AADE-08-DF-HO-35

Novel Liquid-Liquid and Liquid-Solid Mixers Improve Drilling Fluid Performance

Perry Lomond and Mukesh Kapila, M-I SWACO

Abstract

Drilling fluids perform a variety of functions including lubricating the drill bit, transporting cuttings, and enhancing borehole stability. These fluids are formulated using solid and liquid additives to achieve specific chemical and physical properties; however, efficient blending of these bulk chemicals is often overlooked. Most additives require adequate blending and shearing to ensure dispersion and improve reactions during fluid preparation, historically attempted using rig hoppers, gun-lines or agitators.

Because mixing is a multivariate problem, Computational Fluid Dynamics (CFD) has been used to model the fluid mixing process. Through the analysis of fluid boundaries, velocity vectors and volume fraction contours, fluid movement is traced using CFD dispersion models to improve mixing efficiency. This has facilitated development of mixing equipment to enhance dispersion and optimize the physical and chemical properties of the fluid.

An examination of the flow characteristics of solids added to a liquid stream using a multiple-pass mixing hopper/eductor and liquid-liquid blending using single-pass mixing systems is presented. This analysis has created new equipment designs which enhance liquid–solid and liquid–liquid mixing through geometry changes like the addition of flow diverters, static mixers and pressure zones. Optimized mixing equipment improves chemical dispersion, resulting in the realization of designed fluid properties.

The use of techniques like CFD increase the understanding of drilling fluid flow, product dispersion, and shear which promotes the development of novel technology for the mixing of drilling fluids. Other benefits include enhanced fluid performance, decreased chemical consumption, and decreased time and cost while improving overall technical performance.

Introduction

Mixing drilling fluids to maximize the rheological properties requires careful attention to the formulation and use of equipment that promotes the mixing process. This paper reviews the development of two types of mixing equipment. The first is a mixing hopper/eductor table that uses a unique nozzle and diffuser combination to mix bulk-powdered chemicals into a high-flow liquid matrix jet circulating through a closed-loop system. The second device is used to blend fluids of varying densities into a well-mixed slurry through a single-pass process for riserless drilling. Historically, this type of equipment was fabricated on an ad-hoc basis and today some of this first-generation mixing equipment is still used. However, due to costs of chemical constituents and the expected increase of fluid properties, the mixing of these drilling fluids require a more technical approach to optimize mixing capabilities. To create the optimal configuration for mixing equipment, iterative CFD modeling was used to evaluate the mixer geometry and identify or create changes in the flow stream to provide better mixing capabilities.

Equipment Design and CFD Modeling

Designing equipment to mix a dry bulk solid into a liquid stream or to blend two liquid slurries requires an understanding of fluid mechanics and mechanical design. Much of the initial work and concept development can be done using standard empirical design equations; however, the finer points of actual mixing cannot be found in such equations and complex modeling software like Computational Fluid Dynamics (CFD) is needed to complete the design. Material selection, structural support and load distribution can be developed through standard design principles. The nuances of fluid dynamics, fluid mixing and solids dispersion require CFD modeling to determine velocity vectors, turbulence generation, stream degradation, fluid stresses, and shearing that help clarify and evaluate flowline geometry and to optimize mixing effectiveness.

When mixing solids to a liquid jet or combining fluids of differing densities to achieve a specific blend of fluid rheology, the flowline design of the mixing equipment must promote mixing by stressing the fluid, thereby generating shear and turbulence. The combination of high turbulence regions and high shear can be seen in velocity contour plots from a CFD analysis. Peaks in the turbulence occur where rapid changes in the mean velocities are found or the contours tend to eddy.

