**Proceedings of the 7th GACM Colloquium on Computational Mechanics
for Young Scientists from Academia and Industry
October 11-13, 2017 in Stuttgart, Germany**

Simulation of Oil Jets for Piston Cooling Applications Using Mesh Deformation and the Level Set Method

Loïc Wendling[1]*, Karyofyli Violeta[1], Markus Frings[1], Anselm Hopf[2], Elgeti Stefanie[1] and Marek Behr[1]

Micro Abstract
The level set method is used to understand and design oil jets used to cool pistons of internal combustion engines. Different jet configurations ranging from laminar to atomized are presented and compared with experimental results. Exploiting the flexibility of the space-time finite element method, the reciprocating movement of the piston is modeled by using mesh deformation. The resulting multi-physics simulation realistically represents the main flow features.

[1]Chair for Computational Analysis of Technical Systems, RWTH Aachen University, Aachen, Germany
[2]Ford Motor Company, Aachen, Germany
***Corresponding author**: wendling@cats.rwth-aachen.de

1 Introduction

Active cooling strategies for pistons are currently investigated as a way to improve the efficiency of internal combustion engines and reducing harmful emissions. Pistons are notoriously hard to cool because of their high velocity and complex kinematics. One solution consists of using lubricating oil sprayed underneath the piston as a coolant to regulate the temperature. The lubricating oil is preferred due to its wide availability and high specific heat allowing significant heat removal. The new generation of pistons includes a so called cooling gallery to bring the fresh oil even closer to the hot piston rings and the combustion chamber. The challenge is to estimate the efficiency of a set of design parameters including, nozzle diameter, nozzle to piston spacing, cooling gallery geometry and mass flow rate of the cooling oil. The objective of this work is to simulate the resulting oil jets using modern computational fluid dynamics tools like the level set method.

2 Method

In multiphase flows, the complex motion between two heterogeneous phases with steep discontinuities needs to be accurately resolved including the representation of the interface between the two phases. The two main approaches for solving these problems are interface capturing and interface tracking methods [3]. The first implicitly stores the information about the position of the interface into an additional field. The latter uses the mesh as an explicit representation of the interface between the two phases. The level set method [9] is an interface capturing technique where the signed distance to the interface is computed. This technique is suitable because it can handle the large and complex motion of the oil jet in the air domain. The disadvantage is its lack of mass conservation which can be influenced by the mesh resolution and the level set reinitialization. A global volume correction is used to balance the lack of mass preservation. Opposite to the Volume of Fluid method [8], another interesting feature of this method is that the interface is directly available and allows a more accurate estimation of the surface tension force. This force is incorporated into the fluid equations as an additional source term acting

