FloEFD™ for Cyclone Simulation

Air cyclones are a popular method for separating out particulates due to their simple design and low capital and running costs.

By John Murray, Industry Manager, Mentor Graphics

The simple operating principle belies the complexity of the air motion inside the cyclone. This is characterized by high levels of turbulence, strong anisotropy and an unsteady, swirling airflow.

The lack of a stable theory of fluid motion in an air cyclone tends to favor either incremental alteration of existing designs and/or expensive physical prototyping. However, computational fluid dynamics (CFD) can be used to have a better understanding of the intricate flow field structure of the cyclone and help designers understand important features such as hydraulic resistance, central vortex stability and cyclone efficiency, as described by the degree of air purification. Obviously, the usefulness of any such results depends upon the trust that can be placed in the CFD tool for a given application. Mentor Graphics FloEFD addresses this issue directly by continually looking for industrial benchmarks or suitable experimental data against which it can be compared.

This FloEFD CFD study is particularly interesting as the k-ε turbulence model is generally regarded as not being suitable for the swirling flow that obviously plays such a prominent role in a cyclone. It therefore provides an excellent dataset against which to judge FloEFD's enhanced k-ε turbulence model. [1]

Two separate experiments were considered: the first relates to a Stairmand High Efficiency cyclone [2], the second to a cyclone with a bin [3] (see Figure 1 & 2).

As well as the lack of a bin, the Stairmand HE cyclone differs by not having a flow straightening device at the end of the outlet pipe. Beyond geometric considerations, the two experimental datasets emphasized differing aspects and so allowed a broader range of FloEFD outputs to be assessed. Comparison with the Stairmand data will provide a good benchmark of how FloEFD handles the pressure differential between inlet and outlet. The Lorenz cyclone with bin data will allow an assessment of how accurately FloEFD is tracking the motion and settling of particles of various diameters to be simulated.

Stairmand HE Cyclone

A range of cyclone operating conditions were considered, covering inlet velocities of 5-25m/s. A comparison of the FloEFD predictions with the experimental results (Figure 4) show excellent agreement for calculated pressure drop between inlet and outlet.

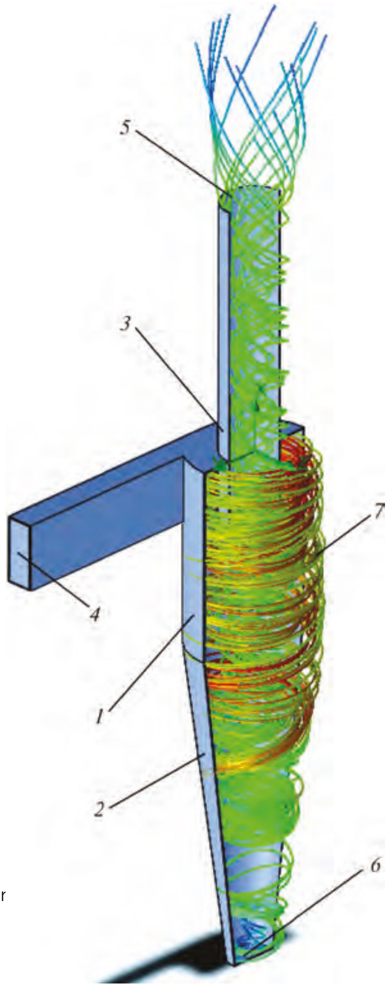

Figure 1. Design of Stairmand HE cyclone: 1) cylindr part; 2) conical part; 3) outlet pipe; 4) inlet pipe; 5) expulsion; 6) dust expulsion; 7) current lines

