11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

FLOW RECIRCULATION IN VHC DESIGNS

Ricardo F. Oliveira[1], Senhorinha F. Teixeira[2]*, Helena C. Marques[3] and José C. Teixeira[4]

[1] CT2M R&D Centre, University of Minho, Guimarães, Portugal, ricardo.falcao.oliveira@gmail.com
[2] CGIT R&D Centre, University of Minho, Guimarães, Portugal, st@dps.uminho.pt
[3] iMed.UL R&D Centre, University of Lisbon, Lisbon, Portugal, hcmarques@ff.ul.pt
[4] CT2M R&D Centre, University of Minho, Guimarães, Portugal, jt@dem.uminho.pt

Key Words: *Computational Fluid Dynamics (CFD), Asthma, Recirculation area, Valved Holding chamber (VHC).*

The pMDI, because of its low cost and acceptable efficiency, is the most used by the medical community for drug delivery. Its major drawbacks are the high spray velocities, which creates the so called “cold-Freon” sensation on the back of the throat, and the necessity of inspiratory coordination with the priming action [1], [2]. In order to solve some of the pMDI problems, spacers, such as the valved holding chamber (VHC) were created. Because the drug spray is released inside, this device leads to the reduction of the high velocity impact of the spray on the throat and the need for inspiratory coordination, allowing the patient to breath normally from the other side of the VHC [1]–[3]. In this way, it is important to study the efficiency of these VHC devices, mainly the internal flow circulation and its influence in the drug delivery. As previous studies have shown [4], the occurrence of recirculation vortexes tend to trap the smaller particles. They represent the most important fraction of the spray distribution because they are most likely to reach the blood stream, being the most effective as a bronchodilator treatment.

The ANSYS® FLUENT 14.0 was used as the solver of choice for the flow simulation inside the VHC. For the turbulence calculations, it was used a two equation model, known as the k-ε Standard. The solution of the differential equations for mass and momentum was carried out in a sequential manner, using the SIMPLE algorithm [5]. The standard discretization scheme was used for the pressure and the second order upwind scheme for the momentum, k and ε equations. Convergence was reached in the simulation by using a criterion value of 1.0E-5. The simulation was obtained in steady state and the fluid was assumed incompressible and Newtonian. There is no energy exchange and the body force was neglected. The parametric influence of design variables was studied for three distinct designs (see Figure 1). For each geometry, taking two design parameters (length and height of point B), several combinations of both variables were tested. For each one, the percentage of recirculation area was evaluated. In all profiles segments AB and BC are connected by an arc with a fixed radius of 40.0 mm. The entrance section (left edge) was defined as a Velocity Inlet, the walls of the geometry behave as No Slip, meaning they are fixed with zero velocity relating to the fluid. Bottom edges were defined as Symmetry Axis and the exit (right edge) as an Outflow (zero derivative for any variable). For the Velocity Inlet condition, an average volumetric flow value corresponding to the Peak of Inspiratory Flow (21.2 L/min) was used. Taking into account inlet cross-section area, the corresponding average velocity was applied to the model boundary through a User’s Defined Function (UDF) describing the turbulent flow profile as a

1/7 power law.

Throughout the combination of both dimensional parameters, the computational results are shown in Figures 2, 3 and 4, as iso-contour plots. These are obtained by interpolation of the computational results of 30 dimensional combinations, for each profile. From the recirculation area plots for each profile (Figure 2, 3 and 4), it is possible to observe that recirculation increases with the height and, in general, inversely with the length. It can also be perceived that the recirculation area is more sensitive to the VHC height than to its length. It can also be noticed that for the same combination of parameters (for example, Length: 70 mm and Height: 50 mm) different values of recirculation are obtained for each profile, being the values Profile 1 > Profile 3 > Profile 2.

