Modelling and Simulation of Single Phase Fluid Flow and Heat Transfer in Packed Beds Using COMSOL Multiphysics

S. Sachdev[1], S. Pareek[1], B. Mahadevan[1], A.Deshpande[1]

[1]Department of Chemical Engineering, BITS Pilani Goa Campus, Zuarinagar, Goa, India

Abstract

Computational fluid dynamics has emerged as an advanced tool for studying detailed behavior of fluid flow and heat transfer characteristics in many of the chemical engineering applications. One of them is fluid flow through packed beds. Packed bed systems play a very important role in most of the chemical industries, especially petroleum, petrochemical and biochemical industries. Hence it is essential to understand the fluid flow behavior and temperature variation in different sections of packed bed. Geometric complexities of packed beds have prevented detailed modeling of their hydrodynamics. With the help of modern CFD codes and the exponential growth of computer power it is now feasible to obtain detailed flow fields in packed beds of low tube-to-particle diameter ratio (N). By using COMSOL Multiphysics for modeling and simulation of packed bed reactors, a detailed description of the flow behavior within the bed can be established. Following models (Physics) were used to model such packed beds in COMSOL Multiphysics:-
1.Incompressible Navier Stokes - To obtain fluid flow characteristics.
2.Turbulent Standard k- model - To incorporate turbulence.
3.Heat Transfer in Fluids (Conduction and Convection Model) - To obtain temperature profiles.
The fluid used was standard air at 300K. To create the N=8 (3D) model, 2 layers of spherical packing were created and staggered over one another to form structured packing (refer to Figure 1). Fluid flow behavior and temperature profiles were attained using the above mentioned models. The following results were obtained for the simulation of N=4 and N=8 models:
1.Whenever there is contraction (decrease in area), velocity through that region increases.
2.Due to high velocities at contractions, turbulent boundary layer grows rapidly and due to adverse pressure gradient, flow separation occurs.
3.Pressure is maximum at the stagnation point (velocity = 0).
4.There is decrease in temperature due to sudden contraction when the fluid flows through the space between two spheres.
5.From the continuity equation, the velocity increases and hence the convective heat flux which further creates a temperature gradient.
6.There is a pressure drop across the length at the expense of increasing in the velocity.
The results obtained for N=4 and N=8 (as shown in Figure 2, 3 & 4) were found to be consistent with those present in the literature and the ones predicted through various equations. COMSOL Multiphysics can also be used to predict the results of much more complex geometries, which are close to reality, such as for N=16 and for various other kinds of packing. Most of the industrial

packing is indeed complicated and COMSOL Multiphysics can help us enhance our understanding of the fluid behavior inside these packing by effectively modeling them.

Reference

Michiel Nijemeisland et al. Anthony G. Dixon, CFD Study of Fluid Flow and Wall Heat Transfer in a Fixed Bed of Spheres, American Institute of Chemical Engineers, AIChE J, 50: 906–921, 2004

Figures used in the abstract

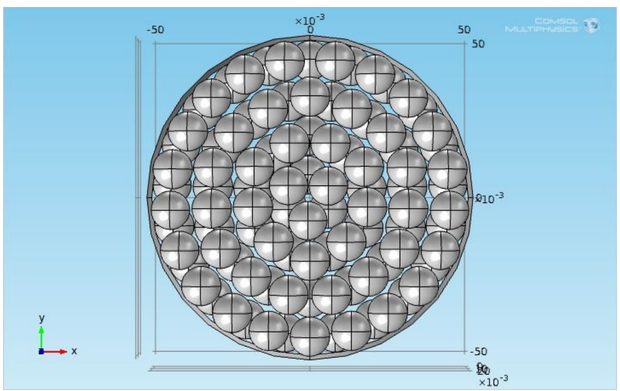

Figure 1: Geometry for N=8 (3D Model).

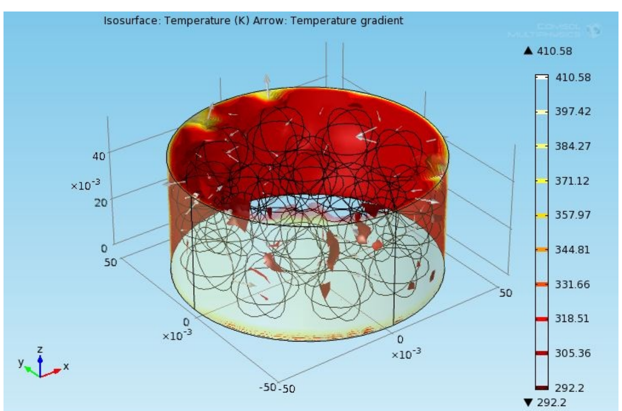

Figure 2: Temperature gradient (arrow) for N=4 (3D model).

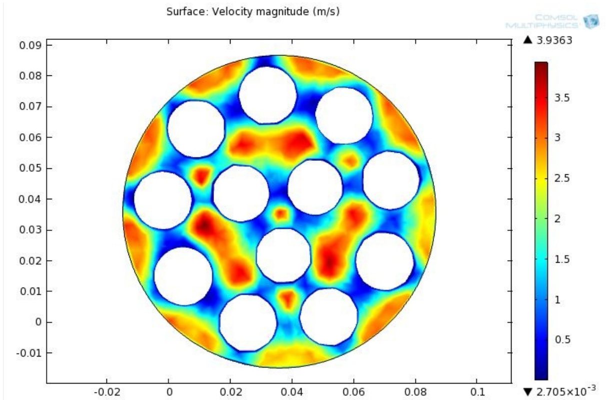

Figure 3: Surface Plot: Velocity Magnitude for N=4 (3D Model).

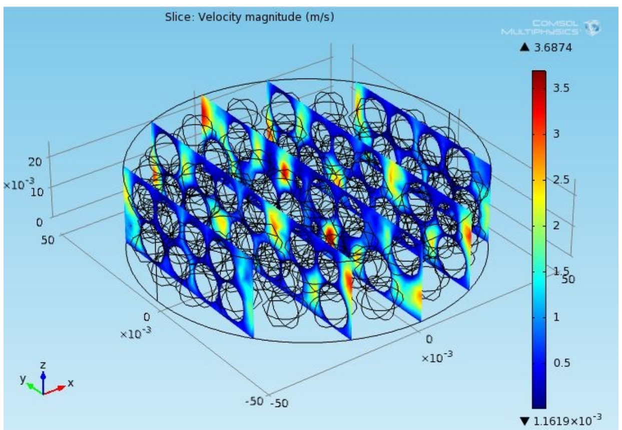

Figure 4: Velocity plot (isometric view) for N=8 (3D Model).