GATHERING DUST

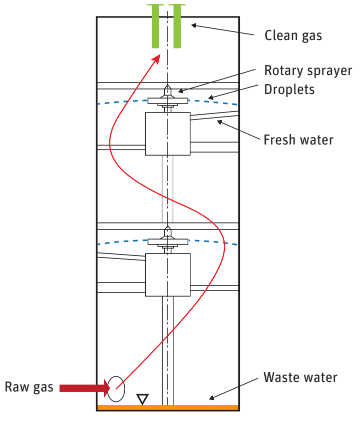
A new simulation approach helps to improve particle removal efficiency of wet scrubbers.

By Damian Pieloth, Physicist, Ph.D. Student; **Gerhard Schaldach,** Chemist, Scientific Assistant, and **Peter Walzel,** Doctor of Technical Sciences, Professor, Chair of Particle Technology Lab, University of Dortmund, Germany

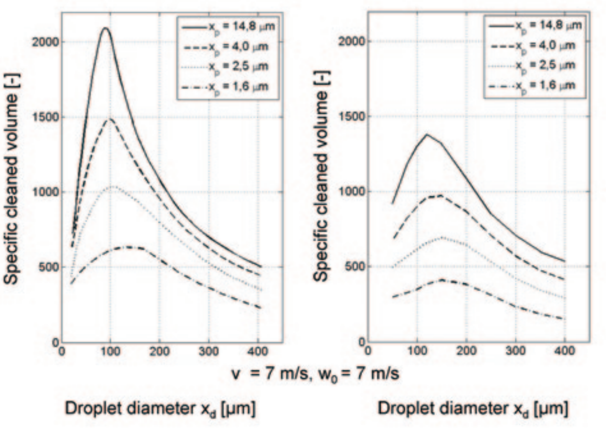
Wet scrubbers are frequently applied in industrial processes, such as removing pollutants from furnace flue gas and sticky particles from process streams. The greatest challenge in designing wet scrubbers is to remove the maximum amount of particles while minimizing energy and washing-liquid consumption. Pilot plant experiments optimizing the design of scrubbers are time-consuming and expensive because of the high cost of building prototypes and the multitude of design parameters. Researchers from the University of Dortmund have developed simulation methods that accurately predict the performance of a wide range of wet scrubber designs[1]. The new approach is based on the Löffler model[2,3] of particle deposition on droplets; it was implemented employing user-defined subroutines within ANSYS computational fluid dynamics (CFD) software. These user-defined subroutines permit the implementation of new user models and the extensive customization of existing ones to provide additional modeling capabilities, proprietary data or specific boundary conditions. The key advantage of the Euler–Lagrange method in simulating wet scrubbers is that it requires far fewer computational resources than the alternate Lagrange–Lagrange approach of treating particle–droplet collisions explicitly.

SCRUBBER DESIGN PRINCIPLES

Wet scrubber devices remove small particles or small droplets from process gases. Their underlying physical working principle is to capture particles in liquid droplets, with the liquid absorbing the pollutants. The liquid droplets have much higher inertia than the particles, which makes them relatively easy to separate from the outlet gas stream using a mist eliminator or entrainment separator. Efficient removal of particles 5 micrometers or less in diameter generally requires devices, such as jet scrubbers, with high gas–liquid relative velocities that provide pressure recovery by proper duct geometries. An advantage of wet scrubbers is that, generally, they are the only air pollution control device that can remove both particles and gases — and in some cases they can achieve high removal efficiency for both pollutants. In addition, wet scrubbers can handle high temperatures with moisture simultaneously cooling the flue gases, thus minimizing



▲ Diagram of rotary wet scrubber



▲ ANSYS CFX numerical modeling results (right) matched Löffler's theoretical calculations and experimental data (left).

The key advantage of the Euler–Lagrange method is that it requires far fewer computational resources than the alternate approach of treating particle–droplet collisions explicitly.

the overall size of the equipment that is required to clean the incoming gas stream.

