Experiment and Numerical Studies on the Atomization of a Swirl Feed Nozzle

Qilong Huang and Jinxian Li
College of Astronautics, Northwestern Polytechnical University, Xi’an, Shaanxi, China
Email: nonuclear911@163.com, lijinxian@nwpu.edu.cn

Abstract—Experimental and numerical investigation of spray characteristic of a swirl fluid catalytic cracking (FCC) feed nozzle was presented. In the experiment, the spray angle and sauter mean droplet (SMD) were measured. Then pressure drop on each section of nozzle was verified. Volume of Fluid (VOF) method was used to simulate the gas-liquid flow process in swirl FCC feed nozzle. The simulation exhibited the process of two-phase flow filling feed nozzle. The nozzle pressure were calculated, which were validated by the experimental data with good agreement. The results show that: the SMD size of feed nozzle was 59-74μm; nozzle with short sheet had the smallest SMD; cross grille in nozzle effectively inhibited the flow rotation.

Index Terms—FCC feed nozzle, SMD, two-phase flow, VOF

I. INTRODUCTION

The liquid petroleum feed is atomized by a gas through a nozzle into the FCC riser reactor in order to process the catalytic cracking reactions. The atomization fluid are admixed in the nozzle mixing chamber and form a spray having a fan-like shape [1]. The challenge of a successful feed nozzle design is to produce the finest feed atomization using the least amount of energy [2]. Many investigations have been conducted to study the SMD of nozzle. Benjamin [3] related the spray characteristics for a range of air to liquid (AIR) ratios. Wei Xiao [4] investigated effects of pressure-swirl nozzle geometry on SMD. With the development of numerical techniques, numerical methods have also become an effective means of research on spray. Especially the development of VOF method can well indicate the nozzle flow field. Ibrahim et al. [5] employed VOF method to study the flow field inside the swirl nozzle and predicted spray angle, film thickness at exit.

In this paper, performance of swirl feed nozzle atomization were studied by experiment. The nozzle flow field was simulated with VOF method. Numerical simulation results accorded with the experiment. The experimental and simulation results can provide reference for this type of nozzle design.

II. EXPERIMENTAL STUDY

A. Swirl Feed Nozzle Structure

As shown in Fig. 1, swirl feed nozzle is composed by center gas inlet, side liquid inlet, mixing chamber, cyclone, injection tube, cross grille and spherical head. Liquid into the mixing chamber is mixed with the gas and become film after the cyclone. Under the crushing of high speed airflow film were fractured. After passing through the injection tube, the nozzle atomization process completes.

![Figure 1. Swirl feed nozzle configuration.](image)

The difference between conventional nozzle and swirl feed nozzle is the latter installed cross grille in injection tube and sheet in nozzle head. The purpose is to suppression spray deflection and to improve the distribution of atomization. The cross grille and sheet are shown in Fig. 2.

![Figure 2. Note how the caption is centered in the column.](image)

B. Experimental System

Nozzle test system is shown in Fig. 3. The test medium is water and compressed air. Using pump-type water supply system and required air from gas source through valve. SMD and particle velocity were measured by a three-dimensional laser Phase Doppler Particle Analyzer (PDPA). It was used a digital camera to shoot the atomization on the side of nozzle. The real-time pressure data of nozzle were collected by the pressure sensors.
should not be excessive. Based on past experience in installation of sheet is larger. Nozzle spray velocity grille. But the particle velocities are contrary, the installed sheet is better than the nozzle installed cross region of atomization, the SMD of nozzle installed of the region of atomization. In comparison, the front position is attention to nozzle design. The position of cross grille is greater than sheet and liquid particle pressure potential is consumed. Overall, the losses caused by the cross grille is little effect on nozzle pressure drop.

