Solid Earth Discuss.,
https://doi.org/10.5194/se-2019-170-AC2, 2020

Solid Earth
Discussions
Open Access
EGU

Interactive comment on "Bilinear pressure diffusion and termination of bilinear flow in a vertically fractured well injecting at constant pressure" *by* Patricio-Ignacio Pérez D. et al.

Patricio-Ignacio Pérez D. et al.

ortiz.adrian@gmail.com

Received and published: 14 April 2020

Dear Mr. Tavassoli:

We thank you for your comment. We want to answer to the points claimed in your comment. First off, we regret that none of the statements made by you is substantiated or backed by any bibliography or extensive revision of the concerned scientific literature as we did in the manuscript.

Your first claim: "The problem statement is very simplified and the analytical and numerical solutions already exist."

Printer-friendly version

Discussion paper

We strongly disagree with your opinion about the already existence of analytical and numerical solutions for the specific problem, for example, of propagation of the isobars or finalization of the bilinear flow for wells injecting/producing at constant pressure into/from a fracture of variable length. The phrase “the problem statement is very simplified” is vague and lacks substantial support. We use a fit-for-purpose model and with the aim of investigating the behavior of isobars along a fracture with finite conductivity the model captures the main physical processes. If you take a look at Referee 1’s comments, you would see that he highlights the novelty of the results. For example:

A) In this paper, it is presented for the first time for injection/production at constant pressure the equation describing the movement of the isobars along the fracture during the bilinear flow regime (Eq. 16).

B) We propose two methodologies to quantitively identify the termination of bilinear flow under constant pressure conditions: (a) considering the transition of flow rate in the well and (b) considering the propagation of isobars P_N along the fracture (highlighted in section 3.2 “Termination of bilinear flow”). In this paper, by using the mentioned methodologies, for the first time expressions are presented for the termination time of bilinear flow when injecting/producing at constant pressure into/from the fracture. The criteria used to quantitatively identify the termination of bilinear flow are explained in detail in sections 3.2.1, 3.2.2, 3.2.3, and 3.2.4.

C) Although the expression for the reciprocal of the flow rate vs. the fourth root of time have already been presented by Guppy et al. (1981b), the proportionality factor obtained by us differs slightly from that presented by them, most likely due to a more robust numerical procedure. In that concern, our investigation may represent a more reliable approach to the theoretical one adopted by Guppy et al. (1981b).

Either in the Abstract or in the Highlights, or more intensively in the Conclusion section, the novelty of these results is emphasized. Based on the previously exposed, we encourage you to read the manuscript more carefully. To sum up, results are presented

in this paper that indeed no one had ever presented before for the case of constant pressure injection/production into/from a fractured reservoir, concerning the isobars behavior along a fracture and the termination of bilinear flow regime. Particularly in section 4.1 “Application to well testing problems”, the practical utility of Eq. (16) to estimate or restrict the fracture length is highlighted.

Your second claim: “Use of a reservoir simulator is recommended than COMSOL. Please check the literature on numerical solutions of reservoir simulations.”

We appreciate your recommendation. However, we do not see why other reservoir simulator should be better. Further, we do not see why other reservoir simulator should do better for the solution of the problem at hand. We want to strengthen the point that there is no such a thing as the all singing and all dancing reservoir simulator. They all have their advantages and disadvantages. We find the use of COMSOL Multiphysics® appropriate for the problem statement we are addressing. The numerical simulation software COMSOL Multiphysics® is widely used to address reservoir processes and it has been tested and validated in innumerable works published. The reservoir process simulator used in our work is COMSOL Multiphysics® as stated in line 115 in the version of the manuscript you read. However, additional information it has now been included in the manuscript about the numerical procedure adopted to look for the solution of the partial differential equations involved. This is now described in much more detail in section 2.4 “Description of the model setup”. This information includes the following “We ran the numerical simulations in the Subsurface Flow Module of COMSOL Multiphysics® software program. The space- and time-dependent balance equations, described in section 2.1, together with their initial and boundary conditions are numerically solved in the entire modeling domain employing the finite-element method (FEM) in a weak formulation. The discretization of the partial differential equations (PDEs) results in a large system of sparse linear algebraic equations, which are solved using the linear system solver MUMPS (MUltifrontal Massively Parallel Sparse direct Solver),

Printer-friendly version

Discussion paper

implemented in the finite element simulation software COMSOL Multiphysics®. Utilizing the Galerkin approach, Lagrange quadratic shape functions have been selected to solve the discretized diffusion equations for the pressure process variable. For the time discretization, a Backward Differentiation Formula (BDF, implicit method) of variable order has been chosen.". We also incorporated two important remarks concerning studies of mesh- and boundary condition-independency of the solution in the modeling domain we are most interested in. The first one "That way, boundary condition-independency of the solution has been guaranteed for in the computational subdomain of most interest" (included in the manuscript in the corresponding place), and the second one at the end of section 2.4 "We performed mesh convergence studies refining the mesh, particularly, in the computational subdomain that contains steep hydraulic gradients, until the solution became mesh-independent."

Your third claim: "The effect of boundary condition can be investigated by changing the boundary condition from close to open and even partially open boundary condition instead of changing the size of the domain."

That is right, it can be investigated the way you propose but it can also be investigated the way we proposed. What matters here is that one makes sure that boundary conditions do not affect the solution in the computational subdomain you are mostly interested in. This can be done one way or another. In scientific research there is no such a thing as "the way" to do that. Normally, one makes sure that the borders of the modeling domain are far enough away from the modeling subdomain one is most interested in, so that the boundary condition does not affect the solution in the simulation subdomain for the simulation time used. Alternatively, one can change boundary conditions and see whether that change does not affect the solution in the concerned modeling subdomain. Both ways are indistinctively used to investigate the issue of solutions being independent on the boundary condition set. Neither way is better than the other. By the way, once it has been checked (as we did) that the edges of the model are far away enough so that the boundary conditions set does not affect the solution

anymore in the modeling domain of most interest, then it does not matter to change boundary conditions. Precisely the boundary condition set at borders of the simulation domain was that of no-flow, as indicated in line 166 of the manuscript version you read. In addition, as indicated in section 2.4, in order for the boundary condition not to affect the solutions after a long simulation time, it is necessary that the edges of the model are far enough away, otherwise when the isobars under study approach the edge, they will not continue to develop driven by the properties and physics of the reservoir but are dominated by the value of the pressure at the edge. This was thoroughly investigated during the simulations performed by progressively enlarging the modeling domains until no change in the solution was observed. Then, that dimensions of the simulation domain were adopted for the simulation time used. It has now been included a remark concerning this issue in the appropriate place of the manuscript.

Thank you again for your comment and we hope we have clarified your observations.

Interactive comment on Solid Earth Discuss., https://doi.org/10.5194/se-2019-170, 2020.

Printer-friendly version

Discussion paper