International Journal on Applications in Mechanical and Production Engineering
Volume 1: Issue 1: January 2015, pp 7-12. www.aetsjournal.com

Investigation on the Effects of Nozzle Geometry in a Di Diesel Engine Fuelled with Methyl Ester Blends

K.Ramesh , S.Nirmalkumar

***Abstract*—:** The aim of the paper is to link nozzle geometry and its influence of spray characteristics, with combustion characteristics in the chamber. The work is to modify the number of holes of the nozzle geometry and to test the performance, emission characteristics fuelled with methyl ester blends. In the present scenario, diesel engine fuel injection system is designed to attain higher injection pressure. The geometry of the nozzle in an injector plays a vital role in controlling diesel spray atomisation. In order to, reduce the fuel droplet size, the nozzle hole size reduced on small momentum. The modified nozzle geometry was designed and analysed using CFD fluent software. The fluid dynamic performances are compared for cylindrical, k hole and Ks hole nozzles. Compared to cylindrical and k hole an improved conditions in terms of flow uniformity, vapour suppression was obtained in Ks-holes and a significant reduction of jet was observed.

Keywords: Spray characteristics, Number of holes, Injection pressure, Fuel droplet size, Flow uniformity, Vapour suppression, Ansys 14.5, Solidworks 2013, ICEM CFD 14.5, Effectiveness of cooling tower.

I. INTRODUCTION

Pollutant emission reduction is currently considered to be one of the most important targets of our society. Legislation about pollution coming from vehicles is getting more and more restrictive. So that research is focused to understand physical processes involved in the engine behavior.

One of the important subjects in these studies on Diesel engines are the behavior of fuel once it is injected in the combustion chamber, and its interaction with air. In these terms, it is well known that nozzle geometry and cavitation strongly affect to evaporation and atomization processes of fuel.

The aim is to link nozzle geometry, and its influence on spray characteristics with combustion development in the chamber. For this purpose, three sac nozzles with different orifices degree of conicity, have been used. In the present work, CH and OH chemiluminescence techniques are used to thoroughly examine combustion process. The analysis of the results allows linking nozzle geometry, spray behavior and combustion development.

In modern Diesel engines, the combustion efficiency is strongly correlated to the spray behavior.Focusing the attention on the geometric layout, one of the most interesting research field has regarded the injector hole shape influence on the Diesel spray characteristics. From this point of view, during the last years many Diesel injector manufacturing companies have switched from the classical cylindrical injector hole shape to the so called K-hole layout characterized by a progressively reduction of the hole cross-sectional area between inlet and outlet sections.

Both K-hole and Ks- hole are classified by the so called K-factor parameter defined as:

K-factor = $(d_{in}-d_{out})/10$

Where d_{in} and d_{out} are inlet and outlet hole diameters respectively. For automotive applications, the K-factor is usually bounded between 1.1 and 2.

The main motivation of the present work resides in the lack of data available in literature about the detailed comparison of the fluid dynamic performance characterizing the cylindrical, K and Ks injector hole geometries. To fill this lack, for each of the identified hole configuration a fully transient multiphase 3D-CFD simulation was performed in order to evaluate the holes fluid dynamic performances under identical operating conditions. All the considered hole geometries were installed on the same injector body.

Brusianiet al (2014)in their paper “Influence of the diesel injector hole geometry on the flow conditions emerging from the nozzle” studied the nozzle layouts and simulation approach of the nozzle design. In this paper, with respect to the cylindrical hole shape, the conical holes showed a quite evident reduction of the mass flow rate mainly linked to the reduction of the hole outlet section.

