Computational Fluid Dynamic Modeling of Geothermal Membrane Distillation

C. Rivera, C. Sanchez, J. Paiz, M. Troyer, F. Huang, Ph.D.

1. Department of Environmental Engineering, New Mexico Institute of Mining and Technology, Socorro, NM, USA

Introduction

A 3-D COMSOL model was established to investigate a CFD model that can be used to predict the performance of full-scale direct contact membrane distillation (DCMD) modules using hollow fiber membrane (HFM) and to optimize the operating conditions for water production and energy consumption. Membrane distillation is a separation process that relies on vapor pressure gradients to drive the production of purified water across a hydrophobic membrane. In direct contact membrane distillation, both the hot water and cold permeate are in direct contact with the membrane. The temperature difference across the hydrophobic membrane induces the gradient of water vapor pressure resulting in desalination and water permeation through diffusion.

Figure 1: Water flow across the membrane

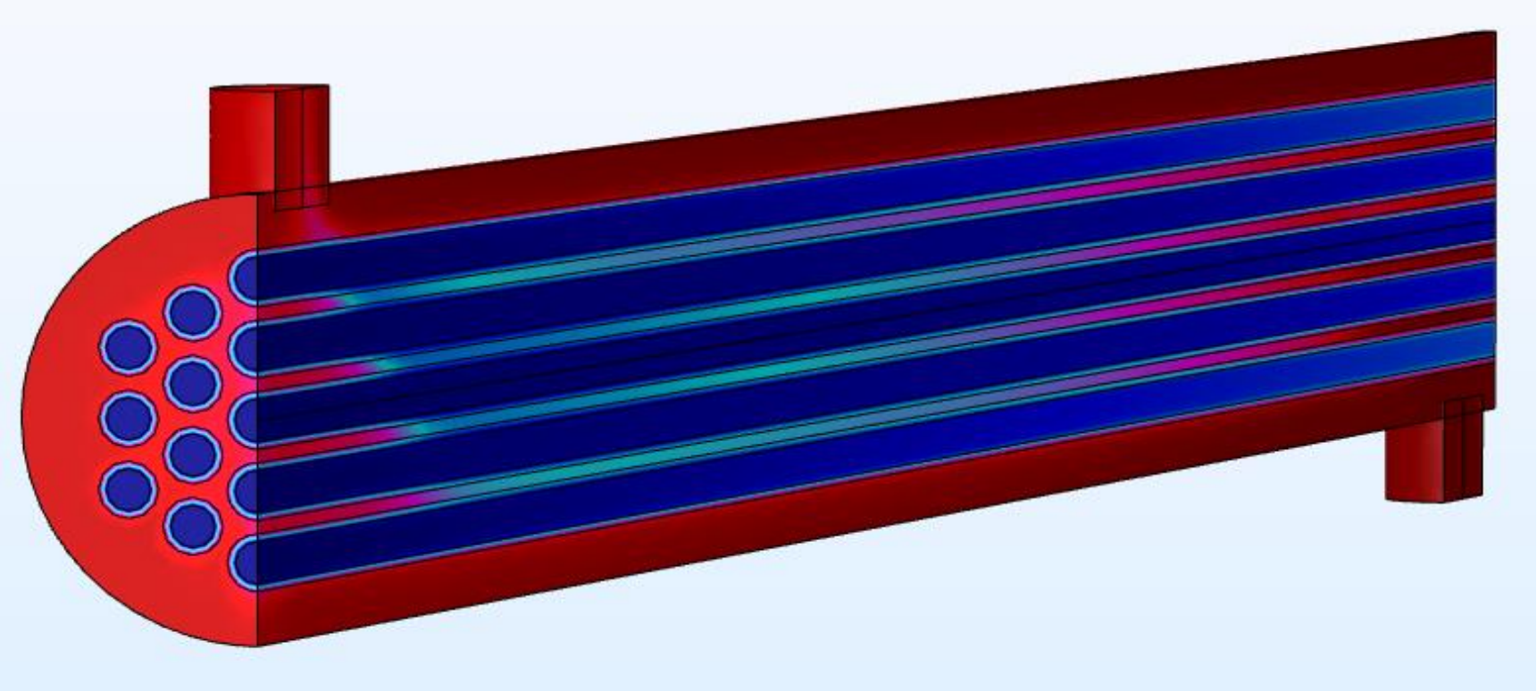

Figure 2: Cross-sectional of CFD module

Computational Methods

COMSOL Multiphysics® was used to couple flow, heat transfer, and the transport of diluted species to simulate the conditions for bench-scale DCMD modules. Membrane properties, such as pore size, porosity, thermal conductivity, etc., were measured and used as the inputs for the CFD simulation. Dusty-Gas Model was employed to describe the trans-membrane mass transfer of water vapor driven by Knudsen-molecular transition diffusion. Bench-scale experiments that employed the same module geometries and baseline operating conditions, were used for model validation.

