Flow instability in an alumina precipitator fitted with a draft tube circulator

Science, Chemistry

This paper 'Flow Instability in an Alumina Precipitator Fitted With a Draft Tube Circulator" is a good example of a case study on chemistry. Bayer process is the industrial method used for the extraction of alumina from bauxite ore. This process involves several steps under which precipitation falls. At the precipitation step that is where alumina is recovered from the pregnant liquor which is supersaturated in sodium aluminate. After the recovery crystallization starts where hydrates are formed.

$Al(OH)4- + Na+ \rightarrow Al(OH)3 + Na+ + OH-$

These crystals grow to form large particles in the process.

This process of precipitation is paramount in the whole Bayer process. Since it entirely depends on the mechanically agitated draft tube precipitator, some factors in the tube have been found to have an influence in the activities of the particles. There is understandably, then, great interest in developing CFD models of the flow field inside the precipitator to support the design of such vessels. It is important to understand the flow of fluid in the tube since it has an impact on the yield.

Computational fluid dynamics (CFD) is a process of modelling that is has gained popularity in mining. There are several models that have been used to try and optimizes the yield of alumina.

k-epsilon, Eddy Viscosity Algebraic Reynolds Stress model, SST (Menter's shear stress transport) large eddy simulation, Scale Adaptive Simulation and Reynolds stress equation are among the major models that were put to test. The important factor here was to see which amongst them will bring a minimum computational cost. Keeping in mind that all the three are turbulent flow models, I preferred wall function to integrate the turbulence model.

As the simulation began it was established that k-epsilon and Eddy Viscosity Algebraic Reynolds Stress model could not exhibit a prediction of the velocity field fluctuation expected from the experimental result. Menter's shear stress transport (SST) and SSG Reynolds stress model, gave inconsistent result predictions and could not be used. Though the scale adaptive simulation model does not predict turbulence due to lack of enough data it was tested. This model is characterized by the use of a transport equation for the turbulent length scale.

This flow is normally treated as Newtonian and incompressible. Using the Reynolds averaging the following equations are derived

Continuity equation 🗝 • (✕U) = 0 (1)

Momentum equation DP/ Dt= Pij+Dij-🎁ij+ ○ ij+⏵ (2)

Where

Rij is the transport term by convection,

Pij is the rate of production,

Dij is the transport by diffusion,

🎁it is the rate of diffusion,

○ it is the transport due to the turbulent pressure-strain interactions and

⏵it is transported due to the rotation.

The SST model is a combination of k-Epson and k-omega and it is used to control the viscosity turbulence in near the wall region. The turbulence viscosity is calculated by

$\mu_t=(\alpha_1 \rho k)/\max(\alpha_1 \omega, SF_2)$

In which the magnitude of the shear strain rate is given by S and α is a constant.

The turbulence length scale derived from SST is given by

$L=\sqrt{K}/(c\ 0.25/\mu\ \omega)$

Where k is the von Karman constant

Model formulation

A good precipitator should mix the slurry well such that its density and temperature should be the same as those of the outflow. The temperature of the outlet of the precipitator should be equal to the temperature of the inlet of the downstream precipitator. The dynamic heat balance in the precipitator is given by this equation

$$CPV\rho\ (dT_i)/dt= C_p F\rho T_i - t - C_P F\rho T_i - C_{p_w} \rho_w F_p (T-t) - Q - \Delta H$$

The dynamic model conforms to this equation where V is the slurry volume i is the precipitator number and the slurry flow rate is given by F.

To achieve a near perfect precipitation rate and quality alumina, the last precipitator should have a constant and stable temperature. The cooling speed should also be constant also the temperature difference of the inlet and outlet should be 10. To achieve this a better like the decentralized adaptive MPC is preferred. Among the many reasons for this choice, it is able to account for measurable turbulences and unmeasurable turbulence for precision.

Draft tube design

A short draft tube is normally the better design compared to the long one. In this study the precipitator tank has a diameter of 14m, the draft tube has a radius of 2. 3m, and height of the liquid is about 46m and a draft tube height of 27m. The height ratio of ≤ 2 ensures that there is minimal energy consumption during agitation. The draft tube is made of straightening vanes and six slot columns. The vanes direct the slurry into the tube and the slot columns are responsible for the communication of the slurry.

Particle suspension

The particle movement in the tube depends on two forces. Gravity is responsible for the downward flow of particles while drag force is responsible for the upward movement. Increase in fluid velocity leads to a decrease in pressure. When velocity increases beyond fluidizing velocity particles tend to move upwards as shown in the diagram below.

How particles are suspended in the precipitator once all conditions are kept constant and achieved.

Mesh configuration

A uniform tetrahedral mesh of 10mm edge length was used for the momentum source. Layers of inflation were used on a 1mm first layer height and a 1. 2 expansion height. The right density for the mesh doesn't exist but research shows that course mesh for SST-SAS yields better results.

Initializing solution

Preliminaries of CFD requires that you come up with the best solution that will ensure that you yield what you desire. Two methods exist for this process either to initialize all the fields or initialize selected cell fields.

Results

CFD predictions and experimental data were compared between 65cm and 115cm from the floor of the vessel. The impeller tip speed is used to normalize the velocities. A uniform velocity and reasonable use of momentum are exhibited at 65cm at 115cm the data shows low velocity, almost steady and angular movement.

SST-SSA model took the least time of 0. 01 seconds to be stable and achieve a bulky number in the vessel. The course mesh provided the ideal prediction at 65cm. The element size also plays a role in the capture of the turbulence flow structure.

It can be concluded from this stud of flow modelling that the flow in the tube is very dynamic and asymmetrical with some complex outcomes such as solid suspension. The SST-SSA model is very sensitive to the size of the mesh and of all the models SST-SSA is the strongest in computational prediction. The draft tube slots are necessary for achieving a good pumping capability at the tank bottom. I can say in totality that CFD has played a big role in cutting down the operational cost.

I can recommend more research to make sure maximum optimization of the process so that industries can maximize yields.

The use of course mesh for SST-SSA for better result should be encouraged.