Comparison of Practical and Analytical Spray Performance in Defouling Process

F. Li*, K. J. Brown, and R. J. Schick
Spray Analysis and Research Services, Spraying Systems Co.
Wheaton, IL 60187-7901 USA

Abstract
Heat exchangers are widely used in heating, air conditioning, refrigeration, industrial plants and refineries, gas processing and water treatment. Fouling is a chronic operating and design problem for retaining highly effective heat exchangers. This leads to countless chemical and mechanical attempts for reducing fouling. Mitigation of fouling and removal of deposits can be addressed efficiently with proper design and gas conditioning modification. All previous mentioned factors and uncertainties can be examined, evaluated and optimized by using computational fluid dynamics (CFD) simulation.

Typical defouling and cleaning process involves chemical solvent sprayed on the heat exchanger surfaces. Manufacturers and operators are facing increasing difficulty to preserve heat exchanger efficiency due to complex structures. Spray behavior and performance cannot be simply predicted due to the many variables involved such as nozzle characteristics, geometry, orientation, gas flow of heat exchanger and pressure resistance. There are tremendous challenges associated with fouling of heat exchangers, where CFD is known as a common tool to predict and resolve those problems. In this work, we will compare the performance of CFD simulation of spray with practical droplet test data in a scaled heat exchanger built and tested in Spraying Systems Co. spray laboratory for a pilot anti-fouling imitation. The spray characteristics based CFD will be used to validate the importance of reasonable and precise spray input simulation in a solvent based defouling process.

* Corresponding author: Fang.Li@spray.com
Introduction

Heat exchangers are process equipment that transfer heat. Through a heat transfer surface that separates the two fluids, heat is continuously or semi-continuously transferred from a hot to a cold fluid directly or indirectly. Heat exchangers are classified by flow arrangement and construction type, primarily bundles of pipes, tubes or plate coils. In processing industries like refineries, fouling is generally the deposition and accumulation of unwanted materials such as scale, algae, suspended solids and insoluble salts on the internal or external surfaces of heat exchangers.

Fouling on process equipment surfaces is a chronic operating problem and can have a significantly negative impact on the operational efficiency of the unit. Fouling could cause a major economic loss on most industries. The total fouling related cost is estimated to exceed $4.4 million dollar yearly, which is about 0.25% to 30% of GDP of industrialized countries \(^1\). According to reports from Harwell Laboratories, maintenance costs of heat exchanger and boilers consist of about 15% of total costs in process plants. Heat exchange fouling associated cost includes costs due to over-design, additional fuel consumption and maintenance, loss of production by efficiency deterioration and shutdown. Also, heat exchanger fouling is responsible for emission of carbon dioxide, data from oil refineries suggested that crude oil fouling accounted for 10% of CO₂ emissions of these plants \(^2\).

Occasionally, heat exchanger surface is replaced rather than renewed. This circumstance arises when reuse is risky or more expensive than replacement, or shutdown is long. This strategy is seldom employed, which indicates the cleaning of heat exchanger is preferred to restore efficiency in process industries. Fouling inside heat exchanger can be reduced by appropriate heat-exchanger design, proper selection of operating conditions and heat exchanger geometry, mitigation/removal methods (mechanical and/or chemical) from the heat transfer surfaces and heat exchanger surface modification/coating.

Numerous studies of the heat exchanger fouling and cleaning have been conducted. Today's oil refineries have to cope with heavier crude oils or densified residuals with higher risk of fouling from sources which until recently have not been economical to process. The production of hydrocarbon fuels from biomass and the co-firing of waste material and straw in power plants are associated with substantial fouling problems \(^3\). Further research on the problem of fouling in heat exchangers and practical methods for predicting the antifouling result, making use in particular of modern digital techniques, are still called for.

