CFD SIMULATION AND VALIDATION OF THE IN-CYLINDER WITHIN A MOTORED TWO STROKE SI ENGINE

M.M.Noor[1], K.Kadirgama[1], Devarajan R.[1], M.S.M.Sani[1],
M.F.M.Nawi[1], T.F.Yusaf[2]

[1]*Faculty of Mechanical Engineering, University Malaysia Pahang,*
26300, Kuantan, Pahang, Malaysia
Email: muhamad@ump.edu.my kumaran@ump.edu.my

[2]*Faculty of Engineering & Surveying, University Southern Queensland,*
Toowoomba 4350 QLD Australia, Email: yusaft@usq.edu.au

Abstract. *The study of in-cylinder characteristic is important to understand the scavenging process of the two-stroke engine. It is important to design a new two-stroke engine to produce low emission that will meet current and future emission regulation. The objective of the paper is to evaluate and analyze a Computational Fluid Dynamic (CFD) simulation of the flow in motored loop scavenged two-stroke engine using commercial software FLUENT and compare with experimental data. The experiment is carried out using the motored method which the rotating engine is generated using motor and without firing. The data is collected at different engine speed from the engine idle speed to maximum engine speed. After the validation, there are relative error between the experiment method and simulation method that need to be observed.*
Keywords: Computational Fluid Dynamic (CFD), two-stroke engine, low emission

INTRODUCTION

The two-stroke engine is used for the lightweight power units which can be employed in various attitudes as handled power tools such as the chainsaws, brush cutter, concrete saws and etc. These tools are manufactured with a view to lightness and high specific power performance.

The first two-stroke engine was a gas engine invented and built by Etienne Lenoir in 1860 and followed the two-stroke diesel engine by Sir Dugald Clark in 1878 [1]. Both design of the early two-stroke engine used a similar cylinder head as the four stroke diesel engine and a supercharger. The gasoline two-stroke engine, and the cylinder ports on which it depends were invented by Joseph Day in 1889 [1]. These cylinder ports were subsequently incorporated into diesel two-stroke engines, replacing either just the inlet valve or both inlet and exhaust valve. A great deal of development of the two-stroke engine was done between the two world wars, particularly by Germany companies such as DKW and Fitchel & Sachs [1].

Internal combustion engine performance and exhaust emissions are governed by unsteady fluid dynamic process [2]. The objective of these processes is fundamental to improving engine characteristics. Computational Fluid Dynamic (CFD) is able to produce detailed information on the flow within an engine. Fundamental Method of Operation of a Simple Two-Stroke Engine, the intake stroke begins with the piston near the top of its travel [3]. As the piston begin its descent, the exhaust valve close fully, the intake valve opens and the volume of combustion chamber begins to increase, creating a vacuum. As the piston descents, an air/fuel mixture drowns from the carburettor into the cylinder through the intake manifold. The intake stroke ends with the intake valve closed just after the piston has begun it upstroke [4]. The process is shown in Figure 1.

As the piston ascends, the fuel/air mixture is forced into the small chamber machined into the cylinder head. This compress the mixture until it occupies ½th to 1/11th of the volume that it did at the time the piston began its ascent. This compression raises the temperature of the mixture and increases its pressure. Increasing the force generated by the expansion of gases during the power stroke. The process is shown in Figure 2. The fuel/air is ignited by the spark plug just before the piston reaches the top if its stroke so that a very large portion of the fuel will have burned by the time the piston begins descending again. The heat produced by combustion increases the pressure in the cylinder, forcing the piston down with great force. The process shown in Figure 3

As the piston approaches the bottom of its stroke, the exhaust valve begins opening and the pressure in the cylinder begins to force the gases out around the valve. The ascent of the piston then forces nearly all the rest of the unburned gases from the cylinder [5]. The cycle begins again as the exhaust valve closes, the intake valve opens and the piston begins descending and bringing a fresh charge of fuel and air into the combustion chamber. The cycle shows in Figure 4.

