Gear interlocking effect study using CFD

Peyman Jafarian M.Sc.

Simulation Engineer

Vicura AB, Sweden

ABSTRACT

Estimating the drag loss and predicting the oil flow distribution within the transmission has always been a challenge in design phase. Trail-and-error methods are time consuming and are not accurate. Moreover, performing tests is an expensive process and not flexible to changes. In this study it is tried to investigate the capabilities of two CFD (Computational Fluid Dynamics) solvers in estimating the drag loss and predicting the flow field. Extensive tests are conducted to validate the simulation results.

1. INTRODUCTION

Splash lubrication is commonly used in low to medium speed transmissions such as automotive gearboxes in which lubricant is projected by the rotation of the gears. The main disadvantages of this type of lubrication are the generation of significant power loss by churning the oil and lack of precise control in term of lubricant supply.

Power loss in drive-train system is one of the major contributors in the total power loss of transmissions. These losses have direct impact on the fuel consumption and the efficiency of the vehicle is in terms of fuel economy and emission levels. Therefore, predicting the power losses accurately within the power-train system and the lubrication system optimization is crucial.

However, it is not an easy task to predict the power loss and design an efficient lubrication system at an early stage of design process due to the complex nature of the flow. Therefore, development of this lubrication technique relies on trial-and-error approach. In automotive applications, for example, transparent housings are frequently used to visualize the oil distribution inside the gearboxes. Such a method is expensive and cannot be utilized at the early design phase of the transmission development. Besides, if any changes are made at the design stage a new hardware is required for testing which makes this method quite inflexible to modifications.

The other challenge is to estimate the drag loss accurately. Many efforts are made to develop mathematical based formula to calculate the power loss within the transmissions. Most of the mathematical formulas are based on empirical calculations and coefficients are calibrated according to the test data. This makes it difficult to use mathematic based drag loss estimation precisely at the early stage of design when hardware is not available and tests cannot be performed

Emphasis in this study is on the investigation of the CFD simulation as a tool to estimate the drag loss and predict the oil distribution within transmissions. The accuracy of the softwares will be examined by comparing the test data with the simulation results.

2. Power loss within transmissions

Total loss in gearbox can be divided into load-dependent and load-independent loss. Exploring the load-dependent loss is out of scope of this study.

Load-independent loss is related to the interaction between the lubricant and the rotating elements. Load-independent loss within the geared system can be divided into power loss due to the lubricant squeezing and power loss due to the oil splashing or churning loss.

Lubricant squeezing power loss occurs due to the gears meshing. During the engagement process, volume between gear teeth is reduced causing lubricant to pressurize and ejects in the axial direction. Churning loss is related to the shearing of oil due to the rotation of gears or other rotating components. Viscous forces and pressure forces acting on the gears and other rotating components form the churning loss in a transmission.

3. Experimental Set up

3.1 Drag Loss Measurements

A simple single-shaft test rig was used to measure the drag loss. The measurements were performed for gear wheel within a box. The box dimensions are 820 × 520 × 320 mm. The gear wheel model is spur gear in SLS, with a diameter of 224mm and thickness of 28mm, and 67 teeth. Schematic drawing of the test rig is shown in Figure 3-1.