Technical Approach

Typical defouling and cleaning process involves chemical solvent sprayed on the heat exchanger surfaces, manually (see Fig.1) or mechanically by spray headers. Manufacturers and operators are facing increasing difficulty to preserve heat exchanger efficiency due to complex structures. Manual antifouling is time consuming, requires significant down time and hard to reach due to heat exchanger structure and safety issues. Injectors are commonly used. There are many variables that contribute to performance, including geometry, orientation, flow of heat exchanger, pressure resistance. Hence spray behavior and performance cannot be simply predicted.

![Figure 1. Heat Exchanger Fouling and Chemical and/or Mechanical Antifouling.](image)

Wilson and Crockford brought up discussion of cleaning and procedure effectiveness incorporating assurance and proof of cleanliness performance, which will dictate the choice of technologies and determine the consistent clean of a particular piece of equipment which cleaning can be verified \(^4,5\).

There are tremendous challenges and requirements associated with defouling of heat exchangers, where CFD is known as a tool to predict and solve those problems. A scaled heat exchanger was built at the extension of a wind tunnel and tested in a spray laboratory as a pilot anti-fouling tube face. Various nozzles and arrangements were applied to this system. Spray performance was captured and recorded in the zone of interest.

The data was then compared with CFD simulation, in which particle tracking was done with Discrete Phase Model. Practical droplet test data for specific nozzles was used as the inlet for model precision. The spray characteristics technology is combined with CFD analysis to ensure and validate the solvent spray behavior in defouling process, both qualitatively and quantitatively.
Equipment and Methods

The experimental setup consisted of spray nozzles, with pump and flow meter, wind tunnel, and PDI with traverse. All tests were carried out with the co-current air flow. The injected fluid was water at ambient temperature ~293.15K. The nozzle was operated with a steady clean water supply for all tests as noted in Table 1. Wind tunnel and arrangement set-ups schematics can be seen in Figures 2-3. Four different types of nozzles at relative layouts were used to complete the tests and compare the corresponding spray performance under the same liquid supply amount.

Wind Tunnel

The subsonic Wenham (blower-style) wind tunnel utilized in these experiments was capable of producing a co-current nominally uniform air flow at a velocity range from 2.5 m/s to greater than 50 m/s; the actual flow velocity generated during these tests was 5 and 10 m/s, respectively, for each arrangement. Before nozzles were placed in the test section, the wind speed and differential pressure was monitored and maintained using a Pitot tube method with calibrated TSI VELOCICALC meter reading at upstream and downstream of test section. This wind speed was chosen as it allowed for a reasonable representation of the scale down defouling systems commonly seen in the industry.

The test section was set to ensure flow was fully developed before entering the test section, and pressure drop created by the heat exchanger was not influenced by the compressed air. During gas baseline stage, experimental results at two flow rates were collected for CFD comparison prior to nozzle installation. Figure 2 provides an image of the wind tunnel with heat exchanger structure (tubes) and the water line pipe for nozzle mounted ahead the test section.

Table 1. Case Arrangement and Spray Nozzle Parameter.

<table>
<thead>
<tr>
<th>per nozzle</th>
<th>Units</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Injector Type</td>
<td></td>
<td>Hollow Cone</td>
<td>Hollow Cone</td>
<td>Hollow Cone</td>
<td>Hollow Cone</td>
</tr>
<tr>
<td>Nozzle ID</td>
<td></td>
<td>1/8BD-3</td>
<td>1/4D3-45</td>
<td>1/4D2-45</td>
<td>1/4D1.5-45</td>
</tr>
<tr>
<td>Air Flow Conditions</td>
<td>m/s</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>m/s</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Operating Pressure/ΔP</td>
<td>Pa</td>
<td>930792</td>
<td>1068690</td>
<td>634318</td>
<td>689476</td>
</tr>
<tr>
<td>Flow Rate</td>
<td>m³/s</td>
<td>5.68e-5</td>
<td>2.84e-5</td>
<td>1.89e-5</td>
<td>1.42e-5</td>
</tr>
<tr>
<td>Dv0.5</td>
<td>m</td>
<td>1.76e-4</td>
<td>1.28e-4</td>
<td>1.4e-4</td>
<td>1.53e-4</td>
</tr>
<tr>
<td>q</td>
<td>-</td>
<td>1.76</td>
<td>1.16</td>
<td>1.38</td>
<td>1.89</td>
</tr>
<tr>
<td>Spray Angle</td>
<td>°</td>
<td>72</td>
<td>55</td>
<td>54</td>
<td>44</td>
</tr>
</tbody>
</table>