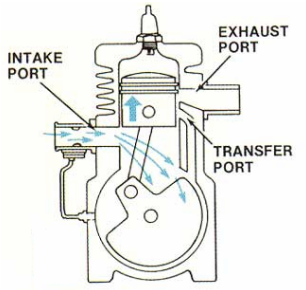

Figure 1: Intake Stroke

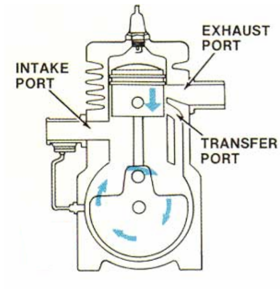

Figure 2: Compression Stroke

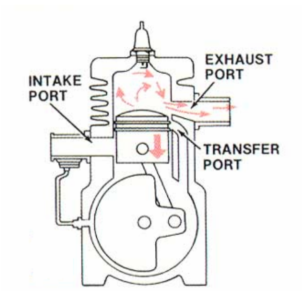

Figure 3: Ignition Stroke

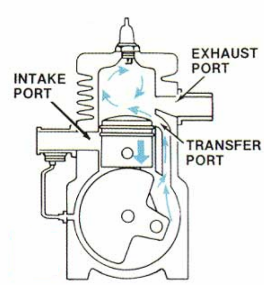

Figure 4: Exhaust Stroke

Computational Fluid Dynamic is the analysis of systems involving fluids flow, heat transfer and associated phenomenon such as chemical reactions by means computer-based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. CFD analysis is a solution of fluid motion equation usually partial differential equations (PDE) type using numerical approximation. This process which is called discretization turns the PDEs into simultaneous algebraic equation which in turn are solved using algebraic solution techniques, usually iterative method.

The approximations are applied to a numbers of small domains in space and times called grid. So, the final results are spatially and temporally resolved. Using CFD, you can build a computational model that represents a system or device that you want to study. Then you apply the fluid flow physics and chemistry to this virtual prototype, and the software will output a prediction of the fluid dynamics and related physical phenomena [6]. Therefore, CFD is a sophisticated computationally-based design and analysis technique.

ENGINE GEOMETRY AND EXPERIMENT

The dimension of the cylinder measured to design the model using SOLIDWORK. From the measurement, the data collected as shown in Table 1.

Table 1: The engine specification

Parameter	Size / Feature
Cylinder Type	Single Cylinder, piston ported
Compression Type	Crankcase compression
Displacement	30.5 cm^3
Scavenged Concept	Multi-port Loop scavenged
Bore x Stroke	35 x 30 mm
Exhaust port opening / closing	101.20 CA/ 259.2 CA ATDC
Scavenged port opening/ closing	241.40 CA/ 155.20 CA ATDC

Figure 5 show the schematic diagram layout of the experimental testing. Measurement for pressure data and crank angle data are using Kistler Pressure Transducer and Crank Angle Encoder respectively. On stage numerical analysis, experiment data totally utilize to study the behaviour of in-cylinder analysis. The measurement location has chosen to determine the main features of the pressure data at the intake port and exhaust port. Pressure transducer (Kistler M8x0.75) was used to measure in-cylinder pressure mounted in the injector hole and (Kistler M5x0.8) mounted in the intake port and exhaust port. The cylinder head was drilled using CNC Milling in order to make a hole and thread to place the pressure transducer.

