

CFD Modelling of a Two Stroke Engine to Predict and Reduce Short Circuit Losses

Manish Garg, Davinder Kumar, and Madani Syed
TVS Motor Company

Siva Nageswara
CD-adapco

ABSTRACT

One of the major reason for lower efficiency and higher unburned hydrocarbon and carbon monoxide emission for two stroke engine is short circuit losses during the scavenging process. An attempt has been made in this study to understand and improve this phenomenon. A three dimensional transient CFD model is developed for a loop scavenged, Schnuller type, 70 cc two stroke engine. Three major processes, namely, blow down (expansion); scavenging and compression have been modelled. The model is validated with PIV measurement done in motoring mode. Model is also validated with experimental data for trapping efficiency with Watson method and for in-cylinder pressure during expansion, blow down and intake events. A good correlation is observed between experimental and simulation results. CFD model is used to quantify various parameters, such as, delivery ratio, trapping efficiency, scavenging efficiency, and amount of fresh mass short circuit at different load and speed points. Around 20 to 25% short circuit losses are observed at all the load cases analyzed. The mechanism for this short circuit loss is understood with the help of a passive scalars based visualization method. Two major root causes identified are, port design and exhaust pressure wave dynamics. Three different designs of ports are analyzed. With the best design, the short circuit loss is reduced by 12%. Same design has been tested on a chassis dynamometer, that resulted into a reduction of 10% in HC (hydro-carbon) emission in a mass emission test on a drive cycle and 10% to 25% improvement in WOT (wide open throttle) wheel force in mid and higher speed operating points.

CITATION: Garg, M., Kumar, D., Syed, M., and Nageswara, S., "CFD Modelling of a Two Stroke Engine to Predict and Reduce Short Circuit Losses," *SAE Int. J. Engines* 9(1):2016.

INTRODUCTION

Mopeds are an important means of personal transport in India and other developing countries. Around 600,000 mopeds are sold per annum in India. And they are mostly powered by a small 70 cc two-stroke engine with loop scavenge, Schnuller type, port arrangement and conventional carburetor. Few of the important reasons for usage of two stroke engine in this class of two wheeler are: low initial cost, low maintenance cost, and ease of operation. However, the major drawback of this design is higher hydrocarbon (HC) emission. It is extensively reported that the main reason for higher HC emission in a two stroke engine is due to short circuit of fresh charge, air-fuel mixture, to exhaust. This happens when both intake and exhaust ports are open during scavenging process [1].

According to next emission norms proposed in India, a significant reduction in the HC emission level has been proposed. A major design change is required in the engine to meet proposed emission targets. There are four possibilities that can reduce the HC emission for this class of engines. These four possibilities are listed below with challenges associated with each of it.

1. Adoption of four stroke technology to this class of two-wheelers. The major challenges associated with this approach are; higher initial and maintenance cost, and engine design to operate with clutch-less single ratio transmission.
2. Use of bigger oxidation catalytic convertor in the existing two-stroke engine. However this will increase the cost to the customer. And back pressure from the bigger catalytic converter can result in performance drop, in terms of torque output and fuel economy
3. The third choice to engine designer is to implement a direct injection in the combustion chamber. Two well-known and proven concept in this line are; air assisted direct injection by Orbital Engines Inc. [2] and low pressure direct injection (LPDI) from Graz University of Technology [3, 4]. Again the major challenges to adopt these sophisticated technologies are initial and operating costs.
4. The fourth possibility is to use latest experimental and computer simulation based methods to improve the current design to fulfil the requirements with minimal increase in product cost.

In this work, authors have attempted to use Computational Fluid Dynamics (CFD) based method to understand the gas exchange process in a two-stroke loop scavenge engine. And later, the CFD model has been used to analyze various port configurations.

CFD modelling technique is a proven method to analyze IC engine processes. It has been used by engine researchers to understand intricate processes, namely gas exchange, air fuel mixing [5, 6], and combustion [7].

In current article, initially CFD modelling approach is explained, which include engine specifications, initial and boundary conditions, then, results pertaining to CFD model validation with PIV (particle image velocimetry) and Watson method are presented. Later to that, base engine results are discussed, this include explanation of various post-processing methods used to analyze the results. Based upon post processing results of base engine, various design iterations are carried out using CFD model. Finally, the best design was selected based upon manufacturing constraints and CFD simulation results. Same was tested on the test bed, the results for the same is presented in the end.

