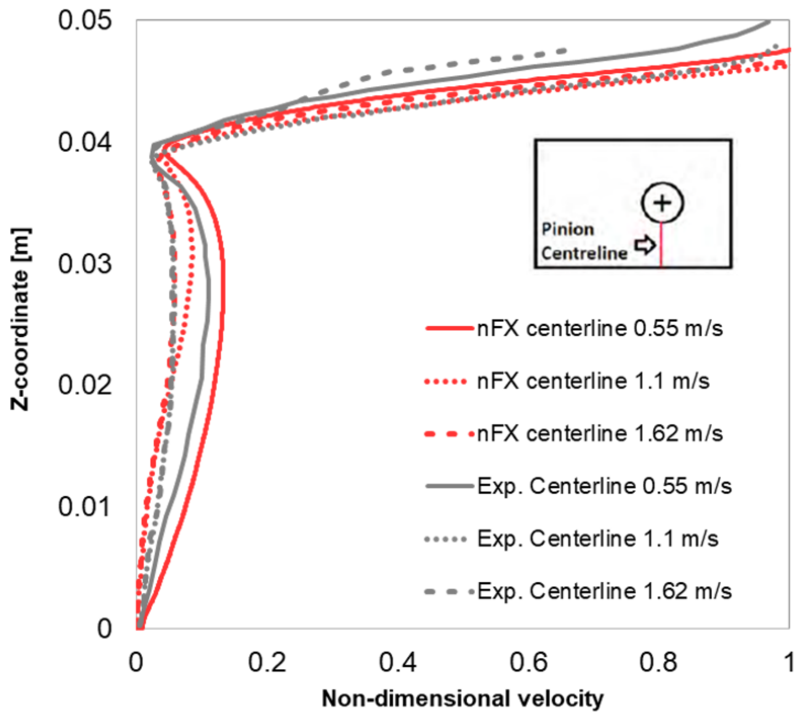


Figure 6 Velocity profile beneath pinion gear [4]

The drag torque results come from the force that is being generated by contact between the fluid and rotating components, multiplied by the component radius. Therefore, reducing the interaction will eventually lead to decreasing the drag torque.

4.2 Multiphase flow, Volume of Fluid (VOF) model

In order to simulate the drainage of the Oil from the container a multiphase simulation is required, which consists of two phases, Oil and Air. A chosen method, that covers all necessary physics is Volume of Fluid (VOF), which uses volume fraction α in mesh cells in order to describe the position of the interface. The two immiscible fluids are defined either with $\alpha = 1$ or $\alpha = 0$ as shown at Figure 7.

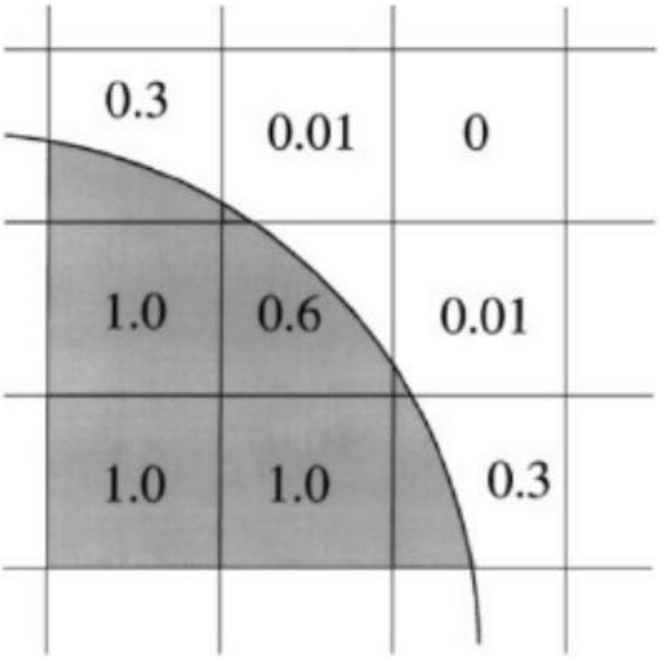


Figure 7 Representation of two immiscible fluids and interface [7]

The governing equations in case of VOF are continuity and momentum equations.

$$\frac{\partial \rho}{\partial t} + \nabla * (\rho u) = 0 \quad 4.6$$

$$\frac{\partial}{\partial t}(\rho u) + \nabla * (\rho u u) = \nabla * T + \rho g + F_{surface\_tension} + F_{wall\_adhesion} \quad 4.7$$

$$\rho = \langle \rho_1 \alpha + \rho_2 (1 - \alpha) \rangle \quad 4.8$$

$$T = \langle T_1 \alpha + T_2 (1 - \alpha) \rangle \quad 4.9$$

Where ρ_1 and ρ_2 represent two phases and u is the velocity of the interface. The wall adhesion force $F_{wall\_adhesion}$ is caused by the adhesive forces between fluid and surface, an important parameter is the contact angle θ, that decides how the fluid wet the surface of the walls.

5 Design of a new part

The new part was designed to fulfill three goals, those are collecting the splash of oil in containers, releasing the oil from the containers and shielding rotating parts from incoming splash. To fulfill these goals the designing process was divided into three steps:

1. Benchmark simulation – a simulation of the original gearbox
2. New part – a new part was designed that fulfill the goals of collecting oil and protect the gears from splash
3. Discharging of oil – dimension holes in the containers that fulfill the goal of releasing the oil

The process begun with a benchmark simulation in nanoFluidX of the original setup in order to visualize the splash inside of the gearbox. From this simulation, problematic areas where found where improvements can be done. The new part was then created in the second step around the findings from the first step, this process was an iterative process. To finish the part, discharging holes were created in the third step with help of simulations in Fluent.

5.1 Methodology

The method of designing a new part consists of three steps. First a benchmark simulation was created, then a new part was designed and in the end a system for discharging the oil was made. Here these methods are presented.

5.1.1 Benchmark simulation

Simulations were performed in nanoFluidX and setup of the simulation was made in a software called Simlab. This setup mostly consisted out of model clean up, meshing and particle creation, parameters such as rotating speeds can later be edited in the text file transferred to nanoFluidX. In order to be able to draw a comparison between various simulations, a consistent particle size of 1.2 mm was used. This particle size provided a good compromise between the required computational time and quality of the results. In addition to that, advanced models such as Adhesion or Surface tension models were not considered mainly due to extensive increase of computational time. Further model setup specification is displayed in Table 1.

Table 1 Values used in nanoFluidX

Phase	Ref. Vel. factor	Timestep factor	Compressibility [-]	Density $[kg/m^3]$	Dyn. viscosity
Oil	1.1	1	0.03	800	0.0123
Air			0.01	1.225	$1.78 * 10^{-5}$

Reference velocity factor is a safety factor added to a max. calculated velocity when calculating the timestep at runtime which ensures the stability of the simulation. Timestep factor manually scales the internally calculated timestep.

The first step after importing the model to Simlab was to create an initial surface-mesh on all the components. This mesh was afterwards modified to meet the requirement of three particles wall thickness in order to avoid problems with particles squeezing through as mentioned in Section 4.1. In Figure 8 one can see the difference between an imported surface and a surface-mesh along with the particles created from the meshed body illustrated on the benchmark oil trap.

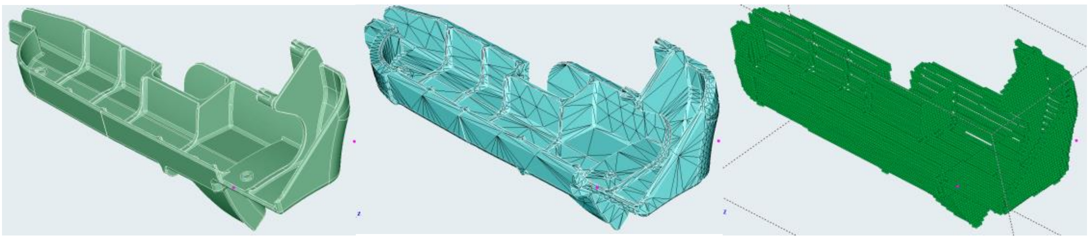


Figure 8 Process from imported part to particles created

There are two different types of boundary conditions that were applied in this simulation setup. Namely, it is the Rotational boundary condition, which was applied onto the rotating objects along with the material Moving wall or just Wall if the part was stationary. Second boundary condition applies to whole domain, it is the gravitational force. These two boundary conditions were preferred to be applied during the mesh creation due to that they would be transferred to the particle bodies. By doing so, it was faster to apply modifications on the meshed bodies and then create new particle bodies due to that motion and material property were already applied.

Furthermore, regarding the speed, it was decided that the simulations used for initial design of the new part would be performed at an approximate car speed of 50km/h, however further testing would be carried out at a variety of speeds. This speed was chosen, since it is the average car speed taken from one of the company cars.

On the differential gear this added up roughly to 403RPM which can be seen specified in Figure 9. As a result of this speed on differential gear, the engine speed added up to about 1500 rpm using gear five, this is a normal operating speed for a combustion engine in a passanger car. When it comes to other rotating parts in the gearbox with preselected sixth gear, the rotational speeds were set according to Table 2, Table 3 and Table 4. In addition to these parts, there were also other smaller parts such as loose rings or synchronizers, which rotate with the speed of the gear next to it or the speed of the gear it is connected to.

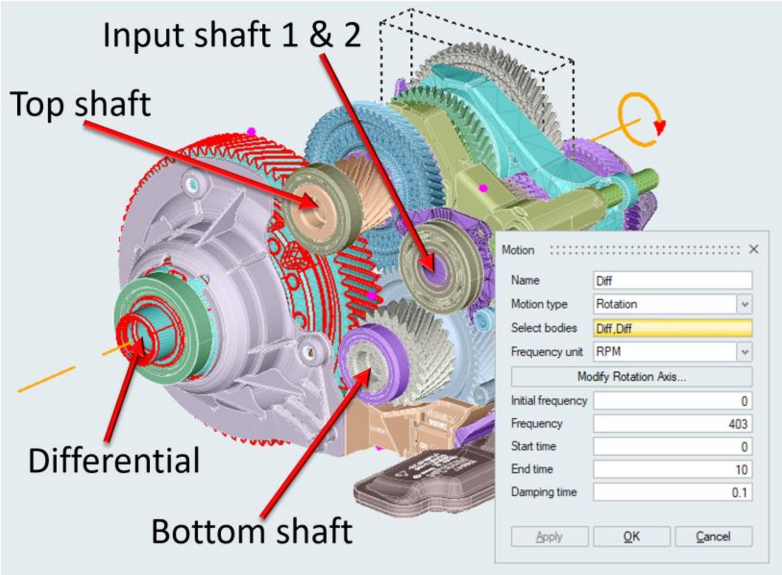


Figure 9 Internals of the 7DCT gearbox in Simlab with speed settings on differential gear and naming of the shafts used in the report

Table 2 The rotational speed on the loose gears on Top shaft

	1st Gear	3rd Gear	4th Gear	Reverse	Shaft
Top shaft [rpm]	425	1042	1138	410	1872

Table 3 The rotational speed on the loose gears on Bottom shaft

	2nd Gear	5th Gear	6th Gear	7th gear	Shaft
Bottom shaft [rpm]	423	1384	1384	2071	1384

Table 4 The rotational speed the input shafts and differential

	Differential	Input shaft 1	Input shaft 2
Other noteworthy structures [rpm]	403	1502	1191

The surface level of the oil was constructed by defining a plane at the specified height provided by CEVT as shown in Figure 10 . Afterwards, fluid properties and their horizontal position relative to the plane were defined. The last step was the generation of the particles representing these fluids in the container.

