Research Report

Investigation of the Interface Conditions Between Nano-Porous Matrix and Fractures of Unconventional Reservoirs

Ali Albinali, Colorado School of Mines

Problem Statement

- Fluid flow from matrix to fractures is commonly addressed by two models:
  - Kazemi de Swaan-O (Transient).
  - Warren-Root (pseudo-steady state).

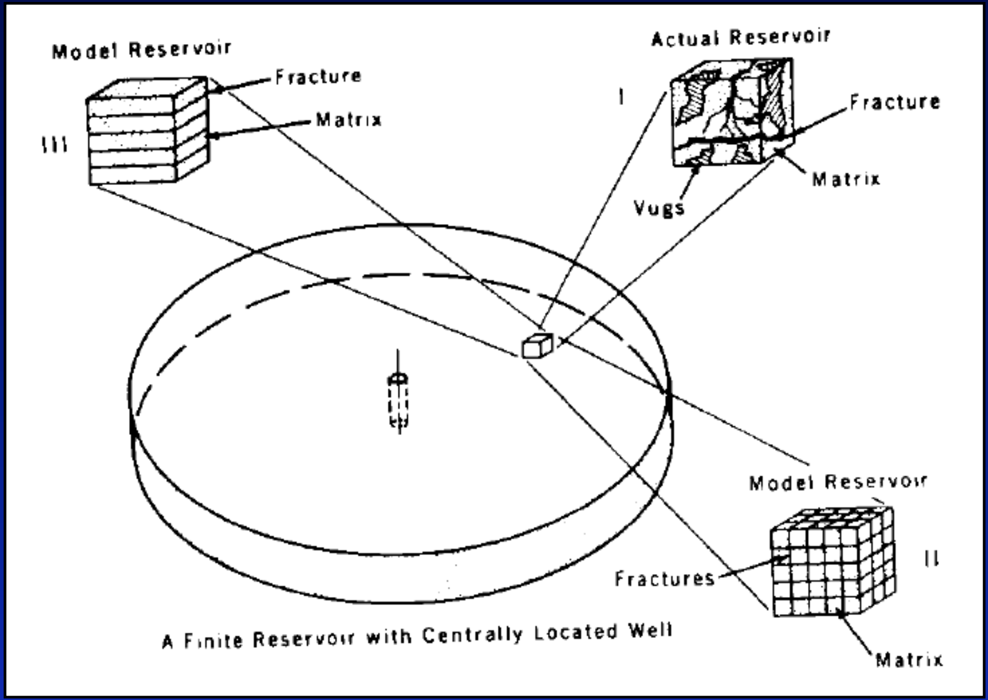


Figure 1: Dual Porosity Idealizations (Kazemi 1969)

Problem Statement

- Transient model is more suitable for tight matrix. One dimensional flow for both regions.

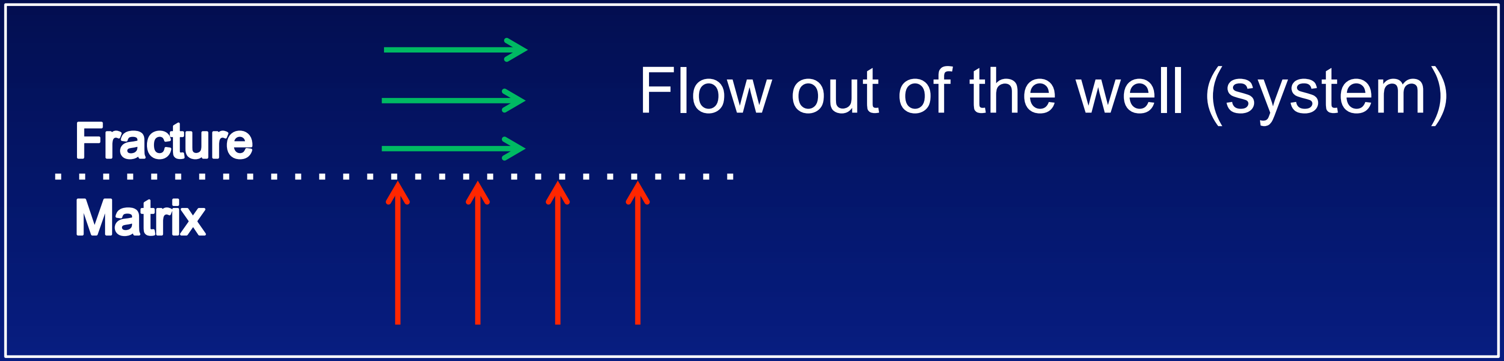


Figure 2: Flow From Matrix to Fracture in Transient Models

- PSS model is zero-dimensional (no matrix shape is required) and does not include the details (direction) of flow from matrix to fracture (matrix provide storage, fracture = provide conductivity).