Zhanget al (2012) **in** their paper “Effects of highly dispersed spray nozzle on fuel injection characteristics and emissions of heavy-duty diesel engine” studied the fuel injection rate , Shorter injection duration , spray tip penetration , Spray cone angle , Spray projected area of conventional nozzle. In this paper, the tested heavy-duty engine equipped with the highly dispersed nozzle has lower emission level and BSFC compared with that equipped with the conventional nozzle, which indicates that the highly-dispersed nozzle can improve the in-cylinder mixture combustion

Payriet al (2009) in their paper “Effects of nozzle geometry on direct injection diesel engine combustion process” studied the nozzle geometry, and its influence on spray characteristics with combustion development in the chamber. Also to check the influence of nozzle geometry on combustion process under real engine conditions.

Herfatmaneshet al(2013) in their paper “ Simultaneous imaging of diesel spray atomization and evaporation processes in a single-cylinder compression ratio diesel engine” studied the spray atomizationand evaporation processes during the

K.Ramesh is Assistant Professor (Senior Grade), Department of Mechanical Engineering, Government College of Technology-Coimbatore, Tamilnadu, India (Email : dr_ramesh@live.com)

S.Nirmalkumar is PG Scholar (Thermal Engineering), Department of Mechanical Engineering, Government College of Technology-Coimbatore, Tamilnadu, India (Email: nirmalmegct@gmail.com)

pre-combustion phase of a diesel engine were measured at an injection pressure of 1200 bar and the engine speed of 1500 rpm.

Winklhofer (1996) in their paper " An experimental database for diesel spray combustion" studied the data describing the influence of fuel injection pressure, nozzle hole diameter , and nozzle type on spray discharge velocities, spray propagation and flame liftoff distance. The study also includes an analysis of the influence of the nozzle hole geometry i.e the inflow conditions into the nozzles holes on spray tip propagation

Solsjoet al (2013) in their paper " Lift-off and stabilization of n-heptane combustion in a diesel engine with a multiple-nozzle injection" studied the injection strategy, employing different nozzle configurations, allows for the quantification of the impact of varying inter-jet angle spacing in the presence of swirl.

PROPERTIES OF PALM OIL METHYL ESTER:

Technical properties	Description
Common name	Biodiesel (bio-diesel)
Common chemical name	Fatty acid (m)ethylester
Chemical formula range	C14 – C24 methyl ester or C15 – C25 H28-48
Kinematic viscosity range (mm^2/s, at 313K)	3.3 – 5.2
Density range (kg/m^3, at 288K)	860 – 894
Boiling point range (K)	➢ 475
Flash point range (K)	430 – 455
Distillation range (K)	470 – 600
Vapour pressure (mm Hg, at 295K)	< 5
Solubility in water	Insoluble in water
Physical appearance	Light to dark yellow, clear liquid
Odor	Light musty or soapy odor
Biodegradability	More biodegradable than petroleum diesel
Reactivity	Stable but avoid strong oxidation agents

II. METHODOLOGY

All the simulations were performed by using the Reynolds-averaged navier-stokes(RANS) simulation approach. The commercial fluent v12 CFD software was considered for the analysis. Exploiting the injector body symmetry, only a sixth of the real fluid dynamic volume was considered.

The initial mesh was generated at a needle lift equal to 5µm(simulation time equal to 0s). The domain was discretized by a fully unstructured hexahedral approach taking particular attention to the mesh structure close to the hole entrance.

Starting from the initial lift(5µm), during the simulation the needle was moved following the profile. To manage the change-in-shape of the computational domain the well-known layering approach was adopted. On the basis of this method, the needle surface and the first four hexahedral layer attached to it were moved rigidly.

The rigid motion produced the increment on the fifth cell layer height till a maximum value defined by the user. Over this maximum cell height value, an additional hexahedral cell layer was added just below the layering plane.

During the needle closing phase the added cell layers were progressively collapsed to allow the needle to reach its initial position (5µm from the nozzle surface at seating zone). The choice to move the first four hexahedral cell layers attached to the needle surface rigidly together the needle itself was laid down the law to imprive the near wall solution also on the needle moving wall.

On the basis of the mesh motion set-up, during the simulation the mesh size was ranging from 300000 cells at needle initial position to 380000 cells at needle maximum lift.

