Volume : 4 | Issue : 7 | July 2015 • ISSN No 2277 - 8179

Research Paper

Cfd Simulation of Injected Fuel Spray in Marine Diesel Engines

Engineering

KEYWORDS : Diesel engines, CFD, Diesel injection, Break-up models

Ph.D. Student Mahran Dawwa

Department of Naval Electromechanics,Constanta Maritime University, Romania, Constanta

ABSTRACT

This paper discusses the simulation of injection process in marine diesel engines by the use of computational fluid dynamics method (CFD). The fuel injection process has a huge impact on the diesel engine parameters, and due to the raise in concern about reducing gas emissions and improving the fuel economy of marine diesel engines, the concentration about improving the fuel injection process has increased. Hence, many researches and studies have been carried out for modeling and simulating the injection process in order to enhance the mixture formation inside the combustion chamber. Diesel spray is subjected to different physical processes (penetration, primary break-up, and secondary break-up) which will be explained and illustrated in details. The simulation in this study was carried out by the use of technical data of Hino W04D high speed marine diesel engine and ANSYS simulation software codes.

INTRODUCTION

The injected diesel inters the combustion chamber space through the injector nozzles under a high pressure; the pressure of injected diesel varies according to the engine type and the type of fuel injection system, it reaches several hundreds of bars.

After the injected fuel inters the combustion chamber space, it spreads inside it forming a cloud of fuel droplets and high pressured hot air, the meet between the hot air with fuel droplets will lead to increase the temperature of fuel droplets, which leads to evaporate the droplets of fuel and ignite the mixture of evaporated fuel and hot air, causing to help in igniting the rest of evaporated fuel. The atomization, evaporation, and ignition processes must be occur in very short time, because of the short time that available to complete the combustion process in internal combustion engines which estimated by just milliseconds. In this section we are interested only in explaining the fuel evaporation process.

The evaporation of injected fuel and the process of transformation from liquid to gas state come before fuel ignition process, and as we mentioned before the heat of compressed air helps in the evaporation process but it is not enough to ensure the completion of fuel evaporation during milliseconds. Hence, the injected fuel must be dispersed to small fuel droplets in order to increase the interface between these droplets and hot compressed air, as well as these droplets should be dispersed away from each other as far as possible to ensure better evaporation processes. The requirements that ensure the good evaporation process can be obtained by two different types of spray breakup, the first type called primary break up, primary breakup occurs near to injector's holes after the fuel spray traveled a short distance in the combustion chamber space it breaks up into a parcel of drops.

After the primary breakup the fuel drops will be broke another breakup which called the secondary breakup, causing to minimize the size of fuel drops and form smaller droplets, the pressure of injected fuel and the degree of turbulence inside the combustion chambers are considered the most important factors that affect the secondary breakup of fuel spray. Then the atomized fuel droplets will evaporate to form a mixture of evaporated fuel and compressed air.

THE SIMULATION OF DIESEL SPRAY

For simulating the breakup process in marine diesel engines, there are many models which used for this purpose such as Taylor Analogy Breakup Model (TAB), Wave Breakup Model, KH-RT Breakup Model, and Stochastic Secondary Droplet Model (SSD). In our study KH-RT Model will be used, KH-RT model is a combination between Kelvin-Helmholtz model and Rayleigh-Taylor model. The spray is divided according to KH-RT method to three regions, first region is the spray liquid core, primary breakup region, and secondary breakup region.

RESULTS DISCUSSION

The simulation of injected diesel was performed only on a sector of engine combustion chamber; the sector area is determined according to the numbers of injector nozzles, where they are 5 in this study. Hence, it can be seen that the sector angle is 72 degree.

The first figure illustrates the mass flux of injected diesel with crank angle; the crank angle 720 degree represents the top dead center, it can be seen that the injection starts at 11 degrees before TDC, and ends at 10.5 degrees after top dead center. The flux reaches its highest value after the start of injection by 5 degrees.

The other figures show the simulation of injection process at different crank angles from just one nozzle, and the structure is the cylinder sector that was decomposed during "Geometry setup". The contour plot shows the velocity magnitude of the diesel particles.

