3D Hydrogeological Modeling - From a Theoretical 2D Model Through a Medium Scale Application Up to a Challenge: Simulations at Basin Scale

E. Cavalli[1], R. Simonetti[1], M. Gorla[1], N. Ceresa[1]

[1]CAP Holding

Abstract

An alluvial quaternary aquifer system, assumed as a high heterogeneous porous media, has probably conceived as a numerical modeling hell, according to most of hydrogeologists. In this work we will show how we tried to manage the whole subsurface physics and, step by step by a simple model at basin scale simulation, how we solved all phenomena concerning hydrogeological simulation. We have chosen COMSOL Multiphysics® for two reasons: 1) FEM methods allow to use very complex geometries and to honor high detailed subsurface geology and surface geomorphology (river, quarry and incised valley, canyon); 2) multiphysics aimed to hydrogeology simulation permits to run a single model with all phenomena resolution at the same time. Firstly, we built a 2D section with these physics: a) Darcy's law (multilayer and heterogeneous saturated aquifer), b) Richards' equation (portion of unsaturated aquifer and capillary fringe), c) ALE (Aleatory Lagrangian-Eulerian) methods to show hydraulic head surface deformation during multi-well water pumping phenomena, d) Inlet from surface (recharge according to hydrogeology terms) with heterogeneous zones, provided by satellite meteorological data, e) solute transport (flux of mass) and inlet from a surface water (2 m incised creek) with a generic solute. We simulated stationary and transient model with good results. After a prime calibration, we observed very good output data according to our own data and bibliography too.
After this "academic" exercise, we applied this modeling work plan in a very critical pollution site, in order to define a capture zone of nine wells with double screens pumping simultaneously. We exactly used the same configuration of 2D model in a 3D domain. We used IGES format to migrate 3D wells technical settings to COMSOL Multiphysics® domain. We meshed all domains with finest mode and forced mapped mesh on the well's screen superimposed a fixed mesh number to honor their cylindrical shape.
Though the domain was increased by 2000% (from a decametric simulation to a kilometric study area) we appreciated how COMSOL Multiphysics® runs very well parallelized in our workstation and found very fastly the convergence although faced with a high areal complexity. Currently, we are developing a very large scale 3D high detailed aquifer model ready to be imported as an interpolation function and to define hydrogeological heterogeneities. We are testing COMSOL Multiphysics® for a very large domain with interesting response. For our test we do not notice a limit (except calculation time) to areal dimension.