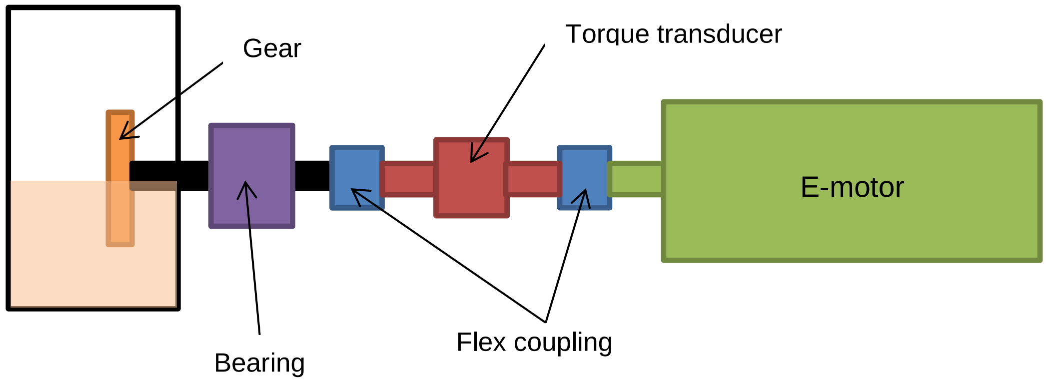

Figure 3-1 schematic of the test rig

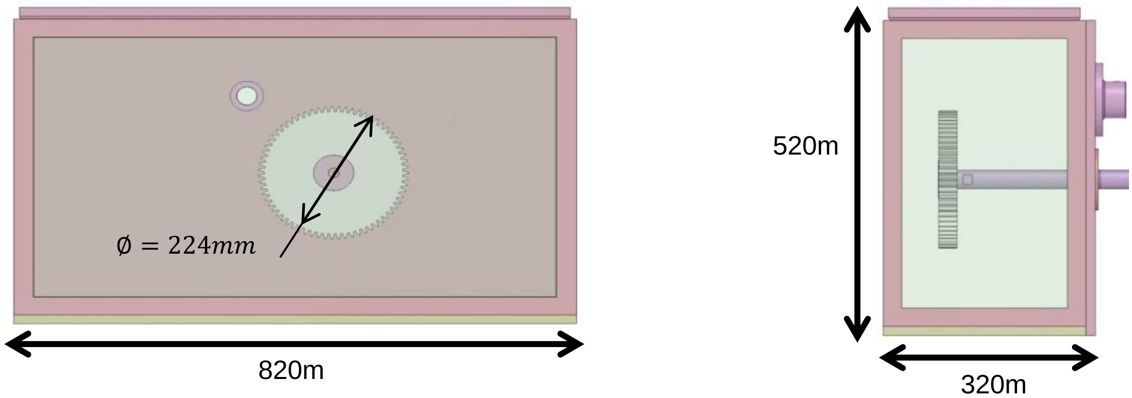

Figure 3-2 dimensions of the test box and gears

To measure the drag loss accurately, torque sensor with nominal range of +/-20Nm and accuracy of 0.1% of full scale is used. Oil temperature is monitored during simulation and it remained constant to ambient temperature, 20+/-1°C. Sampling frequency is 100Hz for high speed and 10Hz for low speed operating conditions. Each test was run for at least 3 minutes to get stabilized drag torque. The final value of the churning loss is obtained by time-averaging the steady part of drag loss signal.

3.2 Flow Field Visualization

To study the flow field and specifically the velocity field calculated by the CFD solvers, tests are performed to visualize the velocity field for the gear pair. The background and basics of this specific type of measurement are explained below.

An optical, non-intrusive method which is related to both the flow visualization and the optical point techniques have been developed over the last 20 years called Particle Image Velocimetry or PIV [1].

This technique can provide an accurate quantitative measurement of the instantaneous flow velocity field across a planar area of a flow field.

PIV measurements were carried out at Chalmers University of Technology for a gear pair. A new simplified model of a gearbox was design and developed based on a standardised FZG back to back test rig. The test rig is suitable for optical measurements. The walls of the test section are transparent. Test gears are made of PMMA plastic which make them transparent too. Severely hydrotreated process oil, Nytex 810, was used as test oil. The refractive index of this oil is close to PMMA. Detail of measurements and test set up can be found at [2]. Tests are done for two pitch velocities of 1.1 m/s and 0.52m/s of the larger gear.

4. Problem Description

Single gear and gear pair rotating in oil bath are the configurations chosen for this study. For single gear case churning loss will be calculated and compared with the test data. For gear pair flow field and specifically velocity field obtained from the simulation will be compared with PIV measurements.

4.1 CFD solvers

Two CFD solvers will be used to perform this study. The first CFD solver is capable of handling the collision and solid contacts between the objects. Moreover, the need for mesh generation is eliminated.