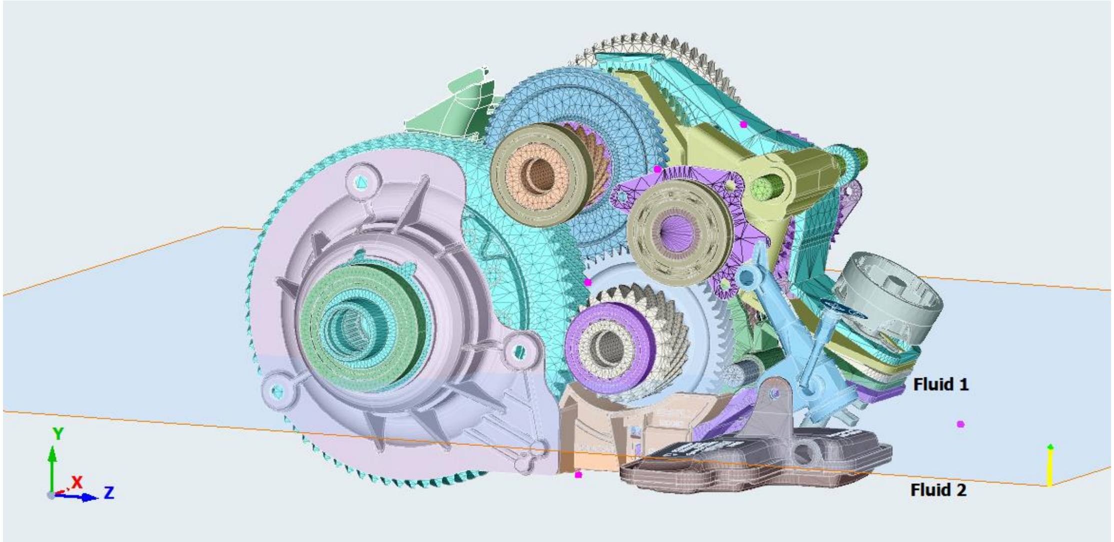


Figure 10 The plane that is defined in Simlab when creating particles of oil and air

5.1.2 New part

A design of a new part was made based upon the results from the benchmark simulation along with the literature study. The goals that it needed to fulfill are:

- Collect the main splash of oil
- Store the oil in containers in the upper region
- Divert the splash of oil from hitting gears moving in the opposite direction
- Fit in between the shifters and the casing
- Use already existing mounting holes

When the new part collects the oil, it must convert the splash into a flow through a channel if it needs to be transported to a place further away. Therefore, it is important to make sure that the splash has the energy to move to this place. The part that collects the oil will be made so it also shields gears from incoming splash, if necessary, some shielding walls will be added. When it is decided what path the oil will take it comes with a challenge of making the new part fit. To do so all the moving components in that area, such as shifters, are moved to the position where they create the extreme case of the smallest space available. Once the new design is done based upon data from the benchmark, the iterative process of simulations and polishing of the new part will follow until the desired results are obtained.

5.1.3 Discharging of oil

When it comes to optimizing the functionality of the new part, it is important to scale the discharge of the containers based on the incoming mass flow. Which means that, when the oil level is at its highest point in the container, the discharge mass flow must always be smaller than the incoming mass flow. That way, it will be possible to retain as much oil in the containers as possible and the Oil level in the lower regions of the gearbox will decrease as intended.

The chosen strategy how to achieve this goal, was to determine the incoming mass flow in extreme circumstances, which constitutes smallest and highest incoming mass flow. Based on the data from the lowest possible incoming mass flow, the positioning and sizing of the hole was determined.

In order to analyse the drainage from the containers, it was necessary to implement two different programs, nanoFluidX and ANSYS Fluent. NanoFluidX uses the SPH method and determines the incoming mass flow to containers, ANSYS Fluent on the other hand is used to simulate the discharge from the containers. The advantage of implementing Fluent is that it provides more in-depth information regarding the multiphase phenomena of the simulation which is crucial.

The pre-processing and geometry clean-up were done in software called ANSA. As displayed on Figure 11, simulated geometry consisted only out of container, the rest of the channel was redundant. No dimensions were altered, the only thing added is the top sealing of the container, which was necessary for the generation of the volume mesh.

The construction of the mesh was divided into two parts. First, the surface mesh generation was executed, with triangular elements, which can be seen on the right picture in Figure 11. The specification of the mesh parameters can be seen at Figure 12.

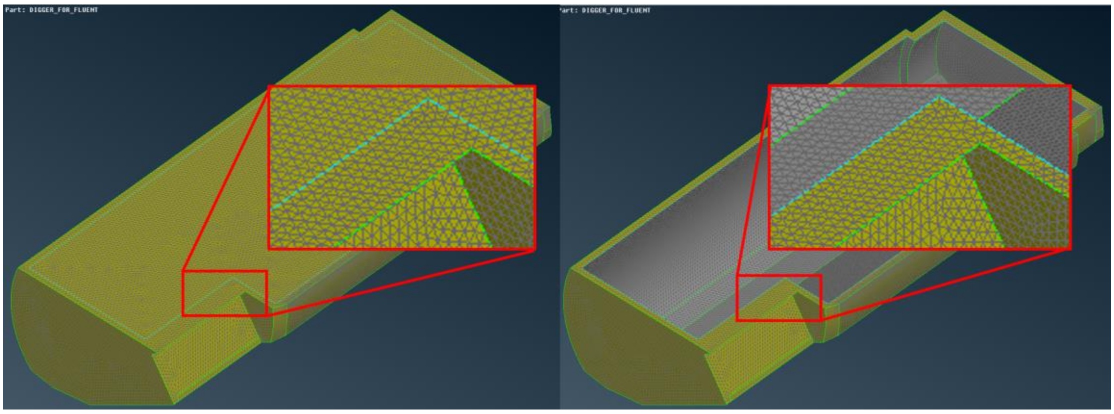


Figure 11 Volume and surface meshed container

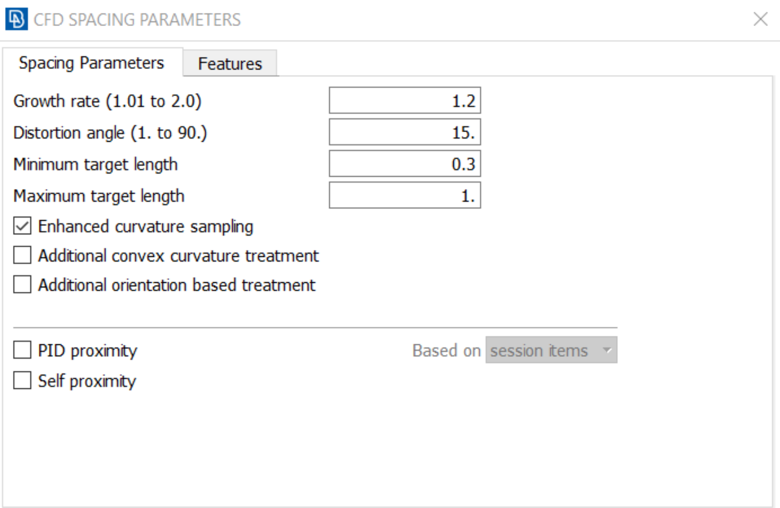


Figure 12 Surface mesh specification of the geometry

The next step of the meshing process was to generate tetragonal elements in the volume of the container which would represent fluid-oil and displayed on the left picture in Figure 11. In this case, it was also necessary to generate the elements even in the volume between the walls, since the geometry in ANSA was regarded as a shell. The settings used for mesh quality correlate with the setting which are being used in Ansys Fluent. The mesh itself, needed to be dense, mainly due to Courant number condition which had to be met since the simulation would be transient.

There were only two types of boundary conditions imposed on the surfaces of the geometry. Namely, it was the pressure outlet which was applied to all displayed surfaces at Figure 13 and impenetrable wall which was the rest of the geometry.

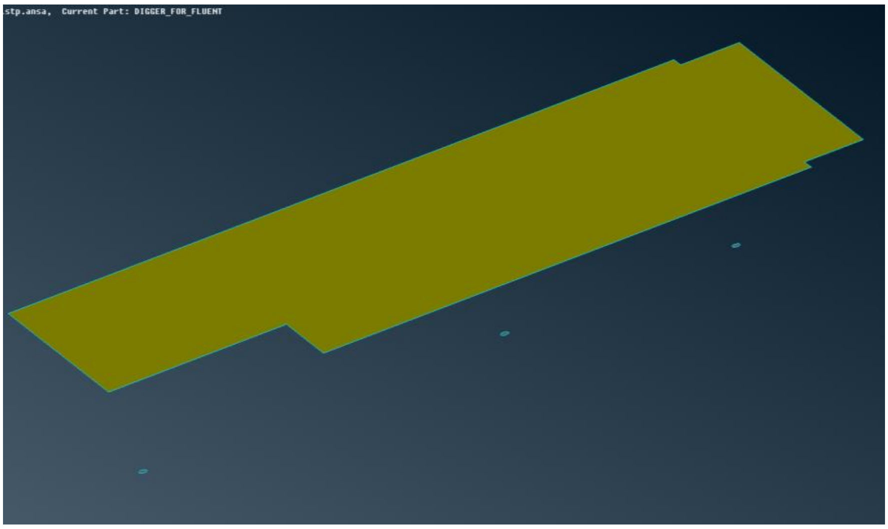


Figure 13 Pressure Outlet boundary surfaces – top and 3 outlet holes

Assigning boundary conditions to surfaces finalised the ANSA setup. Further configuration of the physics and simulation itself were performed in ANSYS Fluent.

