Simulation of Passive Fluid Mixing in Microchannels

Xia Huanming and A/P Shu Chang

(Department of Mechanical Engineering, NUS)

1. Introduction

Fluid mixing in microchannels has a wide application and is very important in microfluidic systems, such as lab-on-a-chip for biochemistry analysis, microreactors for production of organic compounds, etc. However, fluid mixing at the microscopic scale is far more difficult than that in macroscopic fluid devices. In a typical microfluidic device, viscosity dominates the flow and the fluid streams prefer to adopt laminar flow patterns. So the fluid mixing mainly depends on molecular diffusion which is very slow. To achieve good mixing, an efficient micro-mixer usually involves complex 3-dimensional geometries which are utilized to enhance the fluid lamination, stretching and folding. In this study, computational method is applied as an efficient tool to model the micro-mixers and the simulation results can be used as a reference and a prediction of the mixing performance.

2. CFD method and analysis

The following presents a simulation example of a 3-Dimensional serpentine micro-mixer. The commercial CFD code, CFX® is used to perform the simulation. The mixer structure was proposed by Liu, et al. (2000), and its geometry is shown in Fig.1. The mixer is a combination of the “C-shaped” sections that turns the fluid through 180° or rotates the fluid by 90°.

In the simulation, the two different streams are defined by setting the concentration value (voluminal content of one distinguishing fluid per unit volume) 0 at one inlet and 1 at the other. Some simulation results are shown in Fig.2 and 3. Fig.2 (a) and (b) show the concentration distribution at the sampling plane (Fig.1) at different Reynolds numbers. Clearly, the mixing performance is poor at Re=0.2. That because at low Reynolds numbers, viscosity forces
dominate the flow and the velocity in the channel cross-section is 2-dimansinal, as shown in Fig.3 (a). And the mixing between the streams is purely diffusive. However, at Re=60, the effects of the inertial force become important. The transverse momentum around the bends of the channel causes “secondary flows” which increases the interfacial area, and finally leads to a much better mixing. Those observations agree well with the experimental results reported by Liu, et al. (2000).

![Fig.2](image)

Fig.2 Concentration distribution at the sampling plane.

![Fig.3](image)

Fig.3. Velocity vector at the sampling plane.

Note that the fluid mixing shown in Fig.2 is partially due to the mesh diffusion (false diffusion). To reduce the effects of the mesh diffusion demands a finer mesh, which consequently requires more powerful computational resources.

3. Summary

With CFX®, CFD method has been applied to simulate a 3-D micro-mixer. The simulation results present good prediction of the mixing performance. It’s been proven that CFD method can be used as an efficient way to study fluid-mixing problems at the microscopic scale. This will facilitate the design and development of the micro-mixer systems.

Reference