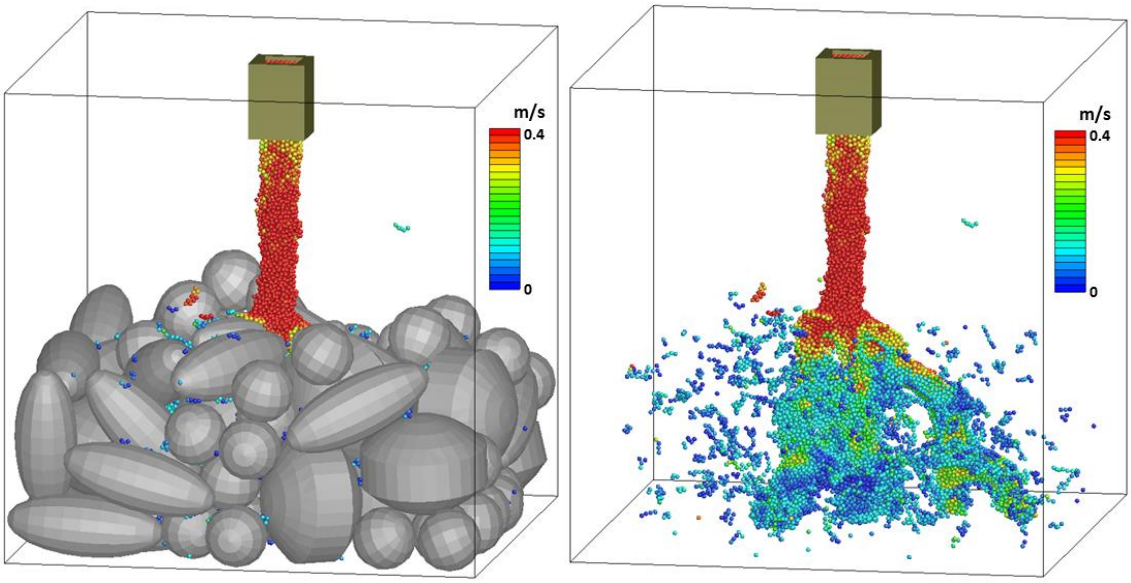
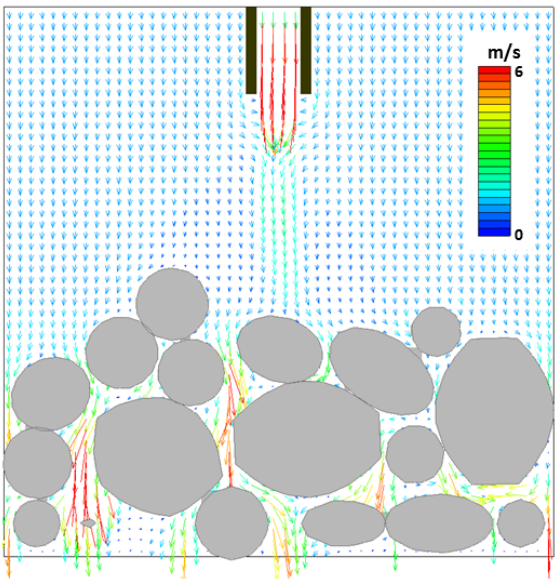
Liquid and Gas Flow Simulation in Trickle Bed Reactor

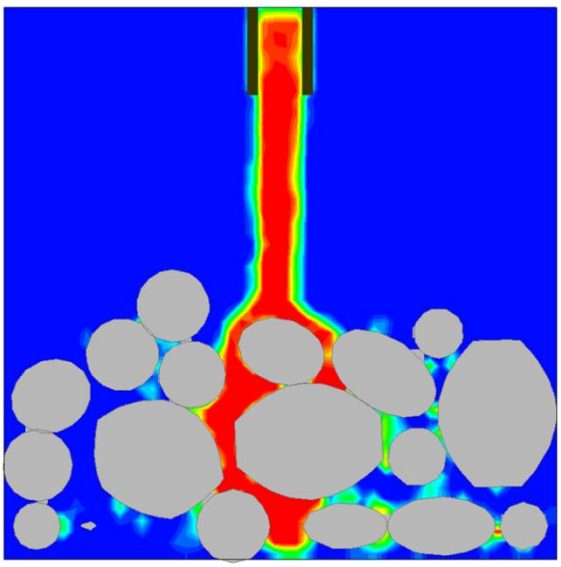
The liquid flow in a trickle bed reactor filled with catalyst particles of various shapes is simulated numerically by the MPS (Moving Particle Semi-implicit) method. The gas flow is simulated by the finite difference (volume) method using computational grids.



Distribution of liquid flow velocity passing between catalyst particles. Particle display color represents the absolute value of MPS particle velocity.



