Investigation of Thin-Film and Lagrangian Models in OpenFOAM to simulate Wing-Mirror Soiling

Awadh Kapoor[1], Ravi Duggirala[2]

[1]Manipal Institute of Technology, Manipal University, KA, INDIA-576014, awadh_kapoor@yahoo.co.in
[2]Mercedes-Benz R & D India Pvt. Ltd., Bangalore, KA, INDIA-560033, ravi.duggirala@daimler.com

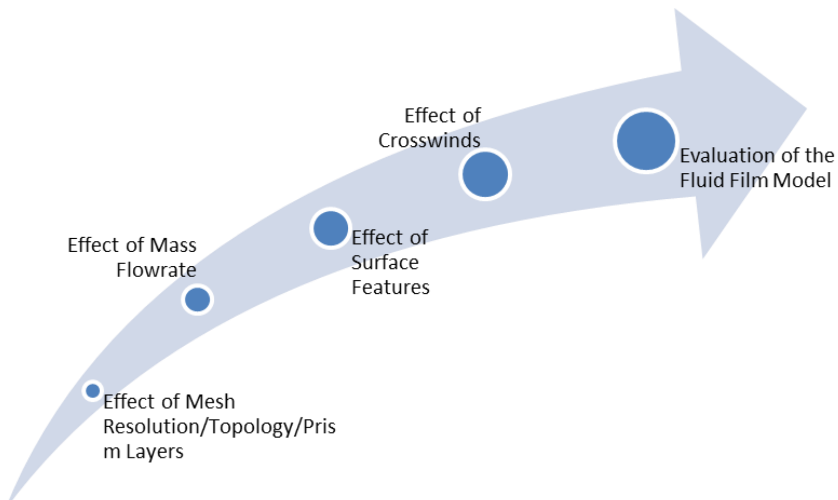
Passenger comfort & safety are the two key elements of interest for a designer. Numerical simulations help designers in the pre-development phase where physical testing is not possible. This study focusses on modelling the water spray impact on a moving car wing mirror, its film formation and consequent film breakup due to instabilities - a phenomenon commonly identified as vehicle soiling in the automobile industry. Such numerical simulations are useful in predicting wiper performance, mirror casing and A-pillar design.

The initial phase of the simulation comprises of decoupled studies using the reactingParcelFilmFoam solver in OpenFOAM. A test case of gravity driven flow on a vertical flat plate is considered, where an inlet with constant mass flow rate is introduced with a film of water. Three different grid resolutions are considered to investigate the mesh sensitivity of the film model. Different mass flow rates have been simulated to observe a variation in the onset of film breakup and rivulet formation. Effect of crosswinds on the rivulet formation is also studied. The simulations are performed for three different OpenFOAM versions viz. V 2.1.1, V 2.2.2 and V 2.3.0.

The second phase comprises of coupled studies of Lagrangian and the fluid film model. For this, wing mirror geometry meshed with SnappyHexMesh is considered with a high velocity spray impinging on it. The flow-field for the continuum is calculated using the Realizable K-Epsilon model of turbulence, an obvious choice because of low mesh resolution requirement of $30<y^+<300$. The spray is modelled using Lagrangian particles injected using a solid cone of semi-cone angle 20 degrees and the flowfield continuum is initialized with a steady state solution with the spray particles. is performed to observe film formation and breakup. The Reitz-Diwakar droplet breakup model was considered for secondary breakup and a constant diameter as well as distribution of droplets was considered. An adjustable time stepping method based on the CFL limitations of 0.9 was chosen for solver stability, accuracy and convergence. The solver was found to be very sensitive to the running step inspite of smaller CFL limits imposed. A close observation of the results suggests realistic film formation patterns.

Adopted Methodology:

The mentioned simulations were run for simple cases of a vertical flat plate with a thin film flow initialized from the top and gravity being the only boundary condition apart from crosswind flow cases. Keeping the complexities of a full vehicle mesh in mind, mesh dependence studies were performed for various resolutions and topologies. Further to evaluate the phenomenon of winds blowing over the vehicle surface, effects of crosswinds and prism layers were studied (Fig 1). To incorporate the surface features at complex locations like around the A-Pillar of a car, the flow over groves/bumps (Fig. 2) on the flat plate was also studied. OpenFOAM was found to be instrumental in capturing all such phenomenon and hence considered apt for modelling vehicle soiling effects.