The general settings for the discharge simulations were the following:

- Transient simulation
- Added gravity
- Pressure based solver

As already mentioned, the primary model employed, which captures the multiphase interaction between phases was VOF. The primary phase was air and the secondary oil. In addition to that, the phase interaction was represented by surface tension and wall adhesion.

Additionally, laminar model was chosen as a viscous model, because it was believed that no substantial turbulent behaviour would be taking place. Coupling between pressure and velocity was done by using the PISO scheme and chosen time step for the simulation was 0.0001.

5.2 Results

The results from the simulations, the iterative process of creating a new part and dimensioning of discharging holes are presented here.

5.2.1 Benchmark simulation

The first performed simulation was on the original model in order to obtain the benchmark data. Representation of the collected data from this simulation can be seen in Figure 14 where distribution of oil is the primary interest. Colours ranging from gold to grey represent the volume fraction of oil. In general, the volume fraction expresses the composition of a mixture and cannot be greater than 1. In this case a volume fraction of 1 consist of pure oil and 0 represents pure air, everything in between is therefore a mixture.

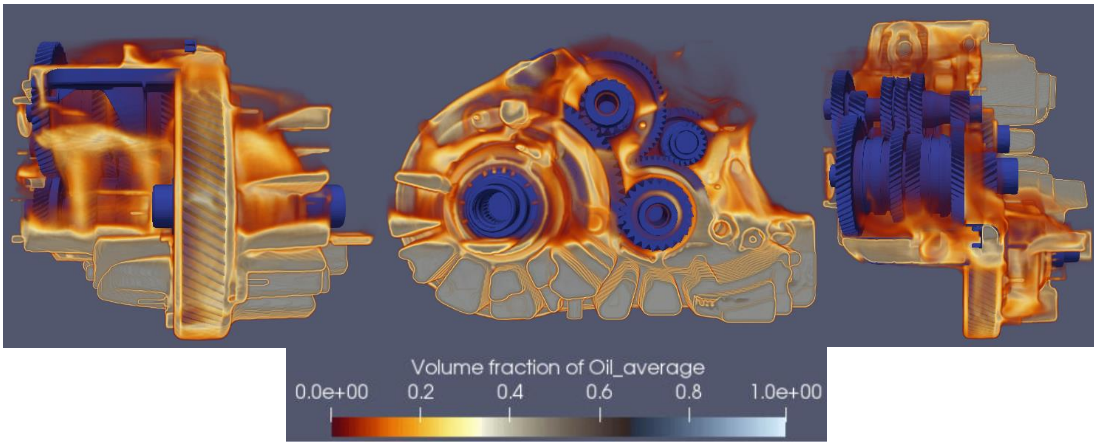


Figure 14 Overview of the gearbox benchmark simulation

The most useful information gained from the benchmark result, is the splash pattern in the gearbox. From Table 5 and Figure 15 it can be concluded that the

Top shaft is more impacted by splash coming from the differential gear in comparison to the other shafts due to the high contribution to drag torque. This is a result of opposite speed of the oil and the gear on Top shaft as seen in Figure 15.

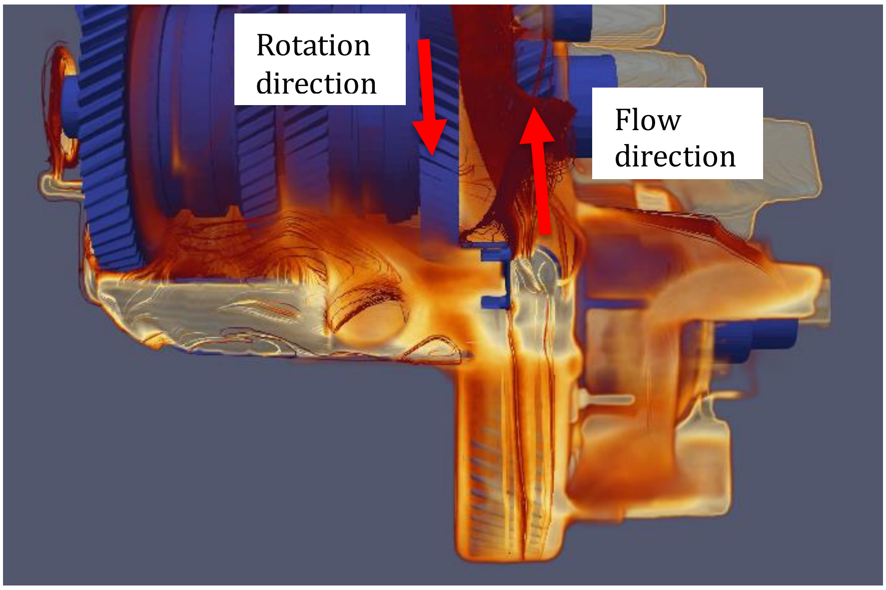
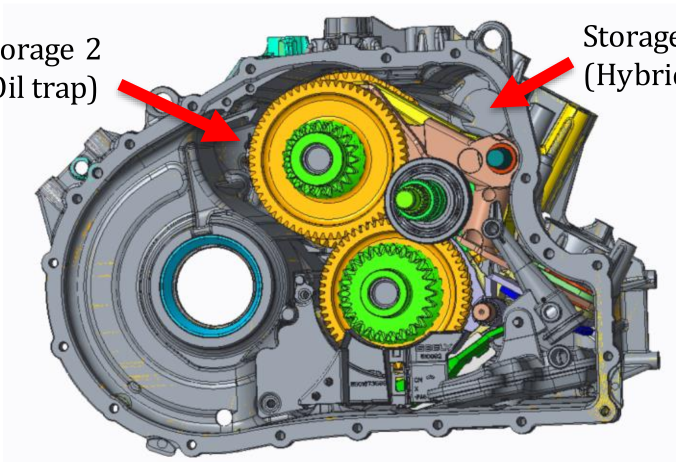


Figure 15 Top view of the gearbox with the differential in the bottom part of the figure transporting oil to the original trap above

Furthermore, from analyzing Table 5 it is apparent that half of the drag torque comes from the differential gear. The cause of this phenomenon can be seen in Figure 14, where almost half of the Differential gear is being submerged in oil. Decreasing the level of the oil in the sump, would cause less interacting between oil and differential gear, which would be beneficial in terms of overall drag torque. Altering the height level of oil can be achieved by storing it in the upper regions of the gearbox as mentioned in Section 5.1.2. As pointed out at Figure 16, two suitable positions were found. First one is located at the position of the hybrid shaft which is not being used and second one is in a space under the original oil trap.

Storage 2
(Oil trap)



Storage 1
(Hybrid shaft)

Figure 16 Two places where oil will be stored, here shown in a CAD picture with differential gear and original oil traps hidden

The duration of the performed simulation was 5 seconds as can be seen from Figure 17. Since the Total drag torque seemed to level out, further simulating would only bring oscillation around the same value, therefore the simulation was deemed to be converged. For further simulation of the benchmark at other speeds the same method will be used to look for convergence, i.e. letting the simulation run until the result looks similar to Figure 17.

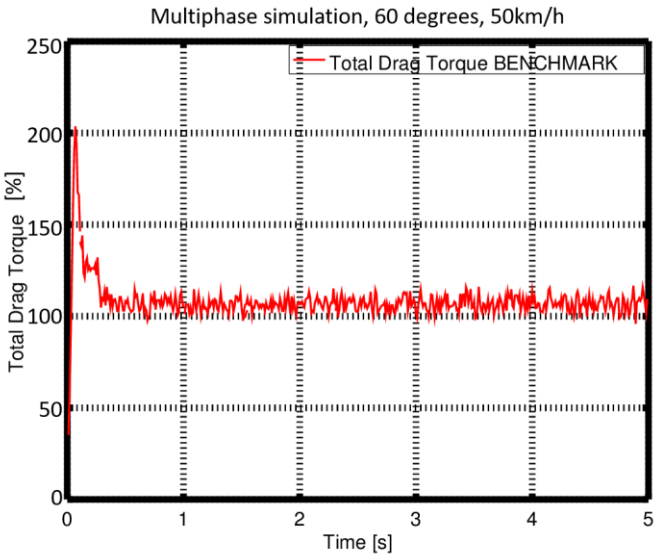


Figure 17 Total drag Torque of the benchmark simulation running for 5 seconds, values changed to percent due to confidentiality

Table 5 Distribution of drag torque coming from churning on rotating parts

	Top shaft	Bottom shaft	Differential	Input shaft 1	Input shaft 2	Total
Drag Torque [%]	30	17.5	50	1.7	0.8	100

In order to validate the physicality of the results, a study regarding density variation of particles was done as well, which can be seen from Figure 18. It is apparent, that the number of particles which exceed its actual density by 10% and 20% respectively, is rather small which indicates no major problems in terms of physicality of the results. It is at most around 400 particles out of 11.6 million used in the simulation.

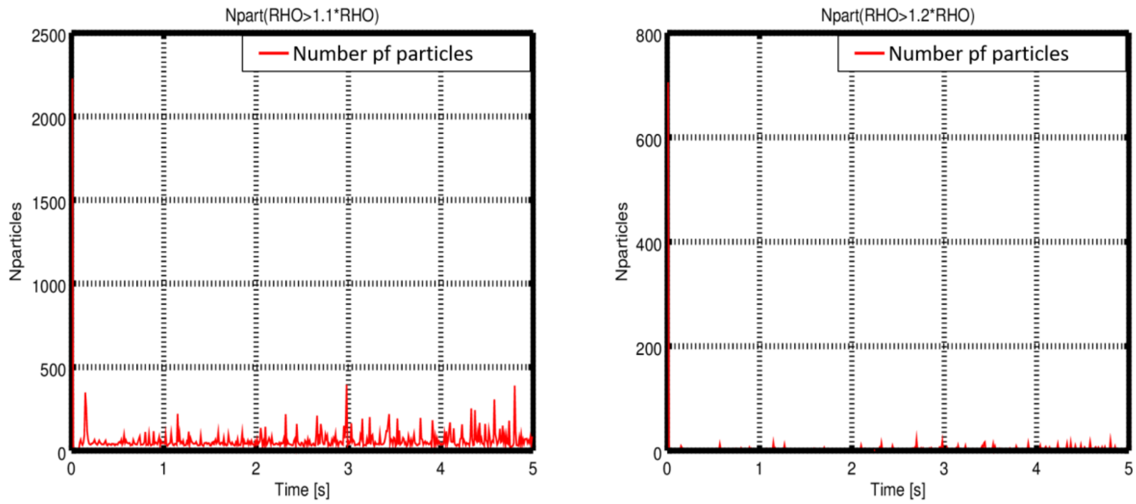


Figure 18 Number of excessing volume fraction particles

5.2.2 New part

Based on the information gained from the simulation of the benchmark at 50km/h a new part was created. In this part more emphasis was put on collecting the oil and shielding oil from hitting gears in the opposite direction of speed. Since it was identified, that the differential gear is responsible for most of the oil being transport to the top, the main purpose of the new part is to collect oil. The collected oil is afterwards guided to two storage containers which are placed in the two locations mentioned in Section 5.2.1 and are pointed out in Figure 19. Apart from storing the oil the new part should also be able to protect the top shaft from the incoming oil. Figure 19 displays the finished design of a part made for simulation and prototype testing.

