Chemical Technology Department

CHULALONGKORN UNIVERSITY

Three-dimension Simulation of Circulating Fluidized Bed Reactor Hydrodynamics

Thailand

www.chemtech.sc.chula.ac.th

ANSYS®

CAD-IT CONSULTANTS

Overview

The chemical technology department of Chulalongkorn university has conducted a research in circulating fluidized bed (CFB) reactor for more than ten years by using computational fluid dynamics (CFD) simulation. We are a leading research group for the development of CFB reactor in Thailand by using finite volume method.

A CFB reactor is widely applied in many industries, including combustion and gasification, because it gives high system efficiency and supports various types of solid fuel. To develop and improve the process, an information of the hydrodynamics inside the CFB reactor is very important. Therefore, a CFD model has been developed and used as a representative of the realistic CFB reactor.

The CFD model has proven to be an useful research tools in the field of multiphase flow fluid mechanics, where it can be used to obtain a better understanding of flow behavior in different phases.

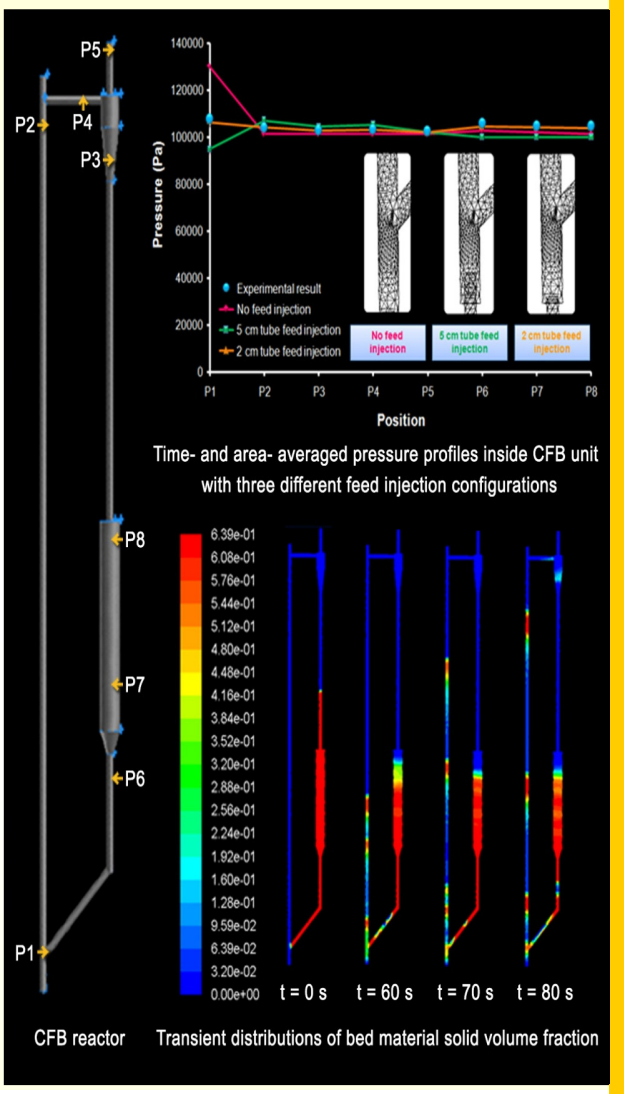
Testimonial

"We used ANSYS Workbench, ANSYS DesignModeler and ANSYS Fluent to study the hydrodynamics in our experimental CFB reactor unit. The simulation successfully computed the flow behavior for gas and solid fuel phases due to the high performance modeling features and low computational time. As a result, all the ANSYS program has now played a significant role in the prosperous growth of our research group."

Pornpote Piumsomboon, Ph.D.

Benjapon Chalermsinsuwan, Ph.D.

Chemical Technology Department, Faculty of Science
Chulalongkorn university



Process challenges

The simulation methodology of CFB reactor can be divided into two main parts. First, a three-dimension mesh generation model was constructed in ANSYS Workbench and DesignModeler. Second, ANSYS Fluent was used to solve the gas-solid flow phenomena. Here, the Eulerian approach with kinetic theory of granular flow was employed. After tuning the proper parameters, the result was compared with experimental data to test the model accuracy.

Solution

In this study, we validated the numerical simulation result with the measured experimental pressure. The simulated pressure profile along the CFB reactor is agreed well with that from the experiment. The transient distributions of bed material solid volume fraction illustrates the slugging fluidization flow pattern in which bed materials separate into slices of gas and solid. Also, the effect of feed injection configurations was explored.

Benefits

The CFD model allows the understanding of CFB reactor system hydrodynamics, while reducing the necessity for expensive cost and time. By using all the ANSYS programs, the reasonable CFB reactor model was obtained. The suitable system hydrodynamics will be able to improve the chemical reaction taking place inside the CFB reactor in order to increase and decrease the chemical composition of the wanted and unwanted gaseous products, respectively.