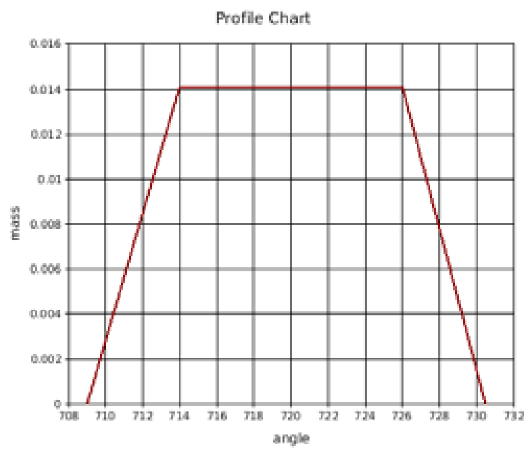

Figure 1: mass flux graph of injected diesel with Crank angle

It can be noticed at figure 2 that the difference in velocity magnitude between diesel particles still small, and the starting of diesel spray breakup phenomenon can be realized as well, which happens in the vicinity of the nozzle. Then the difference in velocity magnitude of diesel particles gets higher and higher with the proceeding of injection process.

The highest velocity magnitude is in the middle of spray cone, and decreases as the spray moves on because of the interaction between the injected diesel and the compressed air inside the combustion chamber, which can be seen clearly at figure 3. The velocity magnitude reaches the highest value at crank angle (722) figure 5.

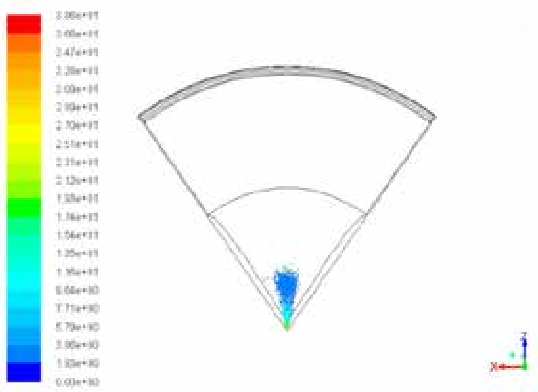

Figure 2: velocity magnitude of particle traces at 710 (deg)

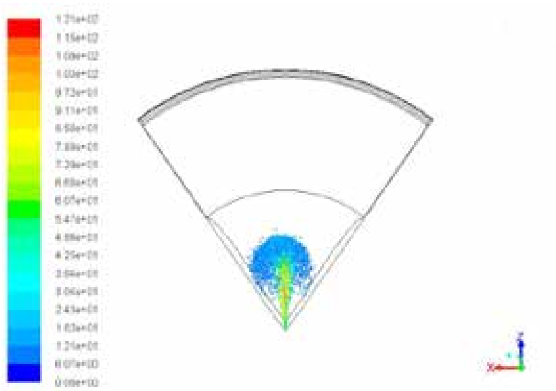

Figure 3: velocity magnitude of particle traces at 716 (deg)

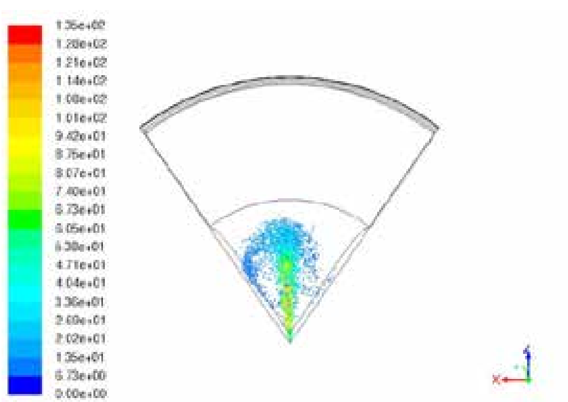

Figure 4: velocity magnitude of particle traces at 718 (deg)

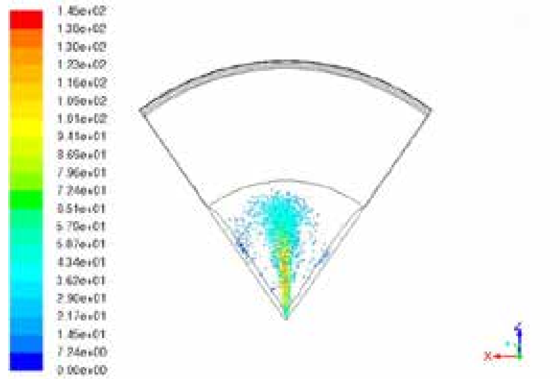

Figure 5: velocity magnitude of particle traces at 722 (deg)

