

OPTIMIZATION FOR INTAKE PORT

LAXMIKANT P. NARKHEDE & ATUL PATIL

Department of Mechanical Engineering, Godavari College of Engineering, Jalgaon, Maharashtra, India

ABSTRACT

The present work deals with the study of the flow within the intake port in both steady and unsteady states and analyze the results to evaluate and improve the ability of the intake port to convey air identically to all cylinders with the least possible pressure losses. Also the effect of engine speed on the volumetric efficiency has been analyzed by 2D CFD model at different engine speeds.

Optimizing airflow performance during intake port process is the main purpose for this project. Here we are going to Analyze in CFD simulation and experimental using academic engine flow model. This analysis could be used to increase efficiency of volumetric flow rate and maximizing usage of air fuel in combustion process, which reduce emission to environment. Even though air flow have been optimized on its intake port, but still intake system could be improve by considering other parts of engine also such as intake manifold, Valve etc..

The present work is related to, two important common fluid flow patterns from computational fluid dynamics (CFD) simulations, namely, effect of steady state and unsteady state analysis and effect on air motion on turbulence inside the cylinder.

The performance of the engine can be improved by efficient design of intake ports. In the process of optimizing the port flow for improving engine performance, Computational fluid dynamics (CFD) simulation plays a very important role by adding cost effectiveness.

KEYWORDS: CFD, Flow Coefficient, Intake Port, Swirl Ratio