CFD MODELLING APPROACH

The purpose of this study is to develop a CFD process which represent gas exchange process for a loop scavenge two stroke engine. This include blow down process of exhaust gases, and displacement and mixing of fresh charge with exhaust gases. To reduce turnaround time for each design iteration, CFD model is prepared and run for exhaust and intake process. CFD calculations are started from 30 degree crank angle ($^{\circ}$ CA) after TDC (top dead center), and it is assumed that most of the combustion process is finished by this crank angle.

CFD model is built using a commercial code STAR-CD by CD-adapco® [8]. Meshing is done in ES-ICE, an expert-system to create mesh for IC (internal combustion) engine with moving boundaries. The various details of CFD modelling are represented in subsequent subsections.

Engine Specifications

The engine used in the current study is a crankcase compression engine, the port arrangement is looped scavenged, Schnurle type. Engine has five scavenged ports (transfer ports) and one exhaust port. Figure 1, shows the layout of the ports, with piston at bottom dead center (BDC), crankcase is not shown in the figure.

Table 1. Engine specifications used in the current study.

Parameter, units	Value
Bore, mm	46
Stroke, mm	42
Displacement volume, cm^3	69.8
Con rod, mm	84
Compression ratio, -	8.0
Exhaust port opening, $^{\circ}$ CA ATDC	115
Exhaust port closing, $^{\circ}$ CA ATDC	244
Scavenge port opening, $^{\circ}$ CA ATDC	134
Scavenge port closing, $^{\circ}$ CA ATDC	226

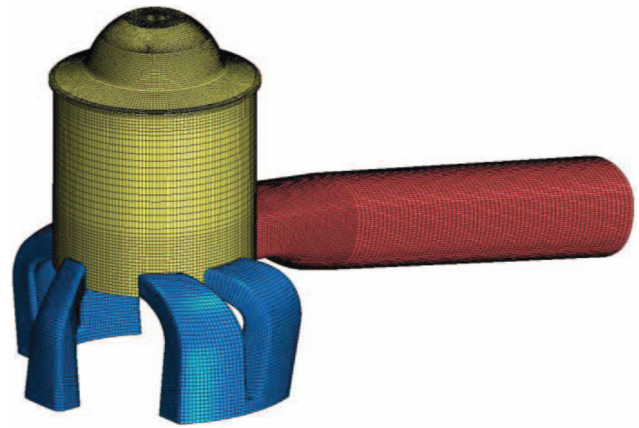


Figure 1. CFD domain, meshed using block structured hexahedral meshing

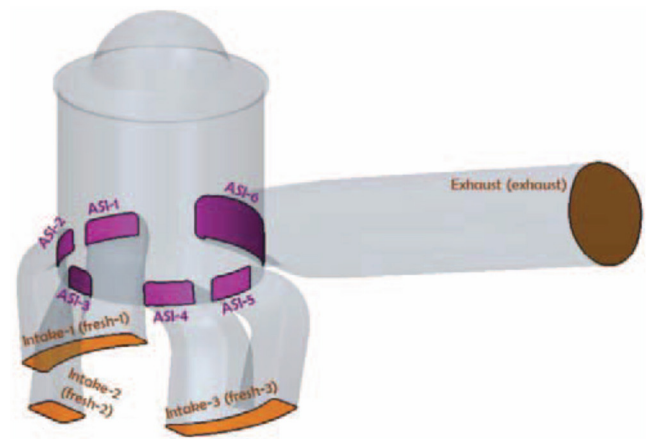


Figure 2. Boundary conditions

The air enters the engine through an air filter, and air fuel mixture is prepared through a conventional carburetor. The fresh charge (air-fuel mixture) enters the crankcase through a reed valve and due to a pressure difference between crankcase and intake system. The detail operation of a crankcase compression looped scavenged engine, similar to the engine used in current study, is explained by Heywood and Sher [1]. The geometric specification and port timings for the engine is listed in Table 1.

CFD Mesh

A block-structured hexahedral meshing is adopted to achieve higher accuracy with minimum turnaround time per design iteration. A template based mesh was built in such a way that different port designs can be meshed using the mapping technique by utilizing the base template. With this method, a very high quality mesh can be created with minimum meshing effort. The average cell size used in the mesh for most of the domain is 0.75 mm and a refined mesh of cell size 0.25 mm is used during the port opening and closing events. The total mesh count for the whole mesh at BDC is 1 million, whereas at TDC is 0.75 million. The snapshot of the mesh is shown in Figure 1.