Numerical Set-Up:

All the multiphase simulations were performed adopting a first order implicit transient formulation and a fixed time step.

The approach was coupled to the Singhal et al. cavitation model to manage the liquid/vapor transition. The singhal cavitation model is based on the assumption of a perfect mixing between the liquid and the vapor phases and takes into account all first order effects of a multiphase flow as : phase change, bubble dynamics, turbulent pressure fluctuations, and non-condensable gases.

The flow-wall interaction was managed by the non-equilibrium wall function based on a law of the wall for mean velocity sensitized to pressure gradients.

To reproduce the turbulence effects on the mean flow the realizable k-£ turbulence model was considered. Pressure, momentum, and volume fraction equations were discretized by using a second-order differencing schemes. All the other equations were discretized adopting the first-order Up-wind scheme.

Boundary Conditions:

On the inlet section a total pressure boundary condition was fixed while on the outlet hole section a static pressure valve was imposed.

The working fluid was represented as a single-component fuel having the same overall physical properties of the ISO-4113 characterized by a saturation pressure equal to 2370 Pa at the considered fuel temperature. This phase-change pressure threshold was corrected during the simulation by the Singhal cavitation model to take into account the effect of the turbulence- induced pressure fluctuations.

The fuel temperature variation inside the injector was considered negligible. Therefore, the nozzle flow was considered as isothermal.

The initial values for the turbulent kinetic energy(k) and its dissipation rate (£) at Inlet/Outlet boundaries were set on the basis of the following equations :

- $K=3/2\ (UI)^2$
- $\sum = C_\mu^{3/4}\,(k^{3/2}/\ L)$

Where,

- I= Turbulent intensity
- L=Nozzle length

The turbulent intensity I was set to 5% while the turbulent length scale l was set to the 7% of the inlet/outlet section characteristic length.

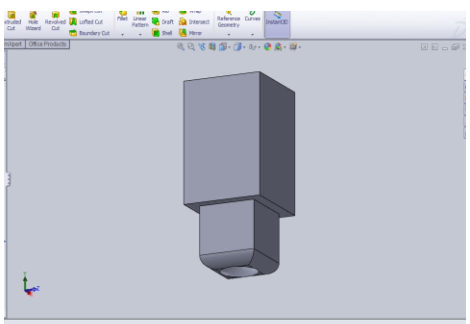
The multiphase simulation strategy presented was extensively validated against the experimental throttle-flow cases concerning three quasi-2D geometries with the same length/ thickness(1000µm/300µm) and different k factors.

III. SOLIDWORKS MODELING

Single Hole Nozzle Modeling

Specifications:

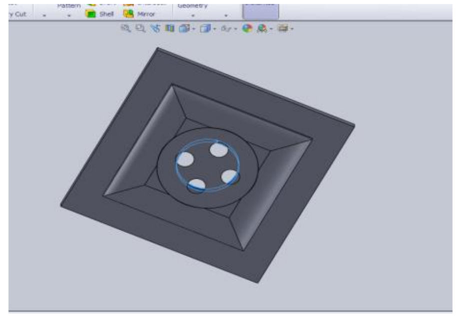
- Common rail fuel injector : solenoid activated
- Fuel injector nominal nozzle
  outlet diameter : 0.090 mm
- Nozzle K factor : 1.5
- Discharge coefficient at
- 10 Mpa pressure drop : 0.86
- Number of holes : 1
- Orifice orientation : Axial



Multi Hole Nozzle Modeling

Specifications:

