A Computational Fluid Dynamics Study of Fluid Catalytic Cracking Cyclones

J. W. McTernan[1], I. Abu-Mahfouz[*2]
[1]Buell Fisher-Klosterman, Lebanon, PA, USA
[2]Pennsylvania State University Harrisburg, PA, USA
*Corresponding author: 777 West Harrisburg Pike, Middletown, PA 17057, Email; iaa2@psu.edu

Abstract: 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. 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. Determining the amount and distribution of catalyst that reaches the cyclone inlets is important in designing cyclones for optimal performance. This paper will examine the aspects of cyclone design using Computational Fluid Dynamics (CFD). CFD analysis has the potential to reduce the cost of cyclone development and to provide a cost effective method for design improvements.

Keywords: CFD, Cyclone, Fluid Catalytic Cracking (FCC).

1. Introduction

Fluid Catalytic Cracking (FCC) is a method used to convert heavy hydrocarbon bonds from crude stock into lighter product with commercial value. Petroleum feedstock is preheated and injected into the reactor riser. The feedstock is vaporized and mixed with a specialized and heated catalyst which initiates the cracking reactions. The cracking reaction produces a carbonaceous material called coke on the catalyst which reduces the reactiveness of the catalyst. In order to separate this catalyst from the product vapors, the gas and catalyst is passed through a series of cyclones. The cyclones create a vortex in which the heavier catalyst particles are tossed out of the gas stream impacting the cyclone wall and then fall to the bottom where they are collected. The percentage of particles collected is considered the efficiency of the cyclone. The gas stream exits the top of the cyclone, often entering a second cyclone for further separation, then continuing through the refinery

Cyclone geometry is critical to its functionality and must be designed properly for maximum collection efficiency and minimal power consumption. Cyclone efficiency and pressure loss are the most important parameters for cyclone design. Cyclones have the additional complication of erosion due to the highly abrasive particles in the gas stream. The complexity of the highly anisotropic turbulent flow causes difficulty in predicting cyclone performance parameters.

Pressure loss of a cyclone is often inversely related to cyclone efficiency. Increasing the inlet velocity of a cyclone will often increase the efficiency, but at the cost of increasing the pressure loss. It has been determined that approximately 80% of the pressure loss is due to the viscous stresses. The remaining loss of pressure occurs at the inlet and outlet of the cyclone and by frictional forces along the cyclone wall [1]. The erosion rate is directly impacted by the gas stream velocity and the particle properties; velocity being the primary variable [2]. Understanding the vortex structure and flow velocities is important when developing cyclone designs that minimize erosion. The focus of this paper will be to determine the ability of commercial CFD programs to accurately model these important aspects of cyclone design. See Fig. 1 for a typical cyclone layout.

Gimbun et al [1] looked at four algebraic models and compared the predicted results of pressure loss to experimental data. Cortes and Gil [3] undertook an extensive study of algebraic prediction methods. The authors suggested that, due to the complexity and instability of flow in cyclones, CFD analysis is the most promising tool for cyclone design and predicting cyclone performance.

Many studies in recent years have examined the ability of CFD tools to examine cyclone design and predict performance. The three most common commercially available turbulent models are the k-ε model, the renormalization group (RNG) k-ε, and the Reynolds Stress Model (RSM). The standard k-ε model equations are developed for the turbulent kinetic energy (k) and the dissipation rate (ε). The RNG k-ε model adds an additional variable in the dissipation equation to take some account for swirl on turbulence. Both the standard and RNG k-ε equations assume isotropic turbulence which is not completely valid for flows in cyclones. The Reynolds stress model incorporates transport equations for each Reynolds stress dissipation component [1]. This provides the ability to account for the anisotropic turbulence inherent in cyclonic flow and to improve model accuracy. Large Eddy Simulation (LES) is a transient approach based on the Navier-Stokes equations. Due to the instability of precessing vortex core of a cyclone, LES is becoming the standard in cyclone modeling [3].