Problem Statement

- We are interested in investigating the possible contribution of the tangential velocity component (v_x) to the mass flux going into the fracture (drag forces).

Suggested Study

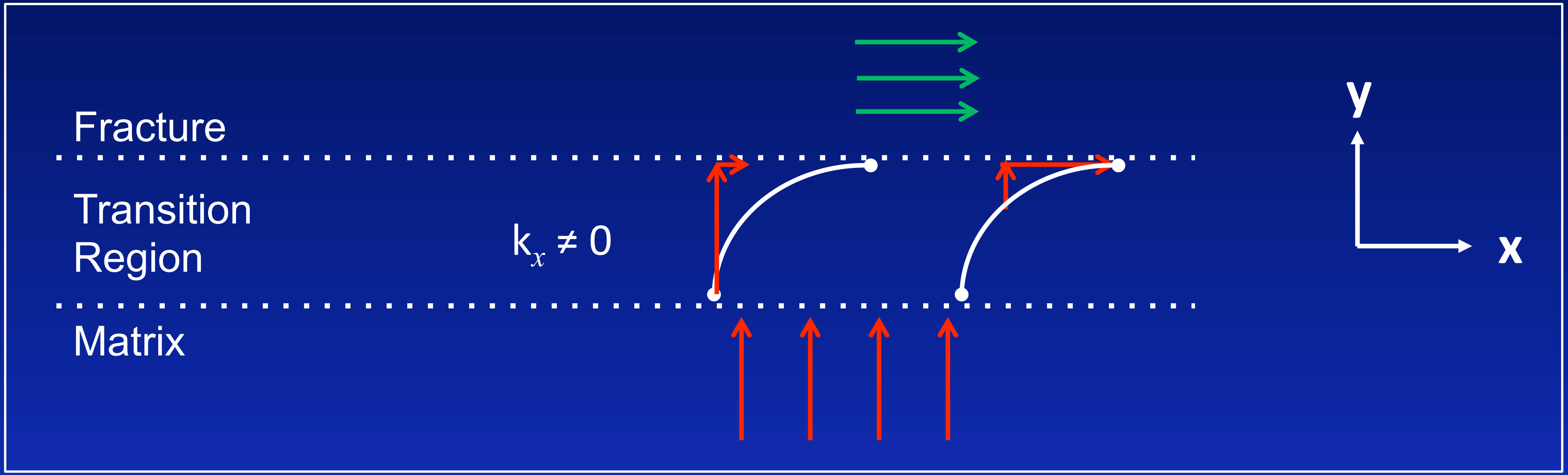


Figure 3: Investigated Behavior for Flow From Matrix to Fracture

Objectives

- Investigate the details of matrix-to-fracture fluid transfer
- Investigate the effect of large velocity contrast between fracture and matrix media
- Demonstrate the effects of confinement on matrix side
- Extend the results to multi-phase flow
- Define a methodology to incorporate the results to large-scale flow models

Importance

- Beavers & Joseph 1967.
- Le Bars and Worster 2006.

You may include some explanation for these references

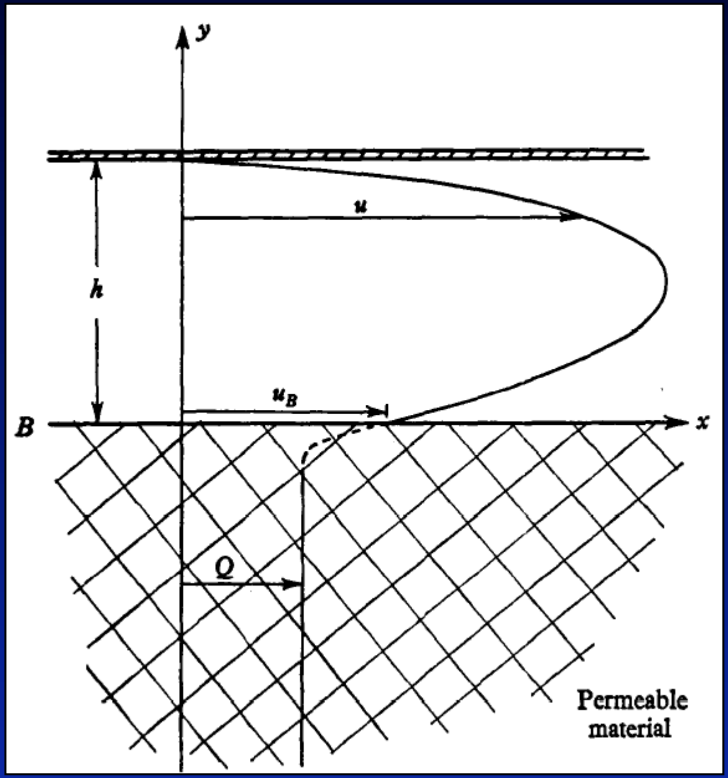


Figure 4: Linear Flow Velocity Profile (Beavers & Joseph 1967)

Importance

- We are looking at the sharp contrast in velocities and proper coupling conditions are necessary (Poiseuille-Darcy or Brinkman-Darcy).