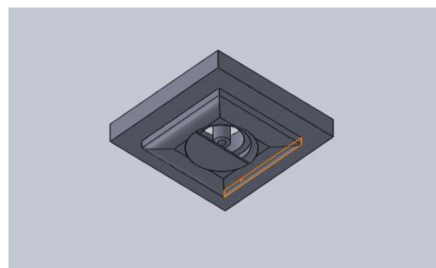
- Common rail fuel injector : solenoid activated
- Fuel injector nominal nozzle
  outlet diameter : 0.130 mm
- Nozzle K factor : 3.4
- Discharge coefficient at
- 10 Mpa pressure drop : 0.92
- Number of holes : 4
- Orifice orientation : Axial



Pintle Hole Nozzle Modeling

Specifications:

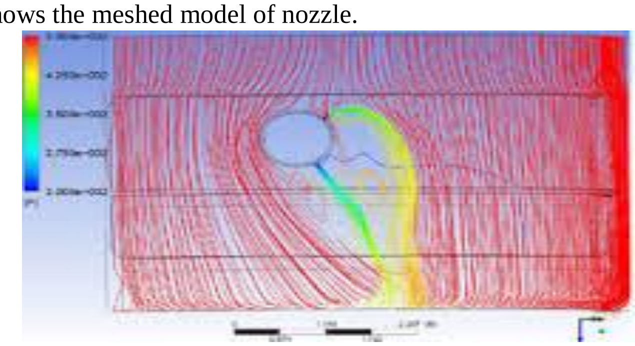
- Pintle angle : 8-30 degree
- Pintle size : Cylindrical/Conical
- Fuel injector nominal nozzle
- outlet diameter : 0.254 mm
- Nozzle K factor : 4.8
- Discharge coefficient at
- 10 Mpa pressure drop : 1.3
- Orifice orientation : Axial



IV. PROCESSING OF COMPUTATIONAL FLUID DYNAMICS

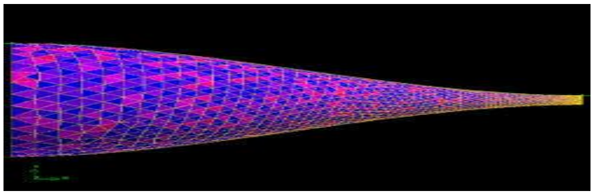
All the nozzle models have been imported as the geometries into IGES (Initial Graphics Exchange Specification) format. Then the models have been meshed using ICEM CFD software.

The mesh is then ready to apply boundary conditions and for writing to the desired solver. By following above steps all the nozzle models are meshed and its geometries are repaired and then imported to FLUENT software and boundary conditions are applied and the solution has been initialized.Fig shows the meshed model of nozzle.

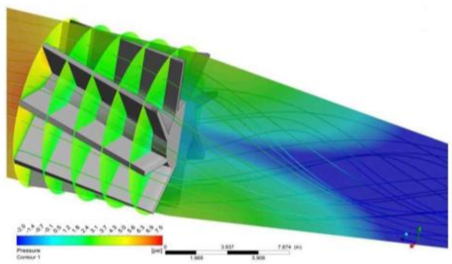


Meshed Model of single hole nozzle(cylindrical hole)

International Journal on Applications in Mechanical and Production Engineering
Volume 1: Issue 1: January 2015, pp 7-12. www.aetsjournal.com



Meshed Model of multi hole nozzle(k hole)



Meshed Model of Pintle hole nozzle(ks- hole)

V. RESULTS AND DISCUSSION

Time Vs Injection velocity

Cylindrical hole (Diesel fuel):

Time(ms)	Injection velocity(m/s)
0.13	450
0.33	520
0.53	520
0.73	520
0.93	520
1.1	410

k hole(Diesel fuel):

Time(ms)	Injection velocity(m/s)
0.13	490
0.33	570
0.53	570
0.73	570
0.93	570
1.1	540

ks hole(Diesel fuel):

Time(ms)	Injection velocity(m/s)
0.13	510
0.33	615
0.53	615
0.73	615
0.93	615
1.1	600

Time Vs Mass flow rate

ks hole(Diesel fuel):

Time(ms)	Mass flow rate (kg/s)
0.13	0.0045
0.33	0.0054
0.53	0.0054
0.73	0.0054
0.93	0.0052
1.1	0.0046