Azadi et al [4] examined the pressure loss and collection efficiency of three different sized cyclones using the Reynolds stress model and the RNG k-ε model. Wang [5] attempted to model a cyclone with low particle flow density and compare the results with empirical data. Flow fields are generated using RSM, and particulate trajectories are generated using a Stochastic Lagrangian model. Some assumptions are also made regarding particle interaction, such as particle collision, that the analysis method is unable to compute [6].

1.1 Model Description

The cyclone models were created using Autodesk Inventor, and were drawn as a solid form according to the dimensions described in Fig. 1. After importing the model into COMSOL, the solid parts were given the properties of air and the boundary shells, with the exception of the inlet and outlet surfaces, were given the properties of stainless steel or wall functions.

The major cyclone dimensions are all proportional to the inside barrel diameter. Cyclone 1 and Cyclone 2 are identical with the exception of the vortex finder diameter. Making this change will provide the ability to examine the effects of this change on pressure drop, vortex structure, velocity profiles, and collection efficiency. Cyclone 3 is considered a high efficiency cyclone. The cyclone is designed to produce an identical pressure drop as Cyclone 1, but with an increase in collection efficiency.

A cyclone series is defined by the barrel area divided by the inlet area. Cyclone 1 and Cyclone 2 are of the same series because each cyclone has an identical inlet and barrel diameter. Cyclone 3 is of different series; designed to have greater collection efficiency without the additional cost of increased pressure loss.

1.2 Simulation Conditions

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 cyclone parameters are shown in **Error! Reference source not found.**.

2. Results and Discussion

The vortex structure is clearly shown in the axial velocity plots (Fig. 2). Comparing the velocity profile of the three inlet velocities demonstrates that the vortex structure stays fairly consistent at various velocities. The velocity profiles are scaled to the inlet geometry, so the intensity of the vortex increase with inlet velocity, but the scaled structure is fairly consistent. As shown in Fig. 2, the velocities in the axial direction are much greater in the inner vortex than in the outer vortex, and that the axial velocities increase rapidly as the center of the cyclone is approached. This would indicate that the closer a particle travels to the center of the cyclone, the higher the probability is that that particle will be lost. It is also shown that, due to the no-slip wall boundary condition, the velocity magnitudes decrease dramatically near the cyclone wall. Another detail worth noting is that the inner vortex is shown to exhibit a sine function fluctuation in the radial direction which appears to be amplified at the vortex tail (Fig. 1). This leads to some evidence that the vortex is slightly unstable and may precess around the cyclones axis. This phenomenon was discussed in detail by Cortez et al [4].

The cyclone pressure loss is calculated as the pressure differential between the inlet and outlet. See Fig. 3 for static pressure distribution. COMSOL produces pressure loss curves that very nearly match empirical data with approximate percent error is 5% (Fig. 4). The percent error of COMSOL exceeds estimates for k-ε models from previous studies [2]. The smaller vortex finder of Cyclone 2 increases pressure loss throughout the cyclone. The restrictive outlet tube increases the outlet velocity and increases the required inlet pressure. The pressure loss due to the decreased outlet tube diameter is in addition to pressure loss associated with the wall drag.

3. Conclusion

The simple geometry of cyclones can falsely lead to assumptions that cyclone design is a simple task. Due to the highly complex turbulent flow, cyclones are actually very complex, and further research is needed to better understand the physics of the fluid flow and particle collection. Several difficulties prohibit CFD analysis from achieving the goals of accurately predicting cyclone performance parameters. Cyclones are highly efficient and potential increases in efficiency will be relatively small; making the task of prediction very difficult. The inability to fully model a cyclone’s anisotropic turbulent flow can lead to inaccuracies of a 5% magnitude - washing out the minute efficiency changes that can be of a 0.1% magnitude. Although the changes in efficiency may only be a fraction of a percent, an increase in efficiency can reduce yearly losses of expensive material by the ton. These obstacles in cyclone modeling are significant, but are also being reduced as CFD solvers are becoming more robust and computation power is increasing. By examining the pressure differentials, vortex structure, and fluid velocities, informed decisions can be made about design and performance.