Fluid Dynamics Equations

CFD analysis using both Reynolds Stress and k-ε forms were used to evaluate the fluid velocities, pressure gradients, and dispersion throughout the mixer boundaries. Equations used are:

Bernoulli's Equation:

$$\frac{p_1}{\rho g}+\frac{u_1^2}{2g}+z_1=\frac{p_2}{\rho g}+\frac{u_2^2}{2g}+z_2 \qquad \text{(Eq. 1)}$$

The continuity equation:

$$Q = A + u \qquad \text{(Eq. 2)}$$

Navier Stokes equation (2-D generalized format):

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=-\frac{\partial p}{\partial x_i}+\mu\frac{\partial^2 u_i}{\partial x_j^2} \qquad \text{(Eq. 3)}$$

Transport equations for k (turbulent kinetic energy) and ε (turbulent dissipation):

$$k=\frac{1}{2}u_i'^2=\frac{1}{2}\left(u_x'u_x'+u_y'u_y'+u_z'u_z'\right) \qquad \text{(Eq. 4)}$$

$$\varepsilon=\nu\left(\frac{\partial u_i}{\partial x_i}\right)^2=\nu\left(\frac{\partial u_x'\partial u_x'}{\partial x\partial x}+\frac{\partial u_y'\partial u_y'}{\partial y\partial y}+\frac{\partial u_z'\partial u_z'}{\partial z\partial z}\right) \qquad \text{(Eq. 5)}$$

Some of the simplifying assumptions imposed for these two projects were:

1. The flow will be in a steady state, *i.e.*, at any location, the pressure, velocity and temperature will not vary with time
2. The fluid will be incompressible, *i.e.*, density does not vary in time or space
3. The fluid will always be turbulent
4. The fluid will behave as a Newtonian fluid at the range of shear rates typically applied for mixing in the field.

Developing a Solid-Liquid Mixing Table

To mix dry bulk solids into a drilling fluid, one must employ a solid-liquid mixing device of which several are available to the industry. Before developing the M-I SWACO HIRIDE* solid-liquid mixing table, several trials were undertaken and a CFD analysis of the various mixing tables was completed to develop baseline information and to evaluate the equipment currently in use. To mix a solid and a liquid, the choice is limited to using either a batch process employing a mix tank and a source of fluid agitation, or a more aggressive mixing table that uses an eductor-type configuration. The solid-liquid mixing table continues to use a mixing tank, but the dry solids are added to a liquid stream that should rapidly mix with the chemical before re-entering the large-volume mixing tank. Mixing large-volume tanks (500 to 1000 bbl) by dumping bulk chemicals on top of a standing fluid column and agitating the liquid with various fan-type impellors has several issues that need to be addressed outside the scope of this paper; however, power, size, efficiency, and time summarize these problems. To minimize some of the issues associated with the batch process described above, avoid large "fish-eyes" and the potentially poor fluid rheology, the eductor-type mixing table is used.

It was determined that mixing tables typically have two common technologies and a multitude of approaches that implement those technologies. The more successful mixing-table design, the mixing-table eductor, utilizes the principle design of a venturi to create a low-pressure zone that helps vacuum powdered solids and combine them with a liquid stream. Typically, an external centrifugal pump provides the motive force for the liquid into the venturi where the shape of the nozzle/orifice accelerates the liquid stream by converting pressure head to velocity head. The high-velocity stream then travels through a primary mixing chamber with controlled volume where a low-pressure region and subsequent vacuum is created. The vacuum draws powder from the hopper to combine with the surface of the liquid and the powder becomes entrained in the liquid stream. The liquid and entrained powder then flow through a diffuser which creates two additional turbulent zones before decelerating the liquid back to a more typical pipe flow.

Fluid motion is predicted in the same manner as solid motion by applying conservation principles, Newton's laws and ultimately Bernoulli's equation. The traditional analysis of this system would require many simplifying assumptions whereas the application of CFD modeling to this system minimizes the assumptions but increases the complexity and accuracy of the calculations performed. CFD modeling depends on millions of calculations performed on many discrete volumes of fluid as they pass through the control volume boundaries. With the aid of 3-D solid modeling, fluid boundaries are much better defined and CFD results have been improved. If fluid flow through a pipe fitting is not ideal, an energy approach (Bernoulli's equation) is only possible if some estimate of the energy loss can be made. In general, theoretical treatment is very difficult so empirical results are commonplace. However, the eductor-based mixing table can be treated theoretically with the abrupt enlargement and orifice solutions outlined below which offer a means of correlating the geometry to simplified equations.