The main factor affecting the spray angle is the expansion angle of nozzle outlet. In addition to that there is gas and liquid flow ratio. The nozzle head outlet is a rectangular slot which expansion angle is 100°. On the experimental situation, the length of sheet and the position of cross grille have less influence on spray angle. In general, the larger spray angle, the sprays are affected more strongly by surrounding air, and the atomization particle diameters become smaller [7]. But FCC process require spray angle is not too large, because the spray will be jet at the edge of riser reactor, so the oil just adhere to the wall that is unfavorable reactions.

Two-phase flow pressure drop in the nozzle is one of the important parameter concerned by designer. The system pressure drop determines the power required. The lower total pressure drop, the energy is consumed fewer. Table III shows the pressure drops of nozzle in test, in each case the total pressure drops are similar. But the pressure drop inside the nozzle segments depending on whether the nozzle install with the cross grille or the sheet. The cross grille results in significantly lower pressure drop on cyclone and increases pressure drop on injection tube and spherical head, while the position of cross grille is little effect on nozzle pressure drop.

C. Experimental Results and Analysis

Table II shows the different configurations of the nozzle SMD and particle velocities. Installation of the cross grille and the sheet in head result in increased of the atomization particle diameters, because the cross grille and the sheet lead to pressure drop, then a portion of the pressure potential is consumed. Overall, the losses caused by the cross grille greater than sheet and liquid particle diameters increase the most. The SMD of nozzle installed of short sheet is smaller than installation of long sheet about 4μm. The longer sheet, the pressure potential is consumed more. Therefore, it is effected on the atomization particle diameters. That should be pay attention to nozzle design. The position of cross grille is also great impact on SMD, particularly in the central region of atomization. In comparison, the front position is best, the end of injection tube is worst. In the central region of atomization, the SMD of nozzle installed of the front of cross grille is smaller than the end of position about 8.5μm. Basically, comparing with SMD, the nozzle installed sheet is better than the nozzle installed cross grille. But the particle velocities are contrary, the installation of sheet is larger. Nozzle spray velocity should not be excessive. Based on past experience in engineering, when the spray particle velocities are higher than 60m/s, the jet will affect catalyst [6].

<table>
<thead>
<tr>
<th>Nozzle number</th>
<th>Nozzle head form</th>
<th>Cross grille position</th>
</tr>
</thead>
<tbody>
<tr>
<td>1#</td>
<td>Short sheet head</td>
<td>None</td>
</tr>
<tr>
<td>2#</td>
<td>Long sheet head</td>
<td>None</td>
</tr>
<tr>
<td>3#</td>
<td>Original head</td>
<td>Front of tube</td>
</tr>
<tr>
<td>4#</td>
<td>Original head</td>
<td>Middle of tube</td>
</tr>
<tr>
<td>5#</td>
<td>Original head</td>
<td>End of tube</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Nozzle number</th>
<th>Liquid inlet (MPa)</th>
<th>Cyclone (MPa)</th>
<th>Injection tube (MPa)</th>
<th>Head (MPa)</th>
<th>Total (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1#</td>
<td>0.077</td>
<td>0.076</td>
<td>0.091</td>
<td>0.155</td>
<td>0.399</td>
</tr>
<tr>
<td>2#</td>
<td>0.076</td>
<td>0.074</td>
<td>0.109</td>
<td>0.135</td>
<td>0.394</td>
</tr>
<tr>
<td>3#</td>
<td>0.074</td>
<td>0.066</td>
<td>0.139</td>
<td>0.132</td>
<td>0.411</td>
</tr>
<tr>
<td>4#</td>
<td>0.074</td>
<td>0.063</td>
<td>0.115</td>
<td>0.135</td>
<td>0.387</td>
</tr>
<tr>
<td>5#</td>
<td>0.079</td>
<td>0.064</td>
<td>0.111</td>
<td>0.131</td>
<td>0.385</td>
</tr>
</tbody>
</table>

A. Physical Model

In the calculation, consider the nozzle spray process about two-phase flow through the cyclone from the mixing chamber, ignoring the structure of the external cavity. In the cyclone, using unstructured tetrahedral meshes, and the remaining parts are structured by hexahedral meshes. Inlet boundary condition is mass flow inlet and outlet boundary condition is pressure outlet. The back pressure set to atmospheric pressure.