Physical tests of wet scrubbers play an important role in the design process, but it is very difficult to design wet scrubbers solely with physical experiments. The design parameters for a typical wet scrubber include the number of sprayers, the droplet-size distribution they generate, sprayer location within the scrubber, the particle-size distribution of particles to remove, the geometry of the scrubber, and the gas mass flow. The designer needs to understand the impact of each of these parameters on the performance of the scrubber to optimize the design. Computer simulation using CFD has long been recognized as a good alternative to pilot trials, but the traditional method of explicitly simulating particle–droplet collisions requires great computational resources and has therefore been simply beyond the reach of most designers.

NEW SIMULATION METHOD

Members of the Particle Technology (PT) group at the University of Dortmund addressed these challenges by utilizing the Löffler model of particle deposition on droplets. Small dust particles tend to follow the streamlines of the gas flow around droplets but leave the streamline due to their inertia and collide with the droplet. The model used here divides the scrubbing process of particles by droplets into three subprocesses.

The first step is calculation of the deposition efficiency of dust particles on a single droplet for a given flow regime. The deposition efficiency of particles on a droplet is a function of process parameters, including particle- and droplet-size distribution as well as gas flow velocities. With larger dust-particle sizes and higher relative velocities between dust particles and droplets, the deposition efficiency of dust particles on droplets increases. Bigger droplets decrease deposition rates due to the weaker inertial forces on the dust, while smaller dust particles are more difficult to deposit because they are more likely to follow the streamline around a droplet. The CFD simulation follows the trajectories of individual droplets. Droplets leaving the spray system collect dust particles along their trajectory through the scrubber, thus increasing the dust load of the droplets.

In the second step, the cleaned gas volume is evaluated by tracking the droplets through the gas flow and calculating the deposition efficiency in every control volume passed by the droplets. Dividing the cleaned volume by the volume of the droplets yields the specific cleaned volume of the droplets. This calculation defines the performance of the scrubber for a given volume of sprayed water.

The third step involves calculating the local dust concentration change by adding up the cleaned volumes of all droplets crossing a given gas flow volume. This model focuses on individual droplets and their deposition efficiency so that the information obtained from the model can be applied to a wide range of scrubber designs and geometries.

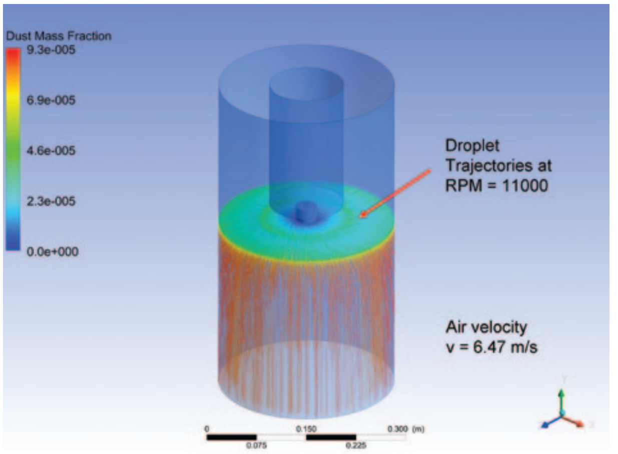
The authors used an Euler–Lagrangian formulation to simulate web scrubbers in ANSYS CFX. The Euler phase represents the two components of the raw gas: air and dust. The Lagrangian phase, which maps the droplets onto the simulation, also has two components: water and dust. During the course of the simulation, the dust component within the Euler phase is transferred to the dust component within the Lagrangian phase. The interchange of dust loading between the two phases is calculated within a CFD user-defined subroutine based on the flow and particle–droplet characteristics. This makes it possible to simulate particle capture by droplets for different particle-size and droplet-size distributions by using several Euler phases for the dust. One limitation of the Löffler model is that it assumes a homogenous distribution of dust particles within the gas flow. Larger dust particles violate this assumption because they tend to segregate from the suspension state under the influence of mass forces such as gravity or inertia. However, in most industrial applications, wet scrubbers are applied to dust particles smaller than 5 microns, so this simplification is widely justified. Also, strands of cleaned gas behind individual droplets are assumed to equalize their concentration quickly.