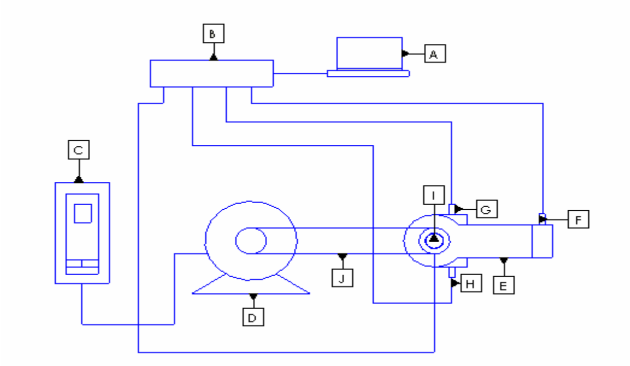

Figure 5: Schematic diagram

The experiment is carried out using the motored method which the engine combustion is generated using motor and without firing. The data is collected at different engine speed from the engine idle speed to maximum engine speed as shown in Table 2.

Table 2: Engine Operation Condition

Frequency (Hz)	Engine Speed (rpm)
20	682
25	754
30	788
35	939
40	1061
45	1239
50	1384
55	1535
60	1655

THE ENGINE MODEL DEVELOPMENT

Computer Aided Design (CAD) model was build based on the Tanika BG-328A engine geometries. The geometries are carried out using Coordinate Measuring Machine (CMM) to make sure the geometry of the model is accurate to the actual engine geometries. This is very important to get the accurate result of the numerical analysis. The In-Cylinder volume as shown in Figure 6 below is generated by COSMOS FloWork and divided into half of the geometry. The In-Cylinder volume is divided to three volume which are Scavenged port volume, Exhaust port volume and Cylinder Clearance volume [7] and saved as .ACIS file. The volume meshed using gambit 2.16 Pre-processor and saved in TGrid to transfer into FLUENT software.

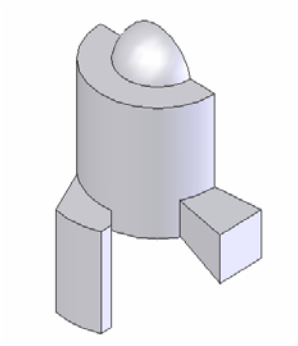

Figure 6: The In-Cylinder Volume Generated by COSMOS FloWork

GRID GENERATION (GAMBIT)

The mesh was constructed using three separated blocks that represent the scavenged port, exhaust port and cylinder [8]. Their cell faces at the interface between each other matched exactly setting in the TGRID stage. The volume is divide and the meshed to fulfil the requirement of the dynamic mesh method. Layered hexahedral meshed is specified for moving part and unstructured tetrahedral element is for stationary region using Gambit 2.16 Pre-processor and TGrid.

DYNAMIC MESH SETUP

The dynamic mesh model in FLUENT can be used to model flow where shape of the domain is changing with time due to motion on the domain boundaries. In this case, dynamic mesh is set to calculate the piston location that is proportional to the crank angle. The update of the volume mesh is handled automatically by FLUENT at each time step based on the new positions of the boundaries. There are three dynamics mesh methods applicable in FLUENT that were Smoothing, Layering and Remeshing. Under in-cylinder, the parameters were set following the data as shown in Table 3.

Table 3: The in-cylinder parameters

Item	Specification
Crank Shaft Speed (rpm)	1061
Starting Crank Angle (deg)	0
Crank Period (deg)	36
Crank Angle Step Size (deg)	1
Piston Stroke (mm)	30
Connecting Rod Length (mm)	57.10
Piston Stroke Cutoff (mm)	10.00
Minimum Valve Lift (mm)	0.100

BOUNDARY CONDITION

Boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are important in the FLUENT simulation and to be specified appropriately during simulations. The boundary condition (BC) at the intake ports were measured from experimental method [9]. The average scavenged pressure for speeds that is 40 Hz (1061 rpm) is used as the boundary condition in the simulation. The average pressure for the scavenged port is shown in Table 4.

The flow at the exit of the exhaust port to atmosphere was set to constant static pressure equal to atmospheric pressure [10]. The cylinder was initialized with an inlet pressure used data as shown in Figure 7. The wall temperature was fixed at 300 K. The turbulence intensity and scale at both scavenged port entrance and the exhaust port exit was set at 10 % of mean velocity and 2 mm, respectively.