In the second CFD code the fluid domain is volume meshed by the software and actual geometry is preserved. In the following section the software capabilities and specifications will be discussed in detail.

4.1.1 First CFD code

This software uses a simple structured grid with the flexibility of deformed, body-fitted grids. The advantage is that it eliminates the complicated mesh generation process and reduces the pre-processing time to set up a case. Moreover, mesh is independent of the geometry in the domain. The consequence is that each cell must be able to handle both arbitrary geometry and flow areas. Software has the capability of the nested block to refine the mesh locally by defining the cell size or total number of cells. Furthermore, mesh can be refined separately for any axis. Setting up the case and assigning the physics is simple and it reduces the pre-processing time.

To perform the simulation, geometry will be inserted into the hexahedral mesh block. To capture the details of the geometry like curved surfaces or holes, software uses a method called FAVOR^{TM} (Fractional Area/Volume Obstacle Representation). The action of interpolating the geometry over the mesh block is called Favorizing. This action is shown in Figure 4-1.

With FAVOR algorithm, each cell has a volume fraction, V_F and 3 area fractions A_F on the faces. Together they define solid surfaces as a plane through the cell. The governing equations are formulated to include the ratios.

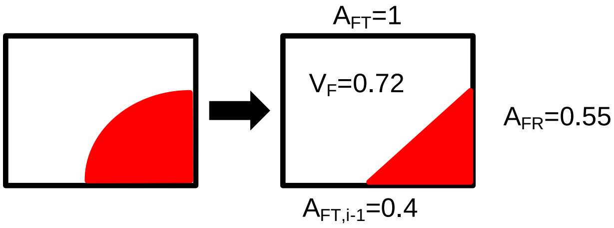

Figure 4-1 conversion of actual geometry with FAVOR to the geometry solver uses for calculation

The advantages of this method are that it is possible to change the geometry without changing the mesh. Moreover, in this method mesh is stationary and will not move (slide or morph). However, the disadvantage is that it is hard to control the mesh near the wall and have a suitable prism layers to solve the turbulent boundary layer. Currently it is possible to have conformal mesh to control the cell size near the wall but it cannot be applied to rotating objects. Moreover, if the geometry has small details like openings with small dimensions, a fine mesh is required to capture those details.

Mesh Setting*.* 5 mesh blocks are used to discretize the geometry. The cell size in the innermost block is 1mm this block is highlighted by light blue color in Figure 4-2. Cell size in the next mesh block which is highlighted with green color in Figure 4-2 is set to 1,5mm to avoid sudden change in cell size between each block. In the next mesh block, which is highlighted by red color in Figure 4-2, cell size is set to 3mm. the cell size in the next two blocks are 6mm and 8mm respectively. As it can be seen from Figure 4-3 cell size is fine enough to capture the gear teeth accurately but it is still not 100% the same as the original CAD file. In this case, however, the tolerance between the original geometry and FAVORized one is in acceptable range.

