Abstract

The Fluid Catalytic Cracking (FCC) feed injection process requires the atomization of a liquid hydrocarbon feed at high capacity. Due to the high viscosity level a secondary, atomizing gas (steam) is used to improve atomization and cracking. The development and application of an enhanced-efficiency nozzle for use in the FCC feed injection process has been conducted and evaluated at Spraying Systems Co. In this application, ‘high-efficiency’ refers to a low relative steam-to-hydrocarbon mass flow rate ratio. The efficacy of this process has been evaluated both experimentally and numerically (CFD) based partially on the surface area to volume ratio of the resulting droplets. A patented [2, 3, 4, 5] Spraying Systems Co. FCC nozzle design which uses a liquid jet impingement pin, an atomizing cross-flow, and slot-like nozzle outlets in pairs is applied.

*Corresponding author
FCC Nozzle Technology, Design, Characterization, and Validation for Optimal Performance

Introduction

FCC Background

The Fluid Catalytic Cracking process utilizes steam in order to more effectively atomize liquid hydrocarbon feed entering the FCC riser. An atomized hydrocarbon feed will more easily provide reaction surface area for the cracking catalyst.

This paper will outline the design considerations and development of an FCC injector and evaluation based on the critical elements of FCC system characteristics. This process provides an overview of the in-depth work which is involved in nozzle design and evaluation. Evaluations will be based on computational fluid dynamics (CFD) simulations of two-fluid mixing (density and temperature) and experimental measurements of drop size, drop size distribution, and drop velocity.

THEORETICAL CONSIDERATIONS

Atomization

The process of atomization is used in many spray applications to produce high surface area to volume ratios of the generated droplets. Often this high ratio provides much more efficient use of the spray droplets in evaporative and/or combustion processes. In general, atomizers which cause the greatest physical interaction between the liquid and vapor are most effective.

Liquid Jet Atomization

The atomization of any liquid jet into a gas region can be characterized into primary and secondary mechanisms. Primary atomization is caused by the initial instabilities within the liquid jet which act to disintegrate the jet internally. Secondary atomization considers the further breakup of drops larger than the critical drop size. In characterizing a jet there are two main properties which are often discussed: the continuous jet length and the drop size which may be used to evaluate the expected breakup of a jet.

Jet Breakup

A liquid jet exhausted into air may do so in a laminar or turbulent state. A laminar jet, which contains fluid particles which are traveling in parallel at the exit plane, may be created by utilizing a rounded inlet, having no mid-flow disturbances, and using a high viscosity liquid. Turbulence in jets, which aides in jet breakup, may be encouraged through high flow velocities, large tube sizes, general surface roughness, rapid cross-sectional changes, and perturbations due to flow obstructions or vibrations. The Reynolds number, \( Re = \frac{\rho U_r D}{\mu} \), which relates pressure and viscous forces, may be used to determine the likelihood of a flow to be laminar (low Re number) or turbulent (high Re number). The critical Reynolds number identifies where laminar flow will undergo the transition to turbulent flow. For pipe flow, the critical Reynolds number is \( Re_{crit} \approx 2300 \). When a flow transitions from laminar to turbulent flow, the mechanisms governing jet breakup change and cause a decrease in jet breakup length.

Liquid Sheet Breakup

Fraser & Eisenklam (1953) defined and described three liquid sheet breakup regimes: Rim, Perforated Sheet, and Wave. Liquid surface tension and viscosity are the primary properties which determine which mode(s) of disintegration occur.

Liquid sheet breakup through “rim” disintegration often occurs with a high viscosity, high surface tension type liquids. In rim disintegration the liquid mass becomes thicker at the free edges which ultimately form liquid threads which breakup into large drops, whereas the internal area disintegrates and forms smaller drops.

In a “perforated sheet” type breakup, many holes are developed in the liquid sheet. The edges of these holes become thicker as the holes grow and more fluid mass is combined at each hole-edge. These holes continued to grow until they encounter other rims and coalesce. Many size drops are created.
“Wave” disintegration occurs when wave motions within the liquid sheet cause fluctuations with distinct wavelengths. These waves break-up into whole or half wavelength sections and surface tension reforms the sections into strands. These strands then disintegrate into drops. Wave disintegration creates drops that vary the most in size.