Transport Type	Transport Equation
Continuity	$\nabla \cdot (\rho \boldsymbol{u}) = 0$
Momentum	$\rho(\boldsymbol{u} \cdot \nabla \boldsymbol{u}) = \nabla \cdot [-p\boldsymbol{I} + \mu(\nabla \boldsymbol{u} + (\nabla)^T)] + \boldsymbol{F}$ $\rho\, \nabla \cdot \boldsymbol{u} = 0$
Energy in fluid	$\rho C_p \boldsymbol{u} \cdot \nabla T + \nabla \cdot \boldsymbol{q} = Q$ $\boldsymbol{q} = -k\nabla T$
Energy through membrane	$\nabla \cdot \boldsymbol{q} = Q$ $\boldsymbol{q} = -k\nabla T$
Mass	$\nabla \cdot (-D_i \nabla c_i) = R_i$ $\boldsymbol{N}_i = -D_i \nabla c_i$

Table 1. CFD module parameters

Parameter	Value
Module Length (L)	6.6 cm
Module inner diameter (D_i)	1.27 cm
Inlet/Outlet inner diameter (D_o)	0.32 cm
Inlet z- position (L_I)	0.2 cm
Outlet z-position (L_O)	6.4 cm
Inlet φ-position (φ_I)	0°
Outlet φ –position (φ_O)	180°

