Design of Hydrocyclone Separation Equipment Using CFD Coupled with Optimization Tools

David Schowalter[1], Rafiqul Khan[1], Therese Polito[2], and Tim Olson[3].

(1) Fluent Inc., 10 Cavendish Court, Lebanon, NH 03766
(2) Engineous Software, 2000 CentreGreen Way, Suite 100, Cary, NC 27513
(3) Krebs Engineers, 5505 W Gillette Rd, Tucson, AZ 85743

As tar sand and low-yield field production becomes more prevalent, performance of equipment used to separate water, gas, and solids from crude oil becomes critical to the profitability of production operations. As computational hardware and multiphase modeling have improved, the commitment of industrial engineers to virtual prototyping has made computational analysis more prevalent in the design and optimization of separation equipment.

One such piece of equipment is the hydrocyclone. In this study, the coupling of CFD to optimization algorithms for a high level of design automation has been demonstrated. Three design parameters are allowed to vary, these being the inlet diameter, the vortex finder diameter, and the vortex finder length. The specified goal is to achieve a minimum D_{50}, the particle size for which half of the particles entering at the inlet are captured. A Design of Experiments tool chooses appropriate conditions to test with CFD. A response surface is then generated and used by the optimization algorithm to determine the optimum design parameters.

Hydrocyclones and CFD

Cyclone separators are devices that utilize centrifugal forces, to separate materials of different density, size and shape. The hydrocyclone is a subcategory of cyclone separator, used for separation of particles or immiscible droplets from liquids. In a typical application, particulate laden flow is introduced tangentially into a stationary vessel consisting of an upper cylindrical section narrowing to form a conical base. This induces a spiral rotation on the fluid, enhancing radial acceleration on the suspended secondary phase. There are two outlets: the underflow, situated at the apex of the cone, and the overflow, an axial tube rising to the vessel top, also called the vortex finder. When the density of the particulate phase is greater than that of the fluid phase, heavier particles migrate quickly towards to the cone wall where the flow is directed downwards. Low density particles migrate more slowly and therefore may be captured in the upward spiral flow and exit from vortex finder via the low pressure center.

Hycrocyclones are used commonly because they are simple and inexpensive to manufacture, require little maintenance, contain no moving parts, and have the ability to operate at high temperatures and pressures. They have been used for many centuries in the process industry, with the greatest proliferation taking place in the last 100 years.

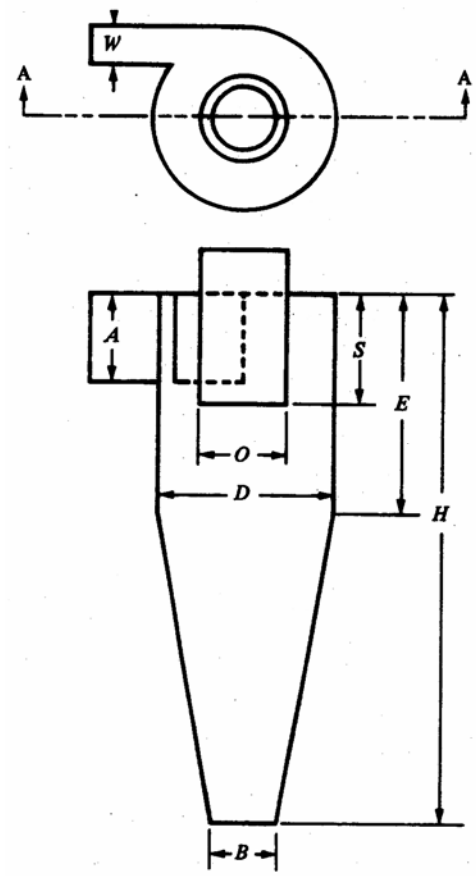

Figure 1. Typical geometry of a hydrocyclone.

Factors affecting separation efficiency include fluid velocity, density, and viscosity as well as the mass, size, and density of the particles. The geometric configuration of the hydrocyclone also plays a critical role, and usually includes the following parameters, which are identified in Figure 1:

- Cyclone diameter (D)
- Inlet width (W) and height (A)
- Overflow diameter (O)
- position of the vortex finder (s)
- Height of the cylindrical chamber (E)
- Total height of the cyclone (H)
- Underflow diameter (B)

The performance of a hydrocyclone is generally characterized by the size of particles for which separation is successful. Specifically, D_{50} is the particle size for which one-half of the injected particles escape through the overflow. A smaller value of D_{50} characterizes an more effective hydrocyclone in general, though pressure drop is also a concern and different hydrocyclones are designed for different applications.

Computational Fluid Dynamics (CFD) has been used extensively over the last ten to fifteen years to design hydrocyclones. CFD analysis can provide valuable guidance in finding the overflow and underflow recovery ratio at given boundary conditions, as well as providing a complete picture of complex hydrodynamics such as radial profiles of mean swirl, axial velocity components, and distribution of turbulent kinetic energy. The method has also been well validated against experimental results, as shown in Figure 2.

