Spring 5-5-2011

Particle-Fluid Flow Simulations of an FCC Regenerator

Sam Clark
CPFD Software, LLC

PARTICLE-FLUID FLOW SIMULATION OF AN FCC REGENERATOR

Sam Clark
CPFD Software, LLC
10899 Montgomery Blvd. NE, Suite A
Albuquerque, NM 87111
sam@cpfd-software.com

ABSTRACT

A particle-fluid flow simulation of a commercial-scale fluidized catalytic cracking unit has been conducted. The simulation was full-scale, three-dimensional, and with complex internal geometries. The focus of the computational model was to predict wear on internal structures. The geometry of the particle feed pipe was found to cause asymmetric flow of high-speed gas which led to significant wear.

INTRODUCTION

A particle-fluid flow simulation of a commercial-scale fluidized catalytic cracking (FCC) unit has been conducted using CPFD's Barracuda software. Barracuda is based on a multi-phase particle-in-cell (MP-PIC) implementation of computational particle fluid dynamics (CPFD), which uses an Eulerian scheme for the fluid field and a Lagrangian scheme for the particles. For the sake of brevity, the current paper does not go into the numerical details of the CPFD method, and the reader interested in such details is directed instead to Andrews and O'Rourke (1) and Snider (2).

The regenerator simulation was full-scale, three-dimensional, and with complex internal geometries. The geometry was generic, i.e. the simulation was not of any actual operating FCC unit. Though chemistry can be included in Barracuda models, as shown by Snider and Banerjee (3) and Snider, Clark, and O'Rourke (4), the current simulation was isothermal, and did not include chemical reaction calculations. The hydrodynamic behavior was of primary interest, so thermal and chemical effects were neglected.

The simulation was run to quasi-steady state operating conditions, and both transient and time-average data were collected. The results of the simulation included prediction of fluidization characteristics, wear on internal structures due to particle impact, and entrainment of solids into cyclones. It was found that an asymmetric feed pipe geometry was the reason for the extreme wear observed on specific internal structures.

GEOMETRY AND OPERATING CONDITIONS

The physical configuration and operating conditions of the FCC regenerator in the current simulation were chosen to be generically representative of what might be found in industry. Fig. 1 shows the geometry of the regenerator, which was a cylindrical vessel with domed bottom and top, a diameter of 15.2 m (50 ft), and an overall height of 29.0 m (95 ft). The bottom portion of the vessel was equipped with three gas distribution rings to provide fluidizing gas. Twelve primary-secondary cyclone pairs were positioned in the top portion of the vessel, and each cyclone had a dipleg extending down to return entrained particles to the fluidized bed. The regenerator was also equipped with a single standpipe, which in an operating system would allow particles to be discharged back to the FCC reactor.

