

Computational Fluid Dynamics (CFD) Study of Boeing 737-800 Propulsion System

Abstract

This research article aims to study the propulsion system of a Boeing 737-800 aircraft by using the Computational Fluid Dynamics (CFD) model to accurately assess its performance. The initial use of SolidWorks is to create an underlying concept of manufacturing and the development of the current features of the propulsion system. The modeling is done based on the known dimensions that are readily available in the open literature.

The setup and analysis of the research is done using multiple tools such as SolidWorks, ANSYS, and MATLAB. In the present study, SolidWorks will be used for the geometry and CAD model of the Boeing 737-800. Then the CAD will be used in ANSYS to simulate combustion and compressible fluid flow through the propulsion system. The ultimate goal is to understand how the propulsion system contributes to the thrust and aerodynamics of the Boeing 737-800 aircraft.