it has both flow momentum vectors in the direction of flow and the normal directions which induces mixing in the bulk flow. If the flow rate was to be increased more from the full flow value, increasingly oblique angles may be needed to create mixing, such as 75° or larger. This result indicates that the flow rate through a channel with surface features affects the mixing in the channel, and that the optimal surface feature angle is dependent upon both the channel dimensions and the design flow rate.

Example

Comparison of Time Spent by a Particle in Surface Feature Compared to Main Channel at Different Reynolds Number

[0388] A case was studied to estimate the time spent by a particle inside the surface features compared to time spent in the main channel (outside surface features) at different Reynolds number. The study was done using computational fluid dynamics tool and the tool used was Fluent V 6.1.22.

[0389] The details of the channel dimensions and surface features are shown in FIG. 3*b* (SFG-1) and described in previous examples. From the point of entrance, the first 3.81 mm section of the channel was without any surface features on any of the walls. The channel cross-section is rectangular in shape and the width and the gap of the channel were 4.57 mm and 1.02 mm. The next 27.94 mm of length contains surface features on the walls with width 4.57 mm and is referred as "surface feature section". The gap of main channel in this section was same as inlet section and was 1.02 mm. The last 5.08 mm length is the exit section and did not have any surface features on any of the walls.

[0390] The mesh for CFD model was built using Gambit V2.2.30. The model was built in a way that the gap of the channel (1.02 mm dimension) was in X-direction, length of the channel (36.83 mm dimension) was in Y direction and width of the channel (4.06 mm dimension) was in Z direction. The X-coordinates of the model varied from (1.53 mm, 0, 0) to (2.95 mm, 0, 0). The Y-coordinates of the model varied from (0, 0, 0) to (0, 36.83 mm, 0). The Z-coordinates of the model varied from (0, 0, −4.57 mm) to (0, 0, 0). FIG. **4** shows the X, Y and Z directions and their co-ordinates.

[0391] The mesh for the computational fluid analysis was developed in Gambit. The total number of cells was 131106, total number of faces was 542409 and total number of nodes was 177006. The mesh was generated to keep it a regular mesh as much as possible. The fluid considered for has following properties and operating conditions:

[0392] i. Viscosity=1.28×10^{-5} kg/m/s

[0393] j. Thermal conductivity=0.087 W/m/K

[0394] k. Specific heat=2768.03 J/kg/K

[0395] l. Density=Using ideal gas law

[0396] m. Molecular weight=17.49 g/mol

[0397] n. Molecular diffusivity=1×10^{-5} m^2/s

The inlet face was divided into four equal quadrants as shown in FIG. **4**. Each zone was assigned different name, however the thermo-physical properties of each zone were same. So zone A is defined as zone with concentration of A=1, B, C, D=0, zone B is defined as zone with concentration of B=1, A, C and D=0 and so on. The molecular diffusivity between four zones was 1×10^{-5} m^2/s. The Reynolds number was calculated as

$$Re = \frac{\rho v D}{\mu}$$

where

[0398] ρ=density of fluid, kg/m^3

[0399] v=Average velocity of fluid at inlet, m/s

[0400] D=Hydraulic diameter of channel, m

[0401] μ=Viscosity of fluid, kg/m/s

Three cases were considered with inlet Reynolds number=10, 100, 1000. The boundary conditions for each case are listed below:

[0402] Operating pressure=2379 kPa

[0403] Outlet pressure=0 psig

[0404] Inlet velocity=0.467 m/s for Re=1000, 0.0467 m/s for Re=10 and 0.00467 m/s for Re=10

[0405] Inlet temperature=300 K

[0406] Wall temperature=350 K

[0407] Zone A mass fraction

[0408] A=1

[0409] B=0

[0410] C=0

[0411] D=0

[0412] Zone B mass fraction

[0413] A=0

[0414] B=1

[0415] C=0

[0416] D=0

[0417] Zone C mass fraction

[0418] A=0

[0419] B=0

[0420] C=1

[0421] D=0

[0422] Zone D mass fraction

[0423] A=0

[0424] B=0

[0425] C=0

[0426] D=1

Model Chosen

K-Omega model (SST type) was chosen for CFD analysis. The values of model constants were default values provided by fluent 6.0. The coefficients of turbulence model were: Alpha*_inf=1; Alpha_inf=