Table 4: The average pressure at scavenged port

Frequency/ Speed	Average Scavenged Pressure
40 Hz / 1061 rpm	2.182 Bar

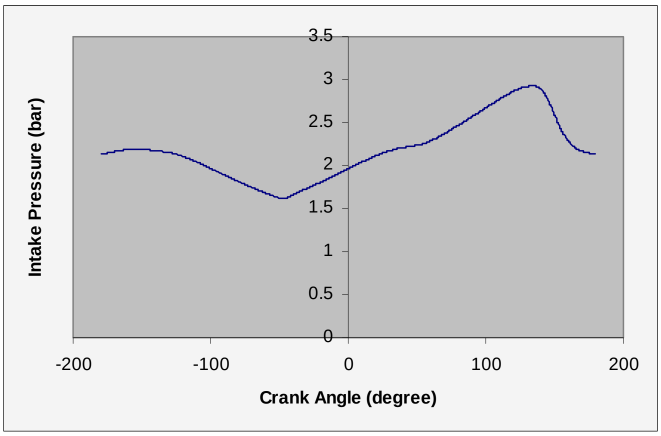

Figure 7: The experiment data as the pressure inlet in simulation for 1060 rpm

RESULT AND DISCUSSION

From the experimental analysis, a set of data is collected when the engine is running at the specific speed. The data will be used as the boundary condition in the numerical analysis. The data result from the experimental analysis for the 1061 rpm speed is shown in Figure 8. The cylinder pressure data was measured from the experiment is shown in Figure 9. The data from the experiment will be compared with the simulation data for the validation.

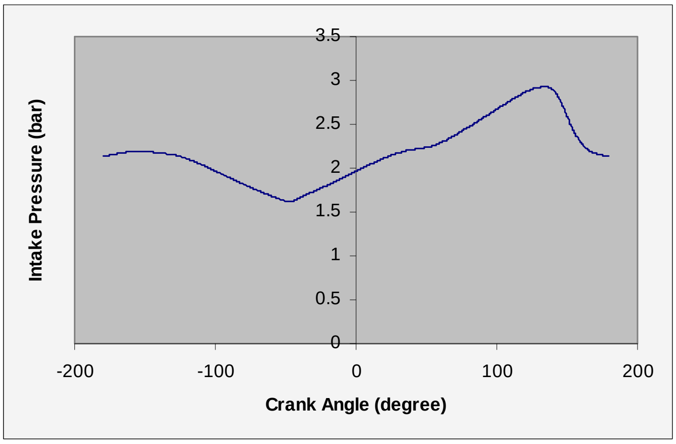

Figure 8: The experiment data as boundary condition in numerical analysis for 1061 rpm speed

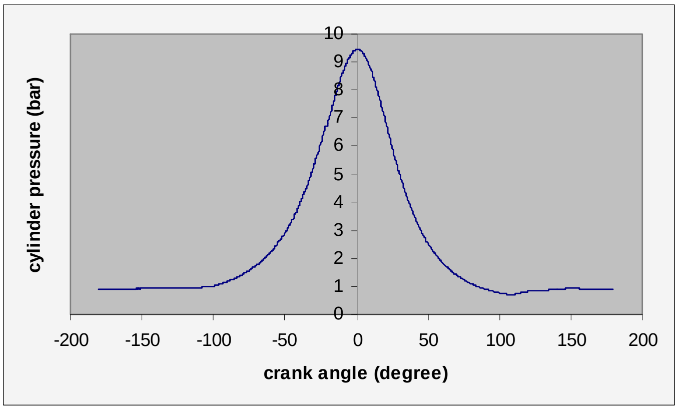

Figure 9: The in-cylinder pressure from the experimental analysis for 1061 rpm speed