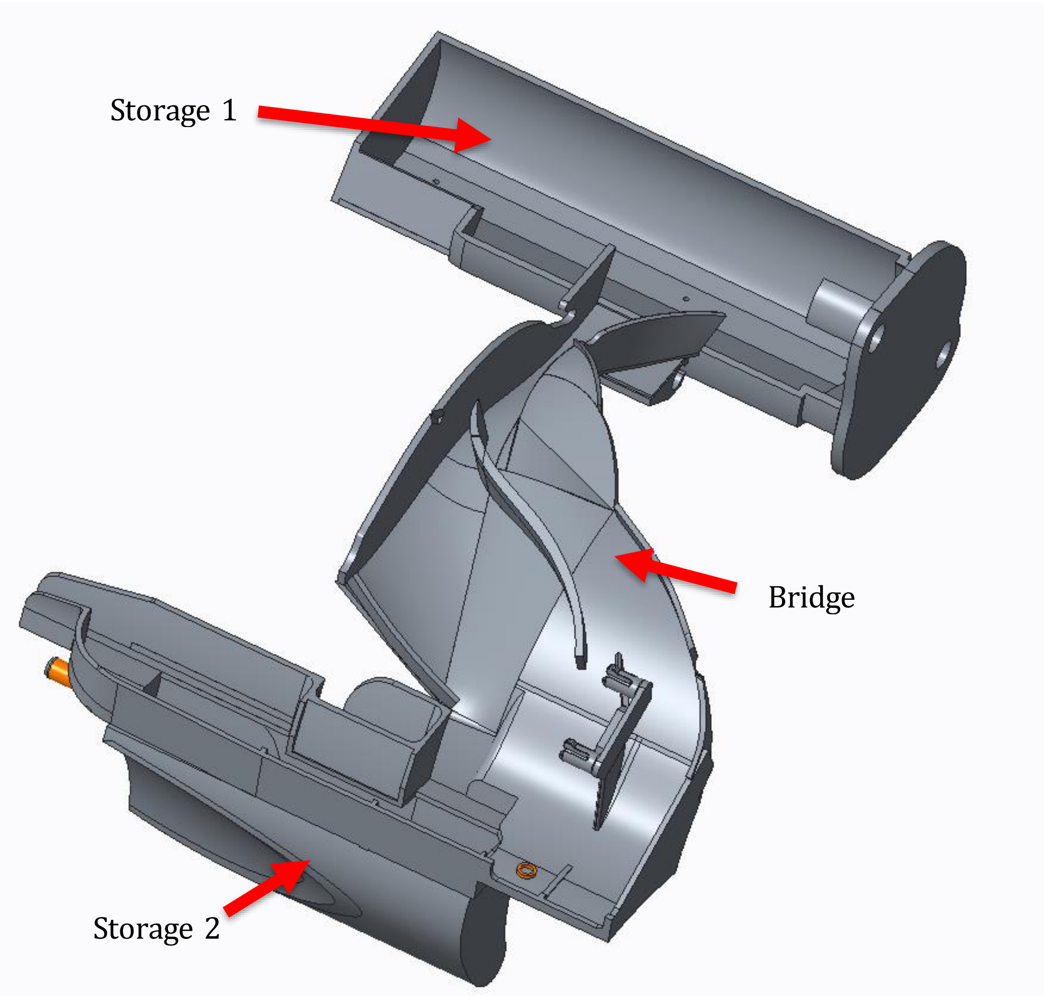


Figure 19 New part that transports oil from the differential and collects it in two different places

As explained in section 5.1.2 the new part needs to leave room for the shifters and gears. Figure 20 shows how the Bridge fits between the shifters when they are in the closest position to the new part. Additionally, the housing of the gearbox restricts the design since it contains parts which strengthen the construction and other parts that support the flow of oil in the original design.

An example of how the new part is designed around obstacles in the housing is shown in Figure 21 where a slot was made in the Bridge. In this case the flow in the Bridge became restricted, due to the part sticking out from the housing. However it also presented an advantage of supporting the Bridge under assembly, which will be more discussed in section 6.1.

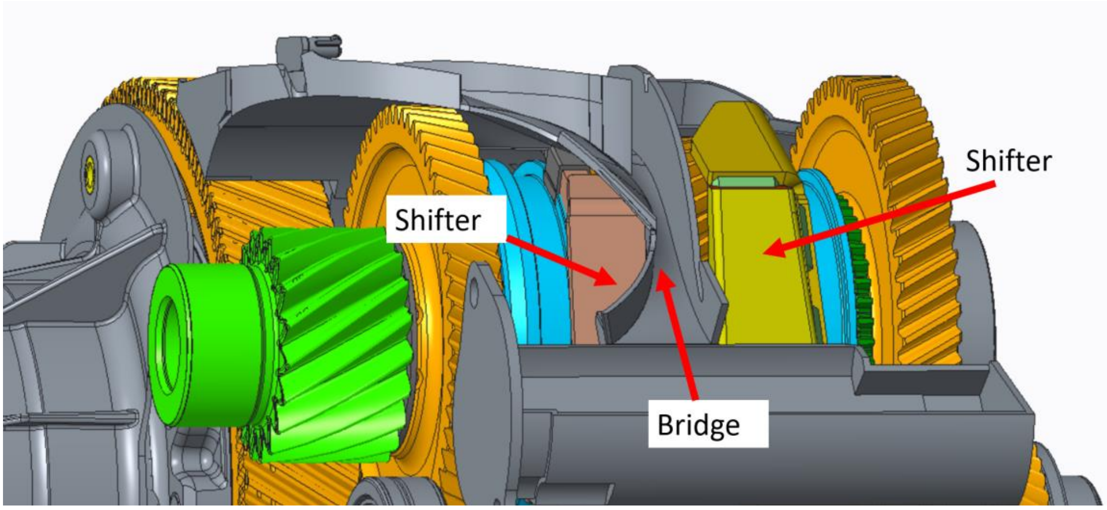


Figure 20 How the Bridge on the new part fits in between the shifters

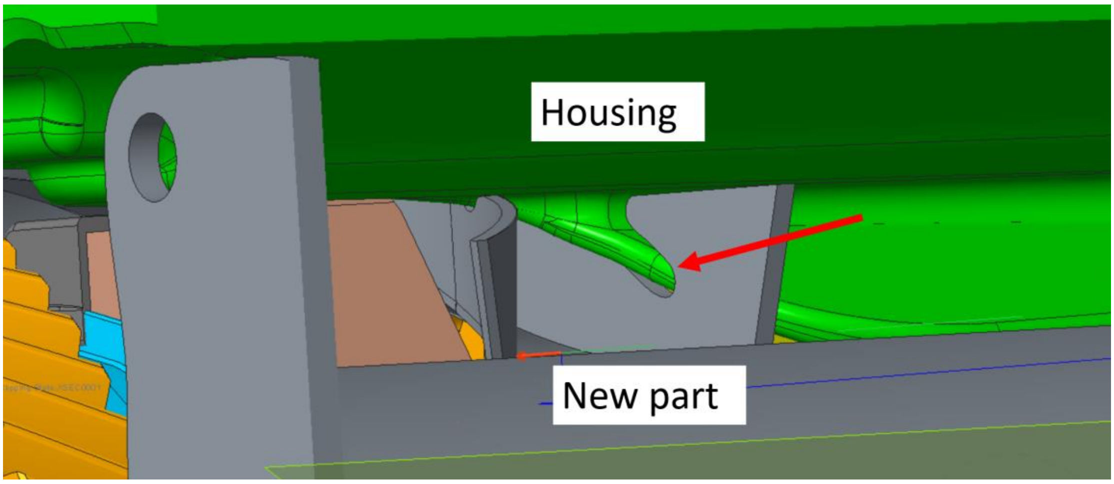


Figure 21 Example on how the new part was designed around obstacles, highlighted with an arrow

An iterative procedure of creating this new part resulted in three simulations before reaching satisfying results. These three simulations contained following:

1. Capturing the splash from differential gear into a flow
2. Directing the flow through a channel into a container
3. Closing gaps and refining the channel

1: The first task was to see if the oil splash coming from the top of the differential can be manipulated into a flow and then guided over the top shaft. An initial part was created that reflected this idea of capturing the splash of oil, it also has a rough design of a bridge connected to storage 1. Results from the first simulation of the new part can be seen in Figure 22, from which it is apparent, that the developed flow from the differential is powerful enough to be transported over the top shaft. At this stage of the new part development, storage 2 was also created and succeeded to be filled up with oil. It was deemed that this storage did not need any further refinement.

