A Computational Fluid Dynamics Study of Fluid Catalytic Cracking Cyclones

I. Abu-Mahfouz[1], J. W. McTernan[2]

[1]Pennsylvania State University - Harrisburg, Middletown, PA, USA
[2]Buell Division of Fisher-Klosterman Inc., Lebanon, PA, USA

Abstract

Cyclone separators are used to remove particulate from a fluid stream. This is accomplished by centrifugal forces that force the particles to dislodge from the fluid flow. The fluid exits through the vortex finder at the top of the cyclone while, due to gravitational forces, the particles fall to the bottom of the cyclone for collection. Pressure loss and collection efficiency are the two most important aspects of a cyclone, because both directly affect the process costs.

Fluidized Catalytic Cracking (FCC) regenerators utilize a fluidized bed to facilitate catalyst regeneration. Cyclones are used to separate the catalyst from the gas stream and return the catalyst to the fluidized bed; as the gas progresses through the system for further processing. Determining the amount and distribution of catalyst that reaches the cyclone inlets is important in designing cyclones for optimal performance.

Methods of cyclone design were previously limited to narrow scope algebraic equations and expensive and time consuming scale modeling. This paper will examine the aspects of cyclone design and the feasibility of Computational Fluid Dynamics (CFD) as a design tool. CFD analysis has the potential to reduce the cost of cyclone development and to provide a cost effective method for design improvements.

Results and Discussion

The three cyclones described in the Model Description section were simulated with COMSOL Multiphysics® software. The results of these analyses were compared with empirical data in order to determine the usefulness and validity of the simulations. Each cyclone geometry was subjected to three inlet velocities: 40 ft/sec, 60 ft/sec, and 80 ft/sec.

The outlet is defined as zero pressure with the assumption that the outlet will not have a pressure to impede the outlet flow. The bottom of the cyclone dipleg is considered sealed for modeling purposes. In operation, the dipleg will be filled with particulate and will have a flapper valve. In some cases the dipleg will be submerged in a higher pressure fluidized bed. This will cause particulate to fill the dipleg until the mass of the particulate overcomes the pressure differential and allows the particulate to flow. In this case the dipleg is also considered closed to gas flow.

The fluidized bed is simulated using the Euler-Euler model from COMSOL Multiphysics. The fluidized bed is contained in a vessel that is 31.5 feet in diameter and 39.5 feet high. The bottom 15 feet of the vessel is considered the dense phase. This area is initially 50% particulate and 50% air before the air grid is initiated. After initiation of the air grid, air will lift the dense phase and begin to fluidize the particulate. Eventually particulate will be carried to the top of the vessel, at an elevation where the cyclone inlets are assumed to be. Any particulate that exits through the top of the vessel is assumed to enter the cyclone. To simulate the particulates travel through the cyclone and return to the bed through the cyclone's dipleg, a function in COMSOL is implemented that returns the particulate by two side inlets at the bottom of the vessel.

Reference

1. Frequently Asked Question - When was the last refinery built in the United States. In: U.S. Energy Information Administration. Available at: http://www.eia.gov/tools/faqs/faq.cfm?id=29&t=6.

2. Gimbun, J., Chuah, T. G., Fakhru'l-Razi, A., Choon, T.: The Influence of Temperature and Inlet Velocity on Cyclone Pressure Drop: a CFD Study. Chemical Engineering and Processing 44, 7-12 (2005).

3. McLean, J.: FCC Catalyst Properties Can Affect Cyclone Erosion. Oil & Gas Journal (2000).

4. Cortes, C., Gil, A.: Modeling the Gas and Particle Flow Inside Cyclone Separators. Progress in Energy and Combustion Science 33, 409-452 (2007).

5. Azadi, M., Azadi, M., Mohebbi, A.: A CFD Study of the Effect of Cyclone Size on its Performance Parameters. Journal of Hazardous Materials 182, 835-841 (2010).

6. Wang, B., Xu, D. L., Xiao, G. X., Chu, K. W., Yu, A. B.: Numerical Study of Gas-Solid Flow in a Cyclone Separator. Third International Conference on CFD in the Minerals and Process Industries CSIRO, 371-376 (2003).