References:

1. 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)

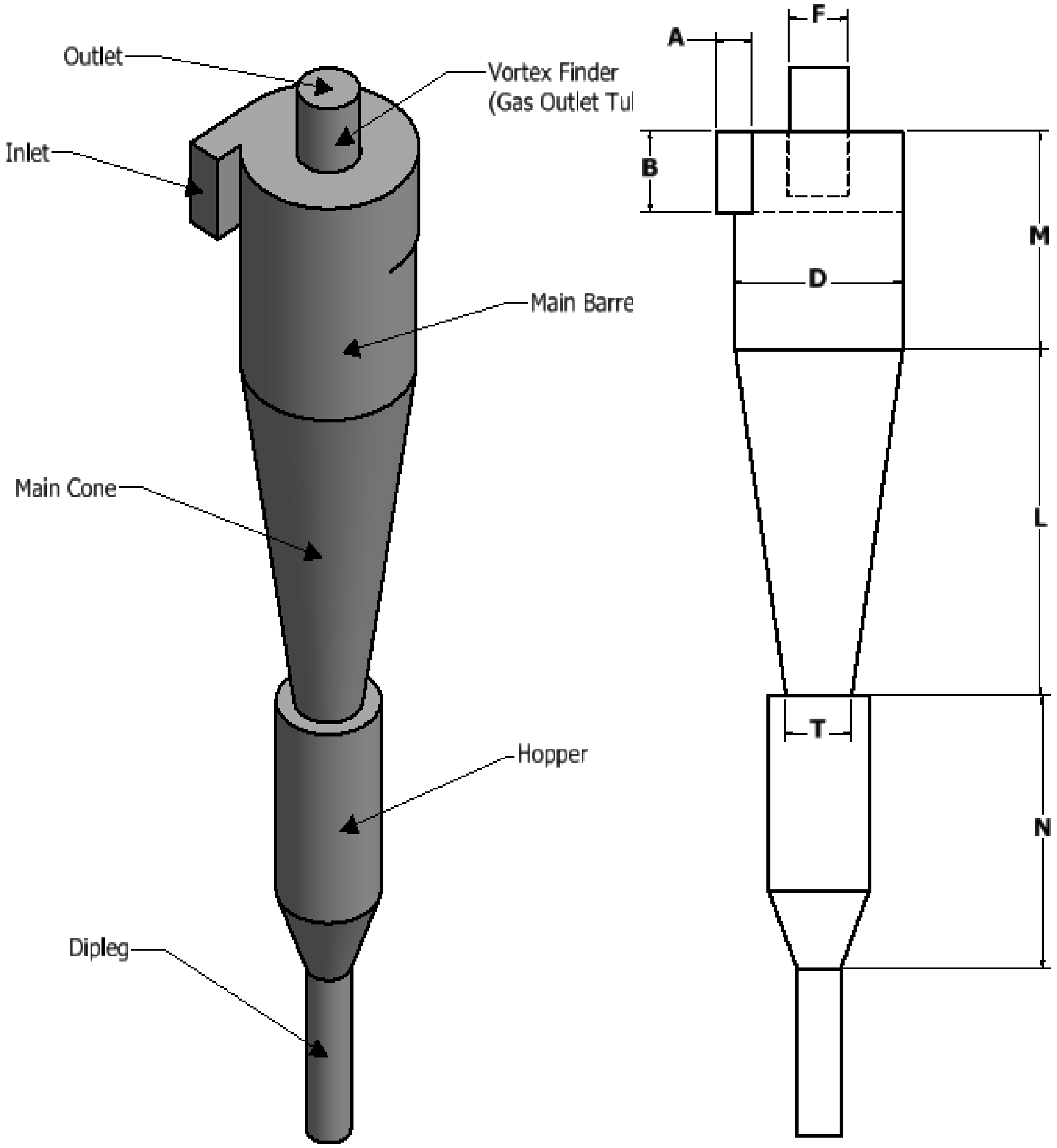
2. McLean, J.: FCC Catalyst Properties Can Affect Cyclone Erosion. Oil & Gas Journal (2000)