Time Vs Turbulent Kinetic Energy

Cylindrical hole(Diesel fuel):

Time(ms)	Turbulent Kinetic Energy(m^2/s^2)
0.13	990
0.33	1750
0.53	1740
0.73	1740
0.93	1730
1.1	1200

K hole (Diesel fuel):

Time(ms)	Turbulent Kinetic Energy(m^2/s^2)
0.13	1200
0.33	1400
0.53	1300
0.73	1400
0.93	1400
1.1	1100

Ks hole (Diesel fuel):

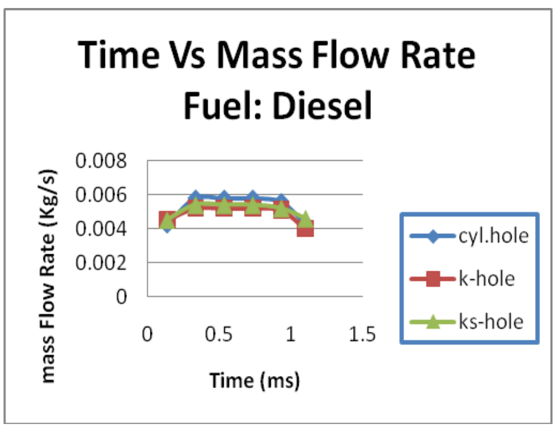
Time(ms)	Turbulent Kinetic Energy(m^2/s^2)
0.13	700
0.33	900
0.53	750
0.73	750
0.93	900
1.1	600

Time Vs Spray Tip Penetration:
Single hole Nozzle (Diesel fuel): Injection Pressure=120 MPa

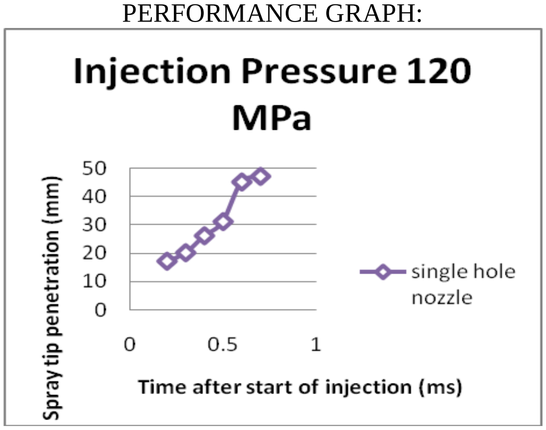
Time(ms)	Spray Tip Penetration (mm)
0.2	17
0.3	20
0.4	26
0.5	31
0.6	45
0.7	47

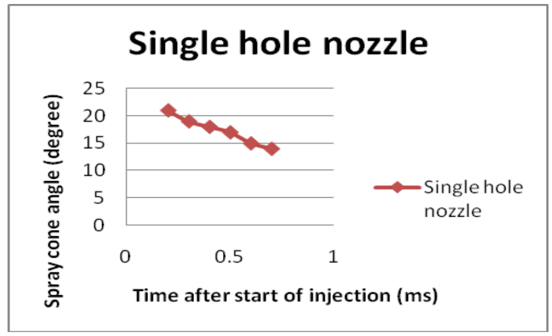
Time Vs Spray cone angle
Single hole Nozzle (Diesel fuel): Injection Pressure=120 MPa

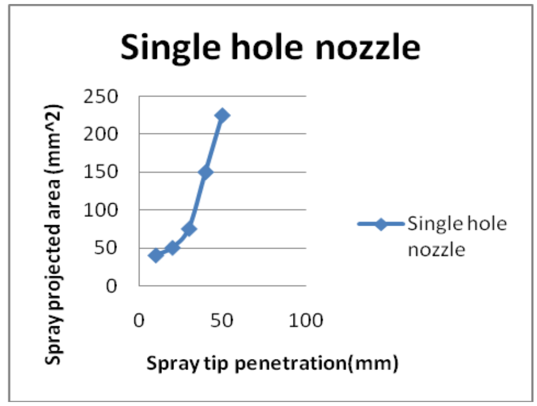
Time(ms)	Spray cone angle(Degree)
0.2	21
0.3	19
0.4	18
0.5	17
0.6	15
0.7	14