Results:

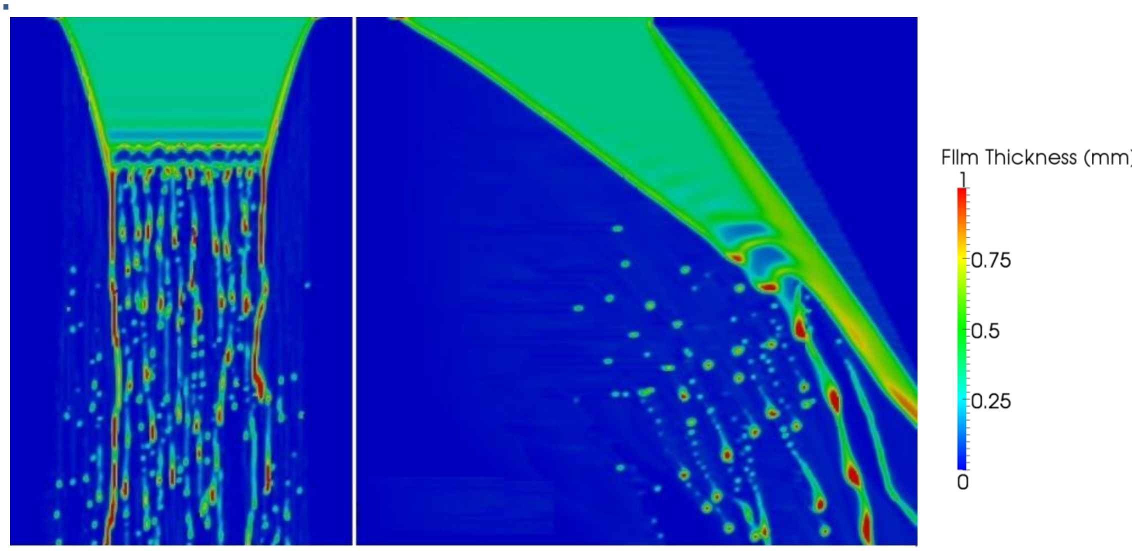


Figure 1 Effect of Crosswinds

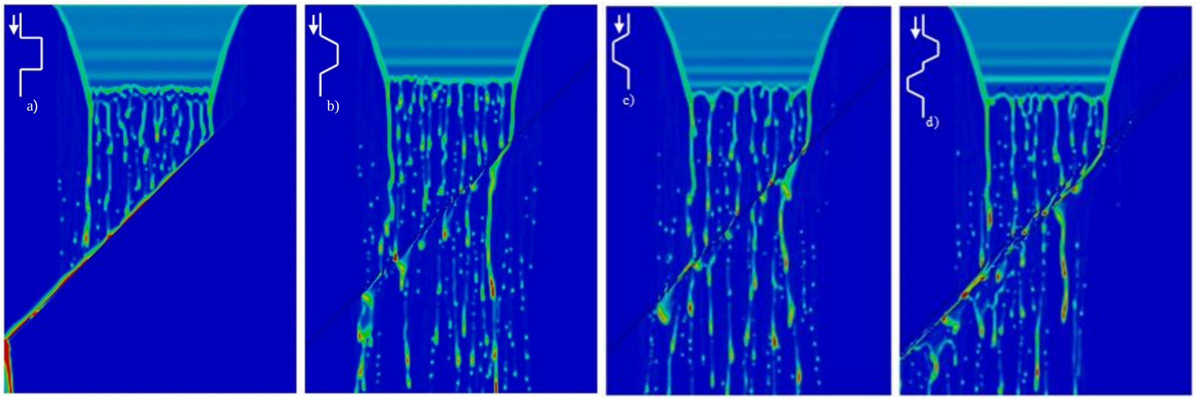


Figure 2 Effect of Surface Groves/Bumps

The effects of film on the vehicle surface could be well accounted for by the thin film model. But the physics of transport of droplets using Lagrangian Particle Tracking was still in the dark. Hence the next step was to study two important phenomenons namely:-

Two-way coupling of Lagrange & Continuum:

The Eulerian-Lagrangian approach is adopted where the air domain is treated as a continuum and the Navier-Stokes equations are solved in the Eulerian frame. The spray droplet dynamics is modelled in the Lagrangian frame, an advantage of which is eliminating the requirement of mesh resolution in the regions of spray particle transport. Accounting for the effect of the presence of Lagrange particles on the field, the two-way coupling – the momentum exchange between the flow field and the spray particles is found to be modelled in the solver physics (Fig 3).

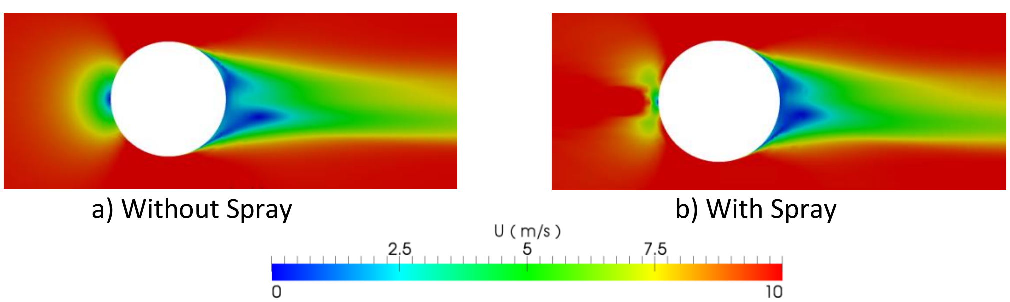


Figure 3 Two way coupling of Lagrange & Continuum

Separation of Film from surface due to film instabilities:

The impingement of spray particles on the surface would lead to increase in the film thickness and liquid accumulation on the surface, which after a certain stability threshold should be stripped off the curvature of the body under consideration. This is done by enabling the curvature separation model in OpenFOAM and specifying the curvature radii.