In a study by Westfall Manufacturing, Gieske (1999) stated:

> *The association of high turbulence intensity with regions of high shear can be seen through inspection of the mean and fluctuation velocity contours. Peaks in the turbulence intensity occur where rapid changes in the mean velocities are found. Reynolds stresses are the correlation between fluctuating velocity components. When the vertical and streamwise velocity components are simultaneously high, a positive combination to the Reynolds stress occurs. High Reynolds stresses are associated with high transport of momentum, temperature, and passive scalars. Because Reynolds stress is well correlated with velocity fluctuation amplitude, transport across the shear layer will also be high in these regions. Consequently, contours of fluctuating velocity can be interpreted as contours of mixing effectiveness, with the greatest mixing occurring where the turbulence intensity is the highest.*

Therefore, the study of various nozzles, eductors, and downstream piping designs can improve the rate of mixing by increasing the contact area between high-speed fluid and low-speed solids. Mixing effectiveness and efficiency are increased due to an increase in turbulence.

Nozzle Analysis

An "abrupt enlargement" is a pipe section that increases in diameter over a negligible length. The flow is unable to follow the abrupt area change and energy-dissipating eddies are set up in the corner of the enlargement.

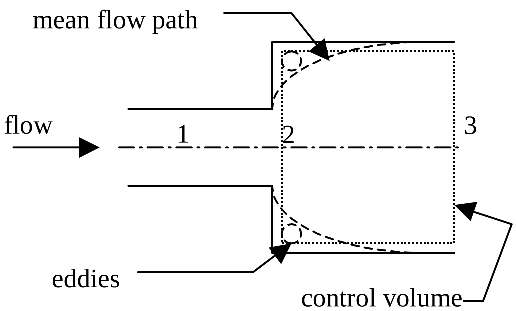

Figure 1 – Flow through an abrupt enlargement.

To determine the fluid flow quantities, a control volume is drawn as shown in Figure 1. The flow can be considered in two regions, 1 to 2 and 2 to 3. In moving from 1 to 2, the fluid has had no opportunity to enter the eddying region so that the flow can be considered ideal. The fluid velocity at 2 is dependent on the flow area and not the physical area. The flow follows the mean path shown so that the flow area at 2 is approximately equal to the pipe area at 1. Thus the velocity at 2 is approximately the same as that at 1. Since energy losses are negligible in this region, the pressure at 2 must be the same as that at 1; however, the pressure at 2 will act over the larger pipe area. The momentum equation can therefore be applied to the control volume in the direction of flow.

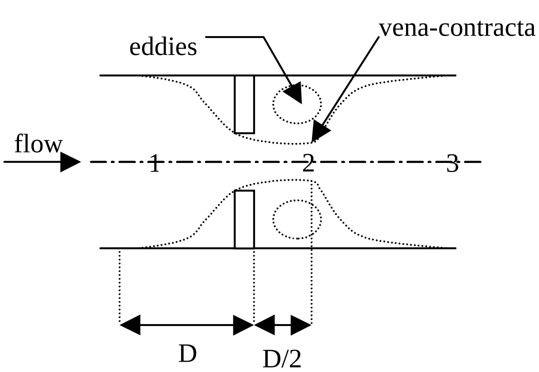

Figure 2 – Flow through an orifice.

Flow through an orifice plate can be considered as shown in Figure 2. Notice that the fluid does not attempt to decelerate until some distance after the orifice plate. The point of minimum flow area is known as the vena-contracta and is located approximately half the pipe diameter (D/2) downstream of the plate.

Mixing Table Results

The Fluent CFD program was used to simulate fluid flow and solids dispersion in the eductor based on the finite volume technique. Fluent is a CFD program for modeling fluid flow, heat transfer, chemical reaction, and the trajectories of dispersed particles/droplets. For flow in the continuous phase, it solves the discretized Reynolds and continuity equations, together with transport equations for the turbulent energy and dissipation rates and the energy equation.