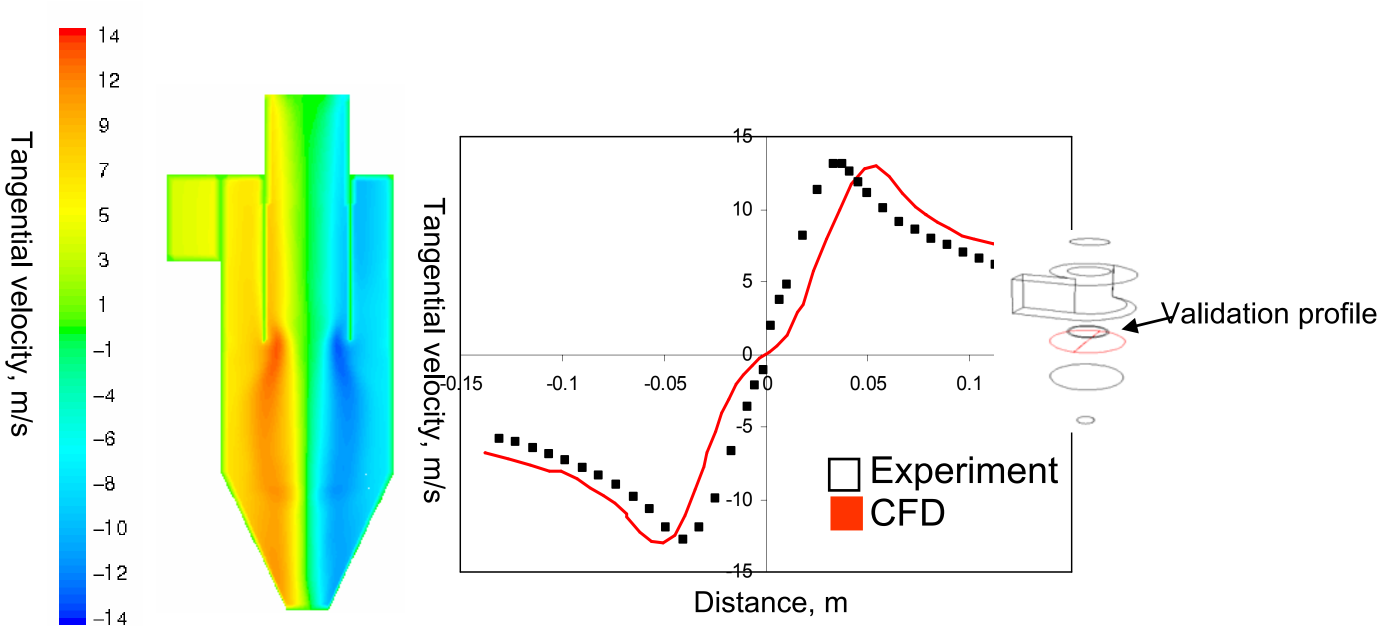

Figure 2. Comparisons made at an axial location 0.33 m downstream of the cyclone top wall.

Design Optimization Integration with CFD

Off-the-shelf software is now available which automates the iterative design process and integrates the numerous relevant analysis codes so that they can be executed in a single run. While the proliferation of engineering analysis tools over the last twenty years has revolutionized design, many engineers continue to perform repeated single point analyses in order to optimize a design. These efforts can now be automated, replacing manual file manipulation. The software captures and automates design strategies and expert knowledge, allowing extensive design exploration that is both probabilistic, and deterministic.

In the demonstration case covered here, the iSIGHT design optimization tool is integrated with the FLUENT CFD code. In order for iSIGHT to query CFD results of varying designs, it uses the capability of changing and remeshing the geometry via parameterized journal files, which direct the CFD software to:

- create the geometry
- generate the mesh
- set the boundary conditions and run the CFD calculation
- write out the desired data from the calculation

In this case, the optimizer is used to detect the combination of inlet diameter (A), vortex finder diameter (O), and vortex finder length (S) that give the minimum value of D_{50} for a fixed mass flow. The constraints placed on these variables are as follows:

- Inlet diameter: 23.4mm < A < 30.3 mm

- Vortex finder diameter: 16.2 mm < O < 34.4 mm
- Vortex finder length: 1 mm < S < 72 mm

Values typical of crude oil were used for the fluid properties, and those of sand were used for particle density.

iSIGHT first performs a Design of Experiments using the Optimal Latin Hypercube method. This resulted in thirty independent designs being evaluated and analyzed with CFD. Each design requires approximately 3000 iterations and 4 CPU hours, so the DOE runs on 2 laptops over a weekend. These discrete results are then used to create a radial basis function (RBF) approximation model, or response surface, which allows the computation of optimal values on a continuous surface. The radial basis function employs a variable power spline: $|| x - x_j ||^{c_j}$, where c_j is a shape function between 0.2 and 3.0. Cuts of this response surface of particle escape percentage as a function of the varied geometric parameters are shown in Figure 3. An optimization on the response surface takes about 2 minutes of CPU time.