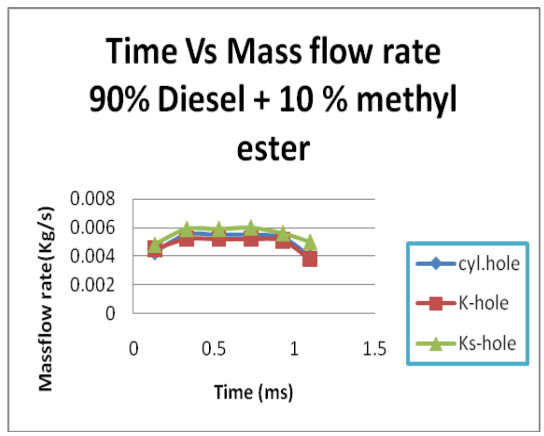


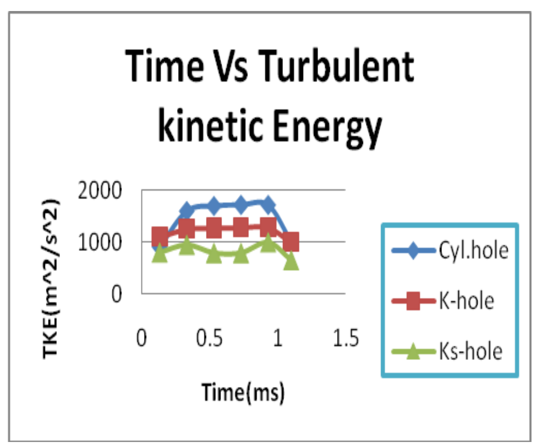
PERFORMANCE GRAPH:

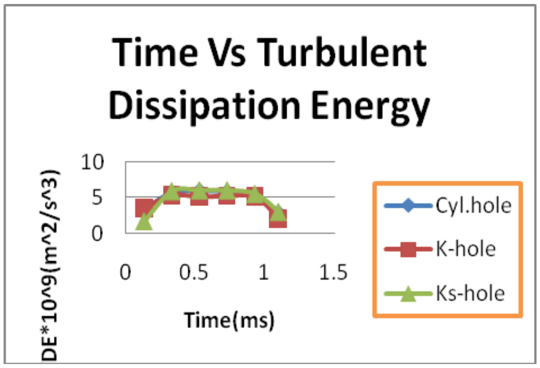


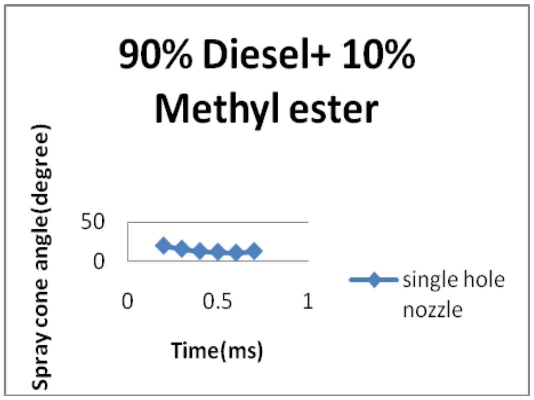












VI. CONCLUSION

In the present work, the fluid dynamic performance of cylindrical hole nozzle, k-hole nozzle and ks-hole nozzles are compared.A 3D-CFD fully transient multiphase approach was adopted for nozzle layouts of the same engine size.When compared with cylindrical hole and k-hole, the ks-hole suppresses the vapour generation over the hole length with an advantage of cavitation erosion risk reduction. In case of flow uniformity ks-hole produced significant reduction of the jet intensity for diesel methyl ester blends.

