Modeling a Nozzle in a Borehole

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

[1]Georg-August Universität Göttingen, Göttingen, Germany

Abstract

Introduction:
Within a borehole a nozzle can be installed in order to increase the efficiency of fluid injection. The position of the nozzle is located near the perforated casing of an injection well. Up to now there are few studies on the turbulent flow building up in the pipe due to the nozzle. The Reynolds number increases due higher fluid velocity at and near the nozzle outlet. The shape of the nozzle and the boundaries of the borehole also play a role. Different modeling methods, settings of meshes, nozzle geometries and variations of outlet flux through perforated casing are discussed using COMSOL Mutiphysics® modeling.

Model Set-up:
A model is set up for a typical nozzle design as applied in practice .Figure 1 shows the 2D model in the axi-symmetric coordinate system. Flow is downward through the nozzle and fluid leaves through the holes in the perforated screen. The model is based on the Navier-Stokes equations. Turbulence closures are the standard two-equation k-ε model and k-ω models. We use reasonable constraints and boundary conditions, and wall functions close to walls. We specify pressure at the outlets. An inlet pipe is used to get the value of k, ε, and ω of turbulent flow at the inlet. By changing modeling methods, fluid rates, setting of mesh and boundary layers, we explored several different numerical settings.

Results:
Figure 2 presents the wall lift-off in viscous units of turbulent flow modeled by k-ω and k-ε approaches in a simplified test model. k-ω model is recommended as it was found to be easier to fit the wall lift-off in viscous units value to be 11.06 along the solid walls. We examined pressure, mean velocity, recirculation length and wall lift-off for various settings of mesh and boundary layer properties. The mesh should be fine enough and the build with corner refinements and proper boundary layers. Figure 3 shows that outlet flux entirely occurs through the three outlet boundaries at the bottom of the pipe. The share on outgoing flux through for each outlet (from bottom to top) is shown in Figure 4. The contribution of each outlet boundary does not change when flow rate changes from 7.5m3/h to 90m3/h.

Conclusion:
Using COMSOL Multiphysics simulations, the turbulent flow modeled by k- ω and k- ε methods through different meshed borehole nozzle have been analyzed by comparing pressure, mean velocity, recirculation length and etc.. The results demonstrate that k- ω method is much better

than k- ε method. Meshing, especially concerning the boundary layers, plays a major role to obtain a high accuracy of the results. Futhermore, the outlet boudaries at the bottom play a dominant role concerning outflow and thus a most relevant for an effective injection.

Reference

Holzbecher E., Jin Y., Ebneth S., Borehole pump & inject: an environmentally sound new method for groundwater lowering, International Journal of Environmental Protection, Vol. 1, No. 4, 53-58, 2011

Jin Y., Holzbecher E., Oberdorfer P., Simulation of a novel groundwater lowering technique using arbitrary Lagrangian-Eulerian method, COMSOL2011, Stuttgart, 2011

Yu Y., Shademan M., Barron R.M., Balanchandar R., CFD study of effects of geometry variations on flow in a nozzle, Eng. Appl. of Comp. Fluid Mech., Vol. 6, No. 3, 412-425, 2012

