PENGARUH VARIASI VISKOSITAS TERHADAP LAJU ALIRAN PIPA PADA UNIT *PRE-HEATING* PT WILMAR NABATI INDONESIA MENGGUNAKAN APLIKASI *SOFTWARE ANSYS*

Nama Mahasiswa   : Muhammad Dzaky Aqil
NIM   : 2204201281
Dosen Pembimbing : Bambang Dwi Haripriadi, ST. M.T

ABSTRAK

Penelitian ini menganalisis pengaruh variasi viskositas terhadap laju aliran pipa pada unit *pre-heating* di PT Wilmar Nabati Indonesia menggunakan *software ansys*. Tujuannya adalah menentukan tekanan fluida berdasarkan viskositas dan suhu serta mengukur tekanan dan laju aliran fluida melalui simulasi CFD (*Computational Fluid Dynamic*).

Metode CFD digunakan untuk menganalisis aliran fluida secara mendetail. Hasil penelitian menunjukkan bahwa, sesuai prinsip Bernoulli, peningkatan kecepatan fluida mengakibatkan penurunan tekanan. Namun, faktor lain seperti viskositas mempengaruhi hasil akhir. Berdasarkan prinsip Bernoulli, viskositas optimal adalah 19 cSt. Namun, dalam aplikasi nyata digunakan viskositas 34 cSt karena adanya pertimbangan lainya.

Kesimpulan penelitian menunjukkan bahwa penggunaan *ansys Fluent* untuk simulasi aliran fluida memberikan hasil yang memenuhi persyaratan analisis. Tekanan dan *mass flux* pada viskositas 34 cSt memperoleh hasil rata-rata simulasi terbesar, sedangkan viskositas 19 cSt memperoleh hasil rata-rata terkecil. Meskipun terdapat perbedaan rata-rata nilai untuk setiap viskositas, perbedaannya tidak signifikan dan masih dalam distribusi normal.

Kata Kunci: Tekanan, *Mass Flux*, *Computational Fluid Dynamic*, Viskositas

THE EFFECT OF VISCOSITY VARIATION ON PIPE FLOW RATE IN THE PRE-HEATING UNIT AT PT WILMAR NABATI INDONESIA USING ANSYS SOFTWARE

Student Name   : Muhammad Dzaky Aqil
NIM   : 2204201281
Thesis Advisor   : Bambang Dwi Haripriadi, ST. M.T

ABSTRACT

This study analyzes the effect of viscosity variation on pipe flow rate in the pre-heating unit at PT Wilmar Nabati Indonesia using ansys software. The objectives are to determine fluid pressure based on viscosity and temperature and to measure pressure and flow rate using CFD (Computational Fluid Dynamic) simulations.

The CFD method was employed to analyze fluid flow in detail. The results indicate that, according to Bernoulli's principle, an increase in fluid velocity results in a decrease in pressure. However, other factors such as viscosity also influence the final outcome. Based on Bernoulli's principle, the optimal viscosity is 19 cSt. Nevertheless, in practical applications, a viscosity of 34 cSt is used due to other considerations.

The conclusion of this study is that using ansys Fluent for simulating fluid flow in pipes provides results that meet analysis requirements. The pressure and mass flux at a viscosity of 34 cSt achieved the highest average simulation results, whereas a viscosity of 19 cSt achieved the lowest average simulation results. Although there are average value differences for each viscosity, these differences are not significant and remain within the normal distribution.

Keywords: Pressure, Mass Flux, Computational Fluid Dynamic, Viscosity