REFERENCES

[1] Federico Brusiani , StefaniaFalfari , Pieropelloni , 2013 , " Influence of the Diesel injector hole geometry on the flow conditions emerging from the nozzle" , Energy Procedia 45 (2014) , 749-758.

[2] Guiyang zhang , XinqiQiao , Xuelong Miao , Jianhai Hong , JinbaoZheng , " Effects of highly dispersed spray nozzle on fuel injection characteristics and emissions of heavy-duty diesel engine", Applied Thermal Engineering 61 (2013) , 121-128.

[3] E.Winklhofer , "An experimental database for diesel spray combustion" , Combustion , 1996 / pp 2541-2547.

[4] Mohammad Reza Herfatmanesh , Mohammad Reza Anbari Attar , Hua Zhao , " Simultaneous imaging of diesel spray atomization and evaporation processes in a single cylinder CR diesel engine" , Experimental Thermal and fluid science 50(2013), 10-20.

[5] J.M.desantes ,J.M.Garcia-oliver , J.M.Pastor , J.G.Ramirez-Hernandez, " Influence of nozzle geometry on ignition and combustion for high-speed direct injection diesel engines under cold start conditions" , Fuel 90 (2011) , 3359-3368.

[6] R.Payri ,F.J.salvador , j.Gimeno , J.de la Morena , " Effects of nozzle geometry on direct injection diesel engine combustion process" , Applied Thermal Engineering 29 (2009), 2051-2060.

[7] R.solsjo ,M.Jangi , C.Chartier , O.Andersson , X.S.Bai , " Lift-off and stabilization of n-heptane combustion in a diesel engine with a multiple-nozzle injection" , Proceedings of the combustion institute 34 (2013), 3031-3038.

[8] EzioMancaruso , Luigi sequino , Bianca M.Vaglieco , " First and second generation biodiesels spray characterization in a diesel engine" , Fuel 90 (2011), 2870-2883.

[9] CenkSayin , MetinGumus , Mustafa Canakci , " Influence of injector hole number on the performance and emissions of a DI diesel engine fuelled with biodiesel-diesel fuel blends", Applied Thermal Engineering 61 (2013), 121-128.

[10] Mikael Lindstrom, " Injector nozzle hole parameters and their influence on real DI diesel performance", TRITA-MMK 2009: 01, ISSN 1400-1179 , ISRN/KTH/MMK/R-09/01-SE.

[11] A.K.Sarma , P.S.Misra , Syed T.Haque, " Combustion analysis of JATROPHA Methyl ester and its ethanol and acetone blends in a diesel engine" , International journal of Emerging Technology and advanced Engineering , volume 3 , (2013), 51-57.

[12] AlirezaValipour, " Experimental Combustion analysis of Biodiesel Spray with Hot surface ignition" , International journal of Mechanical Engineering, volume 2, Issue 1,January 2014, ISSN 2321-6441.

[13] CheahSiew Lee, " Analysis of Engine performance using Palm Oil Methyl Ester" , University of southern Queensland ,Faculty of Engineering and surveying, 2004.

[14] Mahr B, " Future and Potential of Diesel Injection Systems", THIESEL 2002 Conference on Thermo-and Fluid Dynamic Processes in Diesel engines.

[15] Mulemane A, Han J, Lu P, Yoon S.et.al., " A CFD study of the effect of the needle movement on the Cavitation Pattern of Diesel injectors", SAE Technical paper 2009-24-0025, 2009.

[16] Montanaro A, Allocca L , Ettorre D, Lucchini T, Brusiani F, Cazzoli G, " Experimental Characterization of high-pressure impinging sprays for CFD Modeling of GDI engines", 2011-01-0685, SAE World Congress, Detroit,Michigan, USA, April 2011.