Delivery tube – CFD analysis

Ondřej Lajza
Vysoké učení technické v Brně
Technická 2896/2, 616 69 Brno

lajza@fme.vutbr.cz

Abstract. This article describes a CFD analysis for optimization of delivery tube and behavior of droplets. Delivery tube is special device which is use for delivering cooled down droplets to wind tunnel. CFD analysis is incorporated for efficient design of tube and for setting parameters of droplet generator. Multiphase modeling has been setup for droplet analysis. This analysis was used for real design of delivery tube in wind tunnel at EADS Innovation Works.

Keywords. CFD, delivery tube, wind tunnel.

1. Introduction

Particle behavior was calculated in order to modify the first design of delivery tube. The present design of delivery tube is not working correctly in real wind tunnel. Second problem was the change of parameters of droplet generator. First simulation was calculated with previous droplet generator parameters. A simplified calculation model is used for this analysis. Present model is based on the real dimension of the wind tunnel section. Delivery tube is placed in the same distance from the inlet to computational domain like in real wind tunnel. This article describes steps in simplified geometry.

2. Delivery tube design

The present case was the first from series of calculations. New design is based on previous delivery tube design [1]. The delivery tube is a device which is composed from two tubes – a small and a big one. This device is used as a delivery device for the droplets to the wind tunnel. Droplets are cooled down outside the tunnel. Two tubes are used for lower dissipation of droplets and minimization of droplets impact to the wall of the tube. In big tube is created the hole for air inlet. Small tube is shorter that big tube. The exit of the delivery tube is opened to the wind tunnel stream. Droplet generator is placed and sealed in the top of delivery tube. New delivery tube have bigger diameter. Slot is placed close to wind tunnel wall in inner tube. For comparison is used also design with one tube. Length of delivery tube is the same in all computational cases. Geometry was created in CATIA, computational grid in ICEM CFD, calculations were performed by the solver ANSYS FLUENT. Elements number was limited to 1.6 million cells. Figure 1 shows computational domain. Figure 2 shows slot in detail.

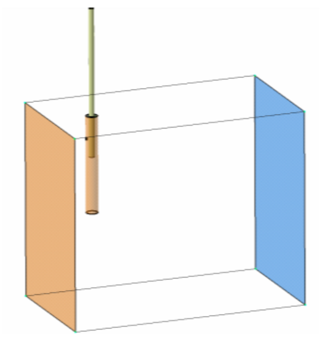

Figure 1: Simplified calculation model

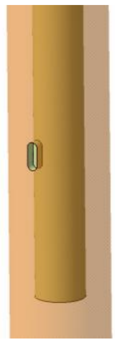

Figure 2: Detail of the slot in the delivery tube

2.1. Calculation setting

Constant velocity profile was used as inlet condition of the computational domain and fixed pressure value was imposed at the exit. Velocity of the flow was set to 11,11m/s. Standard value was used for pressure and temperature. Pressure-based solver was implemented for all calculation. Turbulence is accounted for by the realizable k-ε model. All cases were solved with gravity effect.

Boundary condition trap was used for delivery tube wall. This condition is compatible with the dynamics of the water droplet. Water droplets have the following parameters: initial velocity 9m/s, diameter of droplet 100μm, flow rate 66ml/h.

2.2. Analysis results

In this case was imported flow field in exit from delivery tube. Back flows in delivery tube have big influence to water droplets behavior. Water particle don't have enough speed to overcome back flow. Following pictures are showing droplet behavior for two critical design of the delivery tube.