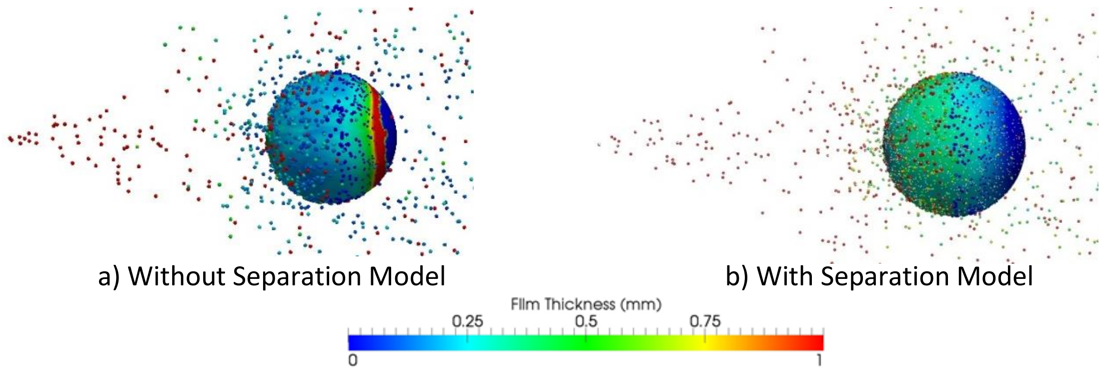


Figure 4 Effect of film separation model

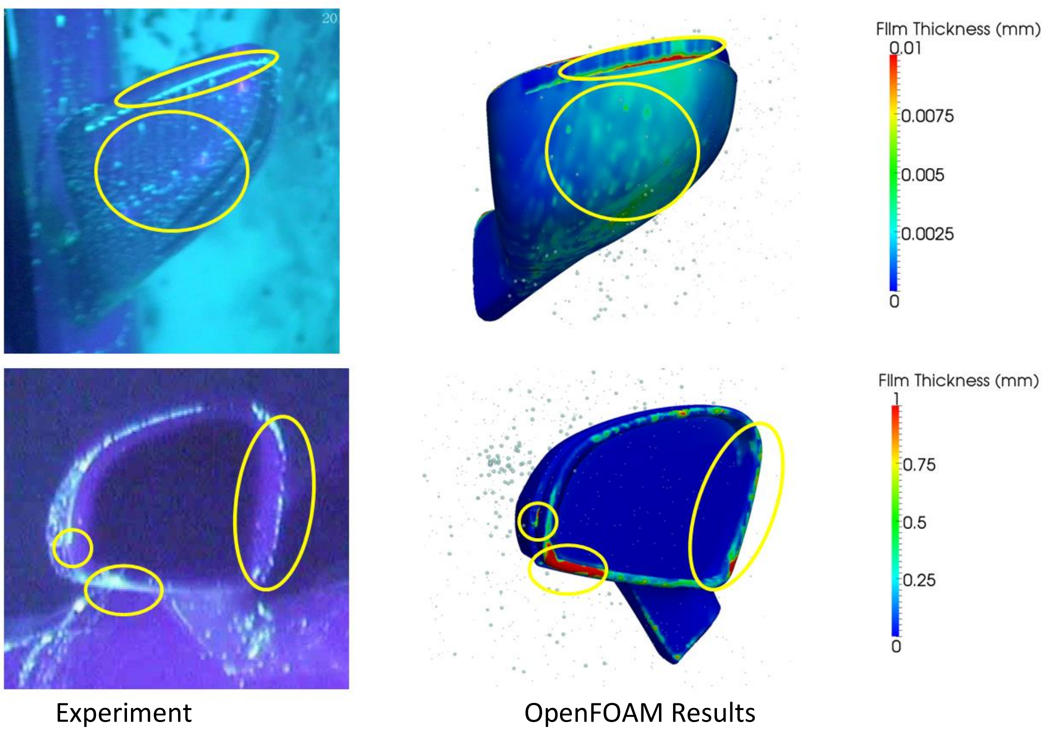


Figure 5 Comparison between experiment & OpenFOAM results

Conclusion: The results obtained from OpenFOAM were qualitatively comparable with experimental tests. However a detailed validation need to be made as the fluid film model is very sensitive to the mesh resolution and topology.