

Abstract

The Venturi effect is used in many applications on merchant marine vessels. The goal of this dissertation is to examine the influence the geometry of a venturi has on its performance. The drop in pressure in a fluid resulting from the flow through a constricted section of a pipe is known as the Venturi effect. In this dissertation, CFD software is used to simulate a laminar flow through ten different venturi models. In each of these models, one particular section differs from the standard model. The pressure, the velocity as well as the turbulence of the flow in the different models are used to compare them to the standard model.

One of the conclusions drawn from the results of the simulations is that an increase in influence on the pressure and the velocity of the flow always coincides with an increase in energy losses in the venturi.