**Drop Breakup**

Atomization is the process by which a liquid jet is disintegrated by aerodynamic forces. These aerodynamic forces which cause the liquid to form into small drops, and often further breakdown into droplets, are created by the relative velocities of the liquid jet to its surroundings. The breakdown of drops in a spray can be summarized with an internal/external force assessment. The external aerodynamic pressure is balanced by the surface tension in order for the internal drop pressure to remain at a constant level, which it must in order to sustain its drop-size.

In the event that the external forces are too large to be balanced though an increase in effective surface tension, the surface tension will be drastically increased through a decrease in the diameter of the drop (drop splitting). The process of drop splitting takes place until the surface tension pressure is large enough to counteract the aerodynamic drag pressure at all points on the drop’s surface. The drop size at this equilibrium level is known as the critical drop size. The mechanisms which cause the breakup of drops can be further identified by considering some of the more complex aspects of ‘real world’ conditions.

In turbulent flow fields the relative velocity between a drop and the surrounding gas will be very high, either locally or on a global scale. The turbulent field will impart a dynamic force on the drop which will determine the largest drop size that may exist in equilibrium due to the energy in the most disruptive turbulence scales \( E \). With dynamic turbulent forces present, the Weber number, \( \text{We} = \rho_\text{L} U^2 R (D/\sigma) \), which for low-viscosity liquids relates the deforming external pressure forces to the reforming surface tension forces of a liquid drop in air, can be evaluated for low-viscosity liquids and used to estimate the maximum drops based on these scales.

In high viscosity (low Reynolds number) flow fields, where dynamic forces no longer control breakup, the surface tension forces and viscous forces work to deform and reform liquid drops. It is generally very difficult to atomize liquids that have a high viscosity ratio of the liquid to air. In these high-viscosity flows, variations in the air viscosity make little difference on the atomization process. Also, high-viscosity liquid phase spray material delays the breakup of drops and impedes atomization which is why more aggressive methods, such as air-blast atomization, are often used.

**THEORETICAL CONSIDERATIONS**

**Hydraulic Nozzles**

Hydraulic nozzles are pressure driven nozzles which spray a single fluid. Many different types of hydraulic nozzle designs exist which aim to accomplish a variety of spray objectives from a continuous stream to a dispersed spray. These nozzles rely solely upon high liquid-to-gas relative velocities at exhaustion to achieve atomization. Liquids with low viscosity/high velocities more readily atomize; therefore hydraulic nozzles may suffice.

**Two-Fluid (Pneumatic) Nozzles**

The mixing of two fluids (usually one liquid phase and one gaseous phase) by a nozzle may be accomplished either internally or externally to the nozzle body. In an internally mixing nozzle, the liquid flow and gas flow interact upstream of the final discharge orifice. In this case, the mixture exits as a single, mixed flow; which widens with a reduced liquid flow velocity due to the pressurized gas. An internally mixing nozzle is optimum for high-viscosity fluids in a low flow rate application since the breakup of this type of flow is
The liquid properties such as (dynamic) viscosity, density, and surface tension directly effect and determine the spray type and quality which is created at a given operating condition. Liquid viscosity is representative of the willingness of a fluid to take the shape of its surroundings. A high viscosity fluid (syrup-like) will resist conforming to its surroundings and will move very slowly. The liquid viscosity directly affects the pattern at which a liquid will spray. High viscosity liquids require a higher pressure to spray due to their resistance to flow and will naturally form narrower sprays. Liquid density is directly proportional to the capacity of a spray. Density represents the mass-to-volume ratio for a liquid and therefore spraying a high-density liquid at a given velocity will result in a higher capacity spray. Liquid surface tension is a property representative of the internal force which holds a liquid together. This internal tension affects the liquid sprays minimum operating pressure, spray angle, and drop size. A higher surface tension will require a higher operating pressure, reduce the spray angle, and produce larger drop sizes. See Table 1 [10] for representative liquid properties and their general affects.

