27/28th International Conference
ENGINEERING MECHANICS 2022
Milovy, Czech Republic, May 9 –12, 2022
Paper **#14**, pp. 65–68, doi: 10.21495/51-2-65

THE NUMERICAL SIMULATION OF THE SPRAY

Česenek J.[*]

Abstract: *In this article we focus on the spray of the liquid which is injected into the cylinder area with gas flow. We work with the non-stationary viscous compressible fluid flow with turbulence model, described by the RANS equations and SST model. Trajectories of the drops are described by the Lagrangian model. Moreover, breakup and trajectory stochastic collision submodels are taken into account. Results of simulations are compared with experimental data.*

Keywords: Compressible gas flow, Spray, RANS, SST turbulence model, Finite volume method, Software, 3D.

1. Introduction

The main topic of this work is to estimate the trajectories of the liquid spray drops, which is injected into the domain with the gas flow. This problem presents a very complex task, which requires the simulation of the gas flow, drops injection, drops trajectories, droplet collision, breakup phenomena, heat transfer, etc. The spray simulations are required in various industrial applications such as the combustion engines and furnace, spray washing, spray cooling, etc.

The physical theory of the compressible fluid motion is based on the principles of conservation laws of mass, momentum, and energy. The mathematical equations describing these fundamental conservation laws form a system of partial differential equations, the so-called Navier-Stokes equations, which we can write in conservative form

$$\frac{\partial w}{\partial t}+\sum_{s=1}^{3}\frac{\partial f_s(w)}{\partial x_s}=\sum_{s=1}^{3}\frac{\partial R_s(w,\nabla w)}{\partial x_s} \tag{1}$$

Here $w(x,t)$ is the state vector, f_s are the inviscid fluxes, R_s are the viscous fluxes. These equations are closed by the SST turbulence model. To this system of equations we have to add the Lagrangian model, which describes movement of the drops

$$\frac{du_p}{dt}=\frac{u_p-U}{\tau_p}+g\,, \tag{2}$$

$$\tau_p=\frac{4}{3}\frac{\rho_p d}{\rho C_d\left|u_p-U\right|}, \tag{3}$$

$$C_d=\begin{cases}\dfrac{24}{Re_p}\left(1+\dfrac{1}{6}Re_p^{\frac{2}{3}}\right), & Re_p\leq 1000\\ 0{,}44, \quad Re_p>1000\end{cases} \tag{4}$$

where u_p is the velocity vector of the drop, U is the speed of the gas flow, g is the vector of gravitational acceleration, ρ_p density of the drop, diameter d and drag coefficient C_d, which depends on the particle´s Reynolds number Re_p.

---

[*] RNDr. Jan Česenek, PhD.: Výzkumný a zkušební letecký ústav, a. s, Beranových 130; 199 05, Prague; CZ, cesenek@vzlu.cz

The finite volume method with implicit time stepping is used for the discretization of the partial differential equations on unstructured meshes. In this research the open-source code OpenFOAM was used to simulate various scenarios.

2. Numerical simulation.

For the numerical simulation of the spray, we use sprayFoam utility from the open-source code OpenFoam. For simplicity of the calculation, we use ReitzDiwakar breakup submodel and trajectory stochastic collision submodel. Others submodels like atomization, heat transfer and so on left to be none.

For a comparison of the numerical simulation results with the experiment we used data from (Liu et al. 1993 or Habchi et al. 1997). Their experiment consists of a liquid drop generator, which injects liquid into the cylindrical area with the flowing gas. The velocity distribution of the drops was measured at two levels 29mm and 47mm from top of the cylinder for two different inlet velocities 59 m/s and 72 m/s. Figure 1 shows positions of drops at a time instant.