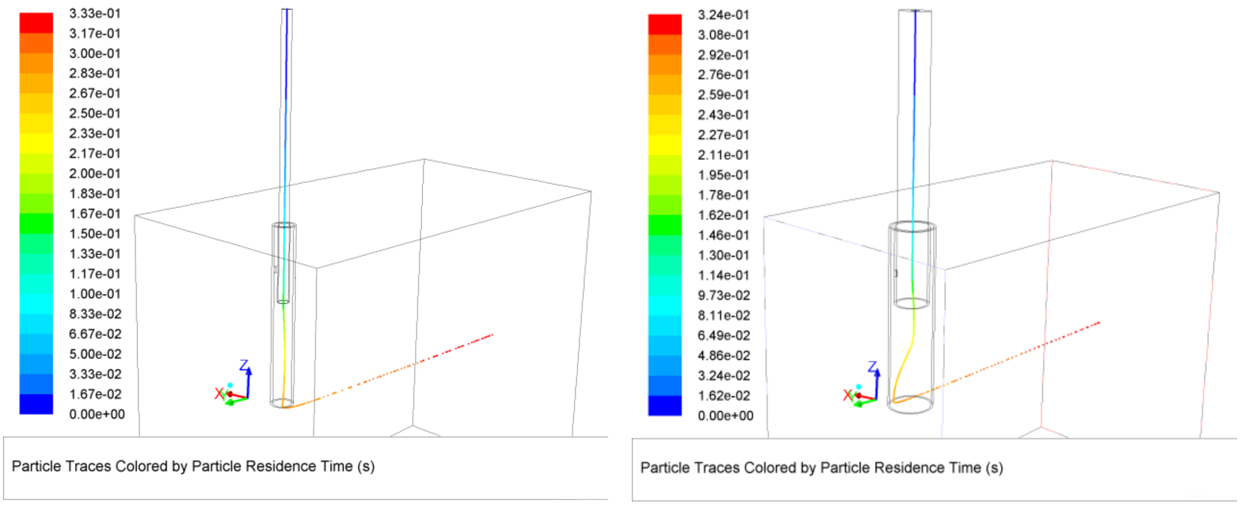

Figure 3: Particle tracking – critical case

Next two pictures show contours of the Discrete Phase Model [DPM] concentration at the exit of the computational domain.

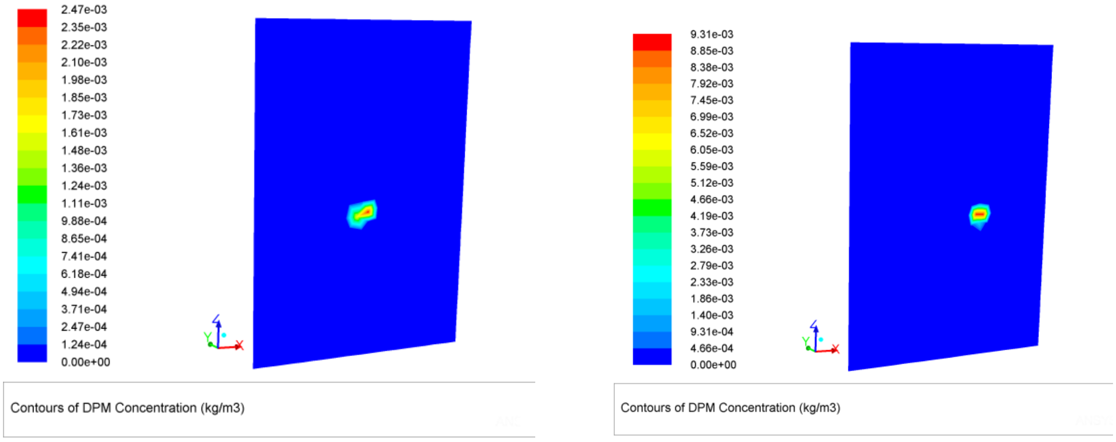

Figure 4: DPM Concentration on exit

2.3. Additional airfoil placed at the exit from the delivery tube

Airfoil at the exit of the delivery tube is placed for reducing the hit area. This is necessary for setting tested airfoil in wind tunnel. Next pictures show difference between delivery tube with and without airfoil in exit.