To create the geometry and ensure a steady state analysis, all flow boundaries were created with additional straight sections upstream and downstream. A fine 3-D mesh was then generated within the boundaries in the flow path. A combination of mesh types were used, more regular sections used quadrilateral mesh (like straight sections of pipe) and the more complex areas of interest used tetrahedral mesh elements. The flow solution was solved after specifying fluid properties and the inlet velocity setting the flow rate. The flow solutions provide the fluid velocities, as well as other parameters such as turbulent kinetic energy (k) and dissipation rate (ε). The turbulence and velocity information was then used to confirm the mixing potential for the eductor geometry.

Concentration on maximizing the turbulence created in the mixing chambers of the HI-RIDE eductor resulted in an initial misstep. The excessive turbulence created in the primary mixing chamber caused hydroscopic powders to prematurely encounter moisture, and subsequently, these wetted solids would plug the mixer body. This contradicted the CFD models, which indicated the increased turbulence improved flow through the eductor as shown in Figure 3 below.

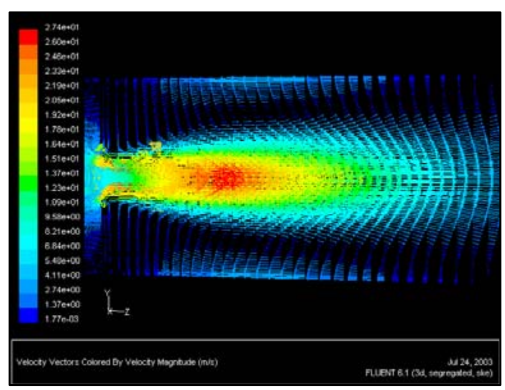

Figure 3 – Flow through an orifice.

Through the use of expanding geometries around the fluid and entrained solids, turbulence is created. This turbulence promotes the mixing of the solid and liquid into a homogeneous slurry. Trials of the revised eductor have verified that minimizing turbulence in the primary mixing chamber improves the ability of the system to entrain powdered solids in the liquid stream. The secondary and tertiary turbulent mixing zones then encapsulate the solids in the flow and thus the mixing efficiency and fluid rheology are improved. Furthermore, the quality of the vacuum generated to assist in the induction of solids corresponds directly with the pressure and velocity of the fluid flow. As the primary mixer pressure approaches a perfect vacuum, the inlet fluid approaches 500 gal/min.

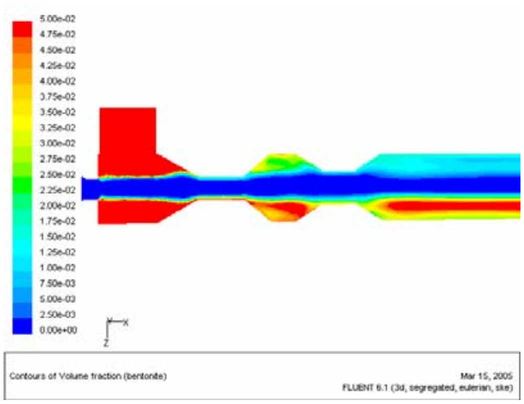

Figure 4 – Vertical cross-section of flow through a mixing table.

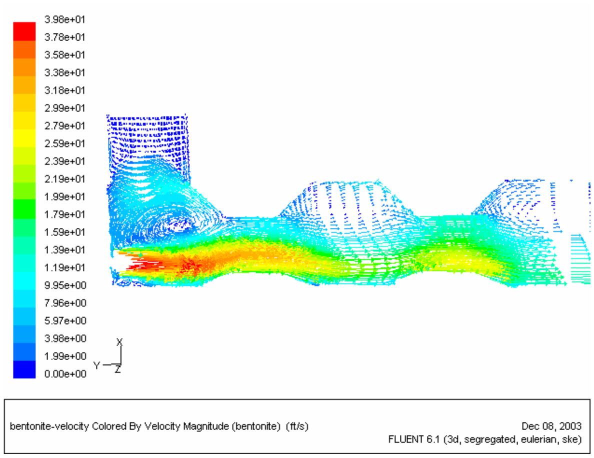

Figure 5 – Vertical cross-section showing velocity vectors.