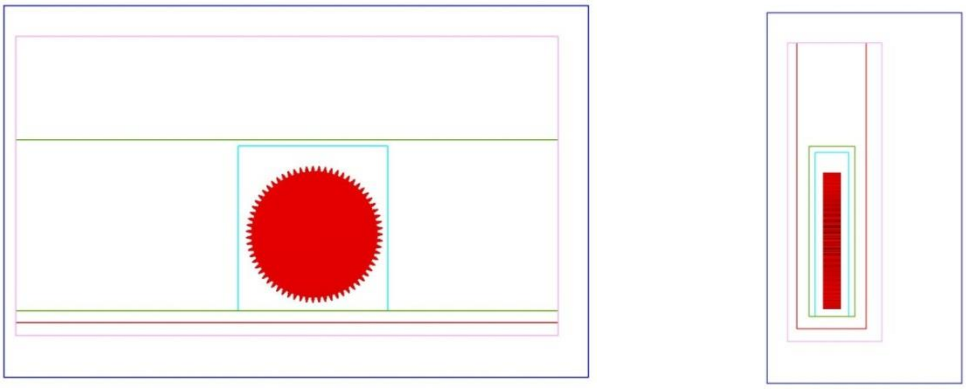

Figure 4-2 mesh blocks use to define cell size

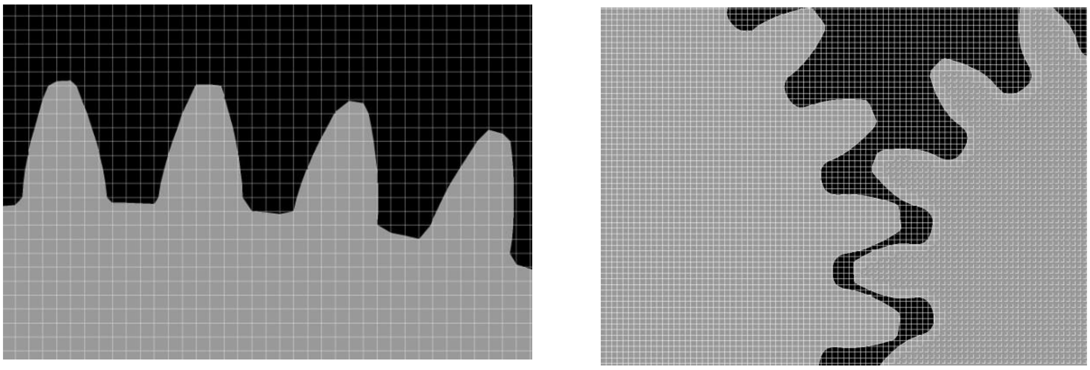

Figure 4-3 plane section showing the mesh and geometry

The simulation model consists of 11.5 million cells. For gear pair model one mesh block with cell size of 0.8mm is used. Total number of cells is 7 million.

Solver Setting. Air entrainment model is activated to consider the effect of air coming to the sump along the gear teeth. Gravity force is activated in the model. Rotation is defined for the gear. Flow is assumed to be a laminar, viscos flow. Implicit solver is used to calculate the pressure field. Explicit solver is used to calculate the viscous stress. Advection and moving objects also use the explicit algorithm.

When densities of the fluid and gas differ significantly, then small variations of pressure within the gas and gas inertia can be neglected compared to those in the fluid. For example, the ratio of water to air densities is about 1000. In these cases each void region can be treated as a uniform pressure region. Such flows are described with the one-fluid model with gas boundaries treated as free-surfaces. If the gas does not undergo volumetric changes (i.e., compression or expansion), then it can be further assumed that the pressure of the void regions is constant in time [3]. Solver settings are the same for the gear pair simulation.

By considering the physics of this problem, one fluid model works well; therefore, CFD code won't calculate any variables in the void region.

4.1.2 Second CFD code

Unlike the first CFD code, the volume mesh in this software will follow the geometry shape. Mesh generation is automated. Both the surface mesh and the volume mesh are generated with the software mesh generator. Polyhedral cell is chosen for this study. Polyhedral meshes provide a balanced solution for complex mesh generation problems. They are relatively easy and efficient to build, requiring no more surface preparation than the equivalent tetrahedral mesh. They also contain approximately five times fewer cells than a tetrahedral mesh for a given starting surface.

Mesh setting. One layer of orthogonal prismatic cells next to wall surfaces is defined. This layer of cells is necessary to improve the accuracy of the flow solution

Volume refiner is used to refine the mesh around the gear region. Cell size of 3mm is chosen for this purpose. Mesh is coarsened further away from the gear to reduce the total number of cells. Section of the mesh is shown in Figure 4-4. Model consists of 284,000 cells for single gear.

Sliding mesh approach is used to model the gear pair case. Interface of each rotating region cannot intersect with the interface of the other rotating region. Therefore, one gear has to be scale down to make this approach works. In this simulation the smaller gear or pinion is scaled down. Figure 4-4 to the right shows the gap between two gears. This model consists of 700,000 cells.