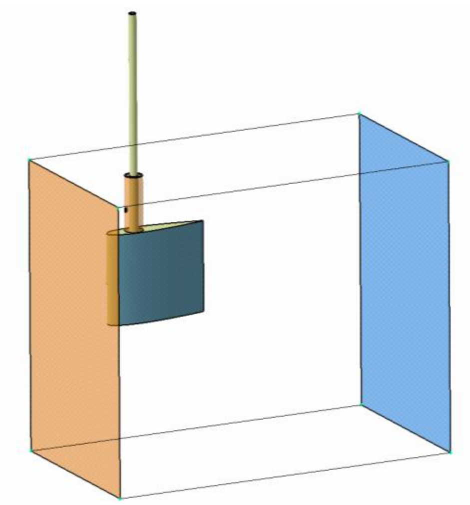

Figure 5: Aditional airfoil placed at the exit from the delivery tube

Figure 6 shows droplet behavior with and without.

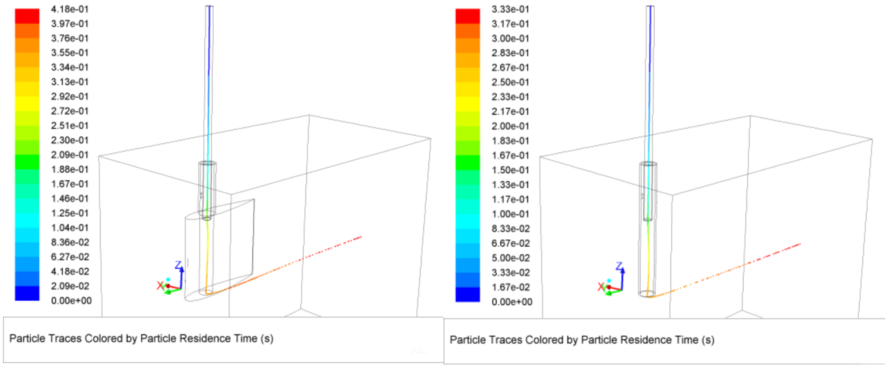

Figure 6: Particle tracking – with and without airfoil

Figure 6 shows concentration at the exit of the computational domain. With airfoil placed at the exit is DPM concentration area smaller.

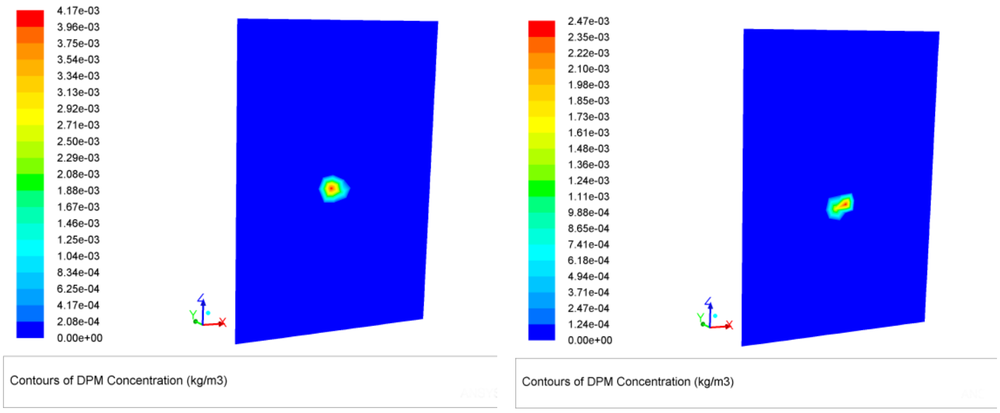

Figure 6: DPM Concentration on exit with (on the left) and without airoil (on the right)

3. Conclusion

CFD analysis with droplet was performed. This analysis shows possibility of calculation droplets on simplified geometry model of wind tunnel. Simplified geometry had good agreement with wind tunnel visualization test. Hit area in the wind tunnel test was placed on the left side in flow direction.

References

[1] Kyle Hutchings, Dominik Raps, Stefan Jung, David Thompson.: Mono-Dispersed Water Droplet Delivery in a Small-Scale Refrigerated Wind Tunnel, 2011, eISBN: 978-1-60086-947-1