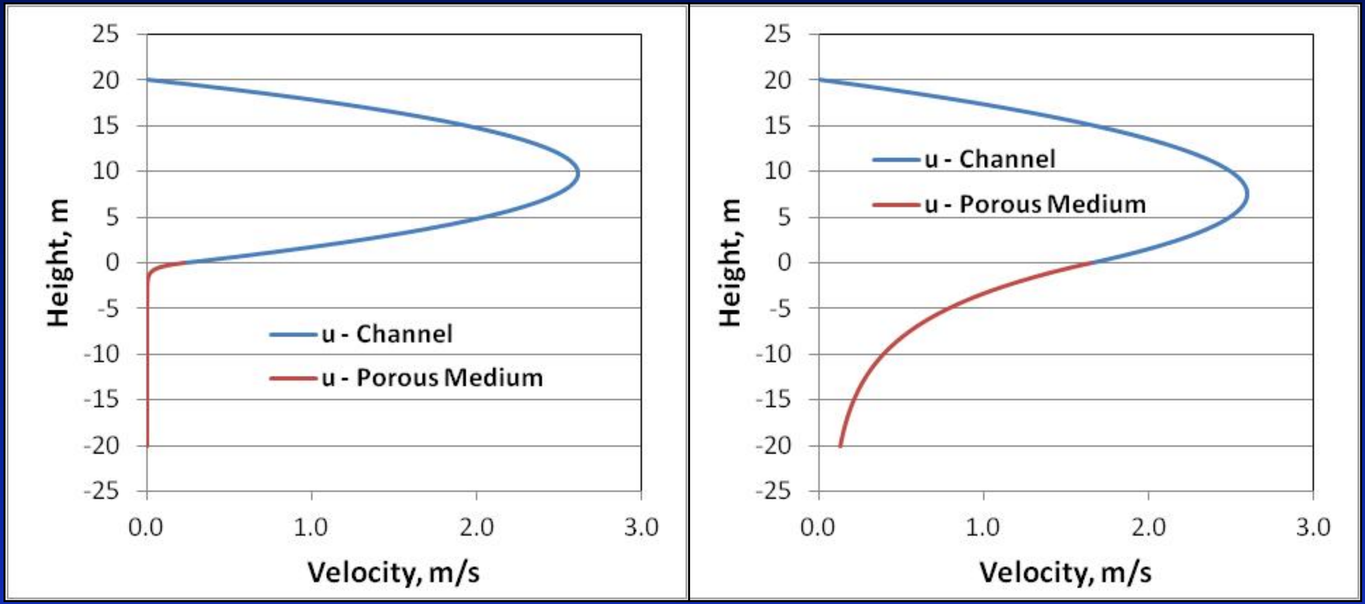


Figure 5: Velocity Profile Under Different Conditions

Importance

- Modeling using LBM and DEM (paper submitted for the 5th Biot Conference on Poromechanics)
- Increase in permeability by 18.8% (drag forces?).