**Table 1. Atomization & Fluid Properties**

<table>
<thead>
<tr>
<th>Fluid Property</th>
<th>Increase in Operating Pressure</th>
<th>Increase in Specific Gravity</th>
<th>Increase in Viscosity</th>
<th>Increase in Liquid Temperature</th>
<th>Increase in Surface Tension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pattern Quality</td>
<td>Improves</td>
<td>Negligible</td>
<td>Worsens</td>
<td>Improves</td>
<td>Negligible</td>
</tr>
<tr>
<td>Capacity</td>
<td>Increases</td>
<td>Decreases</td>
<td>Nozzle Dependent</td>
<td>Fluid and Nozzle Dependent</td>
<td>No Effect</td>
</tr>
<tr>
<td>Spray Angle</td>
<td>Increases/Decreases</td>
<td>Negligible</td>
<td>Decreases</td>
<td>Increases</td>
<td>Decreases</td>
</tr>
<tr>
<td>Drop Size</td>
<td>Decreases</td>
<td>Negligible</td>
<td>Increases</td>
<td>Decreases</td>
<td>Increases</td>
</tr>
<tr>
<td>Velocity</td>
<td>Increases</td>
<td>Decreases</td>
<td>Decreases</td>
<td>Increases</td>
<td>Negligible</td>
</tr>
<tr>
<td>Impact</td>
<td>Increases</td>
<td>Negligible</td>
<td>Decreases</td>
<td>Increases</td>
<td>Negligible</td>
</tr>
<tr>
<td>Wear</td>
<td>Increases</td>
<td>Negligible</td>
<td>Decreases</td>
<td>Fluid and Nozzle Dependent</td>
<td>No Effect</td>
</tr>
</tbody>
</table>
FCC Nozzle Technology, Design, Characterization, and Validation
for Optimal Performance

IN PRACTICE

Nozzle Property Considerations

Nozzle construction materials may vary from lightweight plastics to case hardened metals. The material which is most suited to an application directly depends on the spray substance, spray environment (corrosive, heated, etc.), and desired spray characteristics. There are many readily available standard nozzle materials such as plastic, aluminum, and stainless steel as well as standard nozzle designs; the combination of material and design must be evaluated and optimized for each application.

Nozzle Design (for the current application)

Spraying Systems Co. patented (PAT# 6,098,896) design of an “Enhanced Efficiency Nozzle for use in Fluidized Catalytic Cracking” provides a refined solution to the FCC feed injection process. The use of an impingement pin, along with a transversely intersecting steam flow, greatly improves the efficacy and efficiency of the FCC feed injection process. In addition to these elements, the patented FCC unit provides specification for exit nozzle configurations to improve the post-discharge atomization of the fluid. Figure 1 provides a schematic representation of one of the Spraying Systems Co. FCC unit design configurations.

Through the results presented in the patent, the major design aspects (specifically the dual exit orifice) implemented in the unit are shown to improve the operation of the FCC injection process. The Spraying Systems Co. FCC nozzle offers many advantages, some of which are:

- Flat Spray pattern to fit exact coverage with maximum atomization for a given amount of steam or flow
- Controlled spray velocity
- Each injector nozzle is custom designed to exact specifications to maximize performance
- Ability to provide good atomization at relatively low ΔP (30–40 psi liquid; 50–60 psi stream)
- Large non-clogging passages
- Rugged, durable construction
- On-going research for improved product
- Products are design patent protected
- 25 years of continuous service
- This FCC nozzle design was fabricated and experimentally tested under conditions outlined in the following sections.

These experimental tests, aimed at better understanding the atomization process/system presented by this nozzle, were conducted using a one-eighth scale model nozzle. These tests were conducted using a 2" CS FCC feed, single slot orifice injector nozzle, which was modeled after a commercial unit. The production size unit's liquid inlet flange-to-exit orifice OAL centerline distance is 54". The results of these tests help to characterize the spray created by this nozzle. The numerical simulation conducted for this work helps to characterize the internal mechanisms which could not be investigated experimentally. These results provide a better understanding of the output of this nozzle and the mechanisms which allow it to perform as it does.