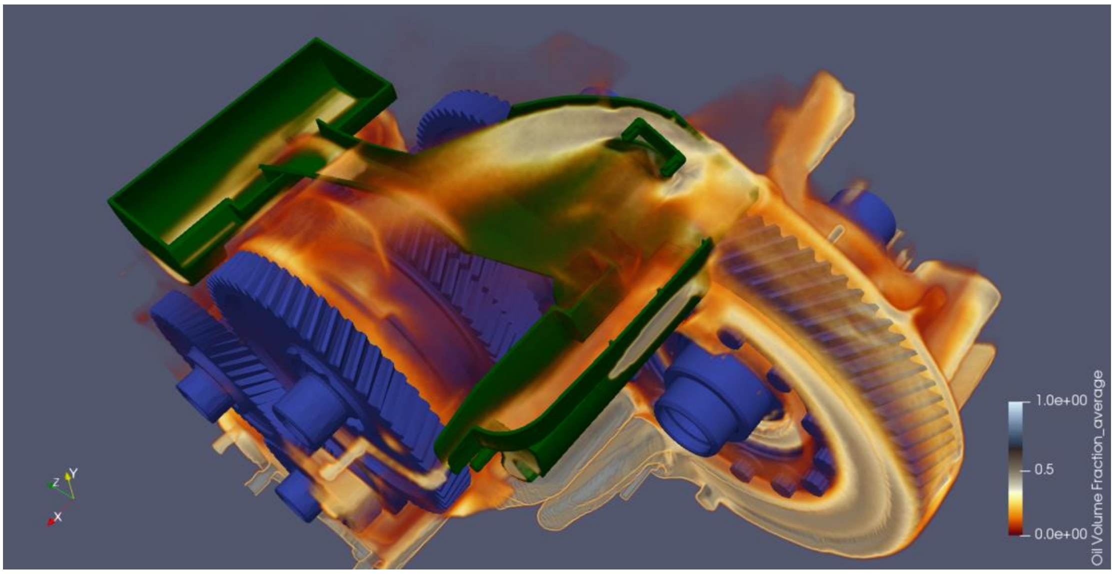


Figure 22 A first design of a new part that proves that the splash from the differential can be redirected

2: Focus in this step was put on transporting the captured flow of oil over the gears and shielding the top shaft from the incoming flow at the same time. From Figure 22 it can be seen, that most of the flow is being shot out along the top shaft. This undesired phenomenon was fixed with adding two additional walls, which can be seen in Figure 23. Wall 1 was positioned at the beginning of the bridge and Wall 2 at the side of the bridge, where it stops the oil from escaping the channel. This resulted in further improvement in amount of oil transported to the container.

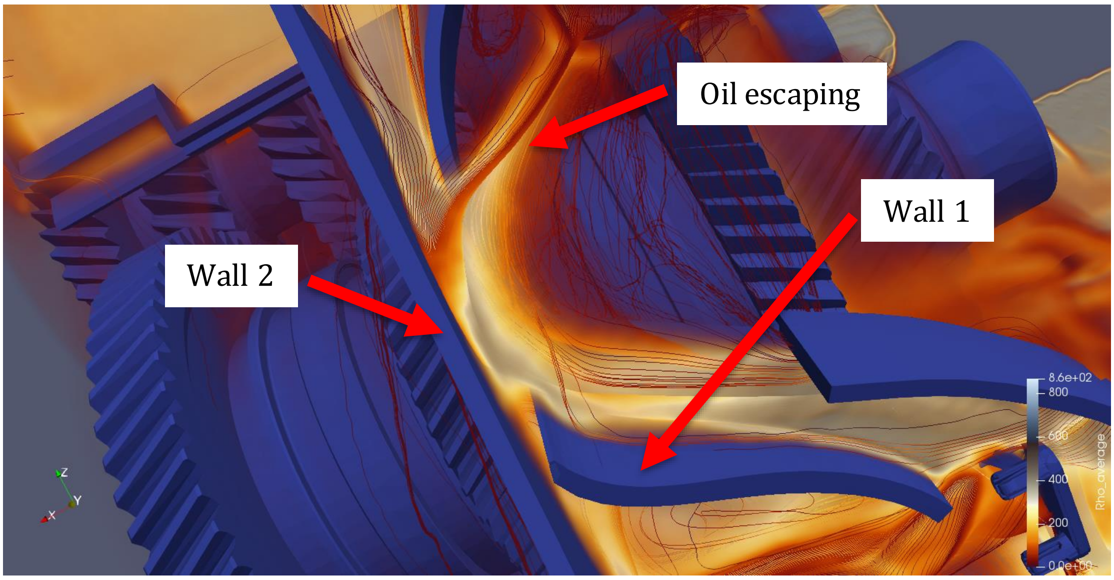


Figure 23 Second design of a new part with two added walls, wall 1 and wall 2

3: The last iteration in the new part focused on capturing the escaping oil as shown in Figure 23. This was achieved by adding a new side wall, Wall 3 as shown in Figure 24. In addition to that, there was also a need for utilizing of the oil which drips down from the rear part of the channel into the sump, therefore another diverting side wall, Wall 4 was added. Due to that, the oil goes straight into storage 2.

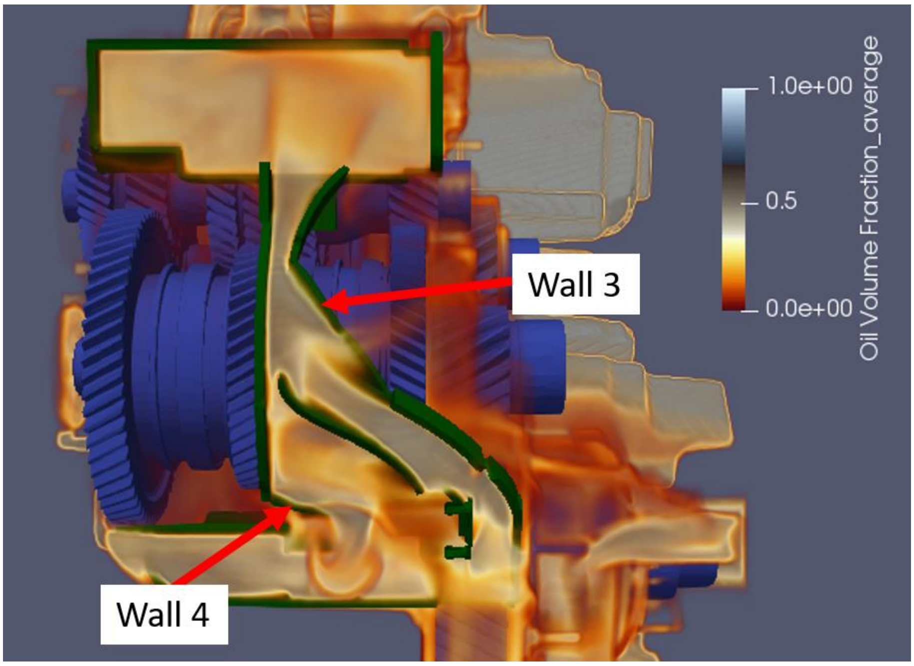


Figure 24 Visualization of the flow to the container with the finished new part were wall 3 and 4 have been added after the results from previous simulation

From the iterative simulations one can conclude, that it is possible to transport and store the oil in desired location and eventually, as a result of that, decrease the oil level in the sump as shown in Figure 26. Furthermore, diverting the oil flow in a clever way leads to protecting the gears from incoming oil, which can be seen from Figure 24.

The setup in the iterative simulations uses only single-phase model which means that air is not present, therefore it cannot be compared to the benchmark data which were obtained from multiphase simulation. In order to see the difference, which the new part makes in terms of numbers, a simulation with the same setup as the benchmark is made. By plotting the total drag torque of the new part over the values from the benchmark simulation as displayed in Figure 25, it is clear that as the two containers fill up, the drag torque decreases. The overall reduction in drag torque adds up to 16.3%.

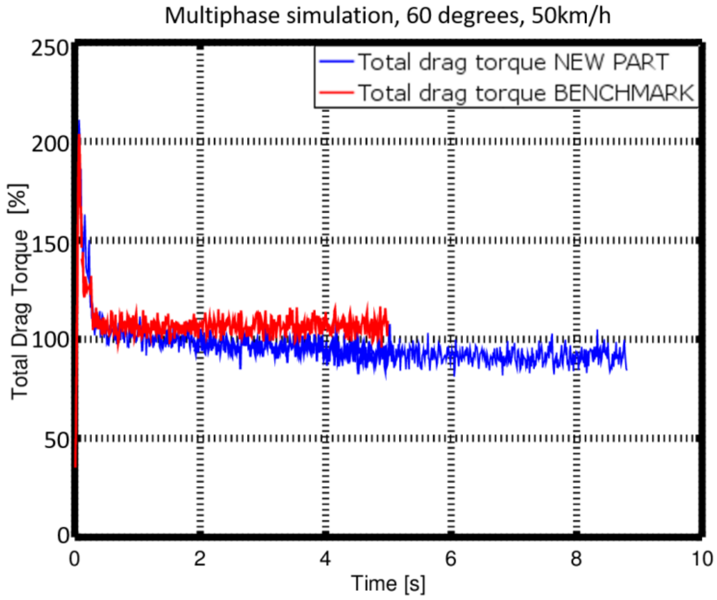


Figure 25 Total drag torque from new part and the benchmark at 50km/h

According to Table 6, the biggest drag torque reduction in percentage is on Input Shaft 2, however in absolute numbers, the biggest reduction is present on Differential gear. The cause of this reduction is probably the decrease in surface level of the oil in the gearbox. That way, the Differential gear is in contact with less oil which eventually leads to drag reduction as explained earlier.

Table 6 Drag Torque reduction in percentage from churning on individual parts at 50km/h and on the complete gearbox

	Top shaft	Bottom shaft	Differential	Input shaft 1	Input shaft 2	Total
Drag Torque Reduction [%]	17.7	18.5	9.2	32.1	41.2	16.3

There is also a substantial reduction on Top shaft which is mainly caused by diverting the splash from Differential gear to Storage 1. When it comes to the reduction on the Bottom shaft, the reasoning is similar as in Differential gear case. Since it is also partially submerged in the sump, the benefit of decreased oil level in the sump is present here as well. Finally, the reduction on Input shaft 1 and 2 is caused mainly by splash reduction coming from the Differential gear.

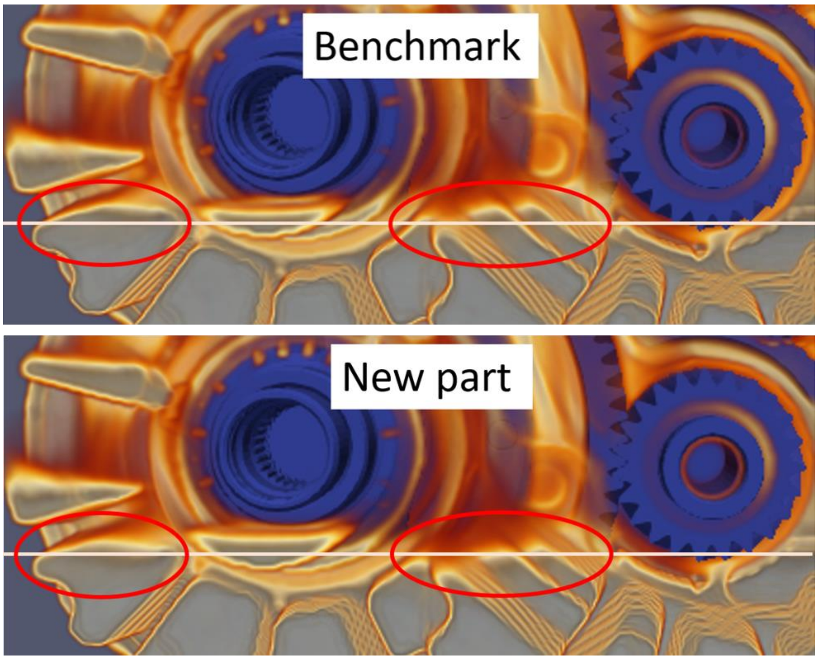


Figure 26 Circled in red, a comparison example between oil level in benchmark and new part case

A conclusion that can be drawn from the 50km/h simulation is, that the new part performs well enough in terms of overall drag torque reduction. Further perfecting of the design would be contra productive because it would not bring better results. Hence instead of that, more emphasis will be put on testing the functionality of the new part at a larger speed interval. A trend study will therefore be made, that analyses the dependency of drag torque reduction versus the car speed. The trend study will be simulated at 20, 50 and 100km/h, that way the whole spectrum of most used speeds is covered. Parameters such as temperature or viscosity of the oil are not changed, otherwise it will not be possible to make a comparison between drag torque trends.