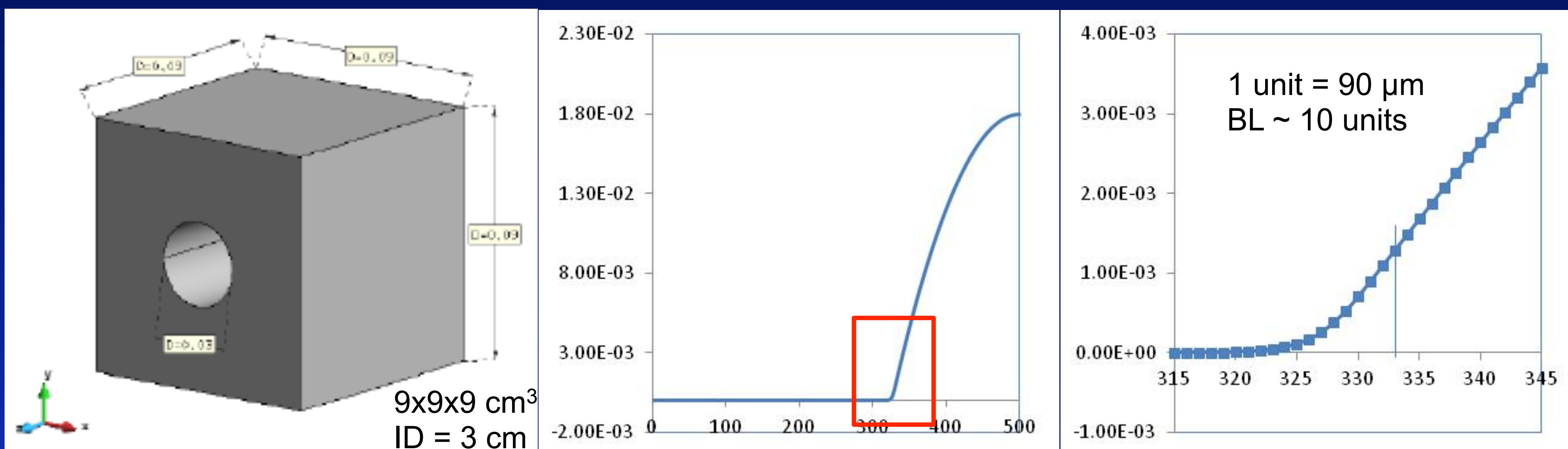


Figure 6: Modeling with LBM

Approach

- Numerical modeling of a 2D grid consisting of 1) low permeability region and 2) high permeability region:
  - Use FD and Darcy law.
  - Scenario 1: $k_{mx} \neq 0$ (under investigation)
  - Scenario 2: $k_{mx} = 0$ (current approach)
  - Compare normal and tangential velocity components → investigate flux.

Modeling

- Constructing the model:
  - Region1 & 2.
  - Initial pressure.
  - Closed boundary (no flow) ——
  - Permeable interface – –
  - Producing well in high permeability region.

Table 1: Input Data Used for Initial Runs

Region 1	Region 2
Permeability = 1 μD	Permeability = 100 mD
Porosity = 5%	Porosity = 35%
Fluid viscosity = 0.5 cP	
Initial Pressure = 7,000 psi	
Producing BHP = 3,000 psi	

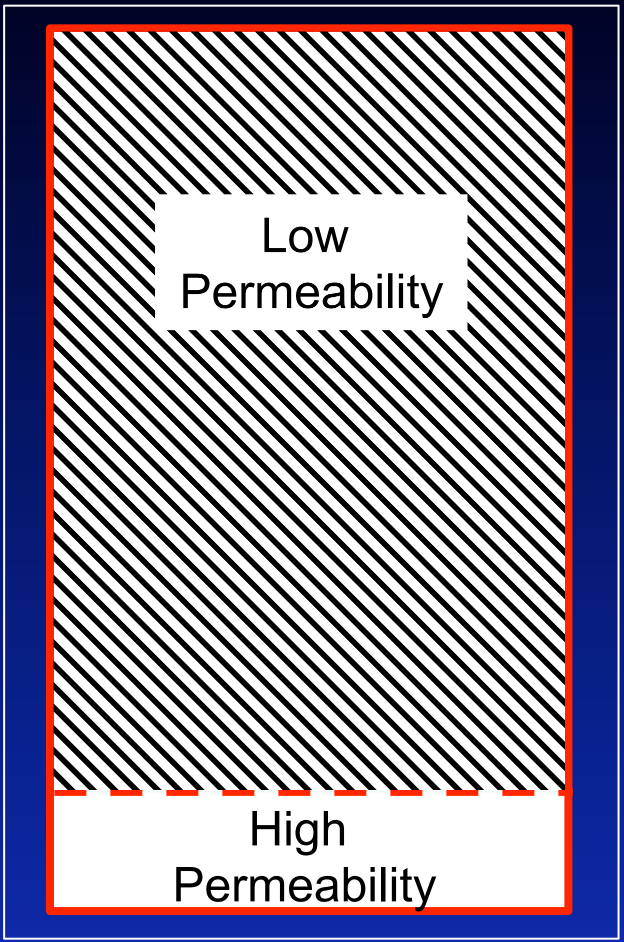


Figure 7: Constructing the Problem

Modeling & Results: 1st Stage

- Modeling I: Results Indicated that for current approach ($k_{mx} = 0$) mass flux is greater, but by insignificant amount (<1%).