Figures used in the abstract

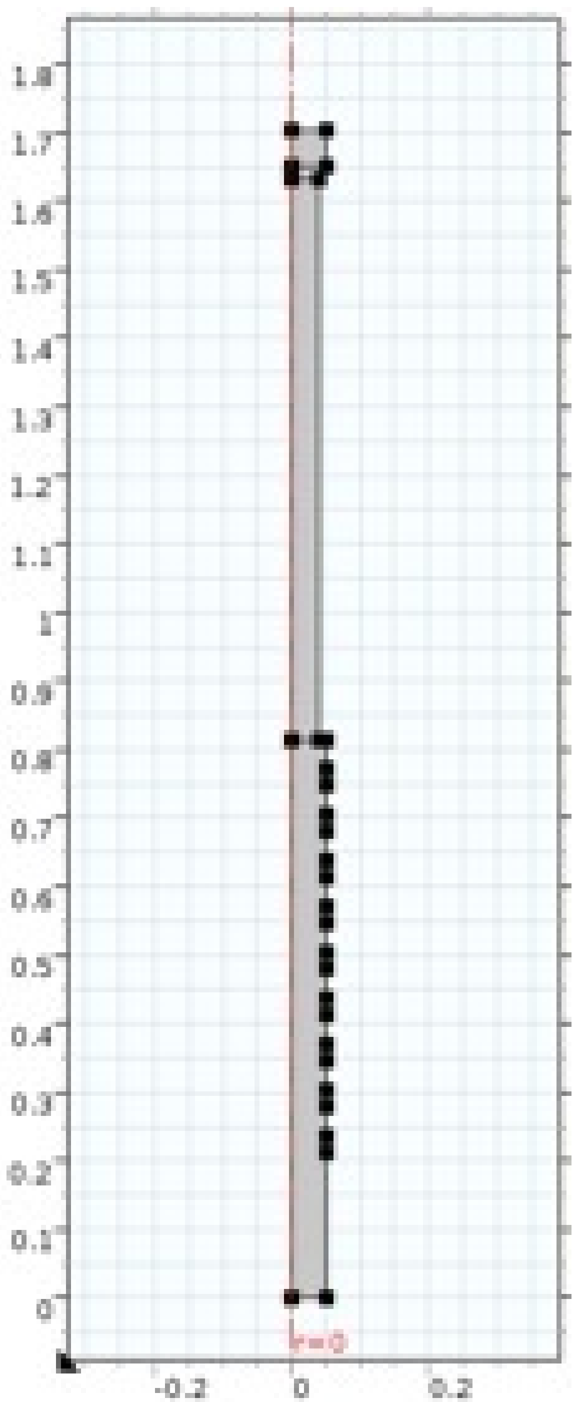

Figure 1: Borehole nozzle geometry in axi-symmetric coordinates

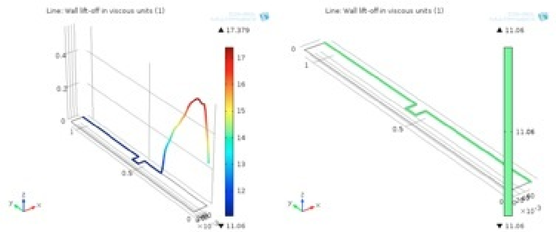

Figure 2: Model test: wall lift-off in viscous units (left: k-ε, right: k-ω model)

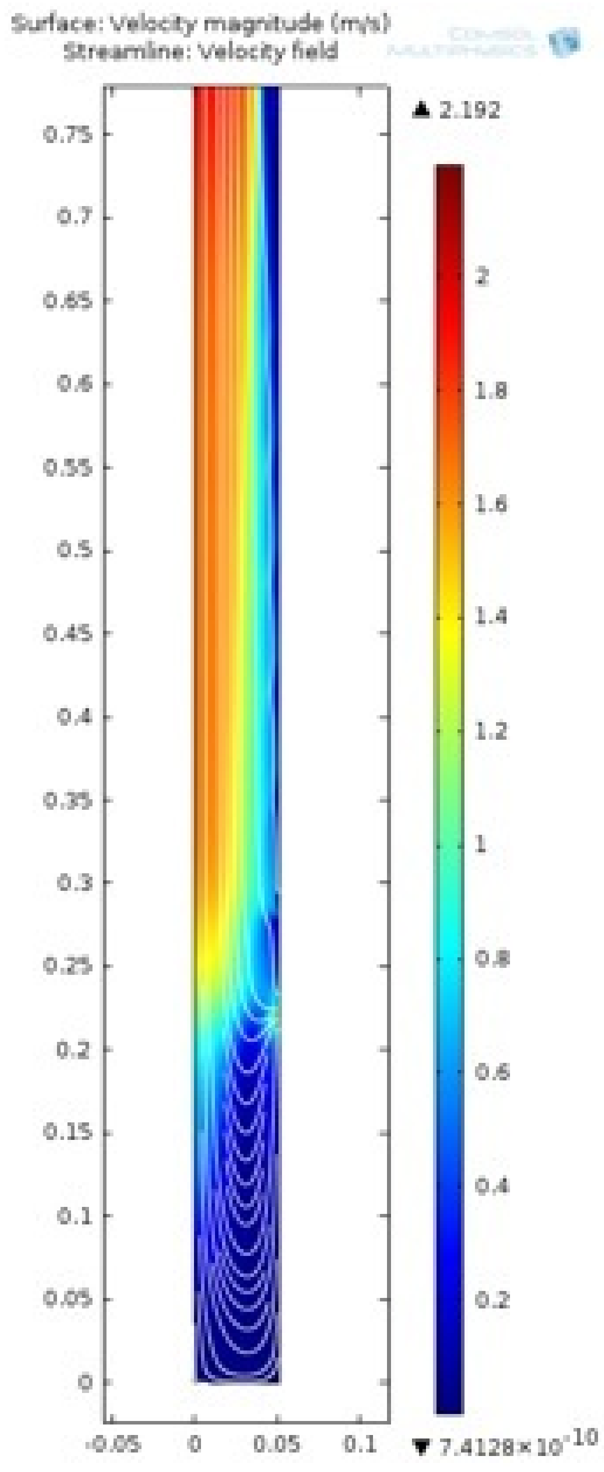

Figure 3: Mean velocity in nozzle and borehole (Q=30 m3/h)

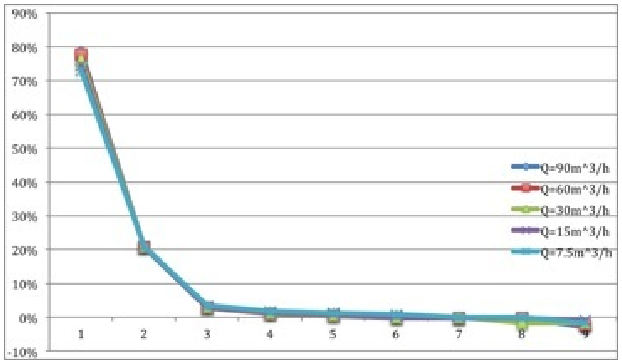

Figure 4: Outlets flux distribution along the pipe (from bottom to top)