Gas flow velocity distribution in the central section of the reactor. Gas flows in from the top front.

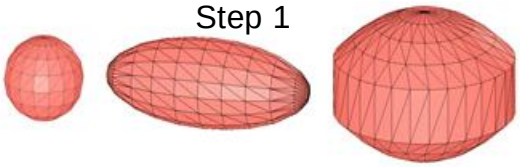


Contour of the liquid distribution in the central section of the reactor calculated from the MPS particle arrangement.

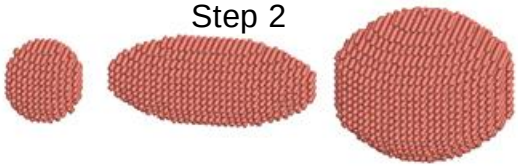
Simulation procedure for trickle-bed reactors

Prior to performing flow simulations in a trickle-bed reactor, it is necessary to reproduce packing conditions of the catalyst particles. Since the catalyst particles have a variety of shapes, the DEM simulation should be performed in several steps using combined particles created from CAD geometry data.

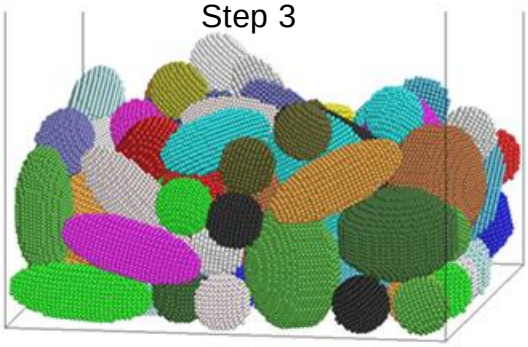
Step 1



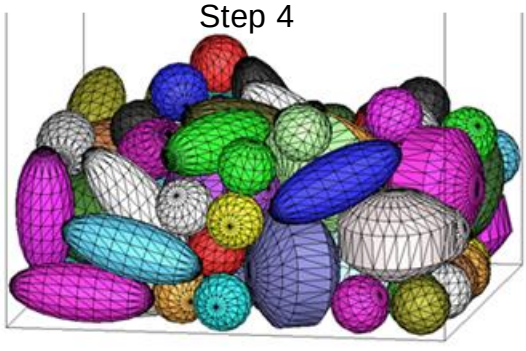
Step 2



Step 3



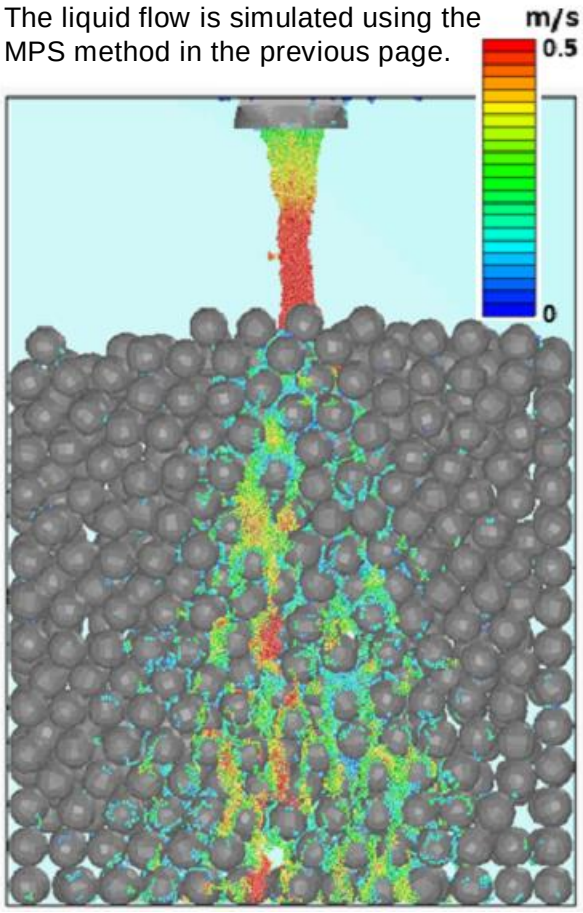
Step 4



Step 5

Computational results for the liquid flow using the MPS method described in the previous page.

Step 1:
Create catalyst particles of various shapes as wall elements. Wall elements can be created from the R-FLOW pre or CAD via STL files.
Step 2:
Generate combined particles within each wall element. Combined particles can be generated by the R-FLOW solver.
Step3:
Perform free-fall DEM simulation using the combined particles until all the particles are stationary.
Step4:
Restore the corresponding wall elements from the stationary combined particles with the R-FLOW solver.
Step5:
The liquid flow is simulated using the MPS method in the previous page.



Liquid flow in a trickle bed reactor filled with spherical catalyst particles obtained by the MPS method simulation.