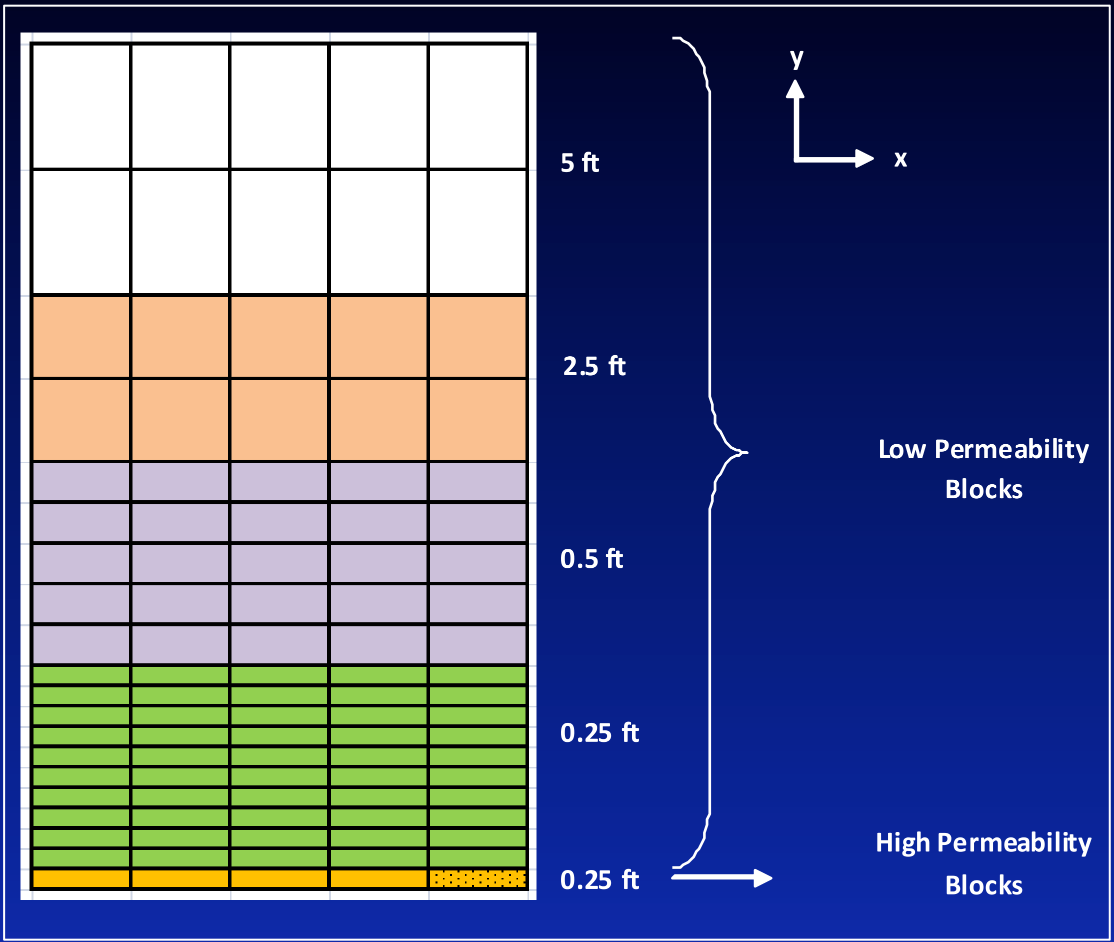


Figure 8: of Initial Modeling Grid

Modeling & Results: 2nd Stage

- Modeling II:
- More refined grid.
- Same conclusion as 1st Stage; scenario 2 (k_{mx} = 0) mass flux is greater, but by insignificant amount (<1%).
- Based on these conclusions, our decision could be STOP.