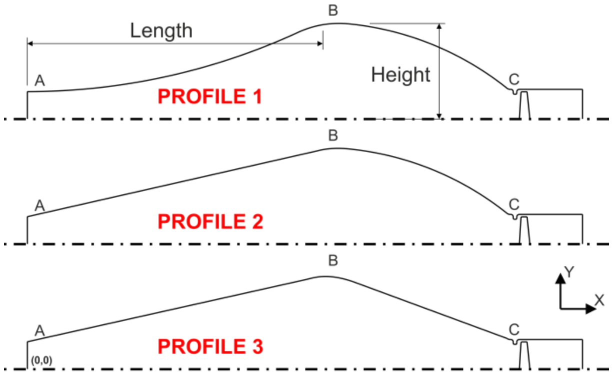

Figure 1 - Geometrical representation of the geometries under numerical evaluation.

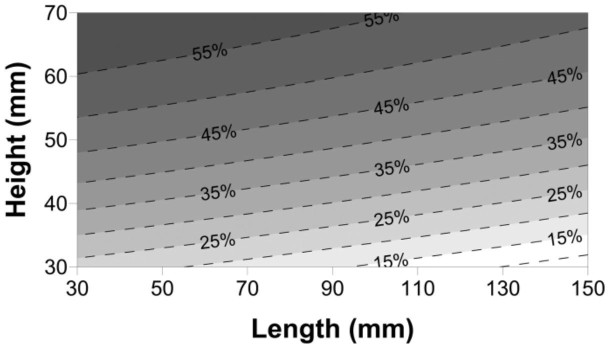

Figure 2 - Iso-contour plot representation of the recirculation area obtained from the computational results for Profile 1.

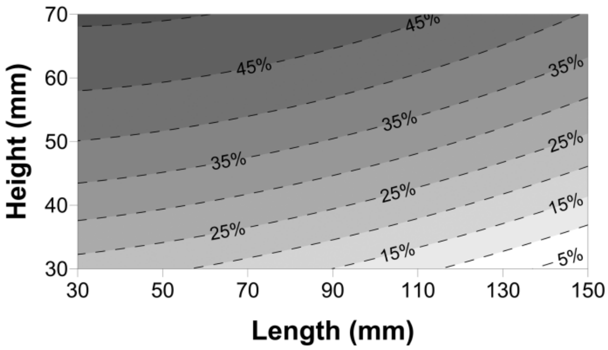

Figure 3 - Iso-contour plot representation of the recirculation area obtained from the computational results for Profile 2.

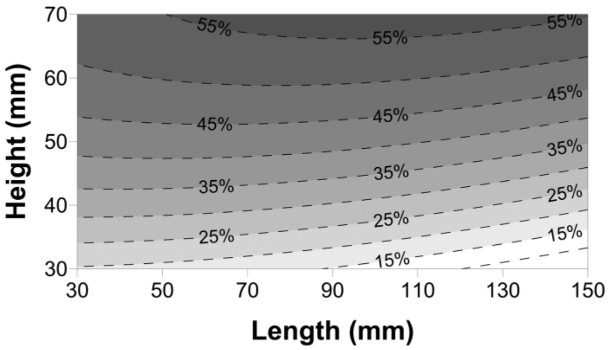

Figure 4 - Iso-contour plot representation of the recirculation area obtained from the computational results for Profile 3.

REFERENCES

[1] M. B. Dolovich and J. B. Fink, "Aerosols and devices," *Respir. Care Clin. N. Am.*, vol. 7, no. 2, pp. 131–173, Jun. 2001.

[2] S. P. Newman, "Aerosols," in *Encyclopedia of Respiratory Medicine*, G. J. Laurent and S. D. Shapiro, Eds. Elsevier, 2006, pp. 58–64.

[3] S. P. Newman, "Spacer devices for metered dose inhalers," *Clin. Pharmacokinet.*, vol. 43, no. 6, pp. 349–360, Jan. 2004.

[4] R. F. Oliveira, S. F. C. F. Teixeira, L. F. Silva, J. C. Teixeira, and H. Antunes, "CFD study of the Volumatic spacer: a realistic approach," in *V European Conference on Computational Fluid Dynamics ECCOMAS CFD 2010*, 2010, vol. 1.

[5] H. K. Versteeg and W. Malalasekera, *An introduction to computational fluid dynamics: the finite volume method*. Harlow, England: Longman, 1995, p. 257.