NUMERICAL ANALYSIS RESULT

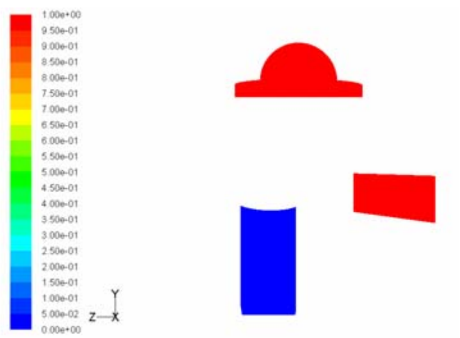

Figure 10: Burned mass fraction for 0 CA

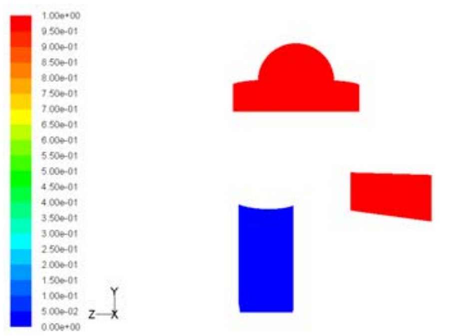

Figure 11: Burned mass fraction for 36 CA

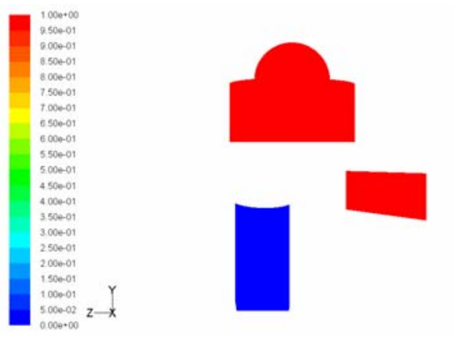

Figure 12: Burned mass fraction for 72 CA

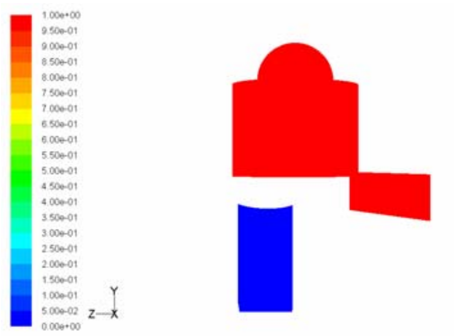

Figure 13: Burned mass fraction for 108 CA

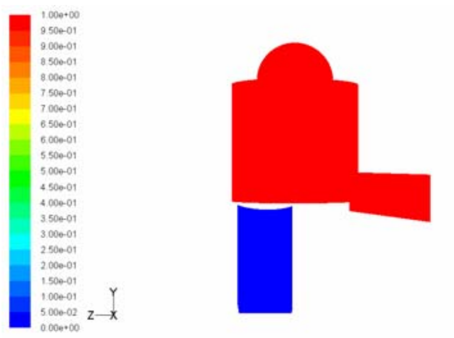

Figure 14: Burned mass fraction for 144 CA

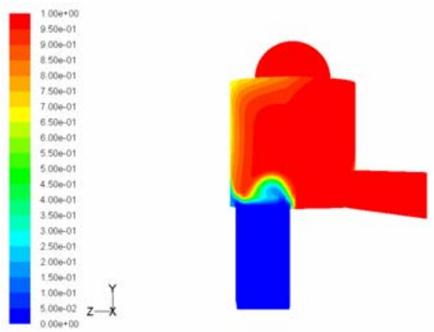

Figure 15: Burned mass fraction for 180 CA

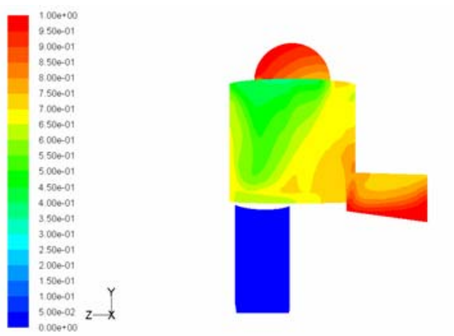

Figure 16: Burned mass fraction for 216 CA

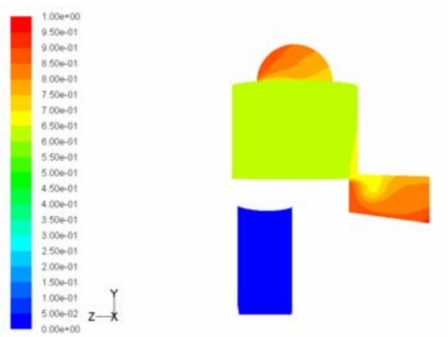

Figure 17: Burned mass fraction for 252 CA

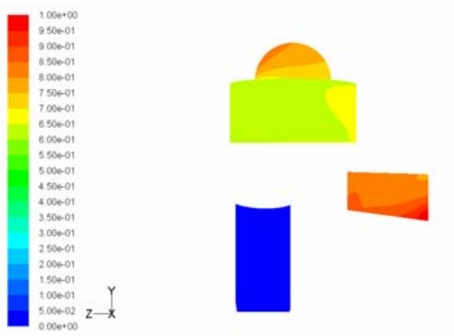

Figure 18: Burned mass fraction for 288 CA

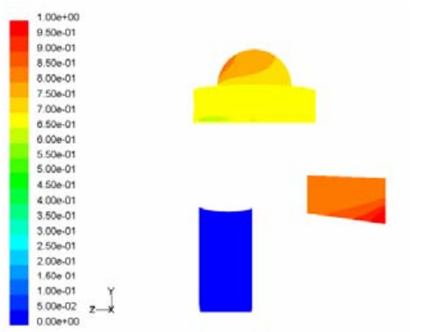

Figure 19: Burned mass fraction for 324 CA

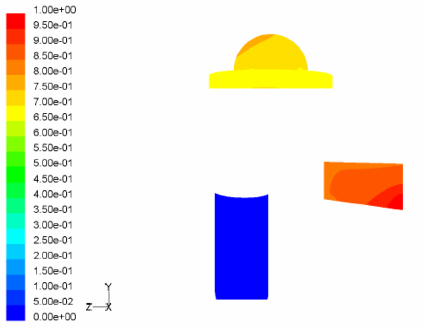

Figure 20: Burned mass fraction for 360 CA

From the experiment, the maximum pressure for the 1061 rpm is 9.282 Bar but for the simulation, the maximum pressure value is 9.571 Bar. There is 3.02 % relative error for the experimental method and the numerical method. The in-cylinder pressure graph for both experimental and simulation is shown in Figure 21. The relative errors are caused by the several factors such as the geometries dimension of the CAD model for the simulation, the experiment condition and the simulation assumptions. The CAD model is not accurate to the actual engine because there is some limitation during measuring the model. Some complex engine geometry needs to be simplified and there is parallax error during measuring the engine model.

The surrounding such as the room temperature and improper method handling the experiment equipment also influences the experiment. During the simulation stages, there is much assumption that is made such as the temperature and the pressure inlet of the simulation is set to constant. This is different to the experiment, which is the pressure, and the temperature is not constant and influenced by the surrounding. During the simulation also, the effect of the surrounding such as the radiations, heat transfer, temperature change and the pressure changed is neglected. Therefore, the experiment result and the simulation result are not accurate because of the several factors as mentioned.

