A new method for the simulation of non-linear parabolic equations in cylindrical coordinates

A. Gil and J. P. G. Galache

CMT Motores Térmicos, Universitat Politècnica de València, e-mail: angime@mot.upv.es

Turbulence is still nowadays one of the most important open problems in physics. Almost any flow relevant for industry is turbulent. In the case of long pipes, turbulence is the main responsible of the energy losses, including some small or narrow pipes, where the flow is also fully turbulent. Fuel in IC engines is usually introduced to the combustion chamber as spray through narrow orifices forming the injector. A complete spray characterization is therefore essential in order to improve the combustion process and thus reduce contaminants and consumption. Several important features of sprays and round jets are known to be consequences of the characteristics of the nozzle flow. Probably one of the most important is the velocity profile at the nozzle exit, but the turbulent spatial structures of the flow at the nozzle are also significant. The link between these structures and those of the spray (or the spray intermittence) is not currently known.

In order to study the physics of the nozzle-space system, the only feasible technique is the Direct Numerical Simulation (DNS) of Navier-Stokes equation. By means of DNS all the relevant scales of the flow are solved. This includes the energy containing scales, the dissipative ones and the inertial range. Of course this leads to highly refined meshes. In the case of pipes, there are two additional problems: firstly the pole at the origin and secondly the big density of mesh points at the center of the nozzle.

In this communication we present a technique to overcome these two difficulties. The code uses a mixed Compact Finite Difference - Fourier method in the spatial variables and a Runge-Kutta method for the temporal discretization. At this stage of the code, 3D convection-diffusion equation has been solved in order to check its validity.