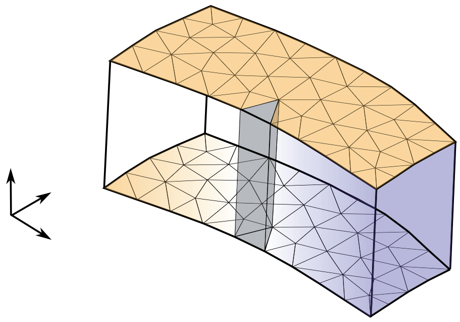


Figure 1. Space-time slab

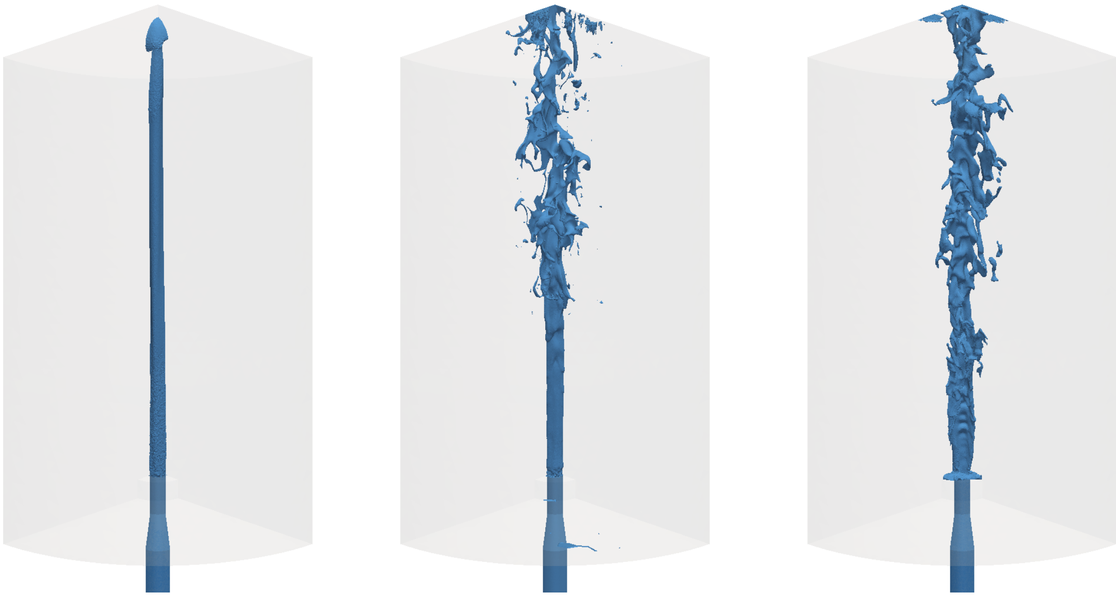
only in the vicinity of the interface. Furthermore, the Laplace-Beltrami approach [5] applied in this work uses differential geometry to avoid the direct computation of the curvature. For the flow, the multiphase Navier-Stokes equations are solved where the density and viscosity depend on the level set function. Both phases are assumed to be composed of Newtonian fluids and do not mix. The discretization used here is the space-time finite elements method [2]. The main feature of this method is the concurrent discretization of space and time using finite elements, forming a so called space-time slab shown in Figure 1. This method is especially useful when using mesh deformation techniques because the time slab can implicitly account for the mesh deformation (Arbitrary Lagrangian-Eulerian approach). The Navier-Stokes equations and the level set advection are solved in a partitioned way and iterated until sufficient convergence is obtained. Steep discontinuity within the velocity and pressure fields are unwanted side effects resulting from multiphase flows. A way to soften these effects is the use of discontinuities capturing methods where diffusion is added to the regions with high gradients [4]. The turbulence at subgrid level is also considered by using the variational multiscale turbulence model [1]. For the second test case, the mesh deformation is handled through the space-time slab. The top of the slab, laying on the next time level, is deformed by considering the mesh at this level as an elastic material. Here, the stiffness of the elements is made inversely proportional to their size. When the mesh is too much deformed, a new mesh is generated and the solution is mapped to it. Here, the periodic movement of the piston is an advantage, because the expensive projection does not need to performed again at the next cycle because the mapping between meshes is stored.

3 Results

Two examples will be presented. The first one with several states of jets from laminar to atomized. The second example shows a laminar jet interacting with a moving piston. Detailed simulation requirements and mesh design will be presented for each example.

3.1 Oil jet atomization

In this first example, the properties of both oil and air are kept constant but the inflow mass flow rate will be increased to obtain a jet breakup and atomization. For the semi-turbulent and turbulent cases, random perturbations mimicking the noise coming from the oil pump and other components are added to the inflow velocity profile to facilitate the onset of atomization. Symmetry boundary conditions are used and only a quarter of the full chamber is simulated. There are two objectives for this test case: guiding the subsequent studies in terms of simulation parameters and understanding the breakup mechanics leading to jet atomization. For classical pistons, oil atomization is used because it creates a spray with numerous droplets increasing the cooling contact area between oil and surface underneath the piston. In Figure 2, the influence of the inflow mass flow rate is shown including the breakup level of the jet core and the disturbances increase by the Reynolds number. The atomization process significantly depends on the mesh refinement, especially in the vicinity of the nozzle exit. This region needs to be sufficiently



(a) Laminar ($1.6L/min$) **(b)** Semi-turbulent ($3L/min$) **(c)** Turbulent ($4.2L/min$)

Figure 2. Influence of the inflow mass flow rate

resolved in order to the oscillations coming from the nozzle to propagate downstream. Those perturbations slowly grow in amplitude and lead to the core breakup. Qualitative agreements are observed by comparing pictures from the experiment and the simulations.

3.2 Laminar jet interacting with a moving piston

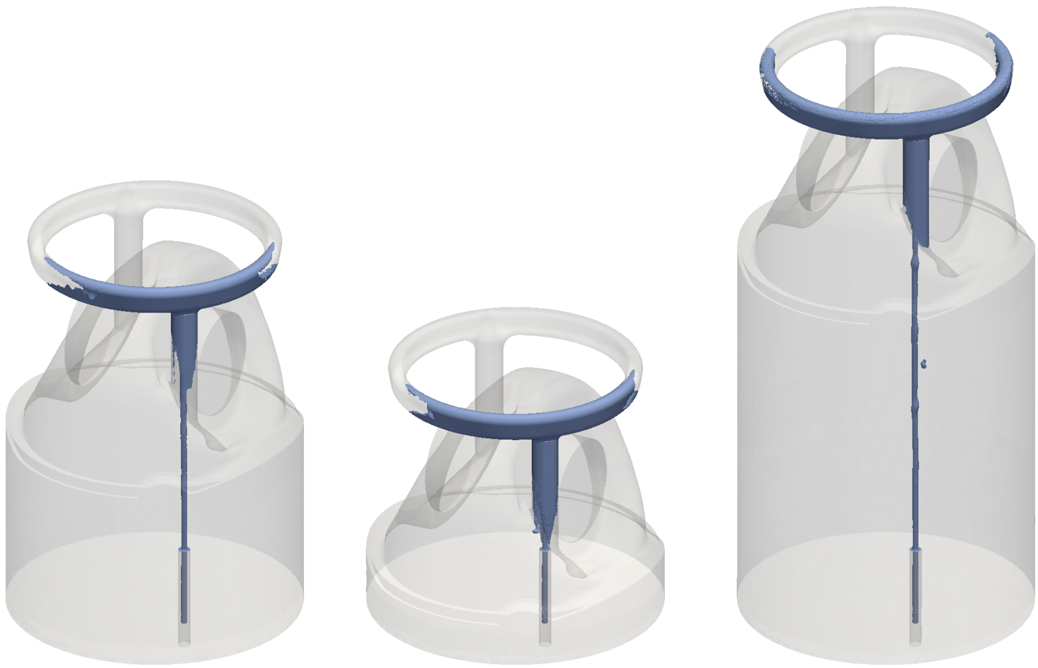
In the second example, a laminar jet is injected into a modern piston with a cooling gallery. The interaction between the rising jet and the oscillating piston is handled by the space-time finite elements method together with appropriate boundary conditions. The displacement of the piston is governed by the equation of motion [7]:

