Comparison of numerical simulation to experiments for a jet nebulizer

N. Lelong * (1), L. Vecellio (1), A. Junqua-Moullet (2), Y. Sommer de Gelicourt (3), C. Tanguy (4), P. Diot (5), H. Suzanne (2)
(1) – DTF Aerodrug, Faculté de Médecine, Tours, France
(2) – CEA Le Ripault, Monts, France
(3) – ANSYS France, Montigny le Bretonneux, France
(4) – Dantec Dynamics, Nozay, France
(5) – INSERM U1100 – EA 6305, Faculté de Médecine, Tours, France
nicolas.lelong@etu.univ-tours.fr, vecellio@med.univ-tours.fr, alexandra.junquamoullet@cea.fr, yann.sommerdegelicourt@ansys.com, christian.tanguy@dantecdynamics.com, diot@med.univ-tours.fr, herve.suzanne@cea.fr

Abstract

The performances of jet nebulizers for medical purposes are limited compared to the new mesh nebulizers, regarding output rate and drug loss. Their experimental development has reached a plateau, however the improvement of the device, cheaper and able to produce smaller droplets, stays an important challenge. The purpose of this study is to design a numerical model simulating the nebulization process and to compare it with experimental data obtained from various measurement methods. Such a model allows a better understanding of the atomization process and a determination of the relevant physical parameters influencing the nebulizer output.

The Updraft nebulizer (Hudson) was chosen to set geometric dimensions of the model and was designed with ANSYS Workbench. The simulation domain covers a wide area with a 4 µm mesh in the center of the device, where the liquid atomization takes place. Two inlet air flow rate were considered: 2 L/min (Re = 4000) and 8 L/min (Re = 16000). Boundary conditions were set with experimental data and the relevant model parameters were determined through a 2D axial simulation. Large Eddy Simulation has been used as the turbulence model to account for the wide range of Reynolds regimes in the process. The transient 3D CFD calculations were then run on a 15° angular sector representing 4 million cells with ANSYS Fluent. Droplet formation times and locations, along with diameters and airflow velocity, were determined thanks to user-defined functions in Fluent. The experimental study consists in characterizing the spray output. Particle size measurements were performed with a laser diffractometer Spraytec (Malvern), particle size and velocity were measured with a Phase Doppler Anemometer (Dantec Dynamics) and aerosol output was determined by a gravimetric method. The behavior of the droplets was visualized with a Fastcam SA1 CCD Camera (Photron).

Observations made with CCD camera showed similar patterns as numerical results. At 2 L/min the liquid periodically formed a film which blew up in drops with various sizes. The larger ones impinged and spread out on the sphere while the faster ones rebounded and split in smaller particles. At 8 L/min, images showed a continuous production of small droplets and the progressive formation of a liquid ring on the sphere surface. For both flow rates, the simulation and camera particle size distributions were similar, especially at 8 L/min. The difference between the mean diameters was about 20 % (26 µm vs. 21 µm; 17 µm vs. 14 µm), which is acceptable given mesh size and image resolution. PDA and laser diffractometry, which take into account spherical droplets with a diameter less than 5 µm, provided smaller values.

CFD predicted atomization phenomena and provided a good estimation for droplet size in the nebulizer compared to CCD camera images. By showing the dynamics in the liquid phase during the fragmentation step, CFD helps understanding the physical processes governing the fragmentation of droplets over 5 µm, which could be extrapolated to micrometric particles obeying to the same dynamics. This model could help predicting nebulizer output with defined geometrical and physical parameters.

* Corresponding author: nicolas.lelong@etu.univ-tours.fr