**CFD simulation of steam distribution in vertical tube bundle at high pressures.**

Arijit Ganguli1

*1 School of Engineering and Applied Sciences, Ahmedabad University, Ahmedabad, Gujarat, 380009*

*\*Corresponding author: arijit.ganguli@ahduni.edu.in*

**Highlights**

* CFD simulations in header and tube assembly to quantify extent of non-uniformity.
* Mal-distribution at high pressures
* Attempt to design innovative distributor to reduce mal distribution

**1. Introduction**

The distribution of a fluid stream into a number of parallel substreams by means of the channels is accompanied by fluid pressure changes owing to the change of fluid momentum in the conduits, while the effect of wall friction in the conduits is negligibly small compared with that in the channels (Bassiouny and Martin, 1984). Such arrangements are found in industrial equipments (heat exchangers) where the flow is distributed from a main header to a number of tubes. Ample amount of research has been done on this problem by researchers over the past (Acrivos et al., 1959; Bassiouny and Martin, 1984). In such cases, the sudden changes in flow direction make the pressure rise in the top header and fall in the bottom header (Gandhi et al. 2011). Gandhi et al. (2011) have carried out simulations at high pressures and eliminated the middle tube for achievement of uniform flow distribution. However, in some situations where the geometry is such that the middle tube is exactly beneath the feed pipe then the use of internals may help in achieving better distribution. CFD studies for pressures around 60-70 atm is difficult and is rarely found in the literature

**2. Methods**

A three dimensional grid has been considered in the study. A non-uniform hexahedral mesh has been created for each configuration. For the geometry the mesh size is 18,634 hexahedral cells with fine cells inside the tubes and walls and uniform mesh at the headers. The grid independency for the geometry was carried out with three different grids namely 128,000, 186,000 and 232,000 and the centerline axial velocity was checked. Since the difference between the magnitude of centerline axial velocity was 1% for the 186000 and 232000 grids, 186000 grids was selected for investigation. The basic governing equations of continuity and momentum in Cartesian co-ordinates have been used. The k-ω turbulence model has been used for modeling the turbulence. The commercial software Ansys FLUENT 18 has been used. In case of k-ω model, Quadratic upstream interpolation for convective kinetics (QUICK) discretization scheme was used for the turbulence parameters. For final sweep over each segment, upwinding has been performed using the QUICK with a second order pressure scheme. The QUICK formulation has a third order accuracy and which helps to mitigate the unfavorable effect of artificial diffusion that can occur when using low order upwinding schemes. All the discretized equations were solved in a segregated manner with the PISO (Pressure Implicit with Splitting of Operators) algorithm. In the present work, steady simulations were performed. All the solutions were considered to be fully converged when the sum of residuals was below 10-5.

**3. Results and discussion**

The pressure drop is calculated as: Pressure Drop = .

The pressure drop profile is shown quantitatively in Figure 1A. Further, the velocity distribution has also been shown in Figure 3B. The dimensionless velocities at the top and the bottom headers show high velocities at the inlet. This indicates that all the fluid passes from the center of the tubes as shown in Figure 1B.



**Figure 1.** (A) Effect of dimensionless pressure drop with dimensionless horizontal distance (B) Effect of non-dimensional velocity on non-dimensional horizontal distance (1) At the inlet of the central tube (2) At the outlet of central tube.

**4. Conclusions**

CFD simulations have been carried out for header tube geometry where inlet and outlet are at the middle of the top and bottom headers. It is seen that at low pressures the extent of non uniformity increases with an increase in pressure when conventional geometry is considered. For very high pressures upto 70 bars the mal-distribution is significantly lower than the parent one

**References [Calibri 10]**

1. Acrivos A., Babcock B.D., Pigford R.L., Chem. Eng. Sci. 10, (1959) 112-124.
2. Bassiouny M.K., Martin H., Chem. 858 Eng. Sci. 39, (1984) 693–700..
3. Gandhi M.S., Ganguli A.A., Joshi J.B., Vijayan P.K., Chem. Eng. Res. Des. 2011.