Table 1. Results of Design Optimization (values in mm)

	Initial Design	Final Design
Inlet Diameter	25.0	25.2
Vortex Finder Diameter	25.0	31.7
Vortex Finder Length	50.0	68.1

Table 1 shows the results for the optimal values of the varied parameters for the hydrocyclone design. The initial values indicate the parameters used in the first CFD model that was run in the design of experiments. Figure 4 shows a visual representation of the geometries outlined in Table 1. Note that the vortex finder extends nearly the full length of the constant diameter section of the hydrocyclone.

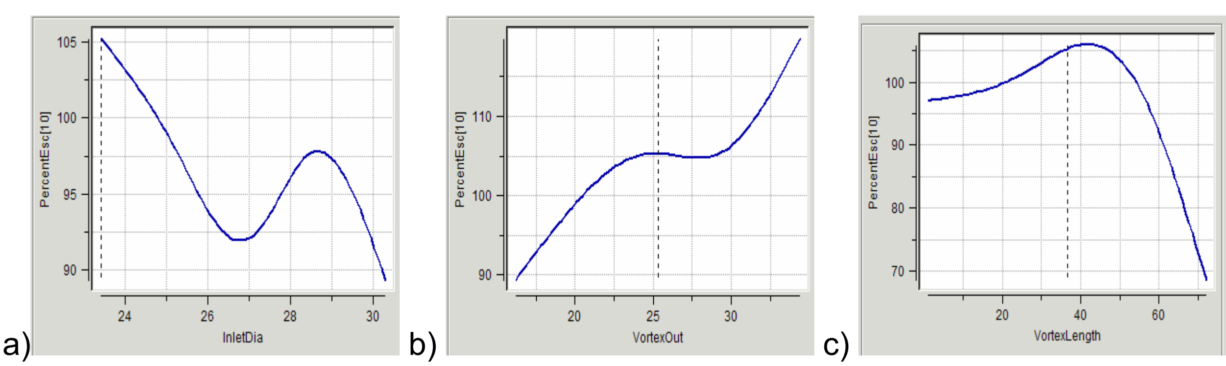

Figure 3. Cuts of the RBF response function showing percent of particle escape versus a) inlet diameter, b) vortex finder diameter, and c) vortex finder length.

Conclusions

A demonstration of the coupling of design optimization software and CFD software for hydrocyclone design has been shown. Optimal values for inlet diameter, vortex finder diameter, and vortex finder length for attaining a minimum value of D_{50} have been determined. All optimal values were within the constraint limits. The process entailed the automated deployment of 30 CFD simulations run over a two day time period.

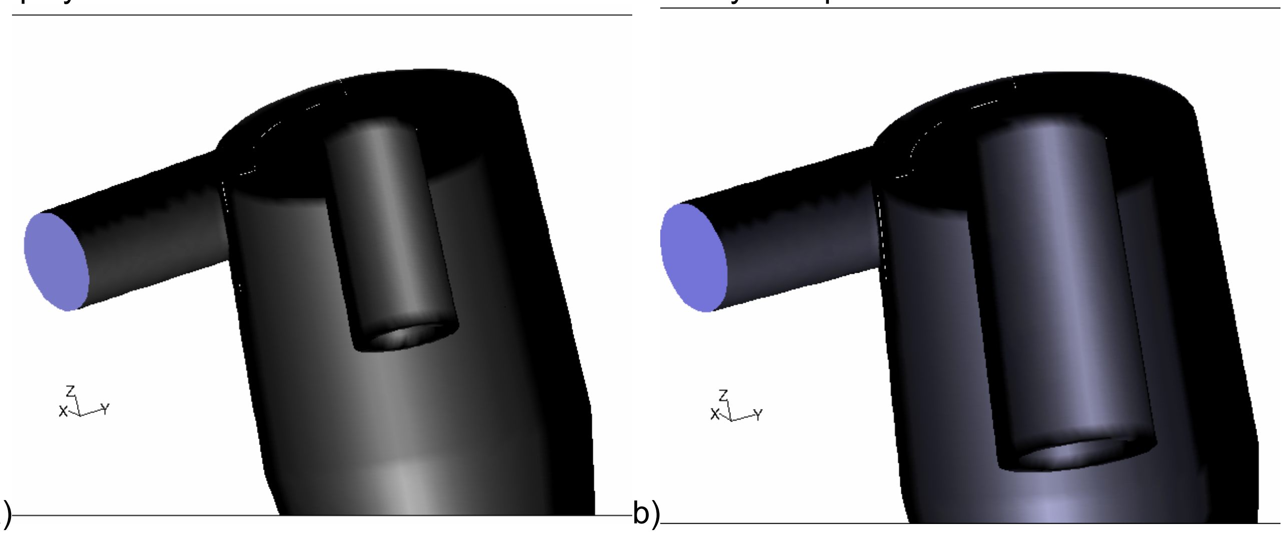

Figure 4. Geometrical configuration of (a) the initial design and (b) the optimized design, as caculated by the coupled CFD and optimization tools.