CFD STUDY OF AN OIL CATCH CAN

A. Abilgaziyev, N. Nogerbek*, L. Rojas-Solorzano

School of Engineering, Nazarbayev University, Astana, Kazakhstan; *nnogerbek@nu.edu.kz

Introduction. Oil catch cans are devices used in some internal combustion engines to prevent oil droplets from entering into the exhaust manifold of the engine. The main purpose of this paper is to study the design of a typical oil catch can used in regular vehicles and analyze the effects caused by varying the length of its inner tube and the relative position of the outlet: either radial or tangential to the can body.

Materials and methods. The setup of the mathematical model is performed using the numerical platform SolidWorks-CFD v.2013-14. The main premises are: steady state, Newtonian fluid, incompressible and turbulent flow. Therefore, the following favre-averaged governing equations and boundary conditions were prescribed: (a) mass conservation; (b) Navier-Stokes (momentum) equations; (c) energy conservation; (d) constitutive relations between p, T and H; and (e) k-H turbulence model with wall functions.

Mesh Sensitivity Analysis (MSA) was performed by evaluating five consecutive steps of mesh refinement, finally adopting a mesh with a total of 112525 cells to fulfill next stages of the analysis. The simulations were carried out using a PC with CPU Intel(R) Core(TM) i7-3517U @ 1.90 GHz, 2.40 GHz and 3 cores of each 8.00 GB RAM were put into the calculation. The time taken for every simulation with the finally chosen mesh to fully convergence starting from previously converged results in coarser mesh was approximately 1 hour.

Results. The results show that the efficiency when the outlet is located tangentially to the can is higher than for the conventional radial outlet catch can. This phenomenon apparently obeys to the fact that the flow is impinged with more rotational momentum and therefore provides more time for the droplets to hit the walls and get stratified in the bottom of the can. Moreover, numerical results show that more droplets are captured when the longest inner-tube configuration is used, demonstrating that when the flow has a more direct impact on walls, particles are more likely to stratify on the bottom of the container, leading to a higher efficiency of the device. However, it was observed a large potential erosion due to a fast jet stream loaded with droplets hitting the can bottom. Therefore, the best solution should conjugate the fact that tangential exhaust favors the wall contact of droplets, while also consider reducing the exposure of the bottom of the container to an extremely large impinging load.

Preliminary Conclusions. One of the most popular models of an oil catch can on the market was studied by using CFD techniques. Further research is currently ongoing to analyze the compromise condition between the location of outlet tube and the length of inner tube such that an optimized model might be obtained. After observing the numerical results, it also resulted obvious the potential improvement that could be obtained by inclining the tangential outlet as it could combine a good extraction capacity with lower pressure drop. These ideas are also currently in consideration within the ongoing research.

Acknowledgments. We would like to acknowledge the support of Nazarbayev University which gave an access to its computer laboratory and our research supervisor Luis Rojas-Solorzano.

References.

1. Bolduc, J. et al. (2013) *Oil-Air Vapor Separator.* Capstone project, Orono, US, University of Maine.
2. Carlson, D., Hoglund, R. (1964) Particle Drag and Heat Transfer in Rocket Nozzles. *AIAA Journal,* 2(11).