When it comes to the simulation of 20km/h, it was expected that the overall drag torque would be smaller than at 50 km/h, however at 100km/h it was expected to be higher. These results can be seen in Figure 27 where 100% represents the same value of drag torque that is in the benchmark simulation.

The time it took to reach converging results in the simulation is as expected, prolonged at low speeds and shorter at higher speeds. Increased speed of the gears results in a higher speed of the oil and makes it faster to fill up the containers. In case of 100km/h it is clearly displayed that the running time of the simulation was sufficient to see converged results but at 20km/h it is difficult to see the improvement. This is due to the small difference in drag torque combined with the oscillations.

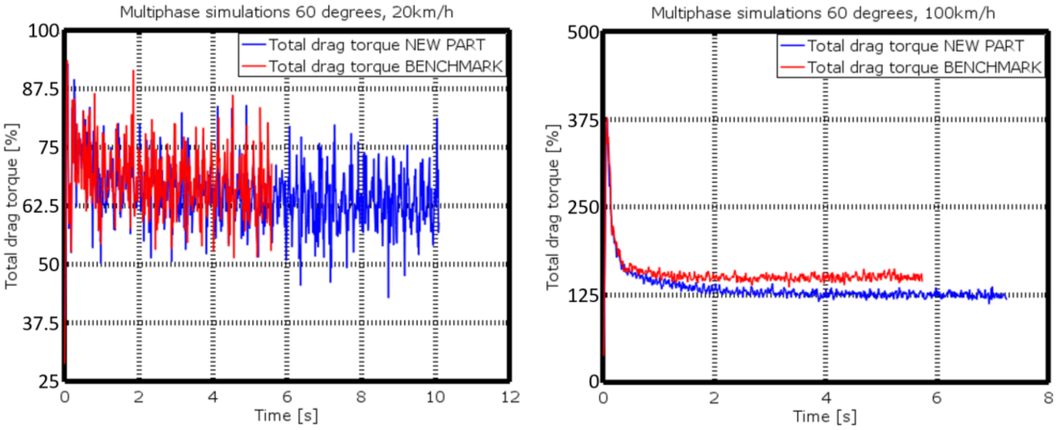


Figure 27 Drag Torque comparison of benchmark and new part at 20 and 100km/h

If the percentual reduction in drag torque is plotted from the three speeds, a trend of reduction versus speed is shown. This trend can be seen in Figure 28 and shows that at higher speeds the new part works better then at low speeds. Between 20 and 50km/h the increase in reduced drag torque is rapidly growing but between 50 and 100 km/h it levels out and is almost constant. The reason why the trend looks this way, is that the splash is not able to reach the gearwheels at the top shaft at low speed and therefore there is not that much that can be improved. Storage 1 and 2 are almost empty due to this as seen in Figure 29 and the top shaft barely gets affected by the oil flow. As the speed increases, more gears start to get effected by splash and can therefore be protected from it. Storage 1 and 2 start to fill up with the increasing speed so the drag coming from the submerged gears is reduced.

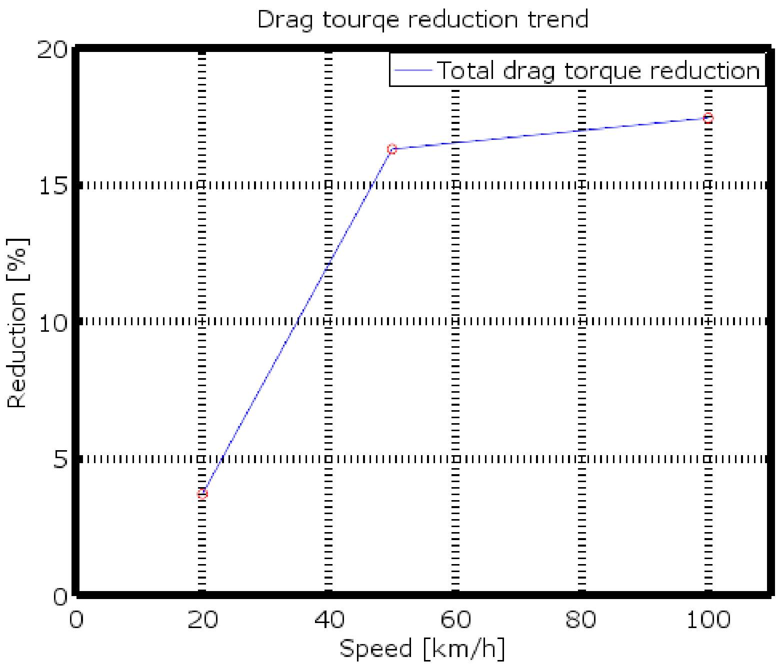


Figure 28 Trend of reduced drag torque with new part at three speeds

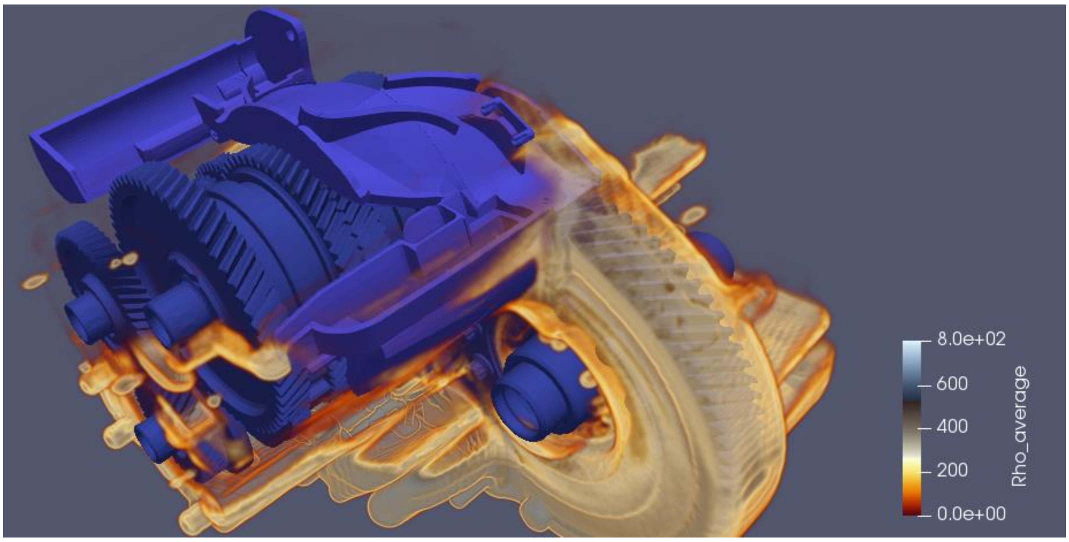


Figure 29 Overview of the gearbox 20km/h simulation results

5.2.3 Discharging of oil

Numerical complexity and available computational resources caused the running time of the simulation to be decreased to 1s. It is apparent from Figure 31 that the total mass flow rate is stabilized at the value of 0.0023kg/s, and its peak was achieved at 0.0024kg/s. In general, the trend of the discharge reaches maximum value in the beginning, when the pressure is at the highest point and afterwards, decreases gradually until it reaches 0kg/s value again. When it comes to density of the mixture which can be seen at Figure 32, in general the trend decreases over time until density of the primary phase is reached, which means that the oil is completely displaced by air at that point. The process of air displacing oil can be seen from Figure 30, where the Volume fraction of Oil in a cut view of the container is being displayed.

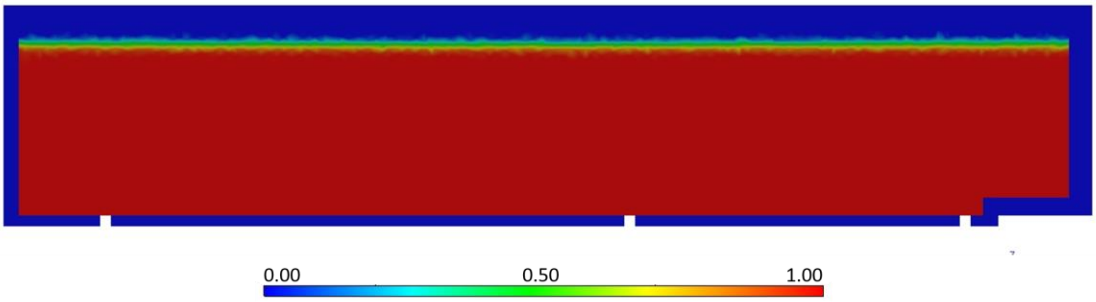


Figure 30 Volume fraction oi Oil

The red colour represents α equal to 1, which refers to pure oil. Region above the oil represented by dark blue colour, with α equal to 0 refers to the air. The sharp interface between these two phases is represented with α values ranging from 0 to 1. The walls of the container are made of solid material, therefore they are also represented with blue colour.