Table 2. CFD module parameters

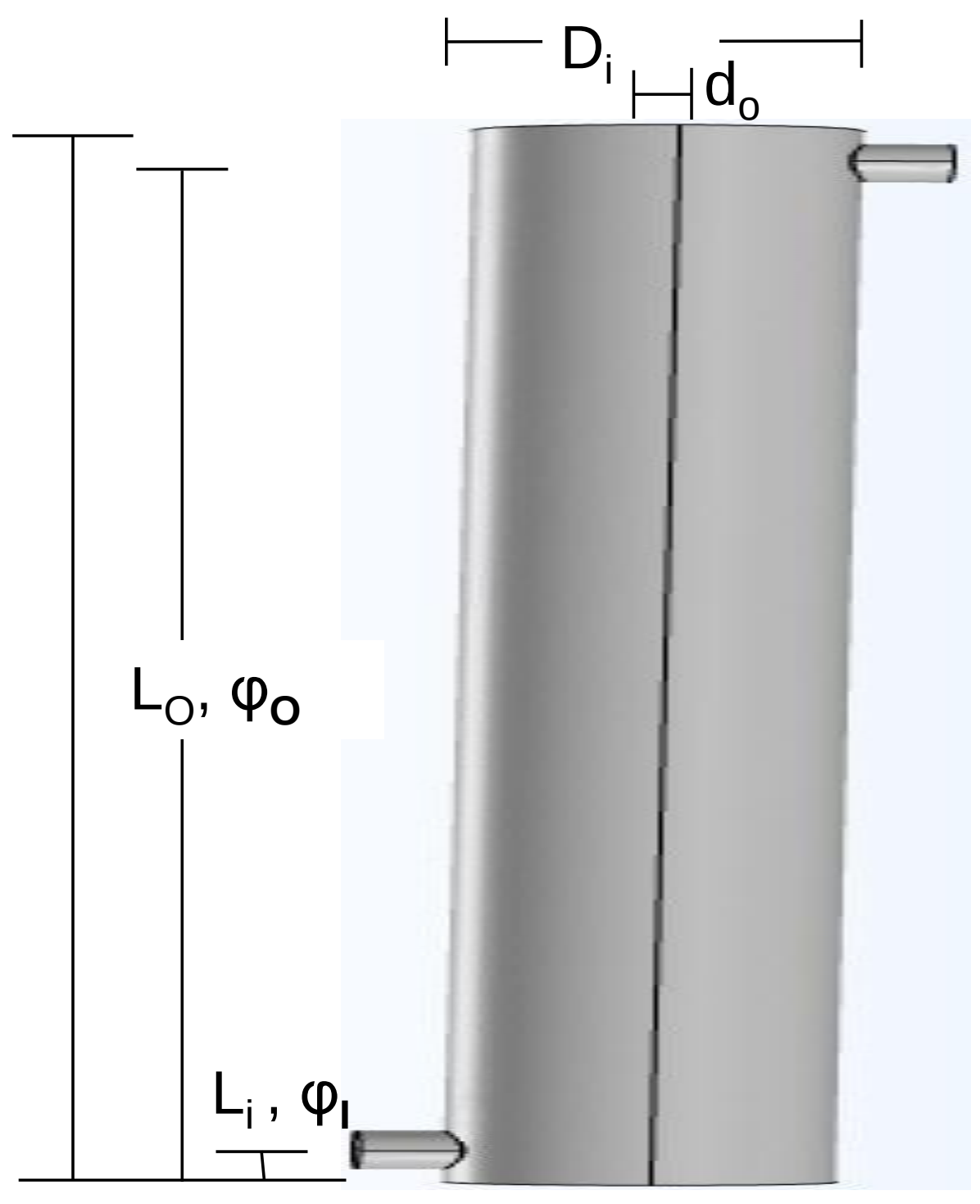

Figure 3. CFD module

Results

The CFD simulations and bench-scale tests were performed varying the fiber packing configuration, packing density, and fluid flow rates. Correlations between these parameters and water flux were made.

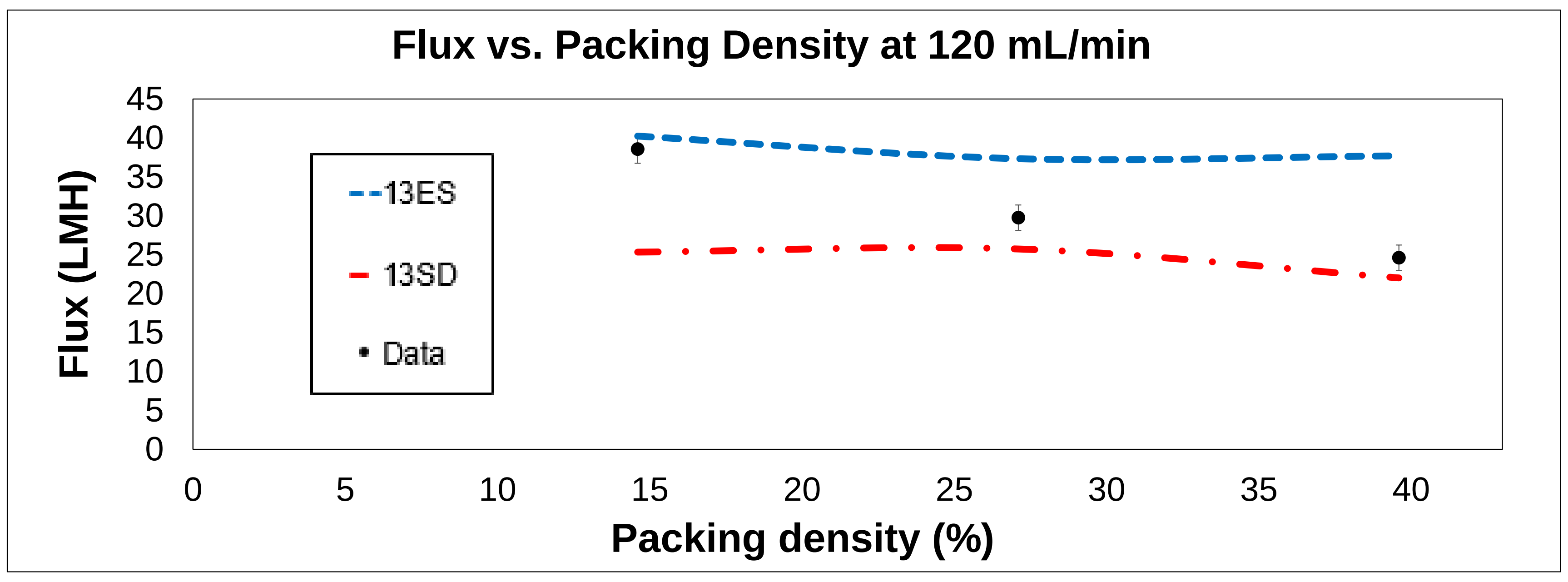

Figure 4. Flux vs. packing density for 13 fibers/module simulations and lab tests.