B. Mathematical Model

Swirl nozzle internal flow is a typical gas-liquid flow. To correct expression of surface tension at the interface of two-phase, gas-liquid interface needs to be tracked and
described. Gas-liquid interface use VOF method to determine.

Assuming the fluid is incompressible, the control equations are

\[ \nabla \cdot \mathbf{v} = 0 \]  
\[ \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = -\frac{1}{\rho} \nabla P + \nu \nabla^2 \mathbf{v} \]  

(1) \hspace{1cm} (2)

In the equations, \( \mathbf{v} \) is velocity vector, \( \rho \) is density, \( P \) is pressure, \( \nu \) is motion viscosity coefficient.

VOF method is introduced parameter C represented the volume percent of liquid in unit control body

\[ C = \begin{cases} 0 & \text{if } C \leq 0 \\ 1 & \text{if } C \geq 1 \\ 0 < C < 1 \end{cases} \]  

(3)

Volume fraction equation

\[ \frac{\partial F}{\partial t} + \mathbf{v} \cdot \nabla F = 0 \]  

(4)

C. Mathematical Model

Nozzle is a unit realizing the liquid atomization, which structure has great impact on the initial atomization parameters. Studying internal flow characteristics inside the nozzle has a lot of sense to atomization initial parameters [8]. Fig. 4 shows the process of two-phase flow filling nozzle. At 20ms initial two-phase flow fills cyclone. After 30ms two-phase flow through the cyclone trough and close to the wall reached at the middle of the injection tube. Affected by the cyclone configuration, there is an annular flow in the injection tube with an air vortex at 40ms. When 60ms the fluid begins to eject from the nozzle. Impacted by hemispherical nozzle head, spray flow converge toward the center. But the nozzle is different from conventional swirler. It will not form a stable annular flow and air vortex, and has variety of fluid state in the injection tube. After 140ms the air vortex gradually disappears, fan-shaped spray has been basically expanded. At 200ms the flow field comes to a steady state and fan-shaped spray fully expands.

Cyclone forces two-phase flow rotation which get the circumferential direction of velocity. Due to the impact of the cyclone, so that the flow in the nozzle is complex. Fig. 5 is flow stream chart after the nozzle flow is steady. Looked forward from the nozzle inlet, two-phase flow inside the nozzle rotates very obviously. Fig. 6 is flow streamlines of nozzle installed cross grille. It can be seen that cross grille inhibit rotational flow clearly, and flow streamlines passed through cross grille are very straight. In the injection tube diameter and tangential velocity basically become zero. Moreover, the sheet in nozzle head also inhibits the rotational flow as shown in Fig. 7.
In the nozzle flow field, pressure distribution in cyclone is the most complex. Fig. 8 is pressure contours of the flow field in each nozzle configuration. There is about 0.1MPa pressure drop on cyclone. Because of greater flow resistance on cyclone it is more pressure gradient. Outside the cyclone, the center of flow field pressure is low, on both sides is high. After installed cross grille, it generates significant pressure gradient at the outlet of the cross grille.

The simulation results are close to experimental data of each nozzle. Table IV shows the values of the mixing chamber experimental measurements and numerical simulation results. The simulation results slightly smaller than the experimental values. This is due to the influence of the experimental prototype machining accuracy, then cyclone actual flow area slightly larger than design value, resulting in less pressure drop on cyclone and large pressure in the mixing chamber.

### IV. CONCLUSION

In this paper, experimental and numerical study of spray characteristic of a swirl feed nozzle was presented. The SMD sizes of feed nozzle were 59-74μm. Nozzle of short sheet structure has the smallest SMD. At 200ms the flow field comes to a steady state, fan-shaped spray fully expanded. And the cross grille inhibit rotational flow obviously. Because of installing cross grille, it generates significant pressure gradient at the outlet of the cross grille.

### REFERENCES


