Three-dimensional two-phase flow full simulation of injector flow and Diesel jet injection into a combustion chamber

C. A. Chasos

Frederick University, Nicosia, Cyprus

Abstract

In direct-injection Diesel internal combustion engines the quality of the air-fuel mixture is highly affected by the characteristics of the injector and the spray jet quality emerging from the injector nozzle, as well as the ambient conditions of the combustion chamber. All these parameters dictate the preparation of the air/fuel mixture to be combusted. The main objective of the present research was to model the full three-dimensional Diesel injector geometry along with the cavitation phenomena occurring in the injector nozzle for predicting the emerging liquid jet structure and the air-fuel mixture pattern in the combustion chamber. From the simulations, the equivalence ratio and the turbulence level fields can be determined in order to assist coupling of the injection phenomena with the modelling of non-premixed and partially-premixed combustion. In the present study, the three-dimensional geometry of a valve-covered orifice sac-less six-hole Diesel injector was designed. The computational domain of the symmetrical one sixth of the injector geometry assembled with a constant volume chamber was generated, and the simulations therein were carried out with the computational fluid dynamics (CFD) code STAR-CD. The Eulerian modelling methodology along with the volume of fluid (VOF) method were utilized. In the VOF method the surface tension of fuel liquid was modelled and the Rayleigh cavitation model accounting for liquid phase change into fuel vapor was used. The conservation equations for mass, momentum and energy were solved for the three phases and the modified high Reynolds number k-ε model for multiphase flow was used for modelling turbulence. Transient-state CFD simulations were performed during the phase of fully open needle of the injector for atmospheric and high pressure and temperature ambient chamber conditions. The simulation in atmospheric chamber was compared against published photographs of the cavitation pattern in the nozzle and there is moderately good qualitative agreement. From the results it was observed that there is asymmetry in the velocity profiles at the nozzle exit and the peak jet velocity is located below the symmetry axis of the nozzle. The air-fuel mixing is more intense in the upper area of the jet, while at the bottom area of the liquid jet there is a more distinct separation of the liquid and air regions. The pattern of the air fuel mixture at the upper region of the jet promotes partially-premixed conditions, while the liquid and air zones at the bottom region of the jet rather sustain non-premixed conditions. From the present research, discussion is carried out on the connection of the spray injection characteristics from a cavitating Diesel injector with the primary jet atomization and the resulting air-fuel mixture and subsequent combustion. Based on the findings, recommendations are provided regarding spray modelling practices and coupling with combustion modelling.

corresponding author: Dr. Charalambos Chasos, eng.cca@frederick.ac.cy

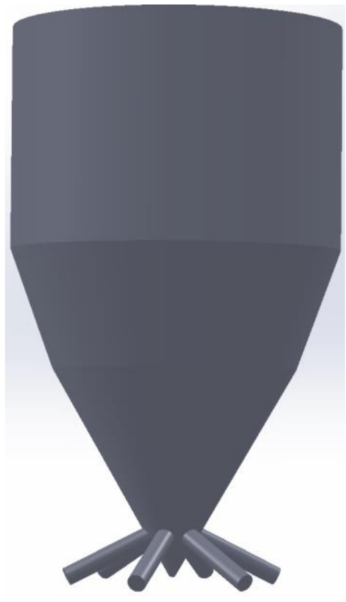

Figure 1 Three-dimensional valve-covered orifice sac-less six-hole Diesel injector design.

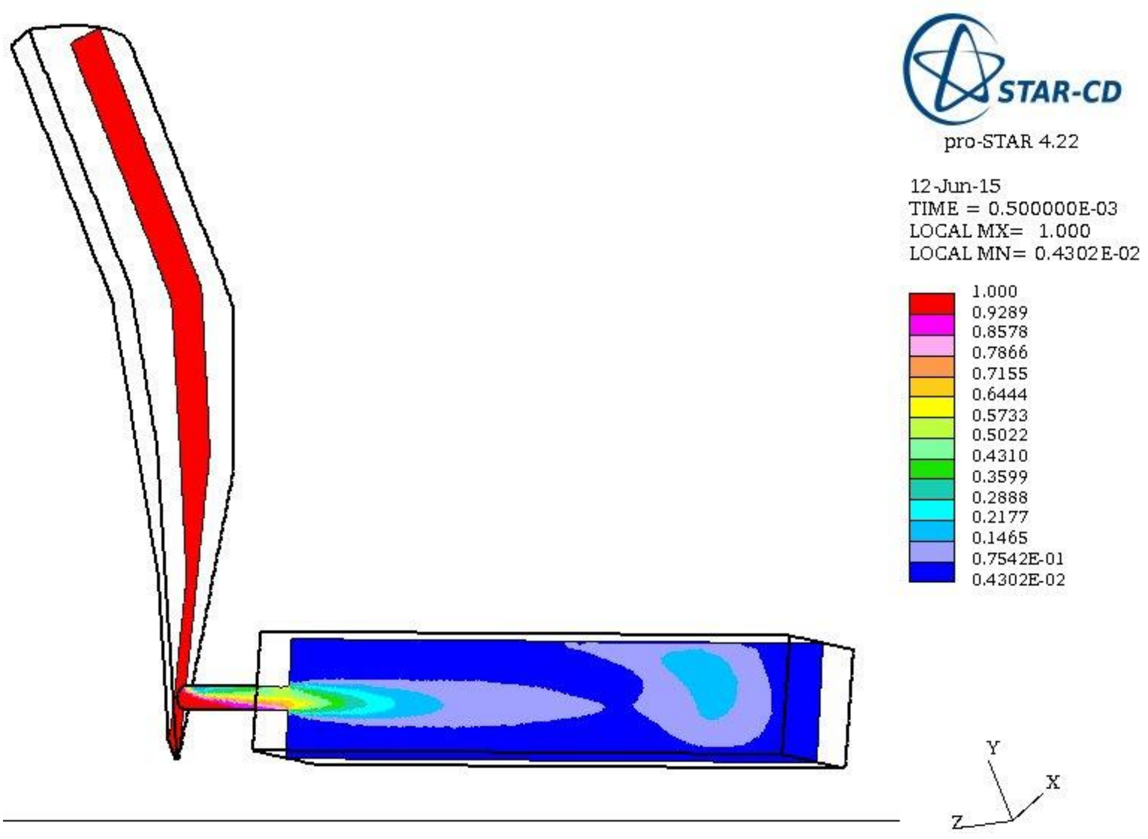

Figure 2 Prediction of the volume of fluid field of the liquid fuel (in red color) and air (in blue color) in the injector and the combustion chamber with atmospheric conditions at 0.5 ms after the start of injection.