Turbulent Flows
Simulation and Modelling

Daniel Ahlman — *ahlman@mech.kth.se*

Olof Grundestam — *olof@mech.kth.se*

Linus Marstorp — *linus@mech.kth.se*

Gustaf Mårtensson — *gustaf@mech.kth.se*

KTH Mekanik

Simulation of fluid flow

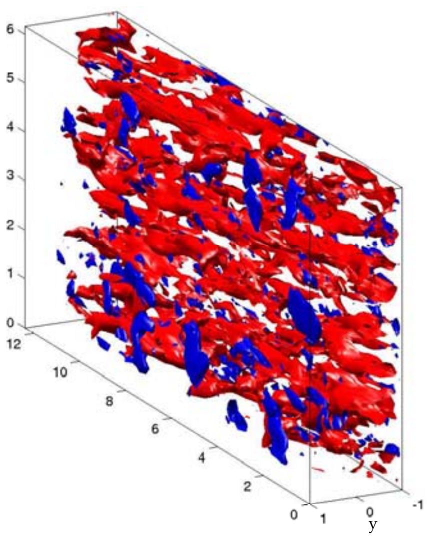
For Newtonian fluids the flow motion is described by the Navier-Stokes equations. In a Direct Numerical Simulation, DNS, the Navier-Stokes equations are solved numerically by discretization in both time and space. DNS of turbulent flows requires very high resolution.

LES of homogeneous shear flow

A Large Eddy Simulation, LES, is an under-resolved numeric simulation of the Navier Stokes equation in which the influence of the non resolved, or subgrid, scales has to be modelled. Homogeneous shear flow is an excellent test case for developing and testing subgrid scale models. The simple geometry of the flow enables the use of the accurate spectral methods which makes it easy to separate model features from the numerical errors.

Rotating channel flow

Rotating channel flow at different rotation numbers are studied through direct numerical simulations. Although the geometry of this case is very simple with the spanwise and streamwise directions that are thought of as infinitely long and wide and with a rotation that is applied in the spanwise direction, the dynamics of the flow are intricate. Even for small rotation rates the response of the flow is strong. The effect on the turbulence stresses is drastic and long structures are formed on especially the suction side of the channel. As the rotation rate is increased the structures get smaller and the damping of the turbulence on the suction side makes the flow laminar-like in this region. This is depicted in the figure below in which isosurfaces of negative wall-normal fluctuating velocity (red) and postive streamwise fluctuating velocity (blue) for rotation number $Ro = 2\Omega/U_b = 1.16$ (based on the mean bulk velocity, U_b) are visualized. Note the laminar-like suction side (positive y).

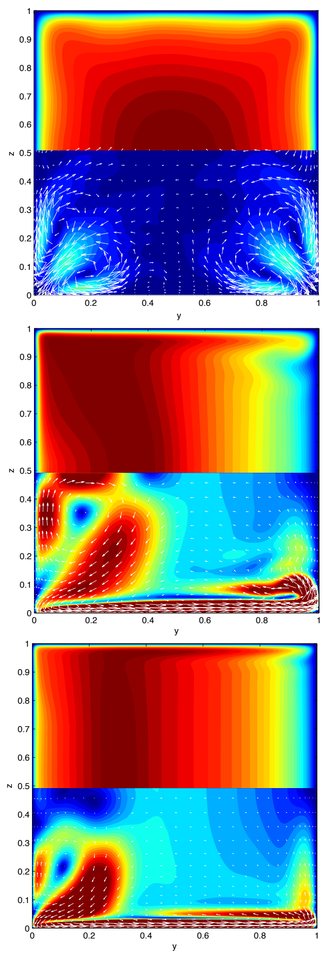


Rotating duct flow

The study of the fluid flow in rotating systems is of considerable academic and industrial interest. One finds rotating flows in industrial applications ranging from turbines to separators to dvd-manufacturing to DNA-replication. A canonical flow situation that is of great interest may be obtained by taking a channel flow and constricting it by introducing two horizontal surfaces. A direct numerical simulation of this flow has been performed to better understand the effect of rotation on turbulent flow, as well as to provide a valuable resource for comparisons with modelling efforts.

In each of the three plots below, which represent flows for rotation rates equivalent to $Ro_w = 0, 0.22$ and 0.77 respectively, three sets of information are illustrated. The box represents a cross-section of the duct normal to the main flow therein. The upper half of the plot contains contours of the mean flow in the direction of the duct. In contrast to the symmetric velocity profile of the non-rotational case, a dramatic skewing of the velocity profile is observed for increasing rotation rate. A tilted velocity profile is also observed in a rotating channel, but it is tilted in the opposite direction. Low speeds are represented by the color blue, while red represents higher speeds.

The lower half of the plot contains a contour plot of the flow in the plane normal to the main flow, as well as a quiver plot that shows the actual direction of the flow in that plane. A powerful secondary flow has developed in response to the rotation. Note that there is a very weak secondary flow consisting of corner vortices even for the stationary case,

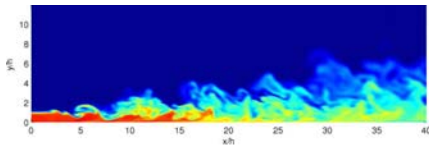


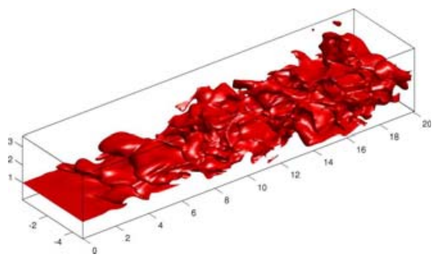
Near-wall mixing and combustion

In many combustion applications, a part of the combustion and species mixing takes place near relatively cold solid walls. The presence of the wall can be argued to have a profound influence on the combustion processes through e.g. the heat flux towards the colder wall. Also the altering of the mixing situation is of relevance. In the vicinity of the wall, turbulent fluctuations will be inhibited, which in turn provides a distinctively different mixing situation. Many of the present combustion models employ scalar mixing in their formulation. However the accuracies of these models are presently not well known in the near-wall region.

The objective of the project is to study near wall mixing and combustion by means of direct numerical simulation. The simulations will provide high order statistics for evaluation and development of mixing and combustion models.

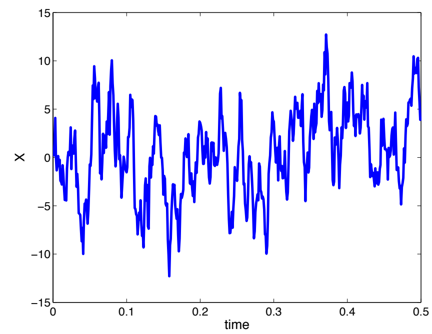
Below are two figures depicting passive scalar mixing in a turbulent plane wall jet. The jet enters in the lower left corner and propagates along a solid wall. The upper figure shows the scalar concentration in the center of the domain, and the lower figure shows the isosurface where the concentration is half the center jet inlet concentration.





Stochastic modelling in LES

A Large Eddy Simulation, LES, is an under-resolved numeric simulation of the Navier Stokes equation in which the influence of the non resolved, or subgrid, scales has to be modelled. Formally one solves the filtered Navier Stokes where the subgrid scale stress tensor has to be modelled in order to close the system of equations.



The nature of the smallest turbulent scales appears as partly random to the resolved scales and the use of stochastic processes can improve subgrid stress models. The stochastic process in the figure has zero mean, constant variance, and constant timescale. It is called "coloured noise" and has recently been used for subgrid modelling.