International J.of Multidispl.Research & Advcs. in Engg.(IJMRAE),
ISSN 0975-7074, Vol. 3, No. II (April 2011), pp. 267-276

A COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF TWO-STROKE INTERNAL COMBUSTION ENGINE

M. A. AFSAR, A. B. DIGHEWAR, A. V. VANALKAR, A. A. BHURA AND M. SOHAIL

Abstract

The main objective of the present work is to simulate the in-cylinder mass flow rate of two stroke spark ignition internal combustion engine. The research describes Computational Fluid Dynamic (CFD) analysis technique to predict in-cylinder flow on scavenging process. The 2-D model is created in GAMBIT 2.2 to generate mesh grids and imported to FLUENT software 6.3. This analysis is using the dynamic mesh approach using unstructured quadrilateral cell is simulated from Top Dead Center (TDC) at 0^0 of crank angle. Parameter of pressure is used as boundary condition collected from the experimental result at the intake port. The engine modeling is based on motored condition for un-firing cases at different engine speeds. Flow structures in the transfer ports and the exhaust port are predicted with and without scavenging. The total pressure and velocity map from computation provided comprehensive information on the scavenging phenomenon and analysis is carried out for the transfer ports flow. An extra port in the transfer port provide for the passage of air to avoid the short circuiting in the engine cylinder. So a proper scavenging process will be carried out with better results to be seen. Turbulence is modeled by high Reynolds number version k-ϵ model. Experimental measurements are made for validating the numerical prediction. The result shows that using dynamic mesh approaches gives confident results compared with experimental. The results have been predicted of in-cylinder pressure of unburned gases on engine cylinder.

Keywords: Computational Fluid Dynamics, In-Cylinder mass flow rate, Pressure simulation.