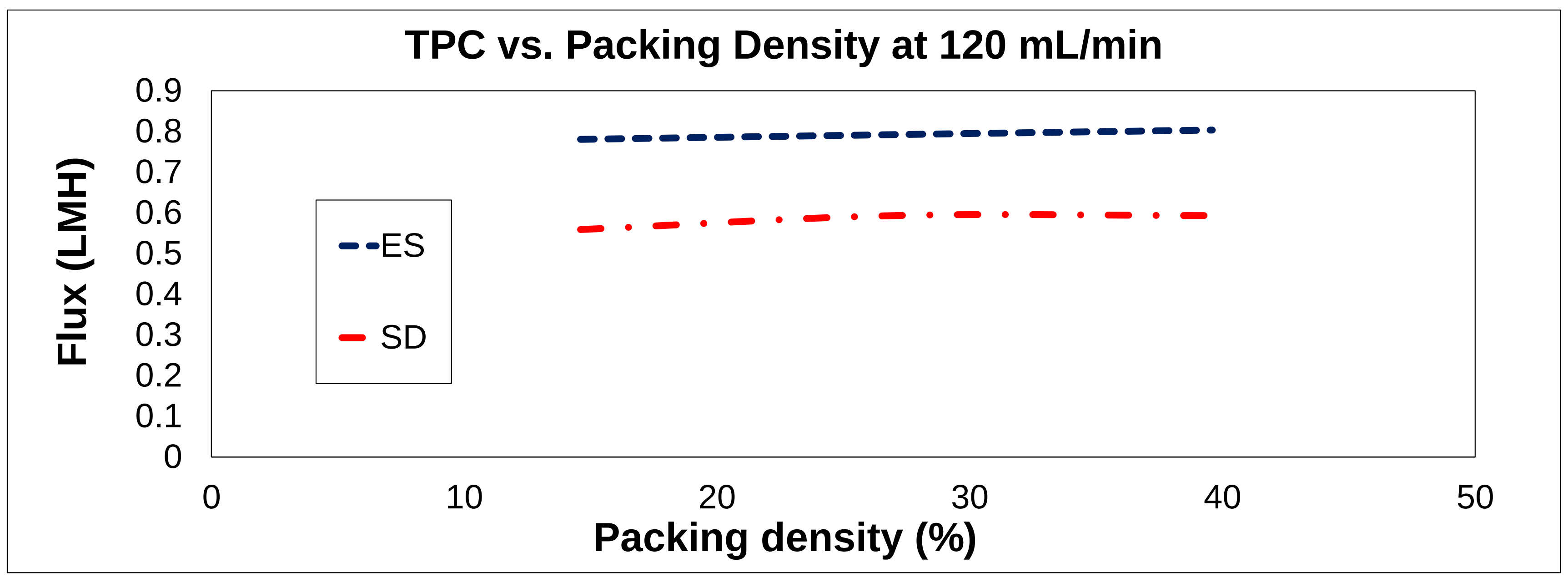

Figure 4. TPC vs. Flow rate for simulated modules containing 13 fibers/module.

