



Computational Fluid Dynamic Simulations on NW-GRID using *Code Saturne*

The *Code Saturne* CFD software

Code Saturne is general purpose CFD software developed at EDF since 1997. It is based on a co-located finite volume approach and can handle any type of cells (tetrahedral, hexahedra, polyhedra) and any type of mesh (structured, unstructured, hybrid, non-conforming). Its basic capabilities enable the handling of either incompressible or expandable flows with or without heat transfer and turbulence (mixing length, 2-equation models, $v2f$, Reynolds stress models, Large Eddy Simulations).

Dedicated modules are available for specific physics such as radiative heat transfer, combustion (gas, coal), magneto-hydro dynamics, compressible flows, two-phase flows (Euler-Lagrange approach with two-way coupling), extensions to specific applications (e.g. for atmospheric environment: code *Mercure Saturne*).

Parallel code coupling capabilities are provided by the FVM library (EDFs FiniteVolume Mesh library, under LGPL licence).

The University of Manchester has been co-developing the code since 2000 focusing on turbulence modelling, and now is the main UK hub for its dissemination. See:

<http://cfd.mace.manchester.ac.uk/Saturne/WebHome> .

The code is now open source under the GPL license and has been installed on the North West Grid.

Flow parallel to tube bundles.

The flow parallel to the fuel rods inside a nuclear reactor has been computed on

the NW-GRID using Large Eddy Simulation. The instantaneous values of cross velocity show the pulsating flow along the tubes length which induces vibrations on the rods.

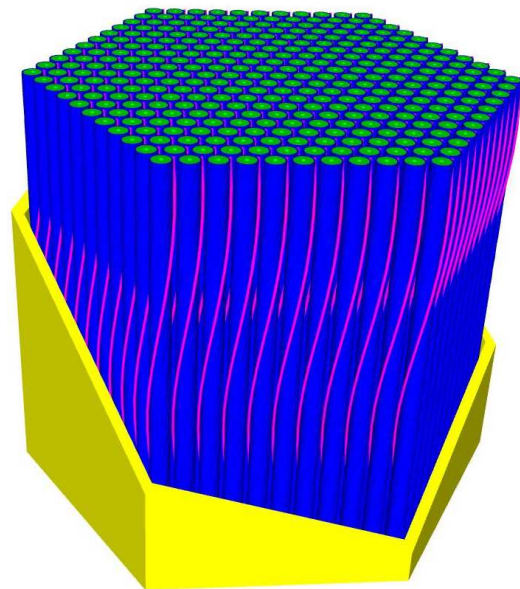


Figure 1: 3D Array of Tubes

Flow over a Trailing Edge

A new hybrid RANS-LES approach for complex turbulent flows has been assessed by considering turbulent boundary-layer flows past an asymmetric trailing-edge. The use of the hybrid method allows for the use of a mesh of about ten times less number of cells when compared to the reference LES. In addition to the hybrid method, a Synthetic Eddy Method is used to recreate the instantaneous turbulent field at the inlet. The SEM proved to be a low-resource alternative for the inlet conditions with only mean average quantities as input. The use of the SEM

saves about 30% of the total computational time when compared to the precursor LES computations.

Flow inside a 3D Diffuser

The flow on a three-dimensional diffuser has been computed. The diffuser has two expanding walls at different angles which creates a three dimensional separation bubble. This is a challenging case for RANS models and the suitability of hybrid approaches is studied.

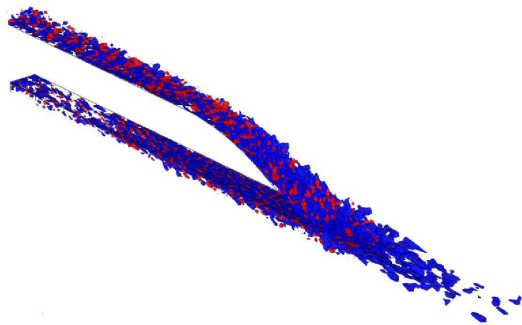


Figure 2: iso-Q surfaces over trailing edge

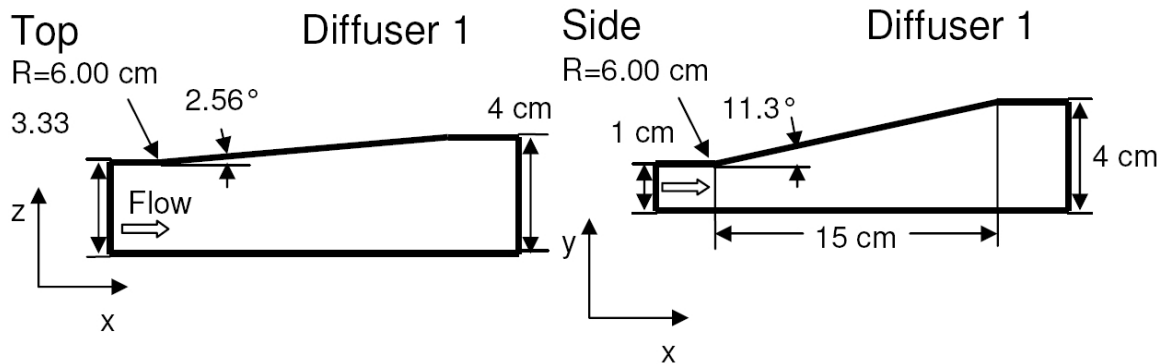


Figure 3: Diffuser Geometry

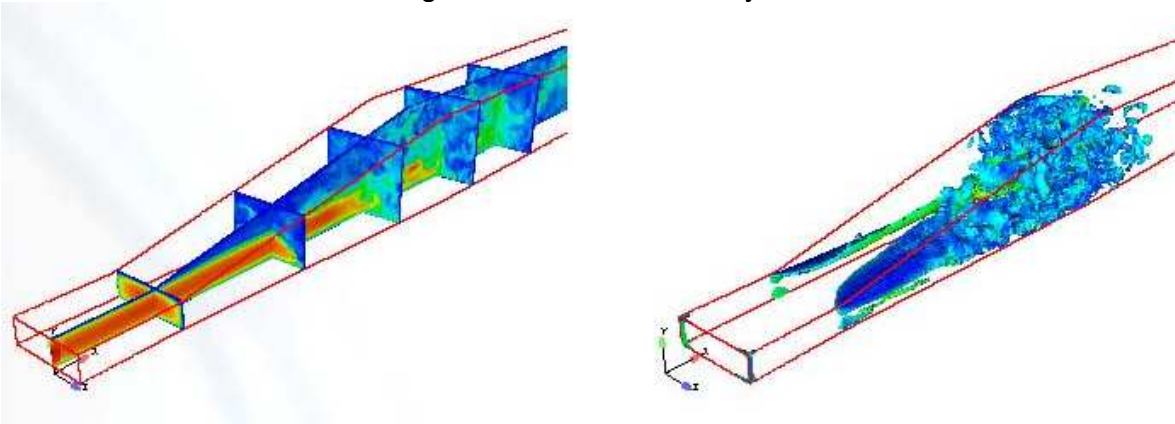


Figure 4: Instantaneous velocity, and iso-surfaces of vorticity in a 3D diffuser.