![Figure 1. Patented Enhanced Efficiency FCC Nozzle (Patent# 6,098,896)](image-url)
**Experimental Testing**

**Test Setup and Data Acquisition**

For drop sizing, the nozzle was mounted on a fixed platform 72” from the floor. A fixed assembly held the nozzle in place and data were acquired at 36” downstream of the nozzle exit. Drop size and velocity data was collected at various operating conditions.

A two-dimensional Artium Technologies PDI-200MD [6, 7, 8, 9] system was used to acquire drop size and velocity measurements. The solid state laser systems (green 532 nm and red 660 nm) used in the PDI-200MD are Class 3B lasers and provide 50-60mWatts of power per beam. The lasers were operated at an adequate power setting to overcome interference due to spray density.

The transmitter and receiver were mounted on a rail assembly with rotary plates; a 40° forward scatter collection angle was used. For this particular test, the choice of lenses was 500mm for the transmitter and 500mm for the receiver unit. This resulted in an ideal size range of about 3.5μm – 409μm diameter drops. The optical setup was used to ensure acquisition of the full range of drop sizes, while maintaining good measurement resolution. The particular range used for these tests was determined by a preliminary test-run where the D<sub>V0.5</sub> and the overall droplet distribution were examined. For each test point, a total of 10,000 samples were acquired. The experimental setup can be seen in Figures 2a and 2b.

The D<sub>V0.1</sub>, D<sub>V0.5</sub>, D<sub>32</sub>, and D<sub>V0.9</sub> diameters were used to evaluate the drop size data. This drop size terminology is as follows:

- **D<sub>V0.1</sub>:** is a value where 10% of the total volume (or mass) of liquid sprayed is made up of drops with diameters smaller or equal to this value.
- **D<sub>32</sub>:** Sauter Mean Diameter (also known as SMD) is a means of expressing the fineness of a spray in terms of the surface area produced by the spray. SMD is the diameter of a drop having the same volume to surface area ratio as the total volume of all the drops to the total surface area of all the drops.
- **D<sub>V0.5</sub>:** 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.
- **D<sub>V0.9</sub>:** is a value where 90% of the total volume (or mass) of liquid sprayed is made up of drops with diameters smaller or equal to this value.

By analyzing drop size based on these standardized drop statistics it is possible to objectively characterize the quality and effectiveness of this atomizing nozzle for the prescribed application.

**Test Fluids and Monitoring Equipment**

All testing was conducted using water and compressed air.

Liquid flow to the system was supplied using a high volume pump at full capacity. The liquid flow rate to the atomizer was monitored with a MicroMotion D6 flow meter and controlled with a large bleed-off valve. The MicroMotion flow meter is a Coriolis Mass flow meter which measures the density of the fluid to determine the volume flow. The meter is accurate to ±0.4% of reading. Air flow rate was controlled with a gate valve and monitored with an in-line Brooks model MT3809 variable-area flow meter, this meter is rated as class 1.6 accurate. Also, liquid and air pressures were monitored upstream of the nozzle with 0-7000Pa, class 3A pressure gauges. Pictures of the setup and operation of the test nozzle were provided in Figures 2a and 2b.
FCC Nozzle Technology, Design, Characterization, and Validation for Optimal Performance

**EXPERIMENTAL TESTING**

**Test Conditions**

Data were acquired for a total of 9 test cases. These tests involved the interaction of various liquid/gas pressure levels and flow rates. Table 3 provides a detailed list of these test conditions. The testing was carried out at these operating conditions in order to characterize the nozzle’s spray at various testing conditions and determine optimal operating conditions for this application.