Solver setting. Segregated Eulerian multiphase flow solver in combination with VoF model is selected to tackle this problem. Order of discretization is 2nd order in space and 1st order in time. Flow is assumed to be a viscous, laminar flow. Implicit algorithm is chosen to march in time.

Time steps. Calculation of the time step size is based on the requirement for CFL number and maximum rotation of the gear per time step. The truncation error associated with the time integration scheme is also considered in the time step size estimation. Number of inner iteration is set to 10. Solver will solve for both air and oil phases and interaction between phases is considered.

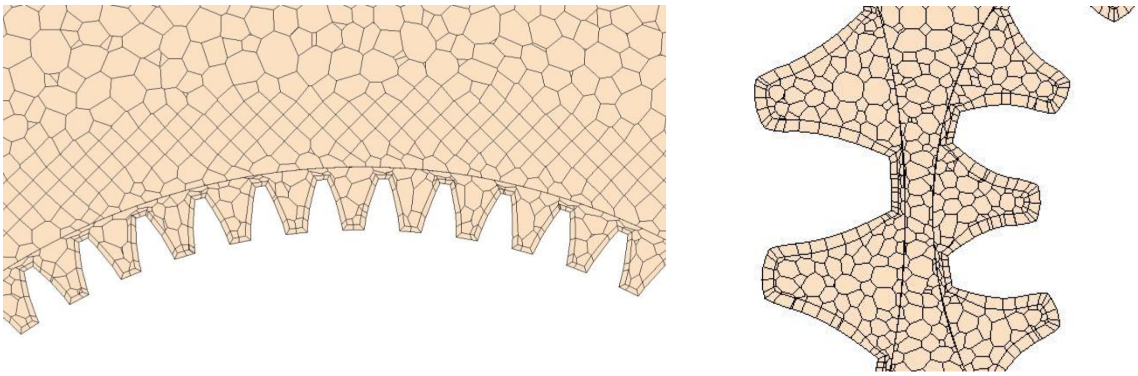

Figure 4-4 section of the mesh for single gear (left) and gear pair (right)

4.2 Oil properties

Multigear® MTF HD oil is an ultra-high performance heavy-duty synthetic automotive gear lubricant. It is designed for use in heavy duty synchromesh manual truck transmissions. Oil properties are shown in Table 4-1.

Density, 15°C [kg/m^3]	Kinematic viscosity,100°C [mm^2/s]	Kinematic viscosity,40°C [mm^2/s]
847	9	54

Table 4-1 oil properties used for drag loss measurements

4.3 Boundary Conditions

All the boundaries were set to a no slip condition. It means that on the wall there is no relative velocity between the walls and the fluid. Pressure is set to ambient conditions.

4.4 Operating Conditions

During the test drag loss is measured for a range of speeds and oil levels. For the sake of simplicity two rotational speeds (500 RPM and 750 RPM) and one oil level (50% of the gear radius) are chosen for this study. For the second comparison pitch velocity of 0.52 m/s and 1.1 m/s of the larger gear are chosen for simulations.

5. Results

5.1 Drag loss comparison for single gear

		Test data	**First CFD solver**	**Error %**	**Second CFD solver**	**Error%**
Churning loss [Nm]	**500 RPM**	1.03	2.39	132	1.06	2.9
	750 RPM	1.75	3.96	126	2.28	30

Table 5-1 comparison of the churning loss

Churning loss obtained from the first CFD solver and the second CFD solver are compared with the test data. It can be seen from Table 5-1 that the churning loss predicted by the second solver for rotational speed of 500RPM is in a good agreement with the test data. This value is obtained on relatively coarse mesh.

The accuracy is reduced for 750 RPM and error is 30% for the second CFD solver. The accuracy can be improved by increasing the mesh resolution close to the wall. However, considering the fact that this result is obtained with a coarse mesh in a short time frame the accuracy is acceptable.