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

4. 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)

5. 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)

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



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 1. Cyclone Dimensions

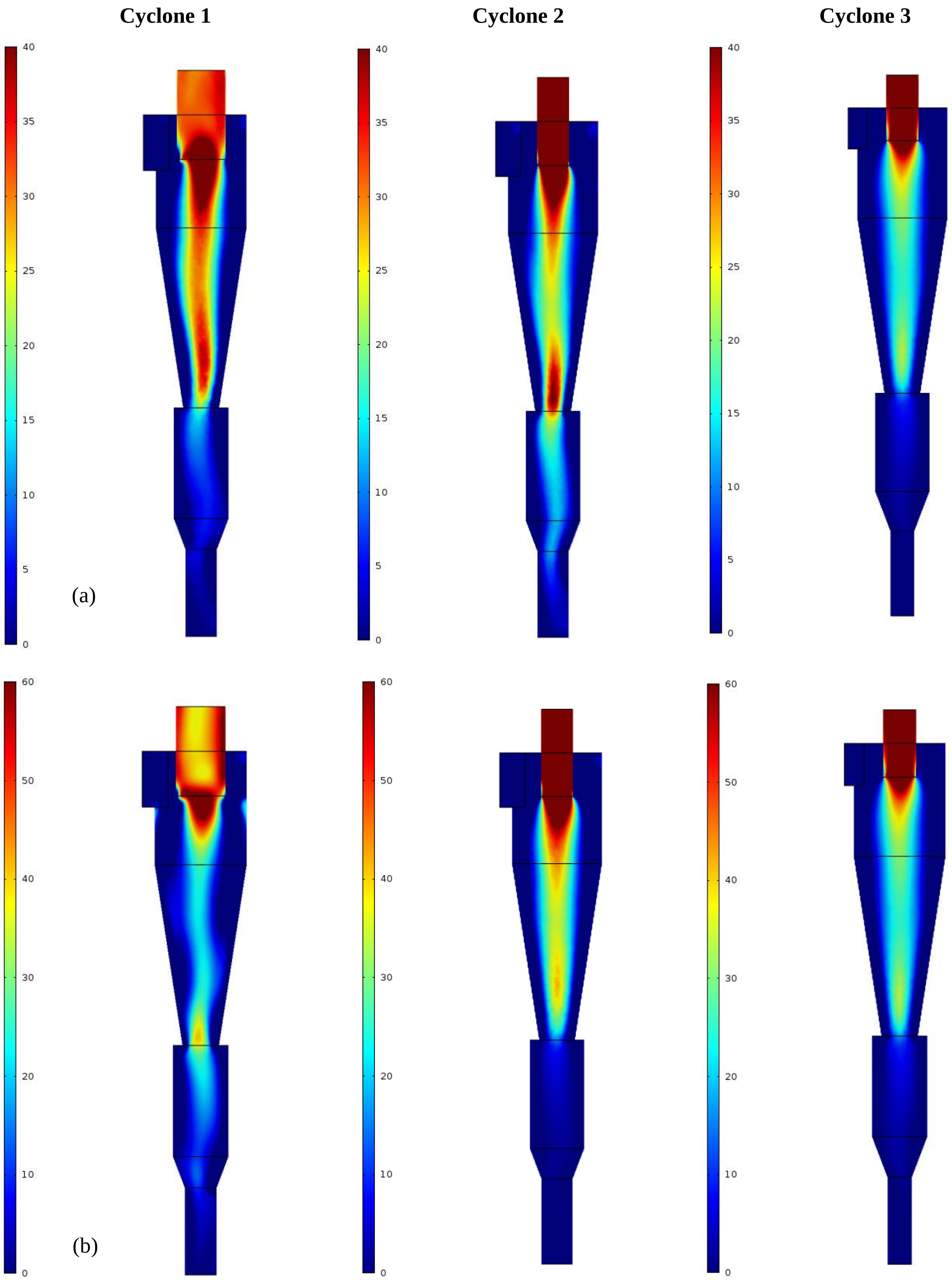


Figure 1. Axial velocity at (a) 40 fps inlet and (b) 60 fps inlet.