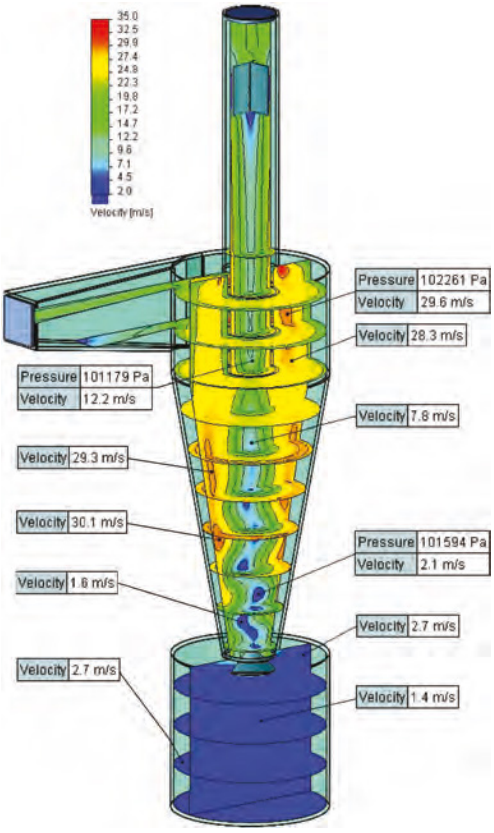

Figure 2. Velocity distribution in cyclone.

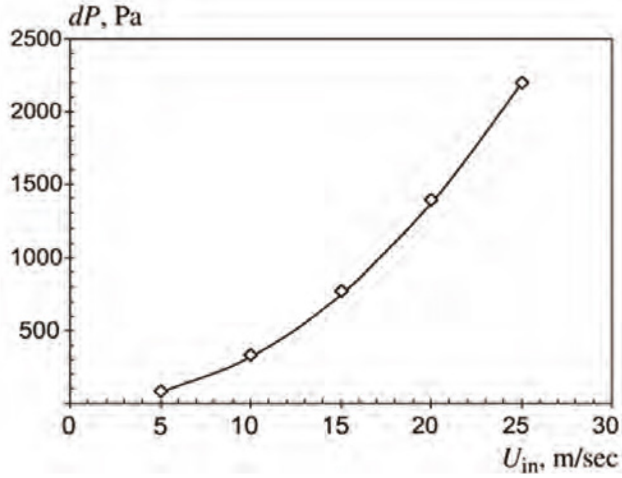

Figure 3. Dependence of hydraulic resistance of Stairmand HE cyclone on air velocity at cyclone inlet

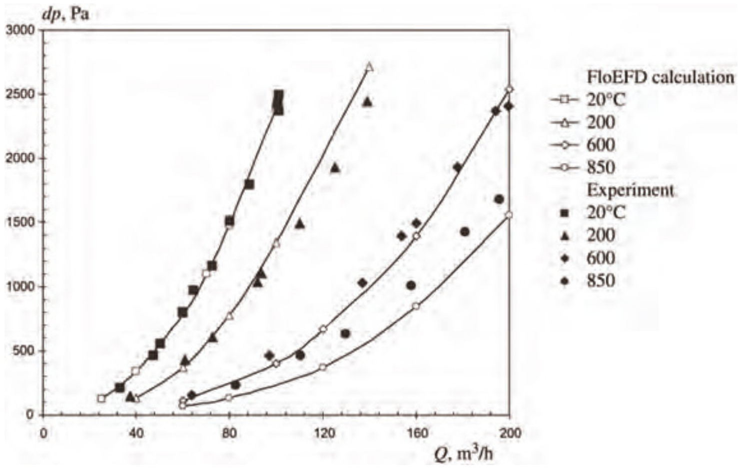

Figure 4. Dependence of hydraulic resistance *dP* of cyclone with bin on throughput *Q* at various gas temperatures

Lorenz Cyclone with a Bin

In contrast to the above, the experimental data for a cyclone with a bin are distinguished by a wide variety of conditions and temperatures. In addition, an evaluation of the cyclone efficiency which FloEFD replicated via an assessment of the motion and settling of particles. In order to do this, the inlet surface discharged 500 particles of a given diameter, across a range of diameters, from the inlet pipe. The trajectory of each particle was then tracked and the cyclone efficiency determined as the ratio of the number of captured to the number of discharged particles of each size. In common with the Stairmand HE case, the first step was to compare predictions of the hydraulic resistance across a range of flow rates with the experimental dataset. In this case, the experimental dataset included points for a range of gas temperatures, which was in turn calculated by FloEFD, (Figure 4).