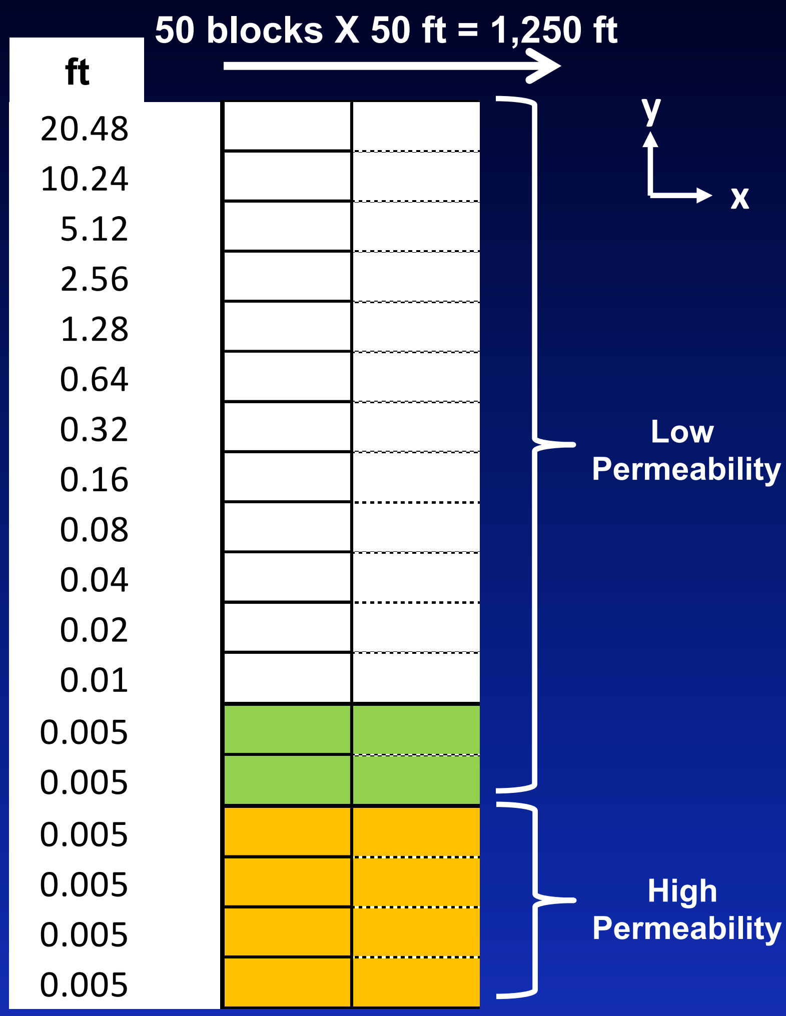


Figure 9: Refined Grid 2nd Stage

Modeling & Results: 2nd Stage

- The investigated transition zone is very thin (small scale).
- Do we need different set of tools (math), scale or geometry to study the problem?
- We are not satisfied with previous conclusions; velocity contribution to flux can be calculated.

Modeling & Results: 3rd Stage

- COMSOL 4.3a: Multi-physics platform using Fluid Flow in Porous Media Module.
- Constructing the model:
- Need to look at a smaller scale

→ Model size: 1”X1”

- Apply similar flow/boundary conditions and input data.

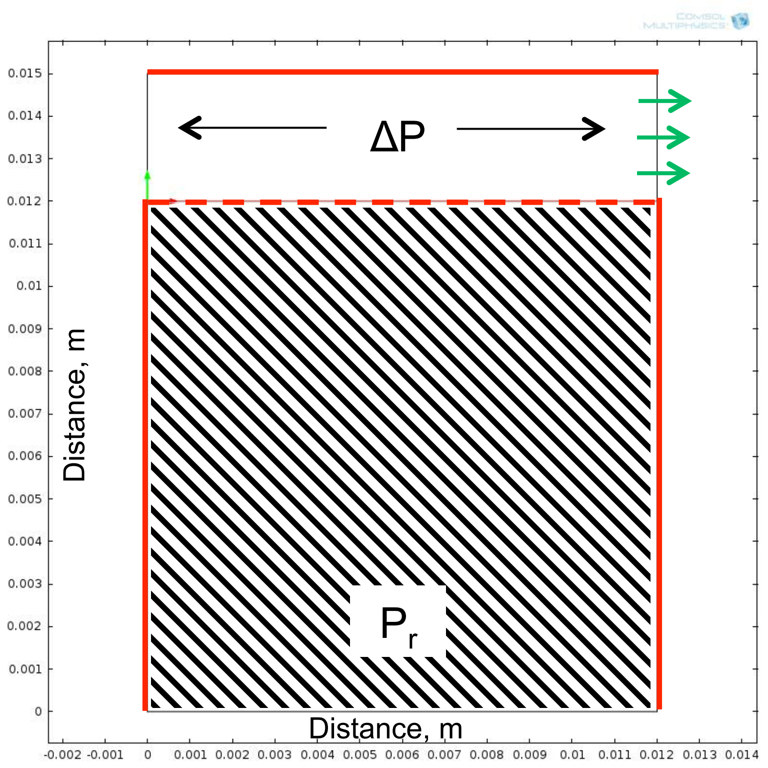


Figure 10: COMSOL Grid

Closed boundary (no flow)

Permeable interface

Modeling & Results: 3rd Stage

- Generate a very dense/fine mesh.
- Results are similar to former modeling cases.