The drag loss obtained from the first CFD solver is not in good agreement with the test data and exhibit relatively higher error. Skin friction drag is one major part of the total churning loss. An accurate estimation of the skin friction drag requires a well resolved boundary layer. As the grid gets less orthogonal to the geometry wall it will result in requirement for huge number of tetrahedral elements to get reasonable accuracy. This demand for high resolution mesh will result in excessively high simulation time. Therefore, the present mesh resolution near gear geometry might be the reason for the error.

5.2 Flow field comparison for gear pair

Figure 5-1 shows the comparison of the velocity field between the PIV data obtained from the test and time-averaged velocity calculated by the second CFD solver

Contours of velocity magnitude are plotted on a plane section passing through the center of the gears. As one can see from this comparison, velocity field is in a good agreement with the test data both quantitatively and qualitatively. Prediction of the boundary layer thickness is comparable. One discrepancy can be seen around the pinion. Extension of boundary layer predicted by the second CFD solver away from the gear terminates earlier than the test data. The reason is that the gear is scaled down to avoid intersection between two rotating regions.

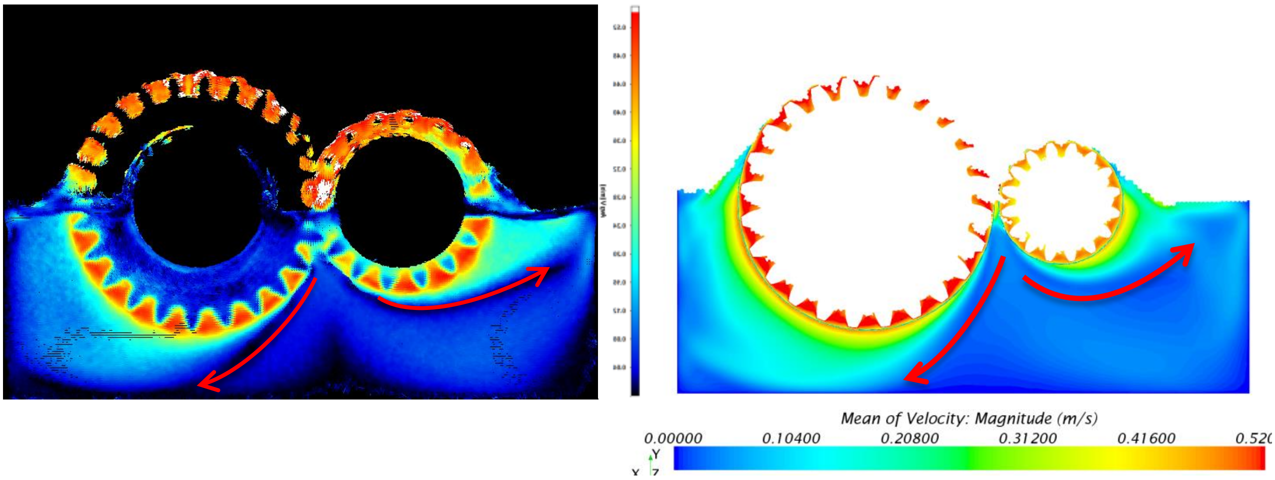

Figure 5-1 Comparison of the PIV data with the second CFD solver results

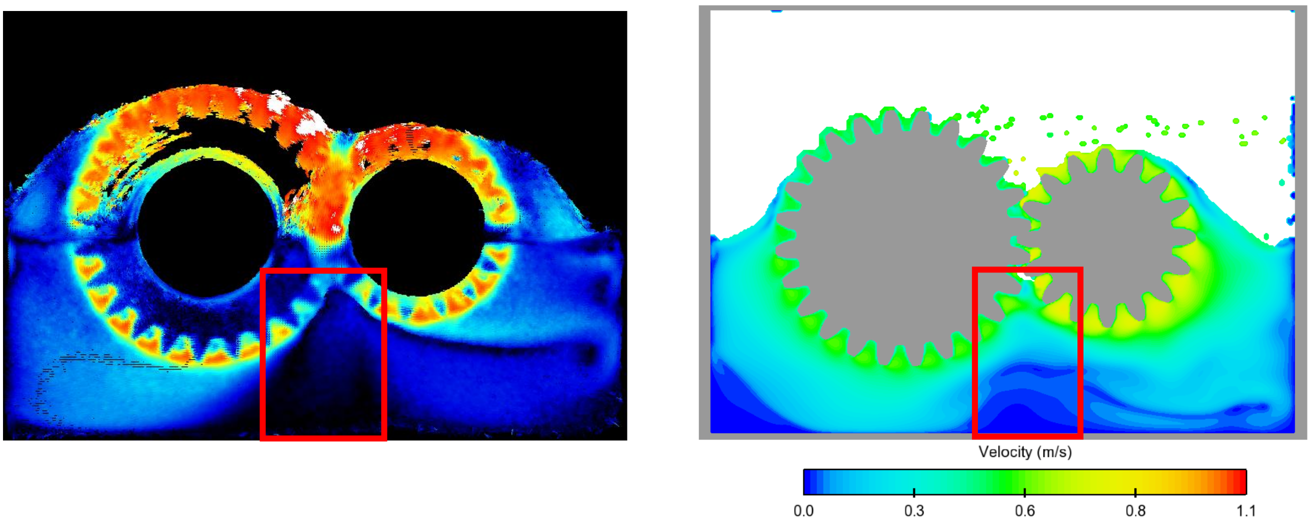

Figure 5-2 Comparison of the PIV data with the Flow-3D results

As it can be seen from Figure 5-2 the first CFD solver over predicts the velocity field in the region highlighted by the red rectangle and it diffuses more in comparison with the test data. Moreover, the splash by the larger gear is not comparable with test results and one can see oil is splashed over the pinion gear to the other side of the box which is not the case in the PIV data.

However, due to the postprocessor limitation it was not possible to make a time-averaged of the velocity field and 2D contour plot of the velocity is instantaneous result. Therefore, the time-averaged velocity field might be slightly different than instantaneous result.

6. Summary and Conclusion