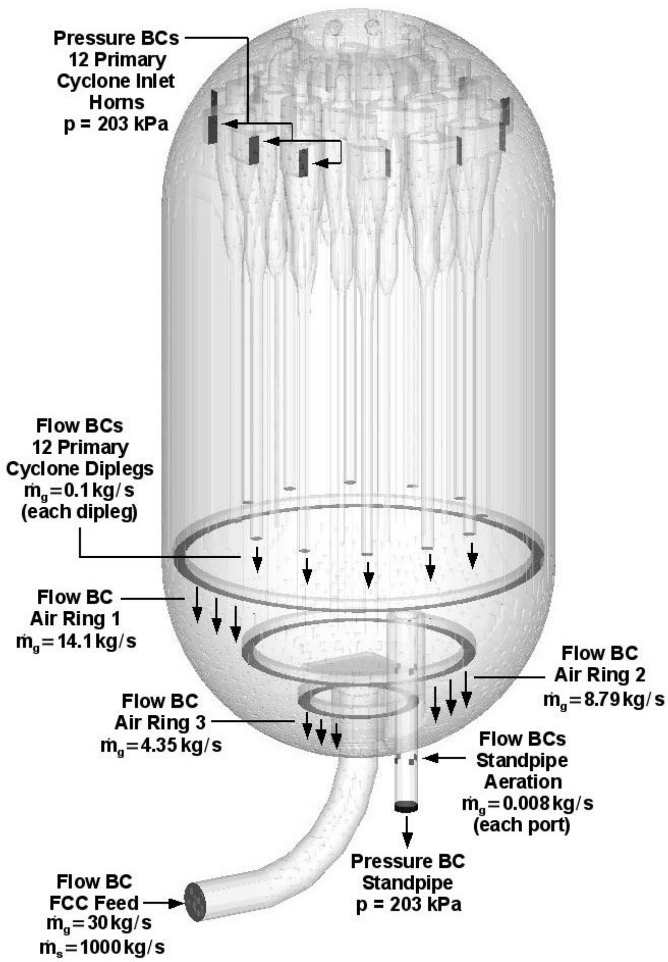

Figure 1: Geometry and boundary conditions for the FCC regenerator

The geometry of the FCC catalyst particle feed pipe was of particular interest in the current simulation. The horizontal pipe led to an upwardly curved “J-bend”, which transitioned to a vertical riser section. This type of “J-bend” configuration is necessary for reactor-regenerator configurations where the units are side-by-side, and the FCC catalyst particles must be transported horizontally from the reactor to the regenerator. In other reactor system designs, the reactor might be positioned directly above the regenerator, and a “J-bend” would not be necessary.

Particles exiting the vertical riser section of the feed pipe impacted a flat plate positioned above the pipe outlet. This plate was used as a crude termination device for the purposes of the current simulation. In operating FCC regenerators, more sophisticated termination devices are used. These more complex designs attempt to distribute particles more evenly into the fluidized bed, which leads to better regeneration of the catalyst. For the purposes of the current generic geometry, it was not necessary to use a more sophisticated termination device.

As approximations of typical operating conditions, the values shown in Table 1 were used in the current simulation.

Table 1: FCC operating conditions

Parameter	**Value**
Freeboard pressure	203 kPa (2 atm)
Temperature (isothermal)	994 K (1330 °F)
FCC catalyst particle density	1,425 kg/m^3 (89 lb/ft^3)
FCC catalyst d_{50} particle diameter	78 microns
Initial bed catalyst particle mass	70,000 kg (77 tons)
Superficial velocity in fluidized bed	0.42 m/s (1.4 ft/s)

SIMULATION SETUP

Computational Grid and Particles

The first step of the simulation process was the definition of a computational grid. The computational cells defined by the grid are used to solve the fluid transport equations, the results of which are strongly coupled with particle momentum equations. A finer grid gives higher fidelity in the computational solution, but requires a longer calculation time. Barracuda uses a regular rectangular grid. The grid used for the current simulation had computational cells with side-lengths ranging from 15 to 46 cm (6 to 18 inches) throughout most of the domain. The computational grid contained 270,000 real cells.

In a large commercial vessel such as the regenerator currently under consideration, there could be on the order of 10^{15} individual particles in the system. With current computers, it is not feasible to model the detailed motion of this many individual particles in a simulation. Barracuda uses so-called computational particles, which allows for useful engineering results in reasonable solution times. Each computational particle represents a group of real particles that share physical properties such as material, radius, and density. The number of computational particles used the simulation affects the accuracy of the results. As with the grid, using more computational particles gives more accuracy, but also requires longer run-times. In the current simulation, about 1.8 million computational particles were used.

Initial and Boundary Conditions

Barracuda solves for the motion of particles and fluid by solving conservation equations for all computational cells and particles in the simulation (2). The initial conditions provide a starting point for all calculations, while the boundary conditions (BCs) specify where fluids and particles are entering or leaving the system. Fig. 1 shows the regenerator geometry with BC definitions. The initial condition for the simulation was specified such that the regenerator was filled to a target level with FCC catalyst particles at rest. The fluid, air, was also at rest. The pressure and temperature of the system were set to match the desired operating conditions.

Flow BCs that brought in only gas were defined for the fluidizing air rings. Flow BCs that brought in both gas and particles were defined at the bottoms of the primary cyclone diplegs and at the FCC catalyst particle feed at the end of the feed pipe. Pressure BCs were defined at the inlet horns of the primary cyclones and at the bottom of the standpipe. Particles were allowed to exit at all pressure BCs, and the overall system mass was maintained at a constant value by using a particle feed controller, which adjusted the flow rate of particles at the cyclone diplegs as needed to maintain a constant system mass.