**NUMERICAL SIMULATIONS**

**CFD Methods**

Computational Fluid Dynamics (CFD) is a numerical method used to analytically solve fluid flow problems. Computers are used to perform the extremely large number of calculations required to simulate the behavior of fluids in complex geometries. CFD is applied to a flow region which is divided into many computational domains or cells. This discretized region sometimes called a computational grid, or more commonly a mesh, is used to breakup the continuous flow domain into a finite number of smaller domains. Within the computational region, CFD solves the Navier-Stokes equations to obtain velocity, pressure, temperature and other quantities that may be desired, see Figure 3 for these discretization equations. These equations form the connection between the physics of fluid flow and the discretized region. Recently CFD became a popular design and optimization tool with the help of commercially available software and advanced computer technology.

The CFD simulation problem called for assessment of the mixing of oil and steam inside the FCC injection nozzle. The nozzle consists of one inlet for the oil, one inlet for the steam, and one outlet through which the mixture exits. Both fluids are to be injected at the inlets at high mass flows and pressures, see Table 2.

The commercially available CFD package Fluent (version 6.3.26) was used for the simulation of the oil/steam flow inside the mixing chamber of the injection nozzle. To generate the mesh for the mixing chamber shown in Figure 4, Gambit (version 2.3.16) was utilized as is demonstrated in Figure 5. The simulation required significant computer power and processing time. The mesh consisted of one-million cells and required little over 20K iterations for an adequately converged solution. The CFD work performed for

![Figure 3. CFD mesh and governing equations](image-url)
these investigations provides valuable information regarding the internal mixing of the two fluids that would be extremely difficult to acquire through experimental testing. The effectiveness of each of the nozzle’s internal elements may be examined.

<table>
<thead>
<tr>
<th>Boundary condition</th>
<th>Oil INLET</th>
<th>Steam INLET</th>
<th>Mixture OUTLET</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass inflow</td>
<td>1.694</td>
<td>0.0983</td>
<td>2.061*</td>
</tr>
<tr>
<td>Mass flow rate (kg/s)</td>
<td>1.117</td>
<td>0.483</td>
<td>–</td>
</tr>
<tr>
<td>Pressure (MPa)</td>
<td>220</td>
<td>200</td>
<td>–</td>
</tr>
<tr>
<td>Temp.(°C)</td>
<td>220</td>
<td>200</td>
<td>–</td>
</tr>
<tr>
<td>Density (kg/m³)</td>
<td>826.3</td>
<td>2.738</td>
<td>54.37*</td>
</tr>
<tr>
<td>Viscosity (cP)</td>
<td>1</td>
<td>0.016</td>
<td>0.078*</td>
</tr>
<tr>
<td>Area (mm²)</td>
<td>47.31#</td>
<td>113.7#</td>
<td>1904.9#</td>
</tr>
<tr>
<td>Volume flow rate (cm³/s)</td>
<td>2.05*+</td>
<td>35.9*+</td>
<td>37.95*+</td>
</tr>
<tr>
<td>Ave. Vel. (m/s)</td>
<td>43.3*+</td>
<td>315.7*+</td>
<td>20.0*+</td>
</tr>
<tr>
<td>Oil (%)</td>
<td>100</td>
<td>0</td>
<td>6.3*</td>
</tr>
<tr>
<td>Steam (%)</td>
<td>0</td>
<td>100</td>
<td>93.7*</td>
</tr>
</tbody>
</table>

*based on CFD results, # based on mesh, + validated with theory

Table 2. CFD Constants & Boundary Conditions

**CFD Boundary Conditions and Assumptions**

In order to perform the CFD analysis, some assumptions had to be made regarding the major flow mechanisms; this is a common practice in CFD which is used to save computing time. The problem consisted of mass flow inlets (oil and steam) and one outflow (mixture) to solve for a steady state flow solution. Both fluids were treated as Newtonian and incompressible. Although the inlet temperatures were slightly different, the energy scheme was not included to save computational time. The walls were assumed rigid, insulated (heat transfer omitted), and the no-slip condition was applied. The flow was modeled with k-epsilon realizable turbulence model with standard wall functions. For the two-phase flow which exits the nozzle, the mixture model was applied.
Experimental Results

The results of the PDI measurements provide a representative characterization of the atomizer effectiveness at the 36” downstream investigation location. As outlined and described in the above sections, the results from testing are provided in Table 3. The average drop velocity and the average volume flux are also calculated and provided in Table 3. The Sauter Mean Diameter (D_{32}) as well as other representative diameter statistics based on the volume flow is presented; also, the diameter distributions are presented in Figure 6. These results allow the evaluation, qualitatively, of the dependence of drop size and drop velocity on the air flow rate as set by the air pressure.