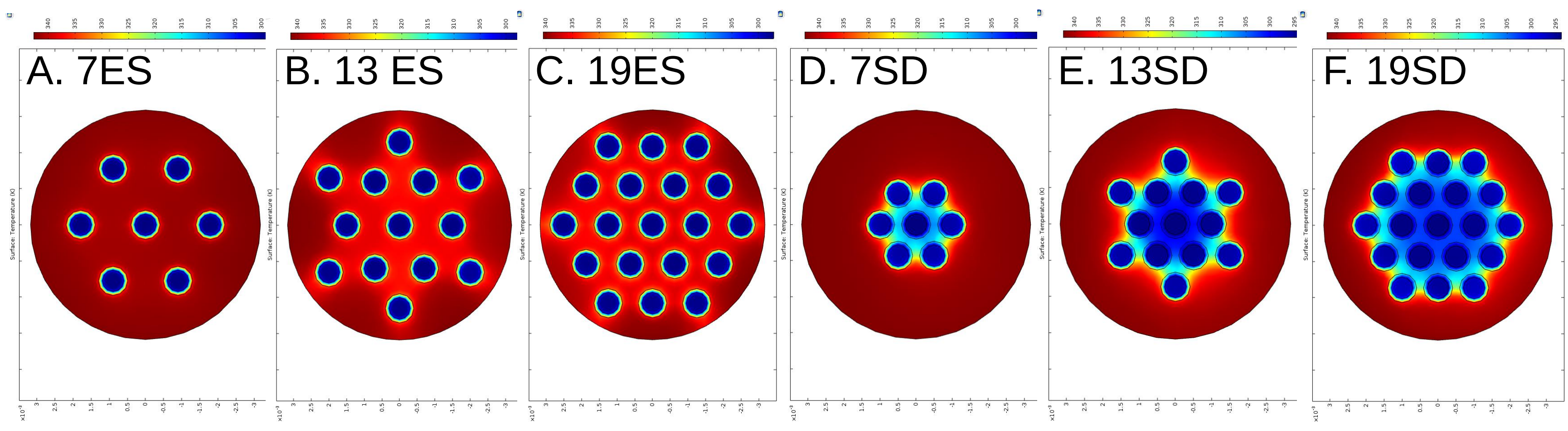

Figure 5. Effect of flow distribution on temperature in module, (A) 7 fibers equally spaced (ES), (B) 13 fibers ES, (C) 19 fibers ES, (D) 7 fibers at set distance(SD), (E) 9 fibers at SD, (F) 19 fibers at SD.

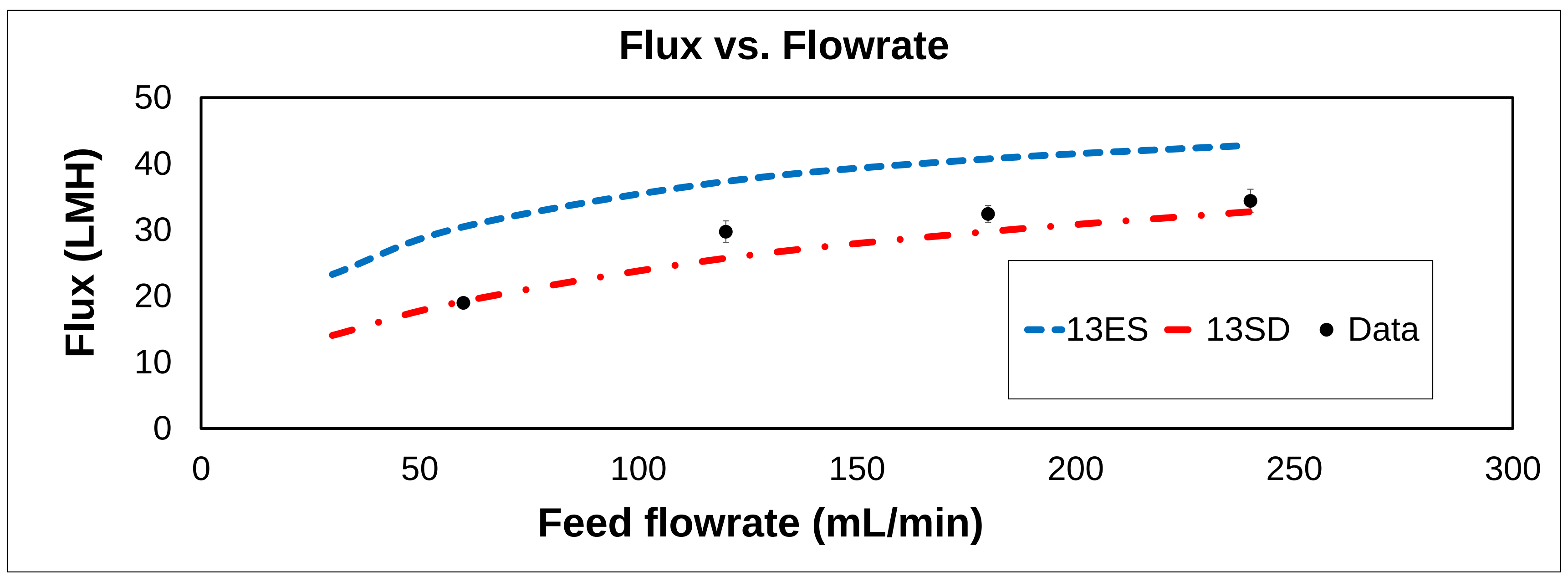

Figure 6. TPC vs. Flow rate for simulated modules containing 13 fibers/module.