Wear Due to Particle Impact

Estimating the expected wear on internal structures was an important goal of the simulation, so the Barracuda wear model was used. The wear model calculates the cumulative wear due to impact of particles on all wall surfaces in the simulation. Research has shown that the damage to a surface from particle impacts is dependent on factors such as particle mass, velocity, and angle of incidence (5). In the Barracuda wear model, when a particle hits a wall, the wear due to this impact is calculated based on the particle mass, velocity, and angle of incidence. The user has control over the dependence on each of these factors, and the appropriate values for each are material-specific. For example, FCC catalyst particles impacting bare metal would have different values for the calculation parameters than FCC catalyst particles impacting refractory. For the current simulation, the particle mass, m, and velocity, u, terms contributed to the calculated wear as $m^{1.5}$ and $u^{3.5}$.

RESULTS

The simulation was set up to collect various types of data, including transient pressure at specific locations, time-averaged gas and particle mass fluxes, and particle residence times for FCC catalyst particles entering through the feed pipe. Many aspects of the system could be examined based on the available data, but this study was focused specifically on two related behaviors shown by the simulation: flow of particles and gas around the “J-bend” in the feed pipe, and high wear on internal cyclone diplegs in one half of the regenerator.

Transporting FCC catalyst particles horizontally is necessary in situations where the reactor and regenerator vessels sit side-by-side. In the current simulation, the FCC catalyst particles being fed to the regenerator traveled through the “J-bend” and then vertically through a riser section before discharging into the regenerator. Ideally, particles should be dispersed uniformly into the fluidized bed. However, the “J-bend” geometry causes the particles to pack against the far wall of the bend, which in turn favors high gas flow rates on the inside portion of the curved section. Figs. 2 and 3 show thin-slice views of particles and gas traveling through the feed pipe. The particles are colored by speed, with faster moving particles having darker colors. The gas vectors have their lengths scaled based on gas velocity and are colored with darker vectors representing higher gas velocity.

The particles which pack up on the far wall of the “J-bend” tend to be low-speed, and gas prefers to avoid the higher solids concentration region, leading to a high-speed condition along the inner portion of the curved pipe. The high-velocity gas continues up the vertical riser section, and exits the feed pipe on the inner-curve side.