As the liquid drops bounce, splash and break up when hitting solid surface, it would be impossible to test exactly at tubeface. For conducting the nozzle tests with acceptable measurement resolution, a slide which was about 0.05m away from the tubeface, with 60% of the circumference length and approximately 0.025m width opening was cut as the test plane. PDI system was mounted at this plane on x- and y-axis traverse and oriented vertically to allow data acquisition at various y-locations.

Figure 2. Wind Tunnel System.

Experimental layouts for specific nozzles are displayed in Figure 3 at the sequence of each line from left to right corresponding to each case. To reduce the repeats and save time in the tests, based on the layout, droplet behavior was assumed symmetrical from the water pipeline. Three horizontal locations, centerline and 0.075m increments, were set up as x-axis of traverse. Each measurement started at 0.025m away from the top of tunnel edge and 0.038m increments per test point moving toward the bottom of tunnel (y-axis of traverse) till no valid point or no laser detection was reached.
**Phase Doppler Interferometry (PDI)**

A two-dimensional Artium PDI-300 system with the integrated AIMS software was used to acquire drop size and velocity measurements. This technique measures angle of trajectory and time of arrival of each particle passing through an optical measurement volume formed by pairs of intersecting laser beams. The ability to measure accurately requires the reliable characterization of the size, velocity, and transit time of each droplet. The PDI system is a validated method for droplet size and velocity measurement; in addition, spray concentration measurements are possible. The technical explanation of the Phase Doppler technique can be reviewed in a number of publications by Bachalo et al. [6].

The Artium PDI system utilizes a unique digital signal burst detection method which reliably detects droplets, even in complex environments. This detection system is also critical to the in situ approach for measuring the effective diameter of the sample volume as a function of drop size. The Fourier transform based signal processor uses quadrature down-mixing to position the signals in an optimum range for processing. The real and imaginary (shifted by 90 degrees) components of the signals are sampled and a full complex Fourier transform is used to obtain the signal frequency and phase. The approach has proven to be very effective in detecting and eliminating sizing errors due to the well-known trajectory problem.

The Artium AIMS software incorporates an auto-setup feature that serves to optimize the frequency and phase shift processing. The auto-setup feature has significantly minimized the user-to-user setup differences to avoid produce varying results and to improve accuracy in PDI data results and less relying upon the operator’s individual experience and understanding of the PDI principals.

In the wind tunnel drop size acquisition process, PDI system was mounted to a vertical plane on two y-axial parallel traverse systems. The laser transmitter lens and receiver focal length was set to 500mm for all tests. The PDI system was oriented at the 40° off-axis forward scatter position, by adjusting transmitter and receiver respectively 20° off horizon level as displayed in Fig 4. Masking was employed as necessary to provide an effective measureable drop size range of 2.8 to 563\(\mu\)m (mask 2, 4.0 to 805\(\mu\)m mask 3). For safety concern, two traverse axes were moved manually on horizontal plane for test location shifting.

