Abstract

In this dissertation, the processes of autoignition and forced ignition occurring in turbulent two-phase flows (gaseous oxidizer/liquid fuel spray) were analysed. The presented results were obtained by numerical simulations using the Large Eddy Simulation (LES) method performed in an in-house LES solver SAILOR. The processes were analysed from the point of view of an influence of modelling methods and the effects of interactions of a turbulent flow field, chemical reactions and the fuel, existing initially in the form of a spray. The analyses were carried out for flow configurations constituting idealised models of phenomena important from the practical point of view, i.e., turbulent temporary mixing layer and turbulent jet.

The first part of the paper contains a comprehensive review of the literature related to the analysed phenomena. Next, the flow model used in the study is presented together with the necessary simplifications and sub-grid models used in the LES method. The following chapters, among others, characterise laminar and turbulent combustion processes and the most important methods for modelling turbulent combustion. This is followed by a description of issues related to modelling the effect of dilute spray on the flow field. The last chapter of the first part presents the numerical methods used to discretise the mathematical model of the analysed phenomena, implemented within the computational code.

In the second part, the obtained results are analysed, starting with those concerning the autoignition in the turbulent temporal mixing layer. The parameter on which attention was focused was the autoignition time. It was characterised by a high sensitivity to the evaporation model used as well as to the discretisation scheme and chemical kinetics mechanism. In the next chapter, forced ignition (spark ignition) was analysed in the same flow configuration but under different conditions. Emphasis was put on investigating the effects of turbulence intensity and spark parameters on ignition probability. The basic conclusion of this part of the study is that in turbulent two-phase flows it is a random process, and its success is largely influenced by the values of local velocity gradients. The last of the analysed cases was autoignition occurring in a two-phase jet. In this configuration the influence of many aspects such as the combustion model, discretisation methods, evaporation models or sub-grid models was analysed. The obtained results were compared with the experimental ones. The changes in the above-mentioned aspects mainly influenced the spatial fields of chemical species, which affected, e.g., the values of maximum temperatures or flame lift-off height. Their influence on the averaged velocity fields was indirect and turned out to be small.

The obtained results clearly prove that the modelling of combustion in two-phase turbulent flows is almost equally sensitive to changes of numerical aspects such as discretisation scheme or numerical implementation as to changes of physical models of phenomena and processes occurring in the flow. This conclusion is particularly emphasised in the present study on two-phase flows, since differences in mixture inhomogeneity due to the presence of fuel droplets, which are significantly influenced by the numerical approach, affect the combustion process to a greater extent than in gaseous mixtures. This fact should be taken into account especially when using computational fluid mechanics methods in the design of practical devices in order to ensure their safe and failure-free operation.