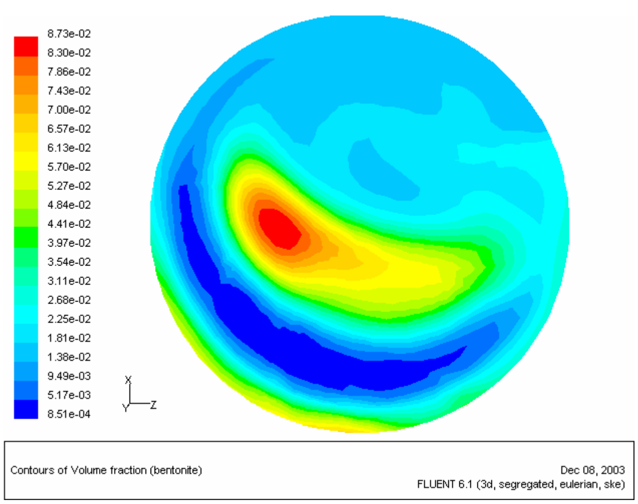

Figure 6 – Cross-section of diffuser outlet showing good mixing with small segments of unmixed material.

Developing a High-Flow, Multi-Density Fluid Blender

The analysis described above was also used to evaluate blending of two and three liquid flows inside a closed pipe conduit. The fluids supplied to the blending equipment primarily varied in density, viscosity, and volumetric flow rate but may have also varied in temperature and other parameters not studied in this system. For the purposes of the design study, variations in density, viscosity and flow rate were examined. The design was intended to blend a high-density fluid (16 lb/gal) with a low-density fluid (9 lb/gal) to create a drilling fluid of a client-specified density or even vary the required density within limits. The blending unit was designed for use with riserless drilling, where large volumes of weighted drilling fluid are required quickly. The current practice is to ship a supersaturated heavy weighted system from shore to the rig where it is blended with sea water in a single pass to the active system.

This type of equipment does not easily conform to the empirical calculations noted above, because the main concern with such a design is to ensure optimum mixing is achieved. Empirically, the blending process should conform to the calculation below; however, the quality of the mix is not seen in this calculation and it is possible to get significantly different test results depending on where the sample is drawn.

$$\rho_{Blended} = \frac{(\rho_{HDF} \times Q_{HFD}) + (\rho_{SW} \times Q_{SW}) + (\rho_{BR} \times Q_{BR})}{(Q_{HDF} + Q_{SW} + Q_{BR})} \quad \text{(Eq. 6)}$$

This calculation works very well with highly turbulent flows, but cannot account for fluids with high energy differentials such that the heavier fluid channels through the lighter fluid and little or no mixing occurs. To address this type of scenario (Figure 7), a CFD analysis was completed to examine the fluid flow lines and determine what mixer geometry will perform best at creating a more homogeneous fluid (Figure 9).

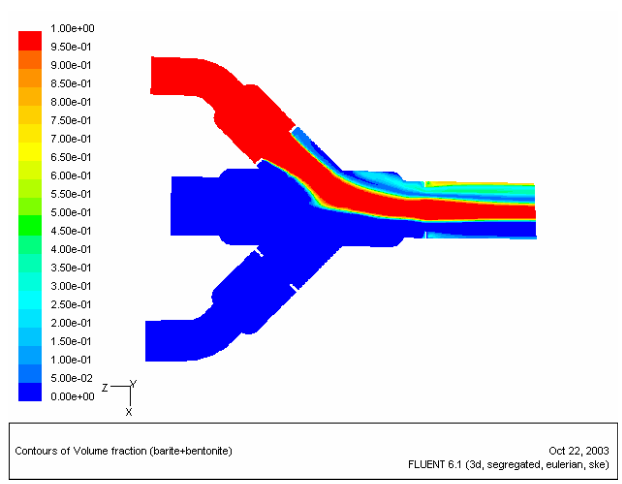

Figure 7 – Early model showing the heavy fluid (shown in red) channeling through seawater (blue).