**Computational Setup and Methods**

For ambient condition drop size acquisition, both the laser transmitting lens and the receiving unit focal length was kept 0.5 meter for consistency; the receiving unit was oriented at the 40° off-axis forward scatter position. Figure 5 illustrates the experimental layout of the PDI system with mounted traverse system (x-, y- and z-axis) at ambient condition.
CFD simulations were performed using ANSYS FLUENT version 16.2. Geometry used for CFD was created according to wind tunnel dimensions and exact heat exchanger structures. As small and multi surfaces structure would result in finer and bigger meshed, in the modeled geometry spray lance was alternated to a simplified pipe version to reduce the relative mesh size. Heat exchanger structure was the primary research objective in gas baseline study and spray performance on the tubeface to compare the experimental data. Hence, the tubeface structure was kept realistic. Meshing was performed within ANSYS Workbench using the automated meshing tool. Dense mesh was incorporated in the tubeface locations for capturing injections. Wall inflation was created on heat exchanger tube walls to ensure enough cells for generating reasonable profiles. Size functions were used to further modifying mesh size. A mesh independent study was done to make sure proper grid. The 3D mesh consisted of mixed elements with approximately 2.6 million cells. Figure 6 provides a general idea of the experimental construction and matched computational mesh.

![Figure 6. CAD model and CFD mesh detail](image)

The CFD model was set up with a uniform mass flow inlet boundary condition (BC) while varying the relative spray injection parameters and location and velocity magnitude in the duct. Table 1 indicates the nozzle operating parameters and drop size parameters input for simulations. The outlet of the duct was defined with a constant pressure boundary condition. The wind tunnel duct and tube walls were specified as rigid with no-slip and adiabatic conditions. Multiple turbulence models were evaluated to determine their suitability. The following models were finally applied in CFD simulation: k-ε Realizable turbulence model, discrete phase model for LaGrangian droplet tracking and species transport model to include mixing of air. In this work, simulations were performed in steady state scheme.

**Results and Discussion**

**Gas Baseline Results**

The two velocity magnitudes, 5m/s and 10m/s, cases were named as low and high flow cases, respectively. Gas velocity, volume flow rate and static pressure were measured by the VELOCICALC meter and used to compare with simulation results. The data comparison can be seen in Figure 7.

![Figure 7. Gas phase only CFD and Experiment Results for Two Flow Cases](image)

Velocity and volume flow rate of both cases on the plane of tubefaces (the closest plane, refer to wind tunnel) and outlet were matched between experiment and CFD. Compared with the pressure across heat exchanger structure, CFD showed bigger pressure drop than experimental values. Pressure loss may occur during testing, as some air leaked out from the Pitot tube insertion port, due to required access ports for instrumentation. However, the overall differential pressure in CFD was not as high as operating pressure and it was proportional with the experimental test pressure.
PDI Results

At ambient conditions, the drop size distributions, velocity, and spray plume angle were obtained at 0.05 m downstream from the nozzle exit orifice with PDI measurements (no wind tunnel), which was used to define the CFD model spray injection parameters. This process of combining the initial velocity characteristics and downstream drop size characteristics was necessary in order to account for the lack of droplet collision and coalescence in the steady state model. The ANSYS Fluent input for drop size distribution was specified using the Rosin-Rammler distribution function, see Equation 1.

\[ F(D) = 1 - \exp\left(-\frac{D}{X}\right)^q \]  \hspace{1cm} (1)

\( F(D) \) is the fraction of total volume of drops with diameter less than \( D \). \( X \) and \( q \) are constants inherent to the Rosin-Rammler function associated with the distribution center and width, respectively \[^7\]. This function is used to convert raw measure drop data into drop size distribution function for CFD. The minimum and maximum diameter input for CFD was specified based on volume flux and area weighted average of \( D_{0.01} \) and \( D_{0.99} \), correspondingly. The \( D_{0.5} \) diameter was used to evaluate the drop size data. This drop size terminology is as follows:

- \( D_{0.5} \): Volume Median Diameter (also known as VMD or MVD). A means of expressing drop size in terms of the volume of liquid sprayed. The VMD is a value where 50% of the total volume (or mass) of liquid sprayed is made up of drops with diameters equal to or smaller than the median value. This diameter is used to compare the change in average drop size between test conditions.

In wind tunnel test section, for quantitative comparison of experimental and computational spray behavior at the heat exchanger face, measurement at \( x = 0, 0.075 \) and 0.15 meter line vertically was planned to give an insight of performance of various injectors.