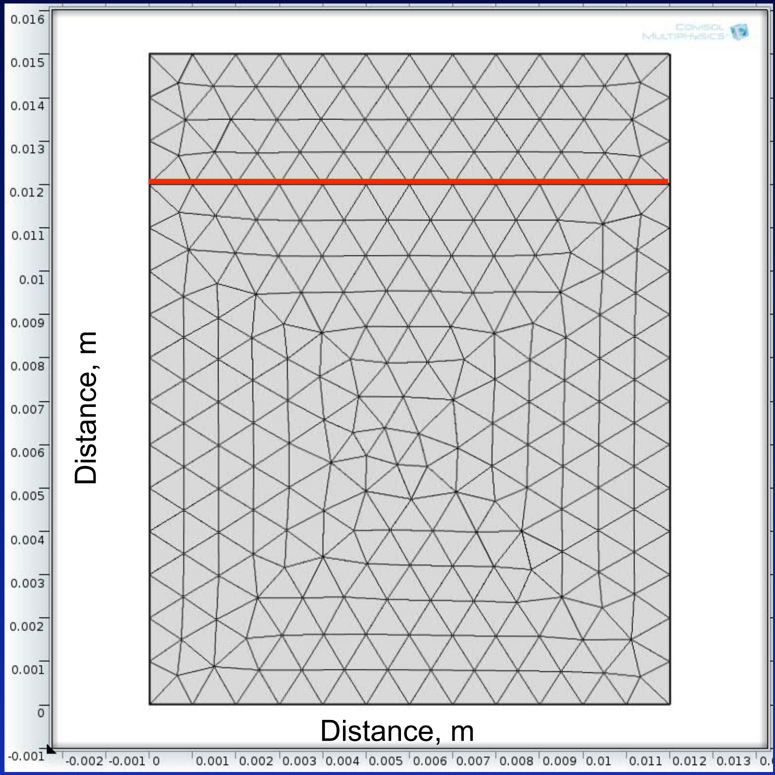


Figure 11: Example of Normal Mesh

Modeling & Results

- Are we doing something wrong?
- Validation run using the same properties (k and ϕ), ~~low and high conductivity regions~~; one single domain.
- Is our suggestion ($k_{mx} \neq 0$) and expectations (drag forces) valid?

Modeling & Results

- Results support our suggestion; average difference of 11.75%.

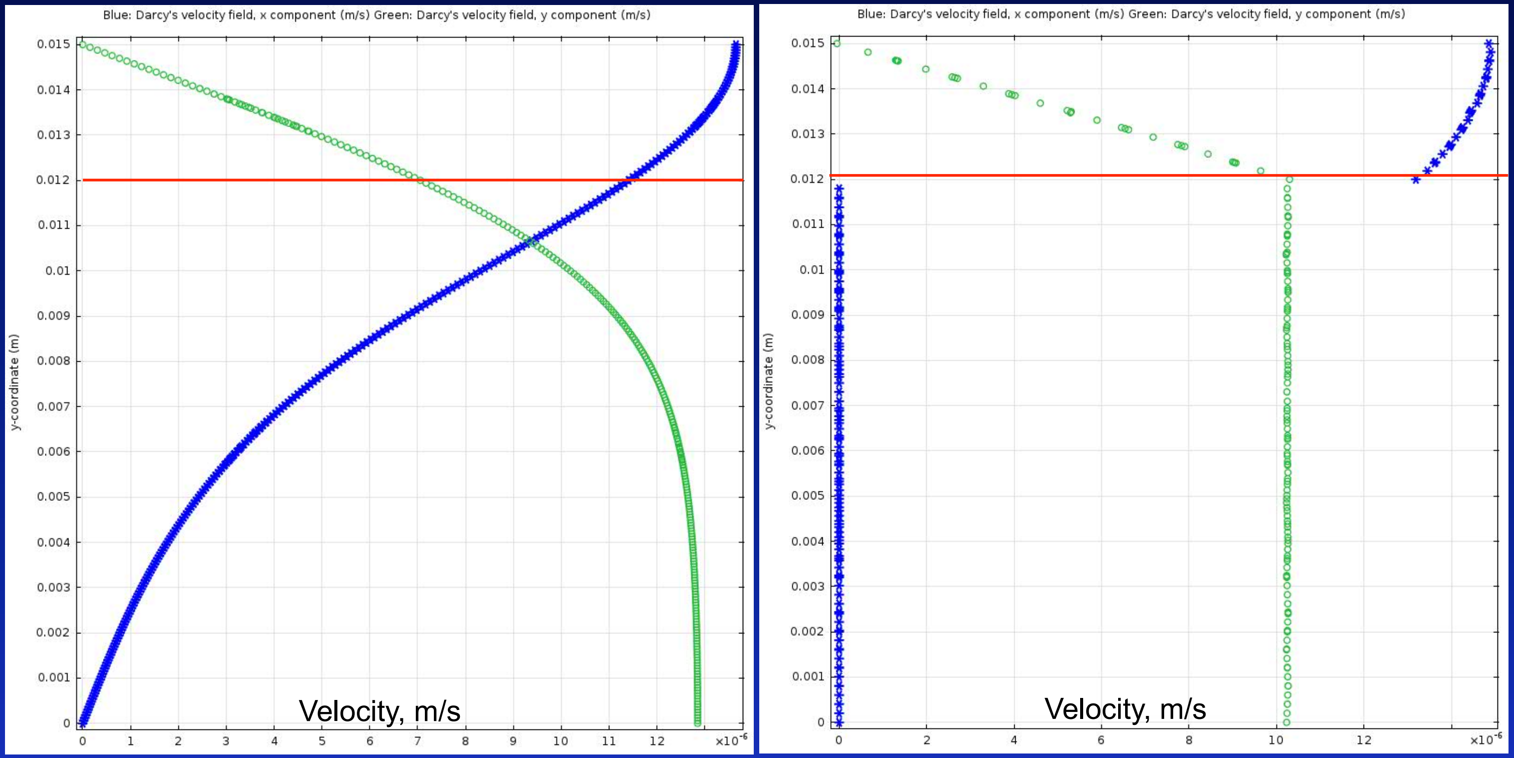
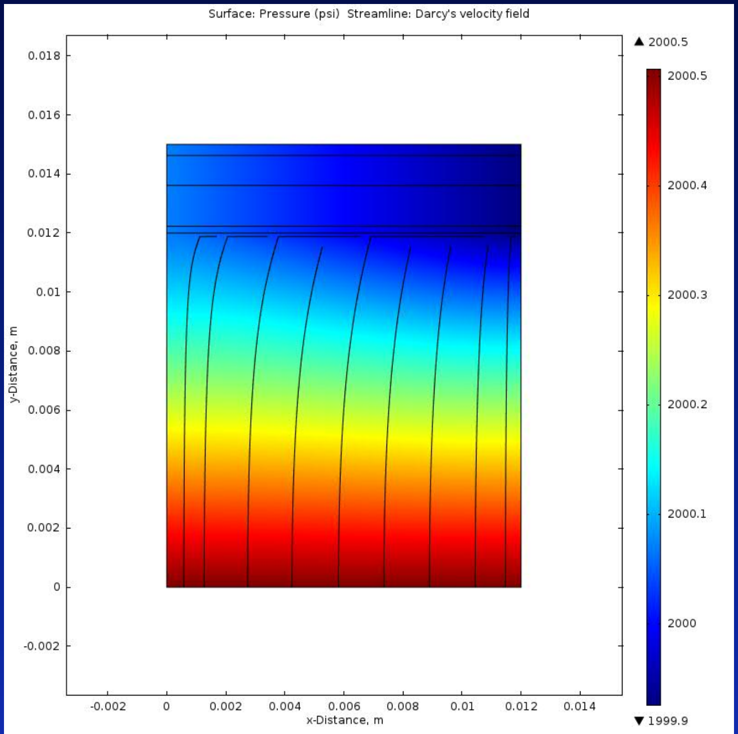


