

Abstract

Amr A. Mohamed Mohamed Hassan

COMPUTATIONAL FLUID DYNAMICS RESULTS FOR AHMED'S CAR MODEL

The well known "Ahmed" car model, has the form of a highly simplified car, consisting of a blunt nose with rounded edges fixed onto a box-like middle section and a rear end that has an upper slanted surface (like a "hatch- back" car), the angle of which can be varied. The main purpose of the present work is to investigate the effect of slant angle on drag coefficient for constant Reynolds number and the effect of Reynolds number on drag coefficient for different slant angles. Computational Fluid Dynamics (CFD) code FLUENT was used to evaluate the flow distribution around Ahmed's car model. The CFD results obtained were also used to evaluate the drag coefficient by calculating the force acting on the body. The results obtained indicate that CFD flow simulation is a useful tool for providing predictions of pressure distribution and forces exerted on the vehicle components.