It can be seen that there is again excellent agreement between experiment and simulation for this cyclone geometry. While the error does increase at higher temperatures, even at 850°C it is little over 10%. In practice, gas flows in cyclones rarely exceed 400°C. The calculated cyclone efficiency is shown Figure 5.

Since the flow pattern in a cyclone is unsteady, the probability of particle drop-out for each diameter was calculated by averaging the results of five discharges of particles. The vertical bars at each point represent the maximum and minimum probability of drop-out of a particle with a definite diameter over five particle discharges. While the calculated cyclone curves have a slightly steeper gradient relative to the experimental data, the agreement between FloEFD and the dataset is very good, particularly below 200°C.

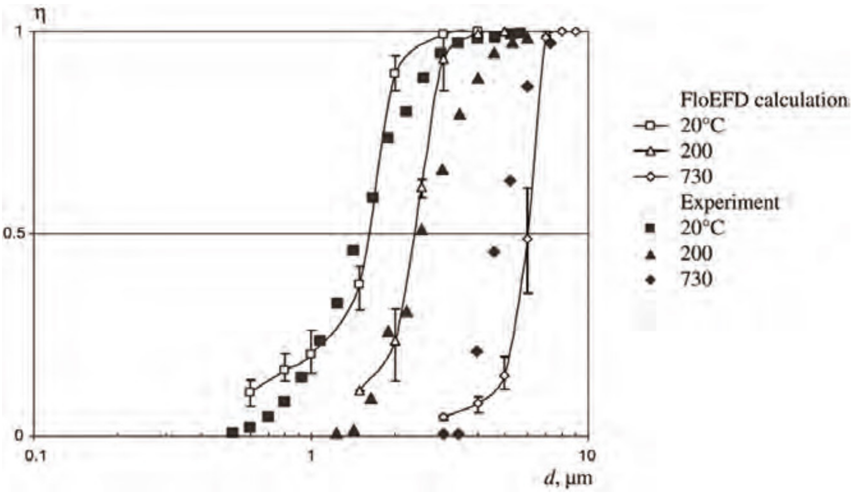

Figure 5. Dependence of gas purification degree η on particle size at various gas temperatures at 60 m³/h cyclone throughput

Conclusions

This work demonstrates that the modified k-ε turbulence model used in FloEFD is well suited to highly swirling flows. Considering the low computational overhead associated with FloEFD (the largest mesh was generated for the cyclone with a bin case, which only came to 380,000 cells) it is clearly the case that FloEFD can offer much to engineers and designers involved in the design of such systems.

References:

[1]. Enhanced Turbulence Modeling in FloEFD™, MGC 02-11, TECH9670-w, Mentor Graphics Corporation (2011).

[2]. F. Boysan, W. H. Ayer, and J. A. Swithenbank, “Fundamental mathematical modeling approach to cyclone design,” Trans. Inst. Chem. Eng., 60, No. 4, 222–230 (1982).

[3]. T. Lorenz, “Heißgas Entstaubung mit Zyklonen,” VDI Fortschritt Berichte, Reihe 3, Verfahrenstechnik, No. 366, VDI-Verlag, Dusseldorf (1994).

This article was adapted from an article published in Petroleum and Engineering Journal, Vol. 49, Nos. 3–4, July, 2013. Computerizing Calculations and Designing a Numerical Modeling Software Package for Computing Aerodynamic Characteristics of Air Cyclones. A. V. Ivanov, G. E. Dumnov, A. V. Muslaev, and M. V. Popov