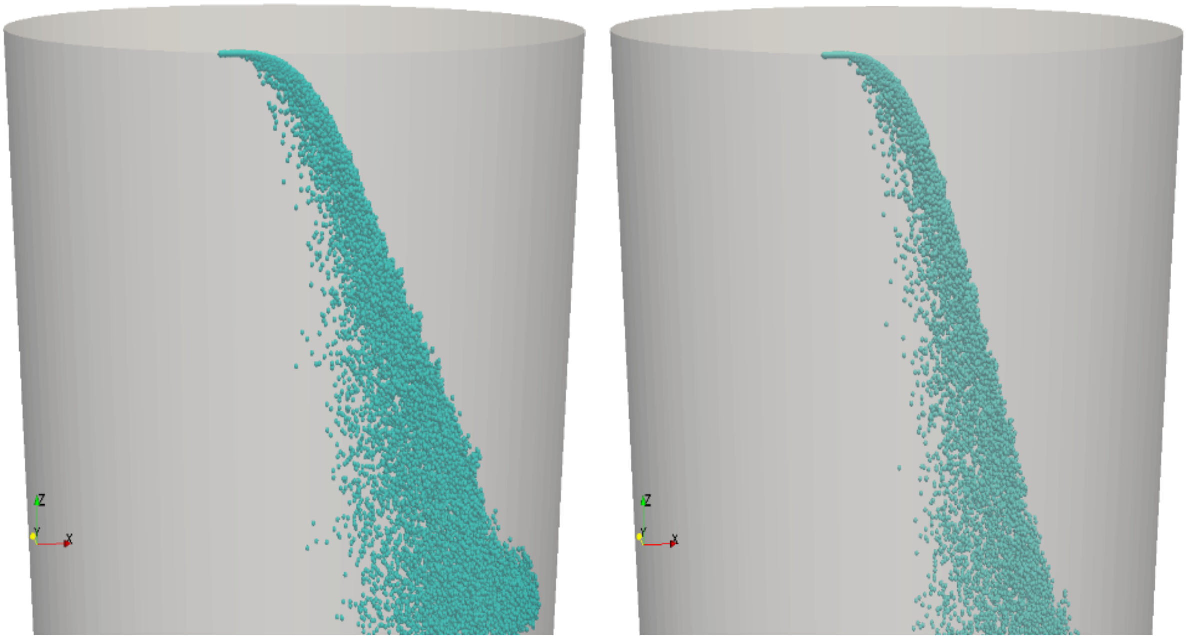

Fig. 1: Drops position of the spray at a time instant. Left for 59 m/s. Right for 72 m/s.

Figure 2 shows the comparison between the calculated velocity distribution across the jet and the experimental data. We can observe a good agreement for the velocity 59m/s and acceptable agreement for the velocity 72 m/s. In this case we can see greater difference between experiment and numerical solution. It is possible that other (more complicated) choice of the breakup and collision models would further improve the solution.

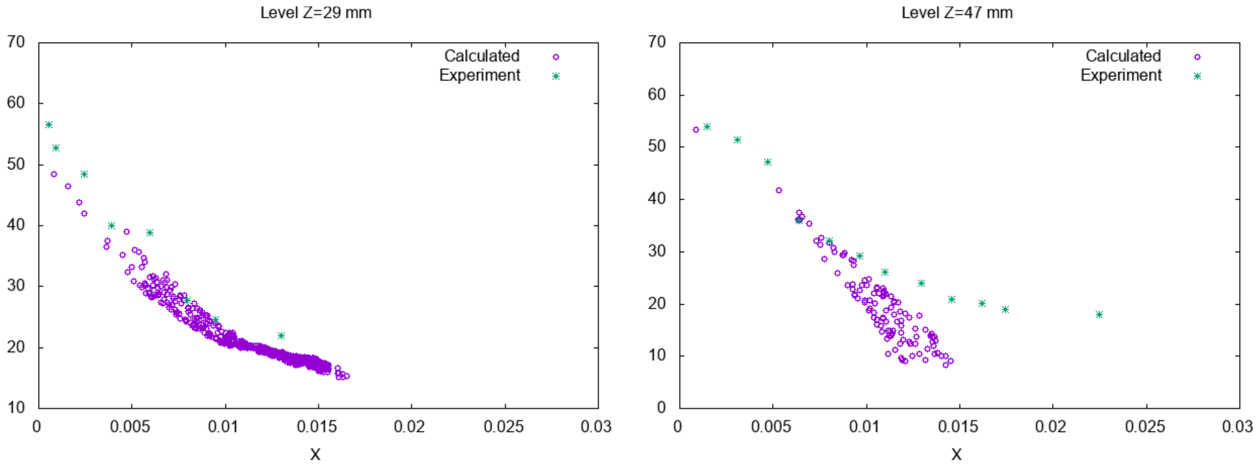

Fig. 2: Drop's velocity distribution for inlet 59 m/s.