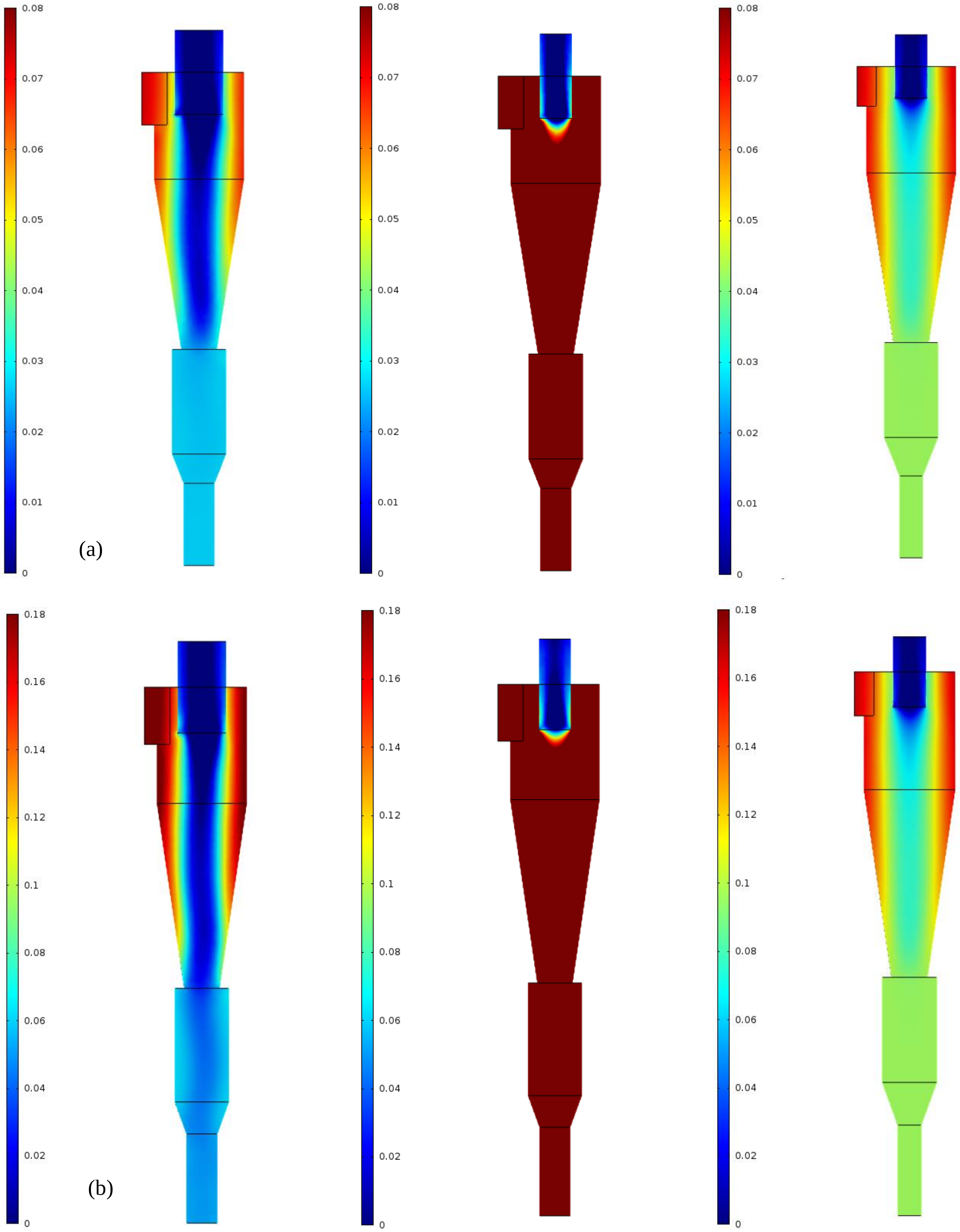


Figure 1. Static pressure at (a) 40 fps and (b) 60 fps.