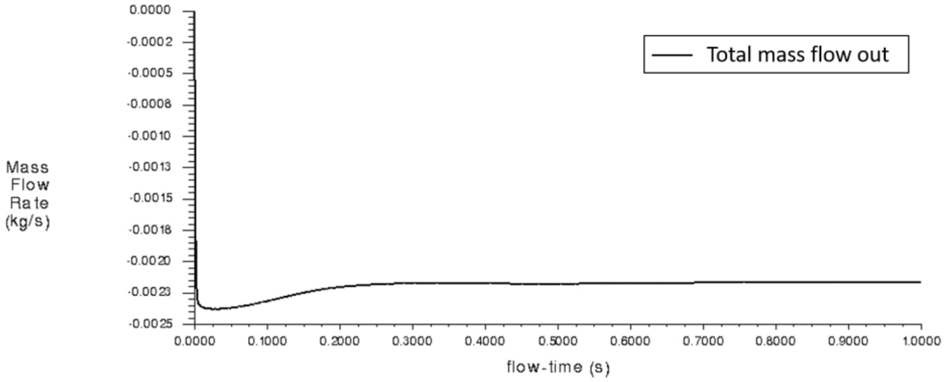


Figure 31 Total mass flow rate with holes 1.5 mm

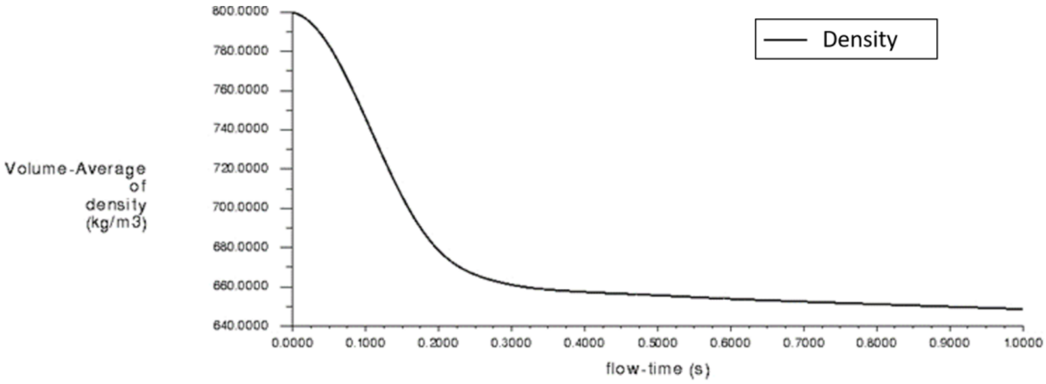


Figure 32 Density of the mixture with holes 1.5 mm

According to Figure 33 one can assume that the incoming mass flow of oil at 50km/h is approximately 0.035kg/s on average. The current design of the container allows the discharge of 0.0023kg/s, which means that the container works in full range of speeds and complete discharge would always be secured.

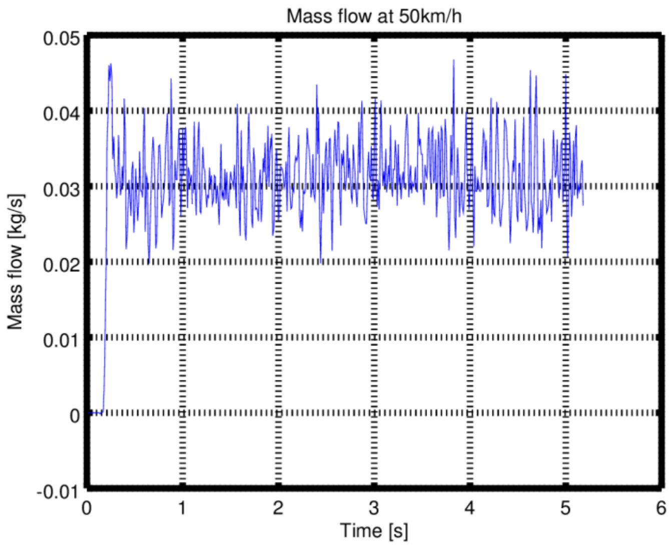


Figure 33 Mass flow entering the container at 50km/h

On the other hand, the overflow would substantially increase at higher speeds. Due to extensive overflow of the oil, it would be necessary to add another distribution channel in the upper regions of the container that would distribute the leakage and increase the diameter of the discharge holes. The results of a simulation with increased holes diameter to 3.5mm can be seen in Figure 34. Other parameters of the geometry as well as mesh and physics were kept intact. As expected, the total discharge mass flow rate increased significantly, up to value of 0.014kg/s. At this point, the overflow would not be as extreme as in previous case, however functionality of the part can be slightly restricted when it comes to lower car speeds.

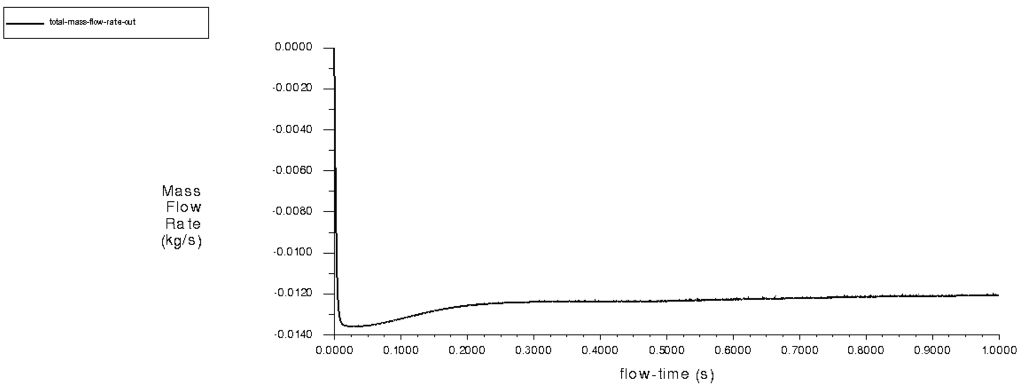


Figure 34 Total mass flow rate with holes 3.5mm

When it comes to validation of the gathered results, mostly investigated parameter will be CFL. Based on its value one can say, if the simulation does not proceed way too fast. It is mostly influenced by the chosen time step and the size of the grid. In ideal case, its target value should be within the range of $0 - 1$, which would indicate that all the important information is being captured inside of the grid elements and nothing is being lost. The results of the CFL from the simulation are being displayed in Figure 35. It is apparent that, the CFL reaches highest values around the discharge holes, which is caused by the fact that the velocity amounts to highest values here as well. Nevertheless, the highest value is approximately 0.8 which is acceptable.

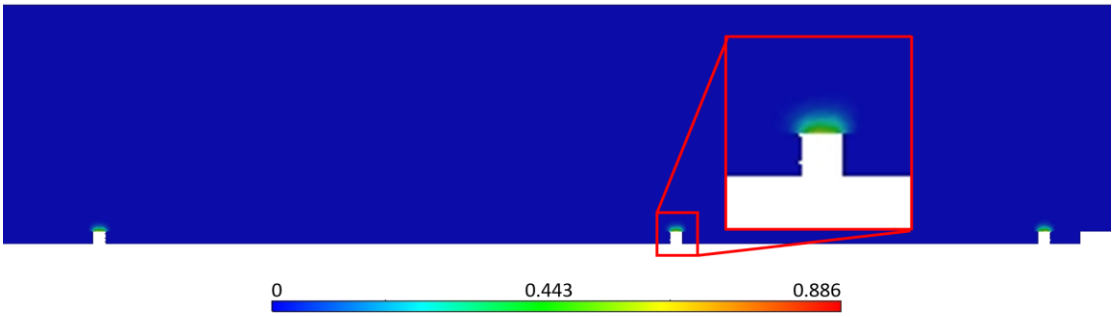


Figure 35 Convective Courant number

5.3 Discussion

Our findings suggest that by creating a new part that redirects the flow of oil, shields gears and stores oil in the upper region, a reduction in drag torque can be accomplished. The reduction with our part is 16.3% of the churning losses at a speed of 50km/h and has a trend of an increase in reduction with an increase in speed. This means that there is a great deal of energy that can be saved with a simple plastic part that is installed in existing mounting holes. The new part has also a very rough design because it was made during a learning period of the CAD-software Creo. With more experience in these software's, even better results could be accomplished.

An interesting result can be seen from the observation of the trend study, where the reduction at 50km/h is almost the same as at 100km/h in percentage. This means that in actual drag torque numbers the reduction in Nm is higher at increased speeds due to the increase in drag torque at higher speeds. Even though this is a good result, there is no data documenting how the reduction looks within this interval. The behaviour of the drag torque reduction trend can be either parabolic with a peak or linear. Further research would be beneficial here because this is the operating interval that most cars are being used within. The limitations in computing power was the reason for why further investigation wasn't done here.

When it comes to speeds below 50km/h, the new part was surprisingly ineffective in reduction of the churning losses. It shows a steep trend from almost no reduction of churning losses at 20km/h to 16.3% reduction at 50km/h, additional investigation in this region would be beneficial.

The investigation of the collected data strongly suggests, that a modification of this kind can be very beneficial for vehicles that are used at higher speeds but for vehicles used below 50km/h in city traffic, it may not be of interest. In case of a company like CEVT, that develops cars which are frequently driven in both cities and highways at higher speeds, further research of how to design a gearbox around the flow of oil can therefore be of interest.

The discoveries from the multiphase simulations part, align with the expectations. With increased diameter of the holes, the discharge mass flow increases as well. The importance of these results is mostly related to the overall functionality of the new part when it comes to storing and releasing the oil.

The biggest limitation of performed simulations is missing incoming mass flow of the oil which would introduce additional turbulence and dynamics. However, due to the complexity of the physics, even the relatively accurate results from the simulations are extremely important and can be used for further development associated with the oil discharge. As an example, future research can be the utilization of the drained oil and its redirection.

6 Investigating experimental tests

The new part, designed and analysed virtually, would benefit from an experimental test in a real gearbox. This comes with several challenges that needs to be investigated and designed for. The two steps of this investigation are:

- Design for assembly and prototype build
- Material properties in the gearbox

A gearbox is complex, and the use of space is limited, therefore the investigation must be made that states an order of assembly. The material available is also not rated [5] for withstanding the oil in the gearbox and has to be investigated if this is also true for a shorter period of time. Here the method and results are presented simultaneously.

6.1 Design for assembly and prototype build

First step in the experimental testing is to make it possible for the new part to be built and installed in the transmission. By doing so, the part is put through the phase of designing for assembly and designing for manufacturing of a prototype. The process of the assembling and taking the gearbox apart is studied along with the personnel in the workshop that have expertise in this area. An order of assembling the components in the gearbox is then created that requires the least amount of modifications of the new part.

After taking the gearbox apart, it was concluded that the new part had to be split in two pieces, because the preferred way of assembling it, is to insert the shafts into one half of the casing and then put the other half of the casing on as a lid. Since the new part lays in between of the gearwheels it must be assembled along with the shafts and afterwards it will be screwed to the other part of the casing. Figure 36 shows the mounting point, labelled 1, with the holes that are used to attach it to the casing in the upper region of the figure. The Bridge that lays in between the gearwheels is labelled 2.