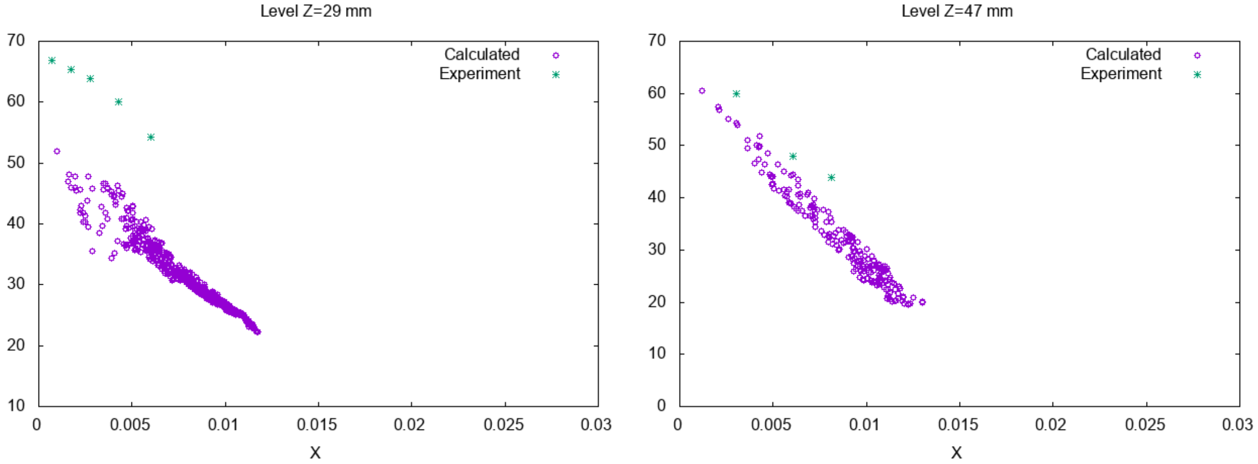

Fig. 3: Drop's velocity distribution for inlet 72 m/s.

3. Conclusions

In this article we worked with the numerical simulation of the spray injected into the gas flow. We show the comparison between numerical simulation and the experimental data. The acceptable agreement was achieved. For the future work we aim to use the numerical simulation of the spray for the prediction of drops trajectories in a real domain in the industry application.

Acknowledgement

The authors are grateful to the European Commission for the support to the present work, performed within the EU CleanSky 2 project TRIcEPS (Grant Agreement No. 831879).

References

Feistauer, M. (1993) *Mathematical Methods in Fluid Dynamics*. Longman Scientific & Technical, Harlow.

Feistauer, M., Felcman, J. and I. Straškraba (2003) *Mathematical and Computational Methods for Compressible Flow*. Clarendon Press, Oxford.

Habchi, C., Verhoeven, D., Huu, C.H., Lambert, L., Vanhemelryck, J.L. and Baritaud, T. (1997), Modeling Atomization and Break up in High-pressure Diesel Sprays, *SAE Transactions*, 106, pp. 1391–1406.

Liu, A.B. and Reitz, R. D. (1993), Mechanism of Air-Assisted Atomization. *Atomization and Spray*, 3, 1, pp. 55–75.

Liu, B., Mather, D. and Reitz, R.D. (1993), Modeling the Effects of Drop Drag and Breakup on Fuel Sprays, *SAE Technical Paper*, 930072.

OpenFoam (2019), *The OpenFOAM-v1906*, url: https://www.openfoam.com/documentation/user-guide.

Reitz, R.D. (1987), Modeling atomization processes in high-pressure vaporizing sprays. *Atomisation and Spray Technology*, 3(4), pp. 309-337.

Reitz, R.D. and Diwakar, R. (1986), Effect of Drop Breakup on Fuel Sprays, *SAE Technical Paper* 860469.

Toro, E.F. (1997) *Riemann Solvers and Numerical Methods for Fluid Dynamics*. Springer, Berlin.

Wilcox, D.C. (1998) *Turbulence Modeling for CFD.* DCW Industries, La Canada, CA.