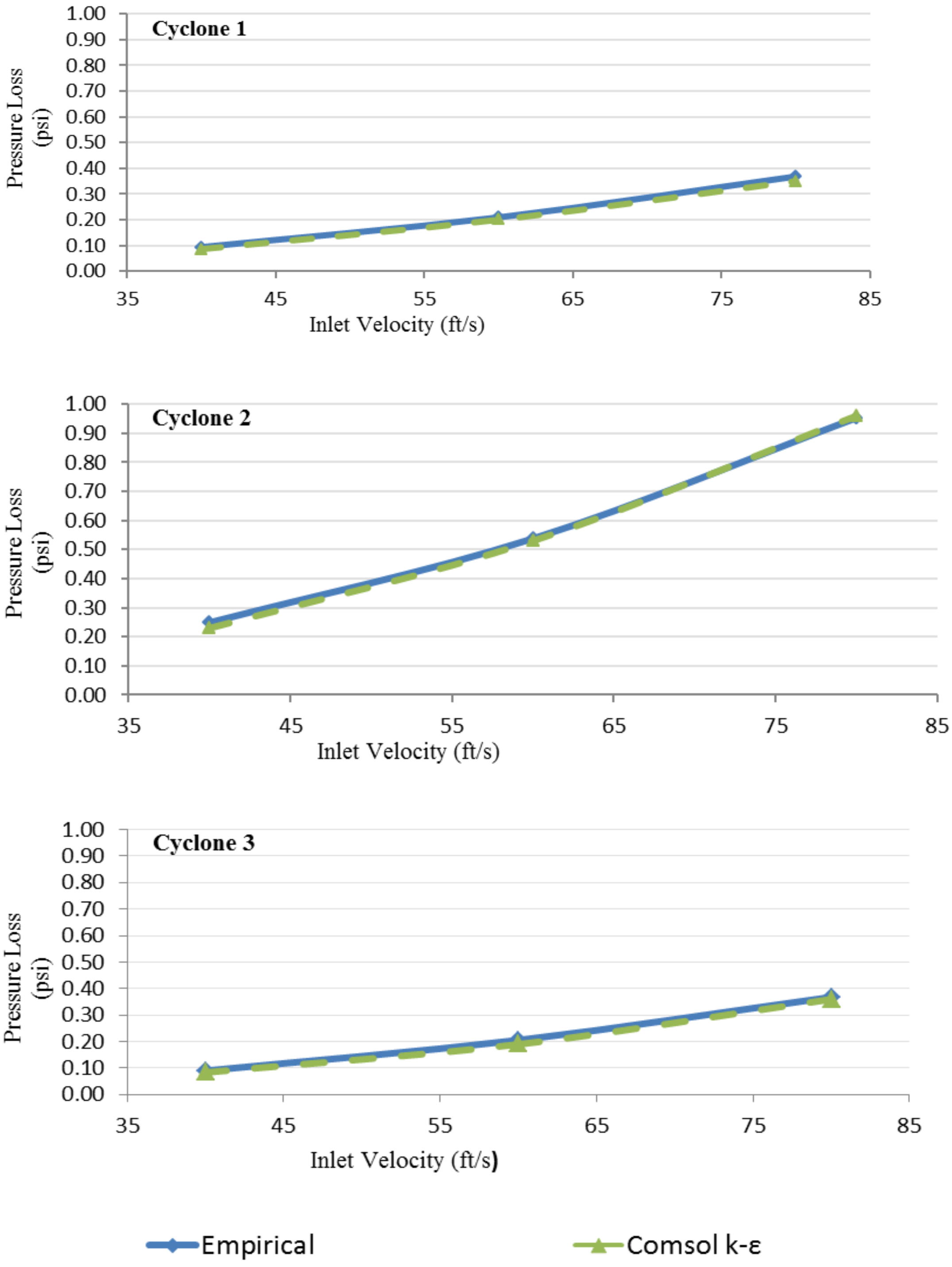


Figure 4. Pressure Loss Curves.