Figure 12: Validation Run Velocity Components

Modeling & Results

- Boundary conditions can influence the flow behavior.



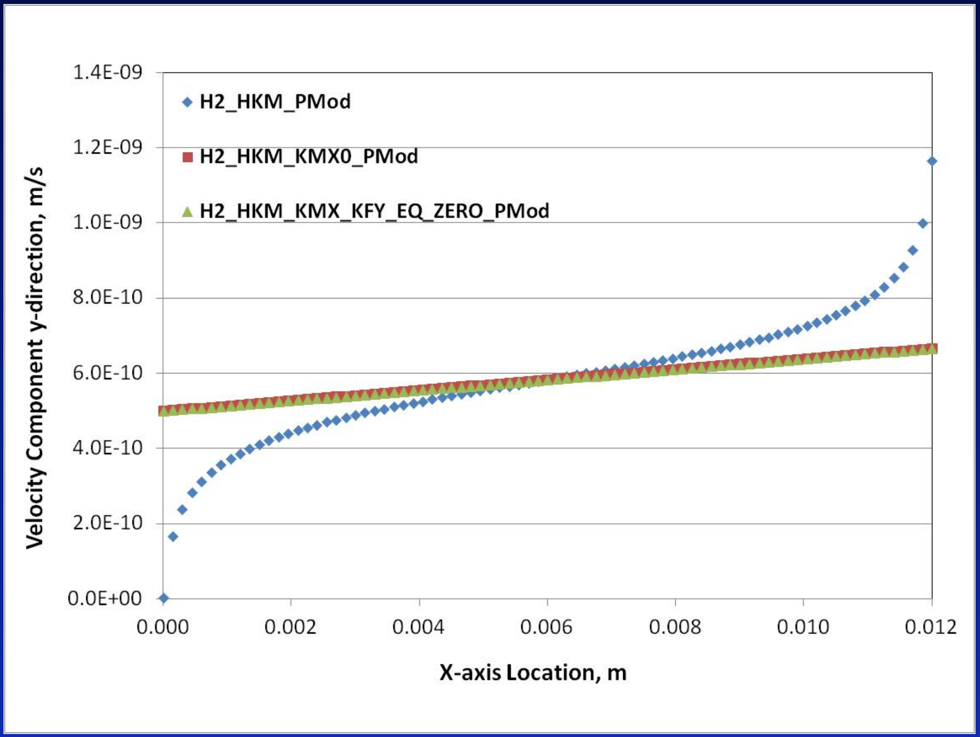


Figure 13: Pressure and Velocity Plots; Various Setup

What's Next?

- We might have possible contribution from tangential velocity.
- Continue investigation:
  - Contrast of properties (k and ϕ) is one issue.
  - Try to justify different results and build appropriate model (boundary)
  - Is it possible to generalize the results in the form of a conventional transfer function?
  - How can we incorporate the results into large-scale flow models

Acknowledgement

I would like to express my appreciation to:

- Dr. Hossein Kazemi
- Dr. Xiaolong Yin
- Johan Sundqvist, COMSOL Technical Support

Thank you