Numerical Model For Hydraulic Performance Of A Perforated Pipe Under-drain Surrounded By Loose Aggregate
Tanjina Afrin¹, Dr. Abdul A. Khan², Dr. Nigel B. Kaye, and Dr. Firat V. Testik²
1. Graduate Student, 2. Associate Professor Of Civil Engineering

Introduction
Urbanization significantly alters the hydrological cycle, leading to reduced infiltration, increased flooding, and reduced water quality. Proper management of storm-water runoff is necessary to mitigate these undesired impacts. The use of Best Management Practices (BMP) and Low Impact Development (LID) is becoming more common day by day for this purpose and perforated pipes are one of the main components of these LIDs and BMPs. This poster presents results from a CFD model that combines both porous media flow and pipe flow. The model was developed in ANSYS Fluent to examine the hydraulic behavior (stage-discharge relationship) of a porous pipe shrouded in loose aggregate for use as an underdrain in storm water management. The model was validated against the experimental data of Murphy et al. (2014) and was then used to undertake a detailed parametric study of porous pipe underdrain performance.

Methodology
A detailed 3D model has been made and solved using the ANSYS workbench and ANSYS Fluent v 14.0 respectively. There were three cell zones in every model; pipe, aggregate and water. The aggregate zone was designated as a porous packed bed zone.

From a mesh sensitivity analysis, it was found that the optimum minimum size of the element is 1.54e-4 m and the maximum face size is 1.3e-2 m. Figures below show the mesh along mid plane (a) and a close view along a cross section of the model (b) respectively.

The figure on the right shows the different boundary conditions using different colors.

The following solution methods were used in developing the model.
• Pressure–velocity coupling : SIMPLE
• Momentum, turbulent kinetic energy and turbulent dissipation rate discretization: Second Order Spatial
• Moment discretization: Second Order Upwind Scheme
• Time : Transient

The Palmetto Cluster at Clemson University was used for all these large scale simulations.

Results
The CFD model constantly over predicted the mass flow rate through the pipe.

• The difference between the CFD model and experimental results was on average 10.6% (upper figure to right).

One possible explanation for this consistent over prediction is that the CFD model does not account for individual pieces of aggregate blocking holes in the pipe sidewall. To correct for this, the wall inlet area was reduced by multiplying it by the aggregate porosity. That is,

\[ A_{\text{inlet corrected}} = \Phi_{\text{aggregate}} A_{\text{inlet}} \]

This reduced the average error to just 6% (lower figure to right).

The CFD model results also indicated that when the water surface is above the top of the aggregate, water flows along the water surface and then down through the aggregate. The aggregate flow is mostly vertical, which is consistent with prior experimental results.

• When the water surface is above the top of the aggregate, water flows along the water surface and then down through the aggregate.

Applications
• This developed model can be used for design and analysis of different LIDs and BMPs for any practical range of parameters.
• The parametric study shows that for draining stormwater, using several parallel pipes instead of a single long pipe of the same diameter and total length will provide better performance.

References