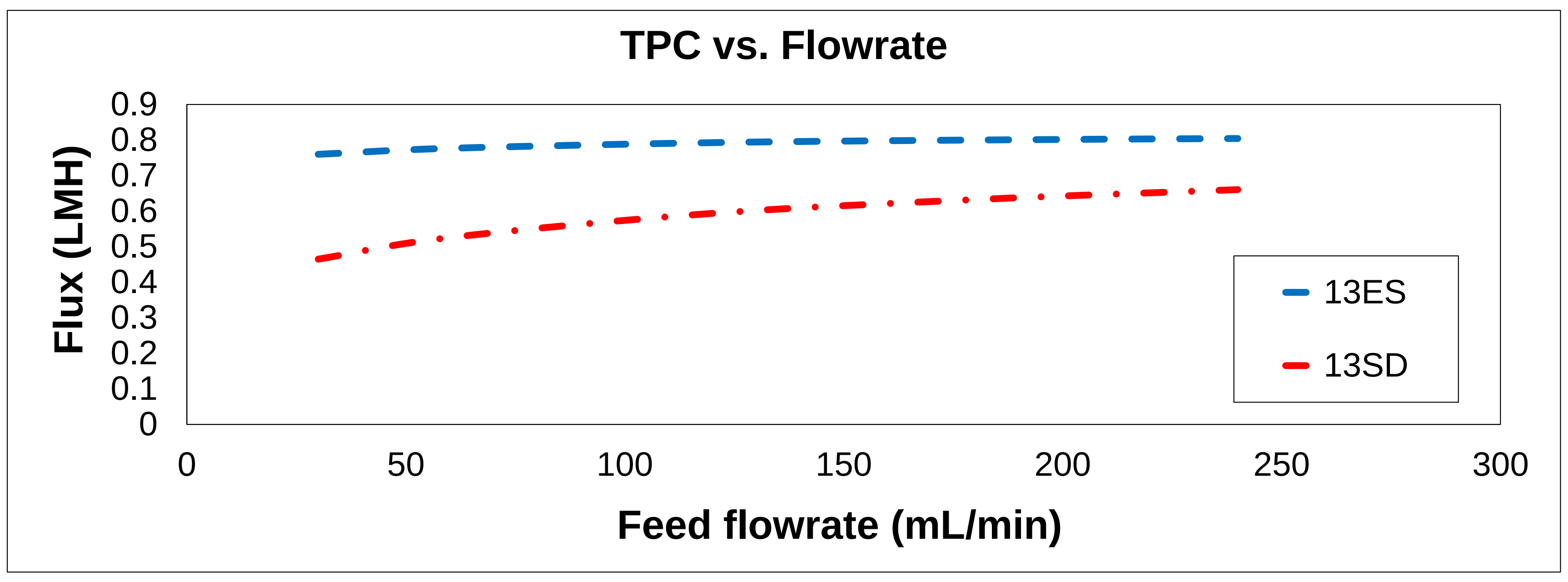

Figure 7. Flux vs. feed flow rate for 13 fibers/module, simulations and lab tests.

Conclusion

Performance of the HFM in terms of water flux depends heavily on fiber packing configuration, packing density, and fluid flow rates. The main findings of the study were:

- Water flux is a strong function of the packing configuration. It doesn't follow a continuous trend with increasing packing density, and is highly impacted by channeling.
- For a set packing density, water flux increases with increasing flow rate. The increase of flux due to flow rate gradually levels out for each packing density and packing configuration. There is a direct correlation between the flow rate and temperature polarization coefficient (TPC). There is a direct correlation between the flow rate and bulk temperature.
- High packing density (≥50%) is necessary to predict the water flux of an actual module using CFD simulations.
- Significance of the entrance and exit effects should be evaluated for modeling large scale modules for water flux predictions. CFD simulations of lab scale modules with and without these effect showed discrepancy of up to 25%.
- Single fiber CFD simulations cannot predict module performance accurately due to difference in velocity distribution and evaporative surface area.

References:

1. Andrjesdóttir, Ó., Ong, C. L., Nabavi, M., Paredes, S., Khalil, A. S. G., Michel, B., Runyu, M. (1997). Membrane distillation. *Journal of Membrane Science*, *124*, 1–25.

Acknowledgements

This work was supported by National Science Foundation award #IIA-1301346 to NM EPSCoR and its Osmotic Power Research Group.

Excerpt from the Proceedings of the 2017 COMSOL Conference in Boston