Computational Fluid Dynamics of Capillary Sample Flow

Phillip G. Allen, Dr. Timothy C. Corcoran
California State Polytechnic University, Pomona

Goals

- Improve the sample detection sensitivity of the Supercontinuum Rapid Excitation-Emission Matrix (ScREEM) Laser Fluorescence Apparatus through Computational Fluid Dynamics (CFD) analysis of the sample flow concentration.
- Maximize sample concentration at the fluorescing zone by varying sample flow rate and sheath flow rate.
- Better understand flow patterns near fluorescing zone.
- Optimum detection zone will be determined and evaluated after considering physical constraints of the assembly and the varying sample concentration along the length of the sample volume.

Method

SolidWorks Flow Simulation is a Computational Fluid Dynamics (CFD) software which solves the Navier-Stokes Equations using an iterative technique. The Navier-Stokes Equations are the governing equations for continuous fluids. The CFD technique discretizes the region of interest into Fluid Cells, Particular Cells, and Solid Cells. Flow parameters are changed iteratively until a predefined goal is reached. The goals are convergence criteria based on desired flow parameters such as velocity. For example, the goal criteria may be a change in velocity of less than 5% of the current velocity value between iterations. Iteration continues until all convergence goals are met.

Navier-Stokes Equations for Incompressible Fluids:

\[ \frac{\partial V}{\partial t} + (V \cdot \nabla) V = -\frac{1}{\rho} \nabla p + \nu \nabla^2 V \]

Where each term is an acceleration, V is the fluid velocity vector, t is time, \( \frac{\partial}{\partial t} \) is acceleration vector due to gravity, \( \rho \) is density, and \( \nu \) is fluid viscosity.

Assumptions

Several modeling parameters have to be assumed in order to run CFD. These parameters strongly influence the results of the simulations and must be consistent with the actual model conditions.

- Internal Flow Analysis
- Incompressible Fluid
- Mesh size used throughout Analysis
- 30,188 Fluid Cells, 29,404 Partial Cells
- Flow Model Geometry – See Figure 3.
- All walls considered Smooth and Fully Insulated
- Initial Conditions
  - Fluid Filling reservoir - Volume Ratio
  - 1 Sheath Fluid: 0 Sample Fluid
  - Standard Temperature
- Boundary Conditions
  - Sample Volumetric Flow Rate Varied
  - Sheath Volumetric Flow Rate Fixed at 0.3125 mm/s
- Constant Flow Rate
- Atmospheric Pressure at flow outlet
- Sample and Sheath Flow Modeled as Water
- Symmetry Over Front Plane B-B

Figure 4. Excitation Emission Matrix (EEM) data output from high-speed CCD

Results & Discussion

SolidWorks Flow Simulation CFD analyses were created for 6 different capillary sample flow rates ranging from .3125 mm/s down to .003125 mm/s as shown below in Figure 5 with respective velocities of 47 mm/s down to 17 mm/s at the detection zone.

- Sample Concentration Gradients for Multiple Flow Rates
- CFD Plots of Streamlines & Velocity Contours

Conclusion

The CFD simulations produced by SolidWorks Flow Simulation helped clarify flow patterns acting in the detection region. A tradeoff between focused diameter and sample concentration was observed. Additionally, a major source of sample concentration reduction was determined to be caused by the squared edges of the modeled capillary.

Future Work:
- Validation of results through physical testing
- CFD of Capillary with rounded or channeled tip
- Validation of Rounded Capillary by etching Capillary and running physical tests

Acknowledgements

We would like to thank the National Science Foundation, Instrument Development in Biological Research, and California State Polytechnic University and the Center for Macromolecular Modeling & Materials Design for financially supporting this work. TC wishes to thank the Provost Teacher-Scholar program for additional support.

A special thanks to the ScREEM Team who contributed to many components of this research: Christopher M. Dettmar, Jacob B. Balthazor, Ivonne P. de la Torre, Alisha J. Lewis, Neda F. Nouri Nassr. Finally, thank you Dr. Hossein Ahmadzadeh for inspiring this in-depth analysis.

References

- Dassault Systemes, SolidWorks Flow Simulation.