On the left side of Figures 8-11 showed the PDI tested average droplet velocity of each nozzle arrangements along y axis at thee lines in both air flow cases, respectively. The results present the trends of droplet velocity per layout. At two different flow velocities results were very similar except a few points in case 3 and case 4. Conversely for two flow cases, average droplet velocity was separated and tended to approach corresponding gas flow velocity in all three lines. Droplet velocity was similar at both center and 0.075m measurements, but smaller near the wall in cases 1 and low air flow 3 and 4. However, the 0.15m line result trends were opposite in high flow cases 2, 3 and 4.

Figures 12-15 presented \( D_{0.5} \) measured by PDI for these four nozzles in their arrangement at both flow rates. In case 2, volume median diameter matched at 5m/s and 10m/s air velocities at all three vertical measurement locations. This was repeated for case 1 and case 3 except for 0.15m off-center location. \( D_{0.5} \) at centerline in case 4 showed similarity but variance in 3 and 0.15m locations. Unlike case 1, where the biggest \( D_{0.5} \) showed up at the center of the test plane, case 2 and 3 exhibited larger \( D_{0.5} \) near the wall of the duct. Case 4 had the most uniform \( D_{0.5} \) along three test locations among all the cases.

The above trend follows the expected results and observed phenomenon in the duct. All drop size and velocity data was defined acceptable based on sufficient droplet concentrations (counts) and valid data rates. In general an acceptable rate was about 30 Hz or 30 droplets per second, threshold may be slightly lower according to test difficulty and resolution. The droplet average velocity was approximately inversely proportional to droplet volume median diameter, particularly to the peaks.

CFD Results

The CFD calculated average droplet velocity per nozzle arrangement at two flow cases was displayed at the right side of Figures 8-11. As shown, the trends in each graph for low and high flow were almost identical, just slightly bigger oscillation in each low flow case. Droplet velocity at all \( x = 0.15m \) locations was smaller than at \( x = 0 \) and 0.075m locations, which were close to

![Figure 16. Velocity of Four Cases under Both Air Flows](image-url)
gas velocity. Excluding case 3, in which two nozzles were placed above the other one (already non-symmetrical structure), all cases showed a symmetrical profile for three different x locations by y direction centerline.

$D_{V0.5}$ results from CFD was acquired from droplet trajectory at heat exchanger face (same location as the experimental test place) and based on volume flux and area weighted, which were shown at the right sides of Figures 12-15. $D_{V0.5}$ data matched in Case 1; trends were approximately similar for the other three layouts at two air flow velocities, but shifting a little along the y direction or at the drop size ranges.

Result Comparisons and analysis

It can be seen from experiment and CFD, droplet velocity and size is significantly influenced by the air flow in the co-current direction spray. Velocity data from empirical and computational cases indicates droplets were likely carried with gas flow once they exited the orifice.

The drop size and velocity trends are mostly consistent from pratical and analytical measurements. Velocity profiles replicated well at $x = 0$ and 0.075m locations for all cases, and $x = 0.15$m location of case 1 in both results. VMD profiles from experimental and CFD have similar trends but did not accurately hit absolute values, particularly obvious in case 4. Overall, CFD predicted wider drop size range than experimental data.

Hollow cone injectors form a wide ring shape distribution, which lead to large drop size at the edge and small drop size toward the center under ambient conditions. In this study spray was contained in a fairly tight space, hence test results could be affected by the droplet splash and bounce from hitting the duct wall or the wall film at the edge of the test plane. It can be explained with a existing theory that the larger droplets along the perimeter are formed due to stripping of the wall film (a wind tunnel water wash study by Brown, et al) [8]. Therefore, larger drop size would be expected along the wall surface of the duct. In this study, large drop size and high velocities were detected close to the wall surface in both PDI and CFD results for each location of all layouts. This phenomenon fits with this injector type and is supported by the previous explanation, which demonstrates the good agreement practically and analytically.