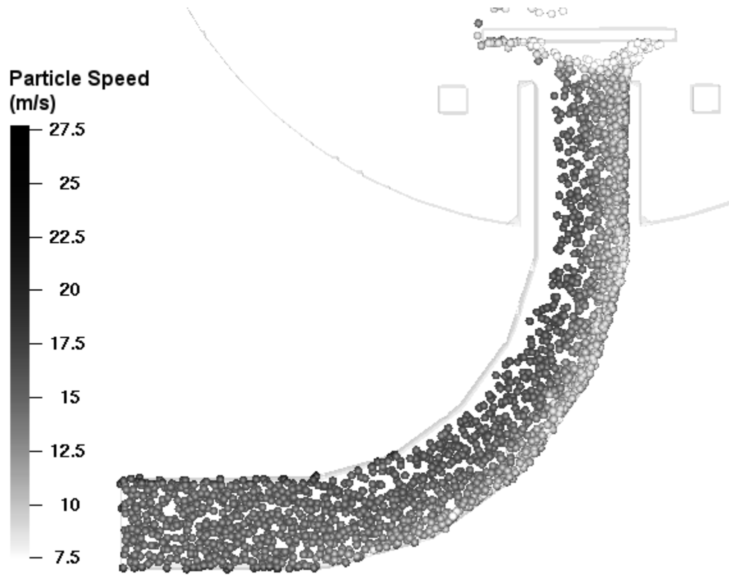

Figure 2: Particles colored by speed in the feed pipe

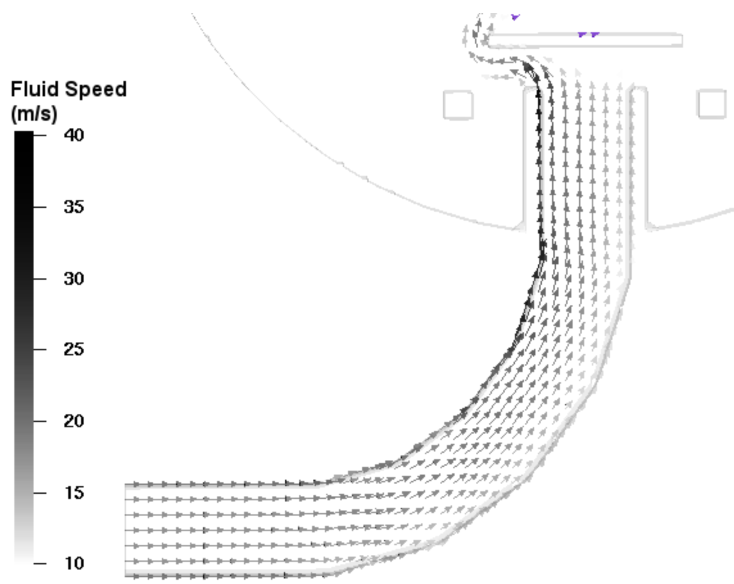

Figure 3: Vectors of gas velocity in the feed pipe

Fig. 4 shows isovolumes of regions where the time-averaged gas velocity is greater than 5 m/s. The tendency of the high-speed gas to stay on the inner-side of the curved pipe is shown by the large isovolume structure emanating from the same side of the feed pipe outlet. Other regions of high fluid velocity include the standpipe area and the inlet horn areas for the primary cyclones. All of these locations are high flux regions, and the relatively small open areas compared with the overall cross-sectional area of the regenerator lead to high gas velocities.

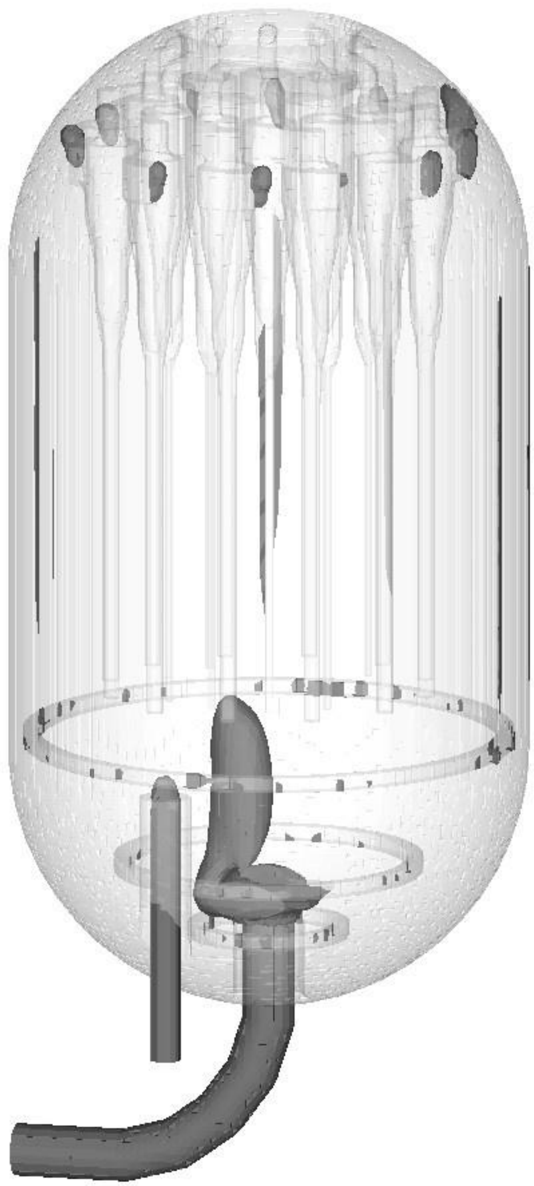

Figure 4: Isovolmues of regions with gas velocity higher than 5 m/s

The high gas velocity on the inner side of the inlet pipe propagates through the fluidized bed, and the effect is significant on the wear calculated for the cyclone diplegs on that side of the regenerator. Fig. 5 shows regions where more and stronger particle impacts occurred. These regions are the most likely to incur damage from wear due to particles hitting the internal structures of the regenerator.