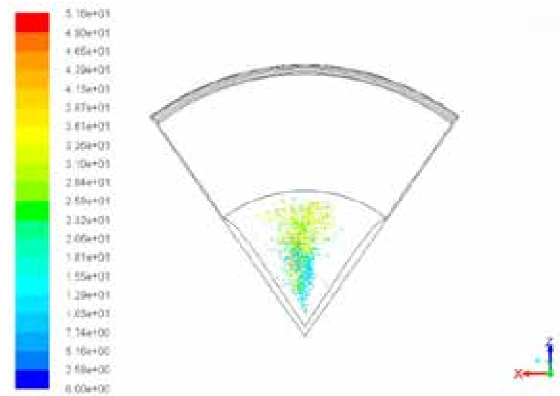

Figure 6: velocity magnitude of particle traces at 732 (deg)

It is obvious in figure 4 that the trajectory of fuel spray is subjected to angular deflection as a result of in-cylinder gases swirl turbulence. Figure 6 which shows the final phase of injection it can be seen that no additional diesel is injected into the combustion chamber, the velocity magnitude of last diesel particles is lower than the velocity of diesel particles that injected previously because of the decrement of injected diesel pressure at the end of injection, which is correspondent with real injection process that happens in marine diesel engines.

The forward front of diesel spray hits the combustion chamber wall at angle 722 deg which can be seen clearly at figure 7. This phenomenon called "spray impingement" which is unavoidable phenomenon in marine diesel engines that affects the emission formation especially hydrocarbons and soot emissions. Also it can be seen the effect of combustion chamber shape on the trajectory of fuel spray after it impinged with piston surface, whereas the spray hits the piston bowl wall then crawls up along the wall of piston bowl taking its shape as figure 8 shows. The spray impingement phenomenon is so related to spray velocity; the higher velocity magnitude means larger amount of diesel will hit the piston walls. It should be mentioned that the temperature of particles are high because the simulation of fuel injection carried out during the combustion process.

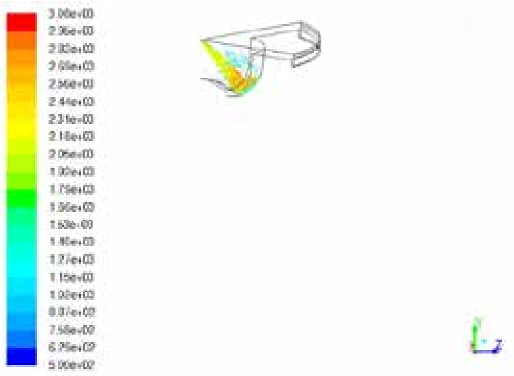

Figure 7: temperature of particle traces at 722 (deg)

Figure 8: temperature of particle traces at 724 (deg)

CONCLUSIONS

Simulating the injection process in marine diesel engines is considered a corner stone for simulating the combustion and emission formation processes in marine diesel engines. In this paper, the injection process in diesel engines have been simulated by the use KH-RT model which was explained, the results of simulation were illustrated and explained from the start of injection till the end of injection at different crank angles. The physical processes such as primary breakup, secondary breakup, and fuel evaporation that take place during diesel injection were explained and demonstrated. The diesel spray impingement phenomenon and the effect of combustion chamber geometry on the spray trajectory after the spray hitting the piston bowl were illustrated as well.

REFERENCE

[1]Ansys company publications and Manuals.(2012). | [2]Chin Heung Bong. (2010), “Diesel Spray Dynamic including the Effects of Fuel Viscosity”. | [3]Gunter P. Merker, Christian Schwarz, RudigerTeichmann. (2012), “Combustion Engines Development Mixture Formation, Combustion, Emissions and Simulation”. | [4]Gunter P. Merker, Christian Schwarz, Gunnar Stiesch, Frank Otto. (2006), “Simulating Combustion Simulation of Combustion and Pollutant Formation for Engine-Development”. | [5]Gunnar Stiesch. (2003), “Modeling Engine Spray and Combustion Processes”. | [6]Hua Zhao. (2010), “Advanced direct injection combustion engine technologies and development Volume 2: Diesel engines”. | [7]Klaus Mollenhauer, Helmut Tschoeke. (2010), “Handbook of Diesel Engines”. | [8]Suresh Kumar J, Ganesan V, MallikarjunaJ. M., Govindarajan S. (2012), “Spray Characteristics of a Fuel Injector: A CFD Study, Proceedings of the FISITA World Automotive Congress”.