Simulation of a Nozzle in a Borehole

E. Holzbecher[1], F. Sun[2]

[1]German University of Technology in Oman, Muscat, Oman, Germany
[2]Georg-August-Universität, Göttingen, Germany

Abstract

In boreholes nozzles have to be found advantageous to increase the infiltration rate of water into the subsurface ground. Studies and practice in the field shows that the infiltration of water into permeable aquifers can be improved, if the flow in the borehole is modified. Due to the nozzle the flow regime turns from linear to turbulent. CFD studies help to understand the physics of the infiltration process. The transition of the flow regime within the borehole and its further effects on flow within the porous medium of the aquifer is examined using COMSOL Multiphysics® software.

We report about first numerical experiments with slightly turbulent flow, as observed in the nozzle. Various options of turbulent flow modeling, as Navier-Stokes k-omega and k-epsilon, are tested. Mesh refinement and boundary layer options are studied in addition. From the experiences in a simplified geometry we draw conclusions about best modelling options for such constellations.

The major work is about a nozzle design that is currently used in practical field work. Parametric model runs were used to examine the effect of the nozzle design (length, radius, shape) placement within the borehole and in relation to the filter screen. Moreover we studied the influence of the pumping-rate as operational parameter on the flow regime within the borehole.

In a further extension we included the porous ground surrounding the borehole in the model. Using parametric runs again, we examined the influence of the porous medium properties (porosity, permeability) on the coupled flow regime. Also the position of the well screen in relation to the boundaries of the aquifer layer was examined.

The figures show the flow field represented by colored surface of absolute values of velocity and by streamlines (white).

COMSOL Multiphysics turned out to be very convenient for such simulations as CFD models for flow of free fluids can be coupled with porous medium approaches.