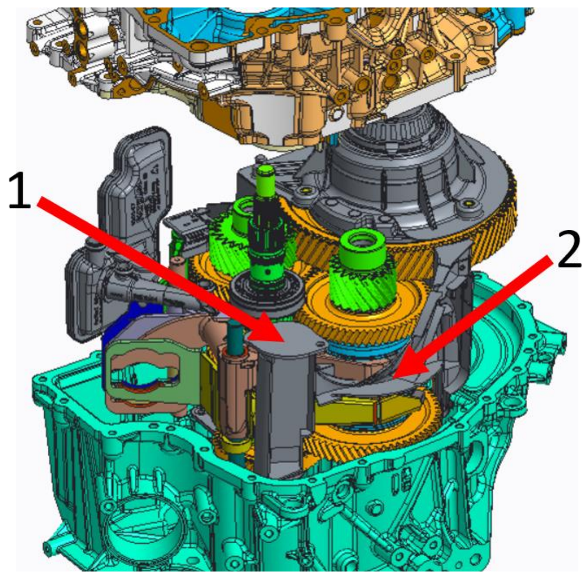


Figure 36 The position of the gearbox under assembly

The chosen mounting solution is to split the new part between Storage 1 and the Bridge. This way, Storage 1 can be screwed to the upper half of the casing and lowered down with it. When lowered down it will lock into the Bridge with help of a pin and matching slot as shown in Figure 37. When being assembled, it is possible that the pin will stick on the edge of the slot and will bend away the Bridge. In order to prevent this from happening a chamfer is made on the pin. Despite these precautions there is still a possibility that the pin will get stuck because of the friction at the walls in the slot. When this happens, there is an extra support from the casing, which is displayed in Figure 21.

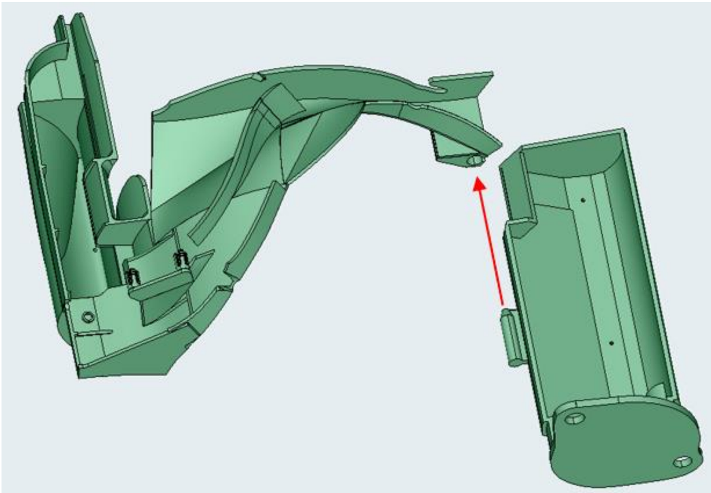


Figure 37 Pin that slides into a slot while assembling the gearbox

Afterwards, the part was divided into three smaller parts as shown in Figure 38, to ensure that the printed surfaces would have the highest possible quality. Then it is converted into a file that the printer can read. The printer uses FDM (Fused Deposition Modelling), a technique, which works with the use of supports for the parts hanging in the air shown with green colour in Figure 38. This technique is not optimal because it creates a rough surface in the area, where the supports are attached to the part. In order to avoid this, the surface which is in direct contact with the oil flow should be facing up in the printer. Material used in the available printer is ABS plastic.

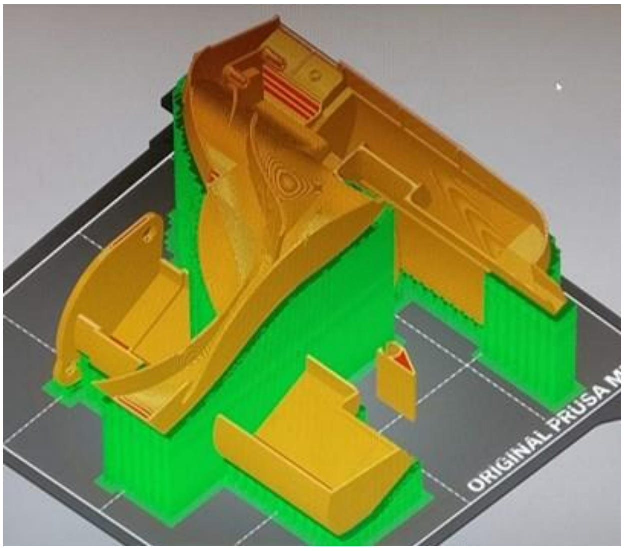


Figure 38 Visualization of supports used in 3D-printing

The result from the printing process is shown in Figure 39. The surface, where the oil flows turned out to be very smooth so no further aftertreatment is required. Tolerances of the pins that go into matching holes in the casing where kept, which was verified when test mounting was performed.

Figure 39 The printed new part

6.2 Material properties in the gearbox

The presence of the vibrations, heat exposure and forces in the gearbox can negatively impact the new part. In order to make sure that the material can withstand these negative effects, several precautions were taken, and tests were conducted. The considered parameters are:

- Withstand the forces acting on the new part
- Withstand the heat
- Withstand the chemicals

Regarding the ability of the new part to withstand forces and vibrations, a reference from the original part is taken that is designed to withstand an entire lifetime of vibrations. Making the new part similar in terms of the walls thickness and fastenings, one can assume the same durability.

When it comes to coping with the heat and the chemical composition of the oil, several tests are performed. First test was conducted to see if the ABS plastic can withstand the oil at room temperature. The test procedure consisted out of submerging a thin plastic wire in a container filled with gearbox oil. The plastic wire is then compared with a reference every ten minutes for one and a half hour by performing a bending test, where two pieces are held horizontally and checked for how much they bend under their own weight.

Finally, in order to test how well does the ABS-plastic withstand the heat combined with the oil, a coil of the plastic wire is placed in heated oil for 30 minutes, which is done for 40, 60 and 80 degrees Celsius. Every test is performed with a new plastic coil that is placed in the oil after the desired temperature has been reached. Same test is then performed with a reference by holding them next to each other and letting the gravity pull out the coil. It was concluded from the results that, at 40 degrees there were no difference compared with the benchmark. At 60 degrees a small difference became visible however it was so small that it would not affect a solid printed part. When performing the final test at 80 degrees the plastic wire started to straighten out already after a few minutes in the heated oil. Compared with the benchmark after 30 minutes it became clear that the plastic cannot withstand this heat, therefore a test with the prototype at this temperature is not possible.

6.3 Discussion

It is stated from this investigation that with the resources available at CEVT, there is a possibility to make experimental tests with the new part. The material used can withstand the environment of a gearbox, be printed strong enough and installed in the gearbox. This means that when making modification within the gearbox that involve the plastic part, it is easy and fast to print it and run a test in a testing rig. One of the reasons to why the oil temperature withstanding tests were conducted was that it was stated [5] that ABS-plastic cannot withstand the oil. It was surprising that the ABS-plastic performed so well in the tests and therefore, with the input from our tests, it can be said that a shorter test can be conducted with this method. Future research should not take this as an indication of that endurance tests can be conducted with a part manufactured this way.

7 Conclusions

In this master thesis project, a way of efficiency optimization in 7DCT transmission was investigated. Since the efficiency of the transmission is affected by many factors, this project focused only on lubrication system due to time limitations. In the end, the project consisted out of three main areas, design, simulations and experimental testing. The main evaluated parameter in terms of lubrication efficiency were the churning losses.

After an extensive work in CFD and CAD software, with analysing the current solution, a new part was created that manages to reduce the churning losses. The information gained from the literature study of previous work is supported by our findings. It shows that by protecting rotating parts from incoming splash and reducing the oil level in the sump by installing oil traps in the upper region, losses from churning will be reduced. In this case the reduction reached around 17% between 50 and 100 km/h and shows that the absolute and the percentual reduction increases with an increase in speed.

To wrap up the simulation part, functionality of the oil storage container was tested by performing VOF simulations of oil drainage in ANSYS Fluent. Collected data from the VOF simulations proved, that it is possible to fill up the containers in the upper region and still have a complete drainage once the transmission stops.

Parallel to the work conducted in the CFD and CAD software, an investigation was conducted focusing on the possibility of making a test of a new part with the available resources at CEVT. The material ABS-plastic used in the 3D-printers was rated through experimental tests to withstand gearbox oil at 60 degrees Celsius for 30 minutes. A new part was then printed and a method of installing it was developed.

Future work should be aimed at researching how to distribute the oil collected in the containers with help of models for cooling and lubricating the gears. A model is already created in this thesis that calculates the rate of discharge from a hole at a given place and size.

8 References

1. Study of power losses in a two-speed dual clutch transmission
   https://www.researchgate.net/publication/263922909_Study_of_Power_Losses_in_a_Two-Speed_Dual_Clutch_Transmission
   Access date: 2020-06-02

2. *Price, D., Smoothed Particle Hydrodynamics: Things I Wish My Mother Taught Me,* *https://arxiv.org/pdf/1111.1259.pdf, accessed 03.06.2016*
   Access date: 2020-06-02

3. *Adami, S., Modelling and Simulation of Multiphase Phenomena with SPH, Doctorate Dissertation, Technical University of Munich, 2014*
   Access date: 2020-06-02

4. *Hartono E A, Golubev M, Chernoray V. PIV Study of Fluid Flow Inside A Gearbox[C]//PIV13; 10th International Symposium on Particle Image Velocimetry, Delft, The Netherlands, July 1-3, 2013. Delft University of Technology, Faculty of Mechanical, Maritime and Materials Engineering, and Faculty of Aerospace Engineering, 2013*
   Access date: 2020-06-02

5. ABS Chemical Resistance http://k-mac-plastics.com/data-sheets/abs_chemical_resistance.htm
   Access date: 2020-06-02

6. http://plaza.ufl.edu/jeffjtd/SitePics/kernelpic.jpg
   Access date: 2020-06-02

7. https://www.researchgate.net/publication/224829233_Volume-of-Fluid_VOF_Simulations_of_Marangoni_Bubbles_Motion_in_Zero_Gravity
   Access date: 2020-06-02

8. Price, D., *Smoothed Particle Hydrodynamics and Magnetohydrodynamics*, Journal of Computational Physics, vol. 231 (**3**), 2010
   Access date: 2020-06-02

9. Detailed Loss Modelling of Vehicle Gearboxes
   https://www.ep.liu.se/ecp/043/048/ecp09430059.pdf
   Access date: 2020-06-02