SIMULATION CORRELATES WELL WITH EXPERIMENTS

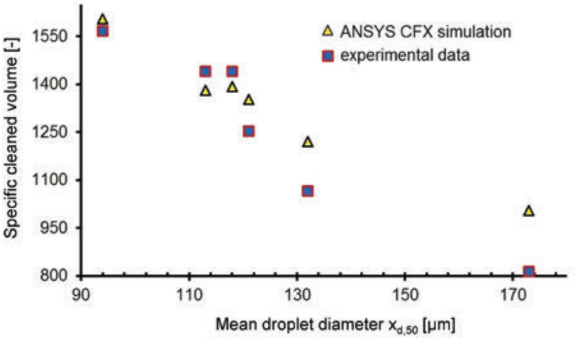
The Dortmund team performed scrubber simulations with ANSYS CFD, compared them with results from literature, and found a very good match. Next, they designed and built a prototype pilot-scale rotary scrubber to generate experimental data and also to compare these results with ANSYS CFX simulations. Salt particles were powdered and screened in order to fit typical dust-particle sizes as in industrial applications of wet scrubbers. By using de-ionized water, they were able to measure the capture of salt particles at droplets by reading the electrical conductivity of the water after the scrubbing process. The droplet-size distribution was varied by changing the rotational

The Dortmund team performed scrubber simulations with ANSYS CFD, compared them with results from literature, and found a very good match.

▲ Pilot-scale prototype



▲ ANSYS CFD wet scrubber simulation



▲ Comparison of CFX predictions to experimental data

velocity of the Lamrot sprayer installed within the device[4]. The specific cleaned gas volume was calculated from mass balance at the scrubber. The team made considerable efforts to match the narrow, well-defined particle- and droplet-size distributions in the physical experiments under different scrubber operating conditions. In addition to this effort, the team simulated a jet scrubber used by an industrial partner and compared the simulations with experimental data. In all cases, the simulation results showed good agreement with experimental data.

The need to efficiently remove dust particles from exhaust gases challenges the design and optimization of wet scrubbers. Equipment must meet strict separation efficiency requirements while also minimizing water and energy consumption. The incorporation of the Löffler model into ANSYS CFD software allows for a quick evaluation of a wide range of possible designs until an optimal design is found. This is done without the expense of building and testing multiple prototypes. The simulation predictions have been correlated with a series of physical experiments and found to accurately predict scrubber performance. The approach described in this article requires considerably less computing power compared to algorithms that treat particle–droplet collisions explicitly. It also can be applied to other applications that require an accurate prediction of capture of particles by droplets, for example powder insertion into a spray or coating of particles[5].

The incorporation of the Löffler model into ANSYS CFD software allows for a quick evaluation of a wide range of possible designs until an optimal design is found.

References

[1] Pieloth, D.; Kohnen, B.; Schaldach, G.; Walzel, P. CFD-Simulation von Nasswäschern, *Chem.-Ing.-Tech.*, **2012**, 84(1-2), pp. 127–137.

[2] Löffler F., Partikelabscheidung an Tropfen und Fasern, *Chem.-Ing.-Tech.*, **1983**, 55(5), pp. 171–178.

[3] Ho, C.; Sommerfeld, M. Modelling of micro-particle agglomeration in turbulent flows, *Chem.-Eng.-Sci.*, **2002**, 57(15), pp. 3073–3084.

[4] Mescher, A.; Möller, A.; Dirks, M.; Walzel, P. Gravity affected breakup of laminar threads at low gas-relative-velocities, *Chem.-Eng.-Sci.*, **2012**, 69(1), pp. 181–192.

[5] Schaldach, G.; Pieloth, D.; Kohnen, B.; Großmann, M.; Walzel, P. CFD-Simulation der Feststoffeinbindung in einen Sprühstrahl, *Chem. Ing. Tech.*, **2011**, 83(6), pp. 893–899.