The results of two CFD solvers with different approach in solving the flow field were compared with the test data. The first CFD solver is the code with capability of modeling the solid contacts and collision in a robust way. Setting up the simulation is quite fast and it eliminates the efforts required for mesh generation. The mesh resolution can be changed easily and capability of nested block makes it possible to refine the mesh locally. This code can give a general overview of the flow field within a short time frame. The second CFD solver on the other hand has difficulty with modeling the solid contacts accurately in a short time frame (current version). However, a volume mesh which follows the geometry gives a better representation of the components boundary. Having the possibility to define prism mesh along the surface boundary, results in improved estimation of the near wall gradients.

Comparison was made by looking at the churning loss (pressure and shear loss) that each code predicted and the test data. Due to the fact that first CFD solver predicted the drag loss for single gear poorly; comparisons were ignored for the gear pair case. The other parameter which was used for comparison was the velocity magnitude. PIV test results were compared with the velocity field obtained from CFD codes plotted on a plane section passing through the center of the gears. Results for both solvers were comparable to test data. However, the first CFD solver provided the data in a shorter time frame as compared to the second CFD solver.

Acknowledgment

The author would like to thank *Flow Science* for providing the license to perform this study. And also XC Engineering and Easysimulation (Former Simtech Systems Inc. Oy) for supporting us with the technical issues. Moreover, I would like to thank Torbjörn Kvist and Usman Afridi for their useful feedbacks.

Bibliography

[1] M. Raffel, C. Willert and J. Kompenhans, “Particle Image Velocimetry A practical guide.,” *Springer,* 1998.

[2] E. Adi Hartono, A. Pavlenko and V. Chernoray, “Stereo-PIV Study of Oil Flow Inside a Model Gearbox,” in *17th International Symposium on Applications of Laser Techniques to Fluid Mechanics*, Lisbon, Portugal, 2014.

[3] Flow Science, “Flow-3D help file,” 2014.