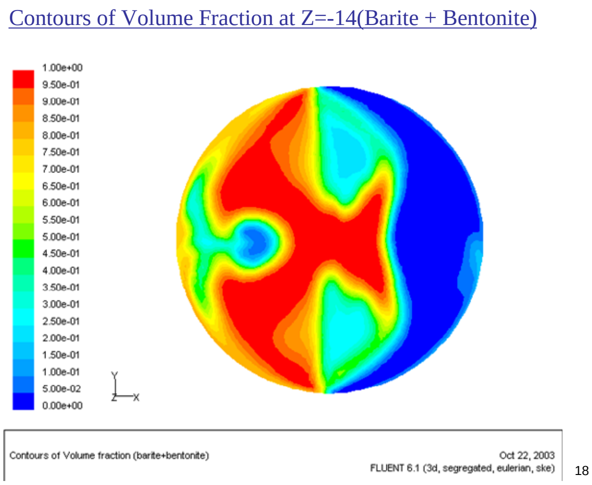

Figure 8 – Cross-section of early model outlet showing the heavy fluid (red) channeling through the seawater (blue).

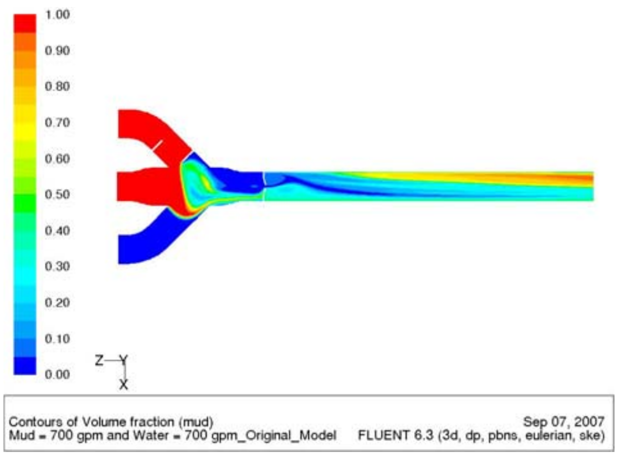

Figure 9 – Later model showing the heavy fluid (red) mixing with the seawater (blue).

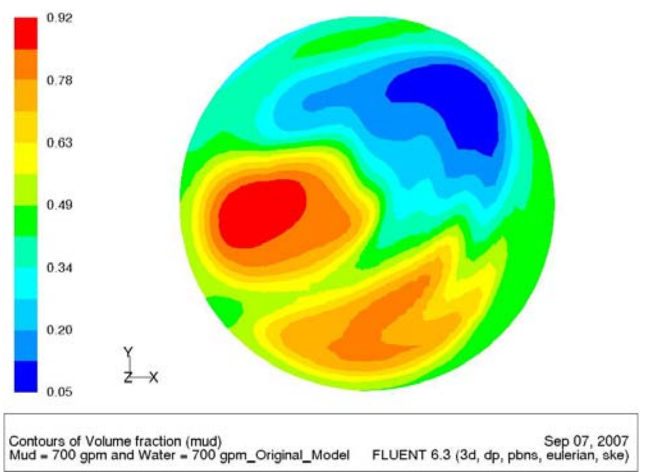

Figure 10 – Later model outlet showing the heavy fluid (red) mixing with the seawater (blue).

Characteristic Flow Curve

The CFD analysis was critical to determine the effect of particle distribution across the flow stream and subsequently the mixing effectiveness of the equipment. By adjusting the fluid streams using strategically placed baffles and flow diverters, the discharge fluid approaches an ideal mix displacement curve. The geometry changes effectively retard the flow of the heavy fluid, reducing its energy due to the increased path. Inherent fluid disturbances reduce the surface stability of the flow stream and further increase turbulence. Once the surface is disrupted and the turbulence is increased, then better mixing follows.

Creating flow-characteristic curves for this equipment proves to be a time and resource consuming process. Because standard empirical formulas do not adequately describe the relationship between pressure drop and volumetric flow through such a multi-directional path, the curves are generated from CFD modeling where each data point requires several days of processing time (Figure 11). To create a formula that can determine a point on any of the characteristic curves shown below, a minimum of four data points must be plotted and the equation of the curve approximated. For example, approximately 70-psi inlet pressure is required to blend a 16-lb/gal drilling fluid and seawater at 1450 gal/min (Figure 11).