With an increase in air flow rate:
- there is a decrease in median drop size,
- and there is an increase in mean drop velocity.

The results also indicate that with an increase in liquid (water) flow rate:
- there is no great dependence on drop size with liquid flow rate,
- and there is an increase in drop velocity.

Further examination of the drop size data may be facilitated by Figure 6 which presents the cumulative drop size distribution for the various test conditions. This cumulative distribution represents the percentage of the total volume flow made up of drops whose diameter is less than or equal to a given size. The nearly constant region (300-400 μm) indicates that the majority of the spray volume is contained within drops of diameter smaller than 300 μm. The drop sizes range from very small diameters (<25 μm) to much larger drops (>400 μm), with the majority of drops at this location falling between 50 μm and 250 μm.

CFD Results

The CFD results provide great insight into the internal mixing mechanisms and two-fluid interactions. The boundary conditions, as described above, mimic the intended real world operation of this FCC feed injection nozzle and provided quality inputs for numerical analyses. These CFD calculations, based on

<table>
<thead>
<tr>
<th>Test</th>
<th>P_{liq}</th>
<th>P_{air}</th>
<th>Q_{liq}</th>
<th>Q_{air}</th>
<th>D_{0.1}</th>
<th>D_{32}</th>
<th>D_{0.5}</th>
<th>D_{0.9}</th>
<th>V_{avg}</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.28</td>
<td>0.41</td>
<td>2145</td>
<td>21240</td>
<td>81</td>
<td>134</td>
<td>146</td>
<td>229</td>
<td>20.87</td>
</tr>
<tr>
<td>2</td>
<td>0.31</td>
<td>0.55</td>
<td>2271</td>
<td>23600</td>
<td>74</td>
<td>127</td>
<td>135</td>
<td>223</td>
<td>20.92</td>
</tr>
<tr>
<td>3</td>
<td>0.34</td>
<td>0.69</td>
<td>2398</td>
<td>26432</td>
<td>72</td>
<td>121</td>
<td>133</td>
<td>226</td>
<td>21.79</td>
</tr>
<tr>
<td>4</td>
<td>0.42</td>
<td>0.55</td>
<td>2524</td>
<td>23600</td>
<td>78</td>
<td>134</td>
<td>146</td>
<td>233</td>
<td>21.59</td>
</tr>
<tr>
<td>5</td>
<td>0.43</td>
<td>0.41</td>
<td>2524</td>
<td>25960</td>
<td>73</td>
<td>122</td>
<td>132</td>
<td>218</td>
<td>23.66</td>
</tr>
<tr>
<td>6</td>
<td>0.42</td>
<td>0.69</td>
<td>2524</td>
<td>27376</td>
<td>73</td>
<td>121</td>
<td>131</td>
<td>217</td>
<td>25.63</td>
</tr>
<tr>
<td>7</td>
<td>0.46</td>
<td>0.41</td>
<td>2776</td>
<td>23600</td>
<td>82</td>
<td>136</td>
<td>147</td>
<td>246</td>
<td>22.61</td>
</tr>
<tr>
<td>8</td>
<td>0.47</td>
<td>0.55</td>
<td>2776</td>
<td>21240</td>
<td>74</td>
<td>125</td>
<td>134</td>
<td>223</td>
<td>24.25</td>
</tr>
<tr>
<td>9</td>
<td>0.50</td>
<td>0.69</td>
<td>2776</td>
<td>26432</td>
<td>70</td>
<td>117</td>
<td>128</td>
<td>205</td>
<td>25.56</td>
</tr>
</tbody>
</table>