Initial and Boundary Conditions

For initial and boundary conditions, in-cylinder, crankcase and exhaust pressures were measured at different load and speed points. The measurement was done on ECD (eddy current dynamometer). For boundary condition details, refer to Figure 2, pressure boundary conditions are applied on boundaries intake-1, intake-2, and intake-3 with a crankcase pressure, and a pressure boundary condition is applied to exhaust boundary with an exhaust pressure. All the other surfaces in the model are considered as no-slip walls. A representative pressure boundary conditions for intake and exhaust are shown in Figure 3. And Table 2, shows the temperature boundary conditions for different wall boundaries.

To understand the gas exchange process in detail and quantify various engine efficiencies associated, number of active and passive scalars are used. Table 3 shows the initial and the boundary conditions for various scalars. Each scavenge port part and its boundary has a scalar associated with it, namely, fresh-1, fresh-2, and fresh-3. Same is shown in Figure 2. The active scalars used in Table 3 are calculated based upon chemical equilibrium conditions at the given, temperature, pressure and air fuel mixture composition [9]. As mentioned earlier, all the initial conditions are representative for a given speed and load condition and these are varied for different load and speed conditions.

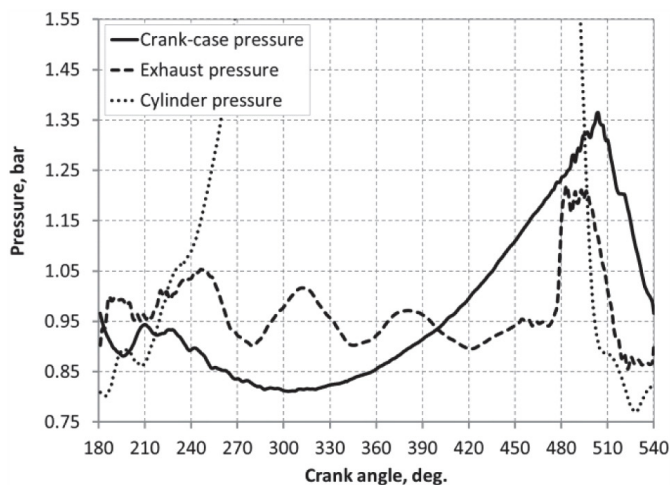


Figure 3. Pressure boundary conditions

Table 2. Temperature boundary condition for domain wall

Wall Boundary	Temperature (K)
Cylinder wall	423
Dome wall	473
Piston wall	473
Intake Port wall	373
Exhaust Port wall	533

Table 3. Initial and boundary conditions for different scalars

Scalars	Mass Fraction		
	Cylinder	Intake Ports boundaries, and domains	Exhaust Port boundary and domain
C_8H_{18} (A)	0	0.086	0
O_2 (A)	0	0.213	0
N_2 (A)	0.706	0.701	0.706
CO_2 (A)	0.117	0	0.117
H_2O (A)	0.093	0	0.093
Intake (P)	0	1	0
Exhaust (P)	1	0	1
H_2 (A)	0.002	0	0.002
CO (A)	0.082	0	0.082

Various CFD Models and Sub-Models

Various models and sub-models used in the study are [8]:

- Finite volume method to discretize the domain.
- Convection term discretisation is done by MARS (Monotone Advection and Reconstruction Scheme) which is multidimensional and second order accurate. MARS operates in two steps, 'reconstruction' and 'advection'. The reconstruction step involves computation of monotone gradients using a multi-dimensional Total Variation Diminishing (TVD) scheme. In the advection step, the reconstructed cell-face flow properties are used to compute the face fluxes for all advected properties using monotone and bounded advection scheme [10].
- PISO (Pressure Implicit Splitting of Operators) for pressure velocity coupling.
- Conjugate Gradient method to solve set of linear algebraic equations.
- Standard High Reynolds Number k-epsilon turbulent model with standard wall function to model turbulence

CFD MODEL VALIDATION

The CFD model is validated for gross parameters namely, in-cylinder pressure and trapping efficiency, and for flow field with a PIV (Particle image velocimetry). The comparison for in-cylinder pressure between experiment and CFD simulation from 30° CA ATDC is shown in Figure 4. A good match is observed for the blow down process and the gas exchange process, these two areas are main area of interest in the current study. The trapping efficiency comparison is shown in Figure 5. The experimental value of trapping efficiency is evaluated from the Watson method [11], and the trapping efficiency for the CFD is evaluated from ratio of fresh scalar mass inside the cylinder to total fresh scalar entered through all the boundaries (refer to Appendix). The trend for the trapping efficiency between the simulation and measurement is captured well, however, the values are not matching very well. The reason for this mismatch can be due to the reason that calculation method between simulation and experiments are fundamentally different.