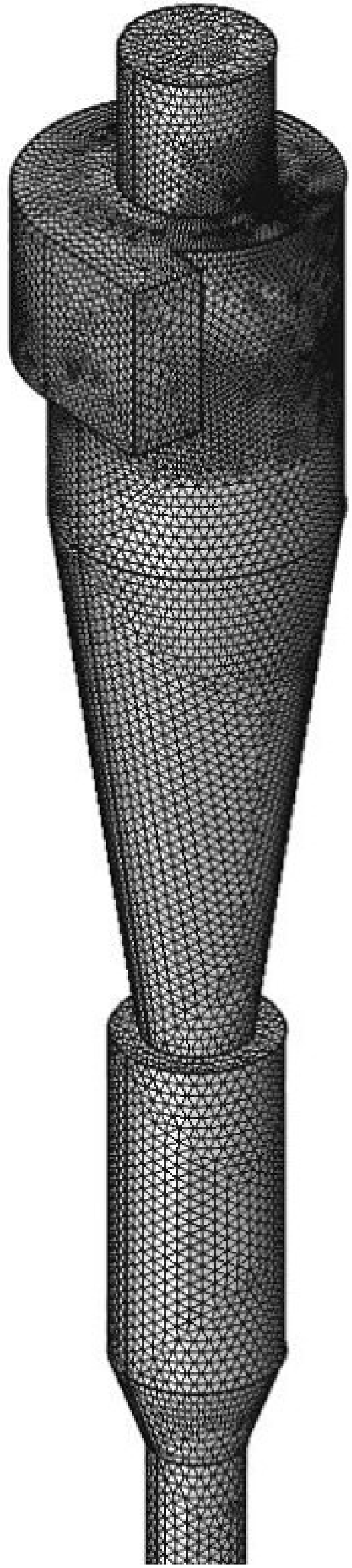
7. Karthik, T. S. D.: Turbulence Models and Their Applications. In: http://www.leb.eei.uni-erlangen.de/winterakademie/2011/report/course01.htm. (Accessed 2011).

8. Zhao, B., Su, Y.: Particle Collection Theory for Cyclone Separators: Summary and Comparison. PPSC, 484-488 (2006).

9. Chu, K. W., Wang, B., Yu, A. B., Vince, A.: CFD-DEM Modelling of Multiphase Flow in Dense Medium Cyclones. Powder Technology 193, 235-247 (2009).

10. Yang, N., Wang, W., Ge, W., Li, J.: CFD Simulation of Concurrent-Up Gas-Solid Flow in Circulating Fluidized Beds with Structure-Dependent Drag Coefficient. Chemical Engineering Journal 96, 71-80 (2003).

Figures used in the abstract



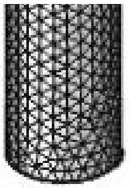


Figure 1: Cyclone mesh.

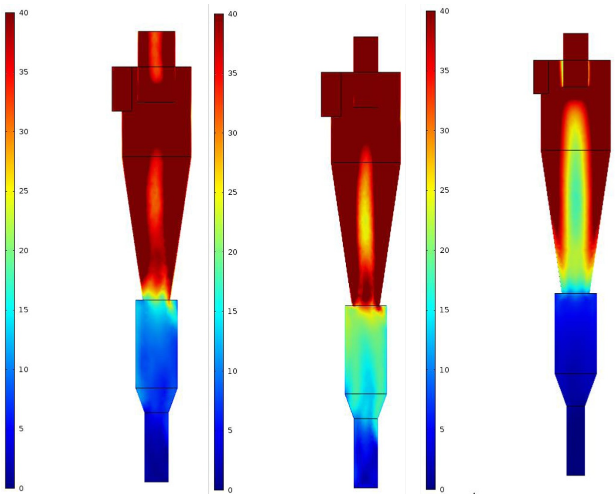
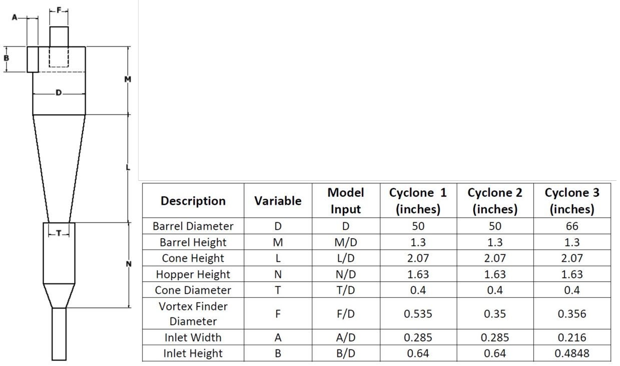


Figure 2: Velocity Magnitude at 40fps Inlet.



Description	Variable	Model Input	Cyclone 1 (inches)	Cyclone 2 (inches)	Cyclone 3 (inches)
Barrel Diameter	D	D	50	50	66
Barrel Height	M	M/D	1.3	1.3	1.3
Cone Height	L	L/D	2.07	2.07	2.07
Hopper Height	N	N/D	1.63	1.63	1.63
Cone Diameter	T	T/D	0.4	0.4	0.4
Vortex Finder Diameter	F	F/D	0.535	0.35	0.356
Inlet Width	A	A/D	0.285	0.285	0.216
Inlet Height	B	B/D	0.64	0.64	0.4848

Figure 3: Cyclone dimensions.