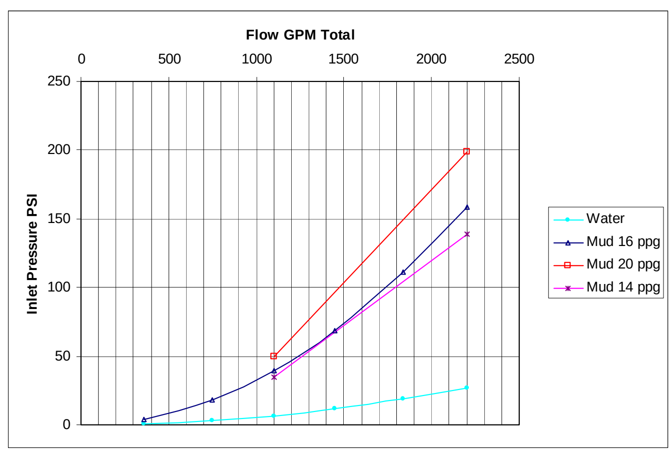

Figure 11 – Flow P/Q curve for a HI-SIDE blending unit.

Conclusions

The mixing of drilling fluids benefits from equipment that can take advantage of subtleties in design to increase dispersion of chemicals and solids throughout the fluid matrix.

Mixing equipment can be designed using simplifying assumptions like abrupt expansion and orifice flow coefficients and empirical equations like Bernoulli, Navier-Stokes, continuity. However, to improve the quality and efficiency of the mixing equipment, a computational fluid dynamics study is required to locate surface disturbances or to position flow path diverters that help create turbulence.

Mixing of chemicals in either powder or liquid form often requires turbulence to create surface disturbances and a transmigration of solids across fluid stream lines.

Mixing bulk powders with a liquid stream usually requires that the solid stay dry until it engages the surface of the liquid and is ultimately entrained. After the initial entrainment occurs, the fluid can be disturbed to generate a desired turbulent mixing environment.

CFD modeling can be used to identify turbulence, eddies, velocities, and particle dispersion to evaluate the mixing equipment geometries and identify or create changes in the flow stream.

CFD modeling has resulted in the design of a mix-on-the-fly blender capable of mixing two or three fluids of varying densities at high flowrates and an eductor mixing table that optimizes the mixing of drilling fluid product additions.

Acknowledgments

The authors thank the management of M-I SWACO for permission to publish this paper. We also thank the M-I SWACO Mixing Technologies team and the crews of the Port Fourchon, Berwick, and Inter-Coastal City, Louisiana, Liquid Mud Plants for their assistance in field testing the equipment developed. Special thanks to Westfall Manufacturing for their assistance in developing the HI-SIDE blender.

Nomenclature

A = *Area*
D = *Diameter*
p = *Pressure*
z = *Height*
g = *Gravitational constant*
u = *Velocity*
Q = *Volumetric flow* (L^3/T)
ρ = *Density* (M/L^3)
k = *Kinetic Energy*
ε = *Turbulent dissipation*
ν = *Viscosity*
lb/gal = *Pounds per U.S. gallon*
bbl = *Barrel(s)*
psi = *Pounds per square inch*
gal/min = *U.S. gallons per minute*
HDF = *High-density fluid*
SW = *Sea water*
BR =*Brine*

References

1. Nakayama, Y. and Boucher, R. F. *Introduction to Fluid Mechanics;* Oxford, UK: Butterworth-Heinemann (1998).
2. Fluent User's Manual. New Hampshire 2006.
3. Gieseke, T. J. *An Evaluation of the Hydrodynamics Mechanisms Which Drive the Performance of the Westfall Static Mixer* (NUWCDIVNPT Code 8233). Bristol, Rhode Island: Westfall Manufacturing (March 29, 1999).
4. Peri, S. and Rogers, B. "Computational Fluid Dynamics (CFD) Erosion Study for Chokes." SPE-110463, SPE Annual Technical Conference, Anaheim, California, November 11-14, 2007.