Abstract: Computational fluid dynamics supporting the optimization of combustion systems of large Marine Diesel engines require sub-models for spray, evaporation and combustion modelling. In this paper, the further development of these sub-models and their application to engine simulations are described. The sub-models are validated against reference data from an experimental setup consisting of an optically accessible, disk-shaped constant volume chamber of 500 mm diameter with peripheral injection into a swirling flow - the Spray Combustion Chamber (SCC) developed in the context of the HERCULES (High Efficiency R&D on Combustion with Ultra Low Emissions for Ships) research program and presented for the first time in CIMAC 2007, Paper No.98. This unique experimental test facility for investigating the spray and combustion behaviour under conditions relevant for large marine diesel engines not only resembles the physical dimensions of large two-stroke engines but also the operational characteristics with regard to thermo- and fluid dynamic conditions at start of injection are reproduced with excellent repeatability. In particular, pressure, temperature and swirl are close to the levels experienced in those engines. Additionally, a wide range of fuel qualities can be used. The highly flexible optical accessibility enables the generation of appropriate reference data and the application of non-intrusive spray and combustion diagnostics contribute to an in-depth understanding of the involved in-cylinder processes. Combustion is particularly sensitive to the fuel vapour distribution, therefore the accurate simulation of spray and evaporation processes is seen as a prerequisite for reliable combustion and emissions formation results. The various experimental investigations that have been performed to gain knowledge about the fuel spray characteristics during injection, the evaporation behaviour and the subsequent ignition and combustion resulted in a reference data set that has been applied to the further development and validation of improved CFD sub-models. The in-nozzle flow and primary breakup of fuel nozzles was analyzed in detail and intensive investigations have been performed to describe the influence of geometrical design features and flow conditions inside the nozzle. As spray and droplet formation in large two-stroke engines is highly unsymmetrical a primary breakup model was developed taking into account asymmetric boundary conditions at the orifice outlet. Finally the improved methodology developed is applied to investigate current issues of engine performance optimization like combustion with high EGR content. The actual CFD supported combustion system product development process is described and exemplified on the basis of the development of the WX35 engine.