$$x(t) = r\cos\left(\omega t + \frac{\pi}{2}\right) + \sqrt{l^2 - \left(r\sin\left(\omega t + \frac{\pi}{2}\right)\right)^2} \quad (1)$$

With x, r, l, ω and t being respectively the current position of the piston, the crankshaft radius, the connecting rod length, the engine rotational speed and the current time. This example takes advantage of the space-time finite elements method by seamlessly combining interface capturing for the jet and interface tracking for the piston movement into one framework. Throughout a piston cycle, the number of element varies from $8.1Mio$ to $8.3Mio$. A small elements size, $0.3mm$ is used inside the cooling gallery where the oil shaker cooling effect takes place [6]. The results in Figure 3 show the evolution of the oil filling when the piston moves toward the nozzle 3(b) and away from the nozzle 3(c). Inside the cooling gallery, the oil hitting the top and the bottom surfaces creating small droplets that are rapidly reabsorbed including the inertia effects. Over several cycles, the filling ratio of the gallery is tracked.

4 Conclusion

The influence of the inflow mass flow rate on the atomization level has been investigated and compared to experiments. This is a first step toward quantifying the sensitivity of each design parameter. Atomization is a complex phenomenon that requires more research to balance the significant computing resources with the desired level of details. Future improvement will include



(a) Initial position **(b)** Bottom Dead Center (BDC) **(c)** Top Dead Center (TDC)

Figure 3. Jet at different piston positions

adaptive mesh refinement based on the interface position to use the available computing resources more optimally. A piston with a cooling gallery has been successfully simulated under moving conditions. The oil in the gallery clearly follows the movement of the piston and sloshes against the wall including inertia effects. Interface tracking and interface capturing may be combined to take advantage of the strengths of each method. The oil properties, especially the viscosity, are sensitive to temperature fluctuations. In order to properly assess the cooling efficiency of a set of design parameters, future simulations will also solve the temperature field. Furthermore, the temperature in the piston will be interacting with the fluid simulation by mean of conjugate heat gradient method. The level set method proved to be powerful and versatile for complex oil motion.

Acknowledgments

This work was supported by the Alliance program between Ford Motor Company and RWTH Aachen University. Computing resources were provided by the Jülich-Aachen Research Alliance JARA-HPC.

References

[1] Y. Bazilevs, V. Calo, J. Cottrell, T. Hughes, A. Reali, and G. Scovazzi. Variational multiscale residual-based turbulence modeling for large eddy simulation of incompressible flows. *Computer Methods in Applied Mechanics and Engineering*, 197(1):173–201, 2007.

[2] M. Behr. Stabilized space-time finite element formulations for free-surface flows. *Communications in numerical methods in engineering*, 17(11):813–819, 2001.

[3] S. Elgeti and H. Sauerland. Deforming fluid domains within the finite element method: five mesh-based tracking methods in comparison. *Archives of Computational Methods in Engineering*, 23(2):323–361, 2016.

[4] T. J. Hughes and M. Mallet. A new finite element formulation for computational fluid dynamics: Iv. a discontinuity-capturing operator for multidimensional advective-diffusive systems. *Computer Methods in Applied Mechanics and Engineering*, 58(3):329–336, 1986.

[5] S.-R. Hysing. *Numerical Simulation of Immiscible Fluids with FEM Level Set Techniques*. PhD thesis, Dortmund University, 2007.

[6] MAHLE. *Pistons and engine testing*. ATZ/MTZ-Fachbuch. Vieweg+Teubner Verlag, 2 edition, 2016.

[7] G. Nasif, R. Barron, and R. Balachandar. Simulation of jet impingement heat transfer onto a moving disc. *International Journal of Heat and Mass Transfer*, 80:539–550, 2015.

[8] G. G. Nasif. *CFD Simulation of Oil Jets with Application to Piston Cooling*. PhD thesis, University of Windsor, 2014.

[9] S. Osher and J. A. Sethian. Fronts propagating with curvature-dependent speed: algorithms based on hamilton-jacobi formulations. *Journal of computational physics*, 79(1):12–49, 1988.