Most deviating points were mainly located on 0.15m location which was close to the duct wall and under the influence of wall wetting and droplet movement, or on the top of each vertical location, which next to the opening (refer to wind tunnel) or at the location bottom was likely affected by turbulence of air flow caused by the pressure loss of the opening and blockage of the tubeface. Unexpectedly, large droplets with velocity approaching the wall would run off or drop out of the air stream. Hence, at those locations the slightly higher velocity value and bigger drop size of empirical
data was understandable and can be interpreted as reasonable data, correlating to computational data.

The fair agreement of DV0.5 and droplet average velocity indicated the capability of using CFD to precisely predict spray behavior and performance at invisible and in situ situations. In case 1, a single centered hollow cone nozzle was applied. Its plume was possibly influenced by the air flow and wall effect. The PDI result showed large drop size at the center. This was not expected but plausible due to poor uniformity of coverage, as demonstrated in CFD by droplet concentration and spray coverage. In addition, bigger droplets tended to have more momentum and higher velocities. Case 4 had the least oscillating droplet velocity and most uniform VMD in the narrowest range. This resulted in a very even and smooth coverage at the tubeface, due to the fact that the smaller droplets remain entrained in the gas stream. While CFD results of case 4 presented the same effects as experiment, the VMD range was much wider than practical measurement. Possible reasons for this trend were (1) large droplets escaped through the opening during trials; (2) secondary break up occurred when spray was overlapped and droplets interfered each other aggressively. Nevertheless, both experimental and computational results exhibited uniformly distributed DV0.5 profile that lead to better spray performance.

In computational modeling, the wall condition models were found to be the key of how accurate droplets to wall effects can be simulated. There was no existing model can completely achieve replication of the empirical results. In this work, the most reliable wall model was selected on the basis of DPM concentration at the walls. It should be noted that CFD overestimated (perfect) spray based on input ambient condition, while experiment was often influenced by test environment and set up. Further transient CFD analysis may add to a more realistic result. Air flow in the wind tunnel could be the issue; drop size of hollow cone spray would break down further to smaller drops, which could cause the results of CFD to estimate larger VMD rather than PDI results.

**Conclusion**

In general, the practical and analytical results demonstrate a good agreement with the spray characteristics of four nozzles with corresponding configurations throughout low and high air flow rates. It demonstrates the capability of computational modeling to validate the spray performance in the defouling process. This would provide greater confidence in CFD applications of conditions which were physically untestable or cost prohibited to collect data. Some future work could be done to strengthen the conclusion: including defouling preferred injectors, different injector types, and additional investigation at set up improvement and combined with qualitatively spray measurement like LSI as well as refinement of the computational model.

**Nomenclature**

- x horizontal axial direction
- y vertical axial direction
- z spray direction

**Subscripts**

- g gas
- l liquid

**References**

5. Crockford, A., *The importance of cleaning validation for CIP systems*, Bioprocess Design XI: Cleaning-in-Place, Design and Validation, IChemE Biochemical Engineering Subject Group, UCL.
Figure 8. Average Velocity of Water Drops from PDI Test and CFD in Case 1 at high and low flow.

Figure 9. Average Velocity of Water Drops from PDI Test and CFD in Case 2 at high and low flow.
Figure 10. Average Velocity of Water Drops from PDI Test and CFD in Case 3 at high and low flow.

Figure 11. Average Velocity of Water Drops from PDI Test and CFD in Case 4 at high and low flow.
Figure 12. VMD of Water Drops from PDI Test and CFD in Case 1 at high and low flow.

Figure 13. VMD of Water Drops from PDI Test and CFD in Case 2 at high and low flow.
Figure 14. VMD of Water Drops from PDI Test and CFD in Case 3 at high and low flow.

Figure 15. VMD of Water Drops from PDI Test and CFD in Case 4 at high and low flow.