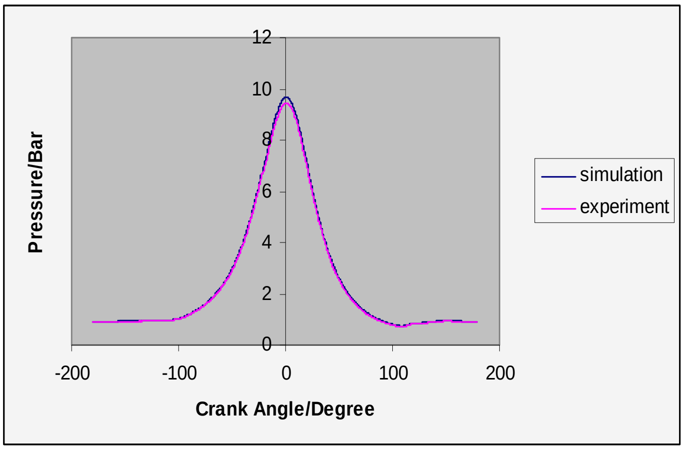

Figure 21: The validation of the in cylinder pressure for the 1061 rpm speed

CONCLUSION

The aim of the research is to do numerical analysis to predict the in-cylinder flow of two-stroke spark ignition engine. The simulation is carried out using computational fluid dynamics (CFD) Fluent. The operating and boundary conditions are measured using the experiment using the motored method, which is no combustion inside the cylinder. The experiment data then used in the simulation that is three-dimensional and unsteady condition. Then the result from the experiment and the simulation is validated.

The study of in-cylinder characteristic is important to understand the scavenging process of the two-stroke engine. This is important to design new two-stroke engine to produce low emission that will meet current and future emission regulation. The research is carried out using the firing method to understand the actual combustion of the two-stroke engine

The research about the effect of the combustion chamber design, scavenged port design, exhaust port design, spark plug location and other parameter of the two-stroke engine to understand the scavenging process of the two-stroke engine. The dimensioning of the actual engine model is carried out using three-dimensional scanner to get the accurate dimension of the engine.

ACKNOWLEDGEMENT

The authors would like to express their thanks to the Universiti Malaysia Pahang for funding this project.

REFERENCES

[1] Blair, G. The Design and Simulation of Two-Stroke Engine. Society of Automotive Engineers Publishers, Ins.Warrendale, PA. 1996.
[2] Marco Nuti, "Emission from Two-Stroke Engine," Senior Staff Scientist Mercury Marine, SAE Order No. R-223, 1998
[3] R. K. Rajput, "A Textbook of Internal Combustion Engine," M.E. (Heat Power Engg.) Hons.-Gold Medallist; Grad. (Mech. Engg. & Elect. Engg.); M.I.E (India); M.S.E.S.I. ; M.I.S.T.E. ; C.E. (India) PATIALA, 2005
[4] Heywood, J.B. Internal Combustion Engine Fundamental. New York, McGraw-Hill, 1988.
[5] R.G Kenny. J.G Smyth, R. Fleck, G.P Blair, "The Scavenging Process in the Two-Stroke Engine Cylinder," Dritte Grazer Zweiradtagung, Technische Universitat, Graz, 13-14 April, 1989.
[6] J.G Smyth, R.G Kenny, G.P Blair, "Steady Flow Analysis of the Scavenging Process in a Loop Scavenged Two-Stroke Cycle Enginea Theoretical and Experimental Study," SAE paper No. 881267, Society of Automotive Engineers, Warrendale, Pa.,1988
[7] Gordon P. Blair, "Design and Simulation of Two-Stroke Engines," The Queen's University of Belfast, SAE Order No. R-161, 1994.
[8] Grasso, I.F, Wey, M.J, Bracco, F.V and Abraham, J., Three Dimensional Computational of Flows in a Stratified Charge Rotary Engine. SAE 870409, 1987
[9] Thornhill, D. Douglas, R. Kenny, R and Fitsimons, B., An Experimental Investigation into the Effect of Bore/Stroke Ratio on a Simple Two-Stroke Cycle Engine. SAE 1999-01-3342, 1999.
[10] B. D. Ragunathan and R. G. Kenny, "CFD Simulation and Validation of the Flow within a Motored Two-Stroke Engine," The Queen's University of Belfast, SAE Technical Paper Series 970359, 1997