Table 3. Experimental Test Conditions and Results
RESULTS

on the k-epsilon turbulence model, were allowed to run through hundreds of iterations until a steady-state solution was realized. This solution was deemed acceptable when the calculation residuals (changes in the results from one iteration to the next) were negligibly low and good convergence was achieved. Once this steady state solution was found, the results could be plotted and assessments were made. Figure 7 provides the results of the flow volume’s velocity magnitude and two-fluid density. Also, cross-sections in 8 locations are provided in Figure 7 (top).

Figure 7. Velocity magnitude and density plots inside the mixing chamber

The velocity profile demonstrates the high velocity magnitude of both fluids as they enter the mixing region. As would be expected, there are very low velocity regions in the mixing chamber near the walls (see lower left contours) and the mixed flow has a fairly uniform, mid-level velocity profile as it moves down the nozzle exit neck (see cross-sections 7 & 8).

Examining the density contours, the steam fluid is represented by the light blue color, and the hydrocarbon fluid is represented by the black color. It is clear (see lower right contours in Figure 7) that both streams retain their fluid densities until they come in close proximity to the impingement pin. The fluid streams then interact very quickly and mix as they flow down the nozzle exit neck. At cross-sectional location 8 the two-fluids are very thoroughly mixed and will be exhausted through the exit.

Figure 8 provides oil particle tracks color coded with fluid material, velocity, or density in the near-impingement region. Figures 9a, 9b, and 9c then provide this information with oil and steam for the entire FCC nozzle flow volume region. The particle tracks represent the path particles would be expected to take if released from the inlet boundary. Figure 8 makes clear the immediate complexity of the particle path upon encountering the impingement pin. Computing ultra-complex trajectories such as these begins to explain why simulation such as these may take days to complete.

Figure 8. Partial paths of oil impingement and its effect on velocity and density

In figure 9a, the particle pathlines are color coded according to the density of the fluid at that location. It is clear that both streams enter the mixing nozzle at very different densities ($\rho_{\text{steam}} = 2.7 \text{ kg/m}^3; \rho_{\text{oil}} = 826 \text{ kg/m}^3$). However, before entering the nozzle exit the fluid density has normalized to a fairly uniform mid-range density ($\rho_{\text{mixture}} = 54.4 \text{ kg/m}^3$) which is one indication of good mixing.

Figure 9b highlights the internal mixing paths of each fluid. The oil (blue) flow clearly stays very intact until encountering the impingement pin at which point the stream diverges immediately towards the side-walls. This divergence demonstrates the effectiveness of
the impingement pin on dispersing the hydrocarbon flow. This aggressive spreading of the oil stream by the impingement pin provides a much thinner oil stream for the steam to break down further.

Figure 9c demonstrates the mostly uniform flow velocity at the nozzle exit. The few pathlines which exhibit very slow velocities are due to the no-slip wall condition. The core of the flow is within a small velocity range, which is a condition of sufficient mixing.

Figure 10 represents the wall shear stress on the internal surfaces of the FCC nozzle. The nearly uniform and lower order wall shear stress over the majority of the internal nozzle volume is expected. The larger levels of wall shear appear to occur where the hydrocarbon stream disperses outwards towards the walls due to the impingement pin. The shape and distribution of this high wall shear region is interesting and without a simulation such as this, may not have been properly assessed. Also, the very low wall shear stress at the centerline of the volume (see 'top view' in Figure 10) is expected with the existence of two counter-rotating flows meeting at the centerline and migrating down the nozzle neck after being split at the impingement pin. The very high wall shear stress (order of magnitude) at the impingement pin is expected and is quantified in the boxed inset of Figure 10.
RESULTS

In Figure 11, the turbulence intensity in each fluid is provided in a 3-dimensional contour. The contour is drawn on the ‘surface’ where each fluid’s velocity is 20m/s. Turbulence intensity is the RMS of the fluctuation in a flow. This quantity is useful in assessing the relative changes in fluid velocity fluctuations. As would be expected the highest turbulence intensities are located where the two streams meet. Due to the much higher viscosity of the hydrocarbon flow, the steam fluid will experience higher turbulence intensity levels.