CONCLUSIONS

The FCC regenerator simulation provided insight into the flow behavior of gas and solids through the “J-bend” in the feeding pipe. The asymmetric flow of high-speed gas into the fluidized bed penetrated into the freeboard, and was found to be responsible for high wear on cyclone diplegs on one particular side of the regenerator. If this were an operating unit, it would be difficult to identify the asymmetric flow behavior through typical pressure taps or thermocouples. If the cyclone diplegs were damaged sufficiently, an emergency shutdown might be required, which would be very costly. This simulation shows that CPFD modeling can be applied to large commercial fluid-particle systems to solve performance issues.

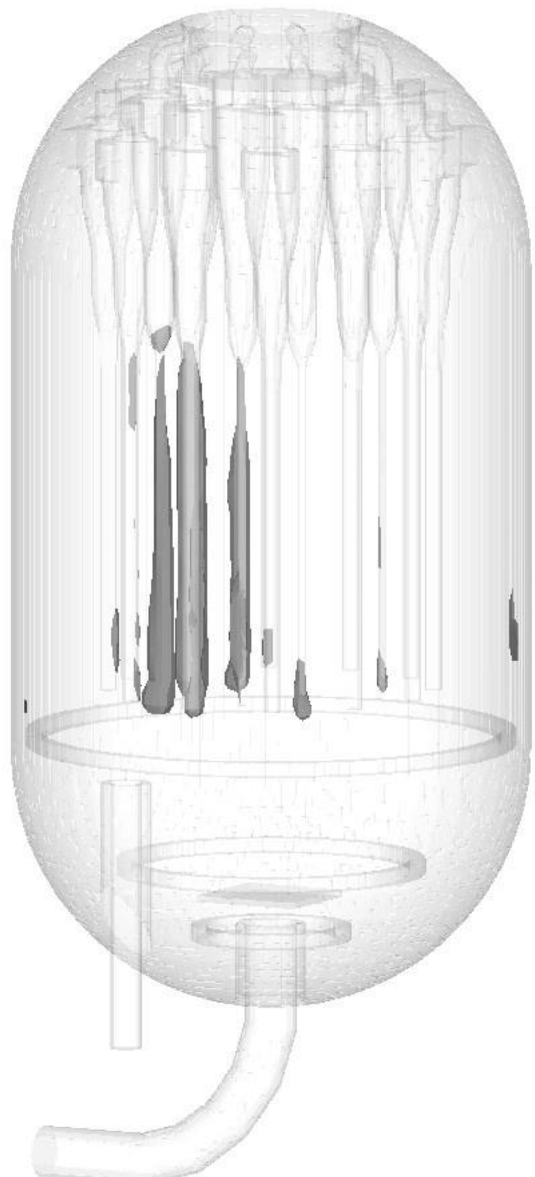

Figure 5: Isovolumes of regions with high predicted wear due to particle impact

REFERENCES

1. Andrews, M. J., and O'Rourke, P. J., 1996. The multiphase particle-in-cell (MP-PIC) method for dense particle flow. Int. J. Multiphase Flow 22, 379-402.

2. Snider, D. M., 2001, An Incompressible three dimensional multiphase particle-in-cell model for dense particle flows. Journal of Computational Physics 170, 523-549.

3. Snider, D. M., Banerjee, S., 2009, Heterogeneous gas chemistry in the CPFD Eulerian–Lagrangian numerical scheme (ozone decomposition). Powder Technology, http://dx.doi.org/10.1016/j.powtec.2009.04.023

4. Snider, D. M., Clark, S. M., and O'Rourke, P.J., 2011, Eulerian-Lagrangian method for three dimensional thermal reacting flow with application to coal gasifiers. Chemical Engineering Science, http://dx.doi.org/10.1016/j.ces.2010.12.042

5. Mills, D., and Mason, J.S., 1979. Evaluating the conveying capacity and service life of pipe bends in pneumatic conveying systems. Journal of Powder & Bulk Solids Technology, 3 (1979) 2; 13-20.