ABSTRACT

The need for an efficient clean water treatment system is becoming increasingly urgent due to the high accumulation of sludge in sedimentation tanks, which hinders the distribution of clean water to the community. To address this issue, PT. Mugi Renes Abadi has designed a high-pressure spray hydrant nozzle system as a tool for breaking sludge. This study analyzes the pressure drop in the spray hydrant nozzle piping system, considering major losses due to fluid friction on straight pipe walls and minor losses from fittings such as elbows, tees, and reducers. The methods used include theoretical calculations using the Darcy-Weisbach equation and K-factor, as well as Computational Fluid Dynamics (CFD) simulations using SolidWorks Flow Simulation. The results show that the fluid flow is turbulent with a Reynolds number > 4000. The minimum pressure drop of 250.092 Pa occurs in valve configuration 3, while the maximum pressure drop of 282.181 Pa occurs in valve configuration 4 at nozzle number six. The theoretical pressure drop values were on average 74% higher than the CFD results, as CFD only evaluated the manifold pipe and pressure at the nozzle. CFD simulations identified high pressure gradients at the manifold pipe elbow due to increased flow velocity during bending, resulting in vortices and turbulence that caused uneven flow distribution among the nozzles, leading to varying spray power depending on the distance between the nozzle and the inlet.
Keywords: Pressure drop, CFD, Piping system, Sediment removal, Spray nozzle

INTISARI

Kebutuhan sistem pengolahan air bersih yang efisien semakin mendesak akibat tingginya akumulasi lumpur pada bak sedimentasi, yang menghambat distribusi air bersih ke masyarakat. Untuk mengatasi masalah ini, PT. Mugi Renes Abadi merancang sistem *spray hydrant nozzle* bertekanan tinggi sebagai alat penghancur lumpur. Penelitian ini menganalisis *pressure drop* pada sistem perpipaan *spray hydrant nozzle*, melalui pertimbangan *major loss* akibat gesekan fluida pada dinding pipa lurus dan *minor loss* dari *fitting* seperti *elbow*, *tee*, dan *reducer*. Metode yang digunakan meliputi perhitungan teoritis dengan persamaan Darcy-Weisbach dan K-factor, serta simulasi Computational Fluid Dynamics (CFD) menggunakan SolidWorks *Flow Simulation*. Hasil menunjukkan aliran fluida bersifat turbulen dengan bilangan Reynolds > 4000. *Pressure drop* minimum sebesar 250.092 Pa terjadi pada konfigurasi valve 3, sedangkan *pressure drop* maksimum sebesar 282.181 Pa pada konfigurasi *valve* 4 di *nozzle* nomor enam. Nilai *pressure drop* perhitungan teoritis rata-rata 74% lebih tinggi dibanding hasil CFD, karena CFD hanya mengevaluasi pipa manifold dan tekanan di *nozzle*. Simulasi CFD mengidentifikasi gradien tekanan tinggi pada *elbow* pipa manifold akibat peningkatan kecepatan aliran saat berbelok, menghasilkan *vortex* dan turbulensi yang menyebabkan distribusi debit tidak merata antar *nozzle*, sehingga daya semprot bervariasi sesuai jarak *nozzle* dengan *inlet*.
Kata kunci: Penurunan tekanan, CFD, Sistem perpipaan, Penghancuran sedimen, Semprotan nosel