Figure 11. Relative Turbulence intensity plotted on the iso-surface based on velocity magnitude of 20 m/s

DISCUSSION & CONCLUSIONS

Multidisciplinary engineering

These results do not consider all the real world conditions which act on this system. The design and optimization of this nozzle was evaluated based primarily on flow mechanics and geometry. Further investigations involving a more inclusive multidisciplinary (fluid dynamics, heat transfer, materials, chemistry, controls) evaluation would provide a more refined optimization. Certain parameters such as temperature and chemistry were simplified for these investigations due to the added complexity of these parameters’ inclusion. However, a good assessment of the primary mechanisms influencing the mixing and atomization in this application was completed in these tests. Full temperature and chemistry testing/simulation is often too costly (in both money and time) to be an effective mode of optimization.

Engineering optimization of FCC injectors

The optimal nozzle design for FCC feed injection incorporates the Spraying Systems Co. patented nozzle optimized based on the testing results presented in this report. The Spraying Systems Co. enhanced-efficiency nozzle can provide the necessary spray characteristics with minimal cracking steam. Drop sizes ranging from less than 25μm to greater than 250μm were created at all operating conditions and the average spray velocity ranged from 20m/s to 25m/s. The total volume flux varied from 5.5cc/cm³/s to 8.5cc/cm³/s; this allows for a broad range of drop size/velocity/volume flux combinations to be created.

The results of the CFD analyses provide insight into the internal mixing mechanics of the two-fluid atomization process and allow for better optimization. By performing high-accuracy, steady-state simulations of the nozzle a better understanding of the governing mixing forces and their relative effect on the internal mixing of the two flow streams. It is clear from these models, see Figures 7–11, that the impingement pin as well as the steam cross-flow create a highly-mixed, very uniform two-fluid flow stream as the mixture exits the nozzle end. Not modeled here is the flow mixing and dispersion downstream of the nozzle exit orifices which will also aide in the complete mixing and atomization of the flow.
The state of the art design employed in the Spraying Systems Co. enhanced-efficiency nozzle for fluidized catalytic cracking provides the highest level of current mixing technology to most effectively crack this hydrocarbon flow. The aspects of this design allow the Spraying Systems Co. nozzle to be both effective and efficient by using only a minimal amount of secondary flow (steam). This FCC nozzle design is the latest result in a line of many patents ranging from the initial design in 1982 to the highest refinement patented issued in August of 2000. These patents [2, 3, 4, 5] include specifications for each aspect of the nozzle including:

- Primary flow entrance parameters
- Secondary flow entrance parameters
- Impingement pin design, position, and function
- Internal mixing regions
- Nozzle mixing neck geometry and function
- Nozzle exit orifice(s) geometry, position, and function

The in-depth analyses of these, and previous, tests provide experimental, computational, and analytical basis for the optimization parameters which are employed in the use of this FCC nozzle. With improved knowledge of the internal mechanics and the external spray pattern, optimization of the nozzle at these operating conditions can be done very effectively.

**Final conclusions**

The advanced design of the Spraying Systems Co. enhanced efficiency nozzle for FCC feed injection has been refined multiple times for increased performance. The current design effectively and efficiently mixes the two fluid streams prior to exiting the nozzle and, upon exhaustion, atomizes the nearly uniform flow in either a diverging or converging jet. Further investigation based on multidisciplinary interactions would be effective in refining the optimization more precisely.

**Acknowledgements**

The authors would like to acknowledge Mr. Kyle Bade, Mr. Wojciech Kalata, Mr. Jim Haruch, Mr. Robert Jett, Mr. Dave Hauffman, and Mr. Dan Filicicchia of Spraying Systems Co. for their assistance with this project.
FCC Nozzle Technology, Design, Characterization, and Validation for Optimal Performance

References