Figures used in the abstract

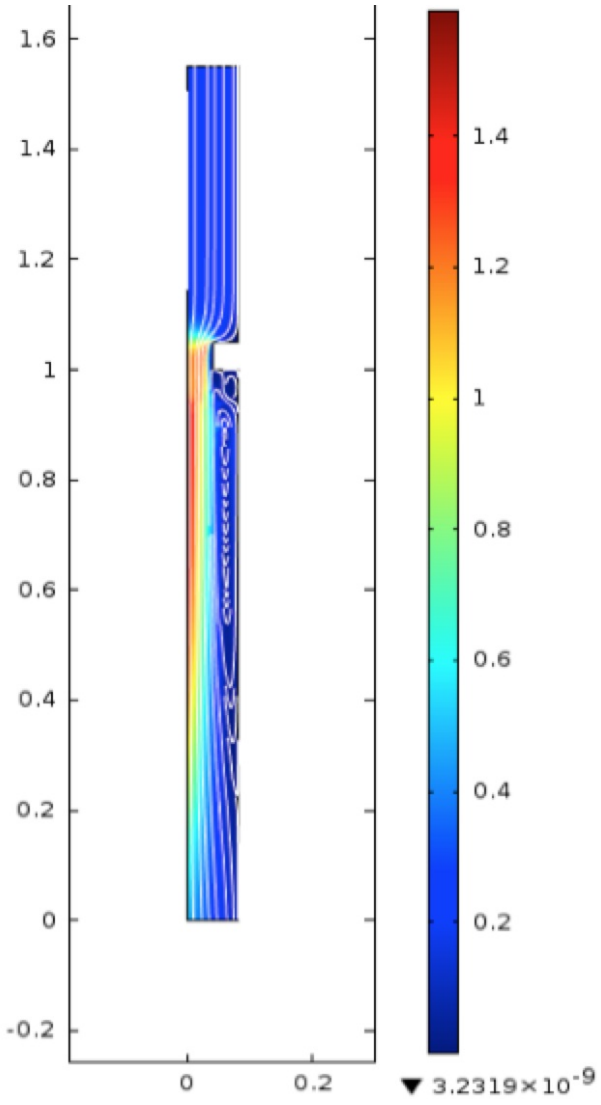

Figure 1: Flow field in a simple nozzle design

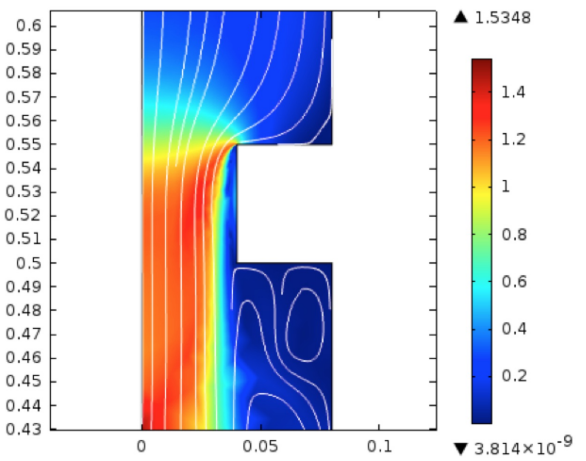

Figure 2: Flow field detail (of Figure 1)

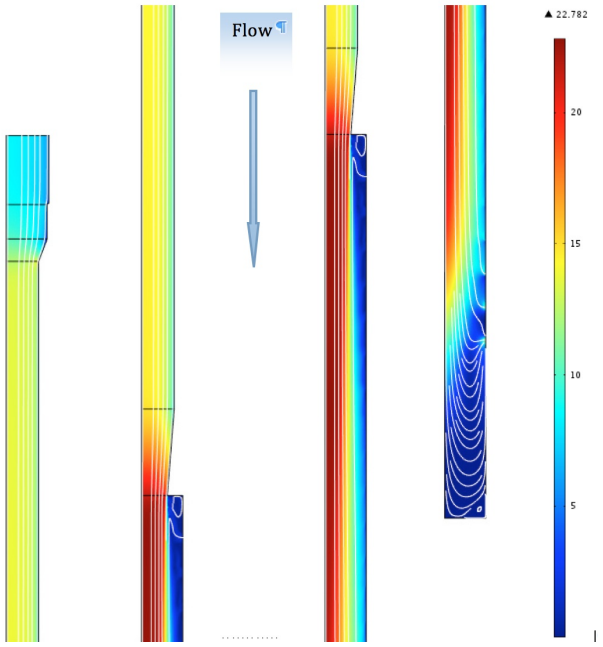

Figure 3: Flow field in a borehole with nozzle

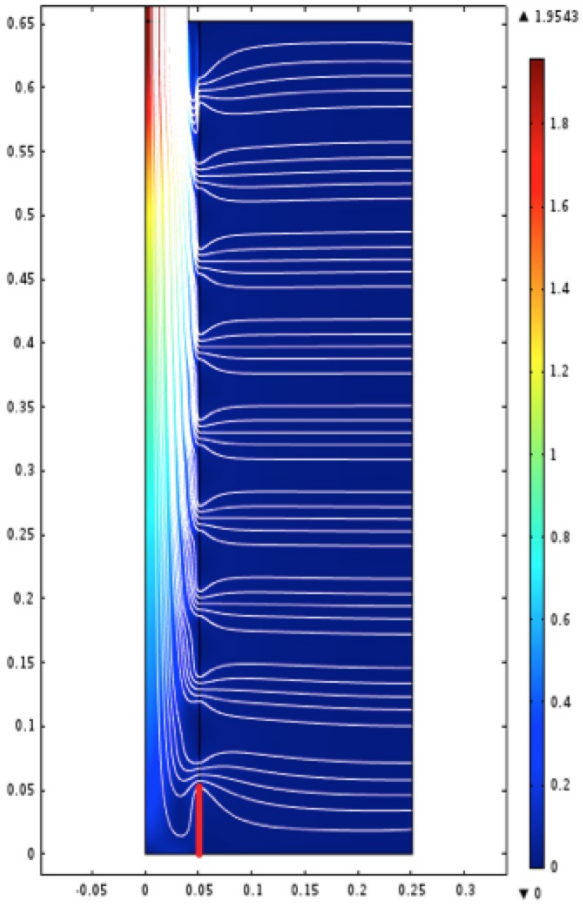

Figure 4: Flow field in borehole and aquifer