Numerical simulations of massive separated flows: flow over a stalled NACA0012 airfoil

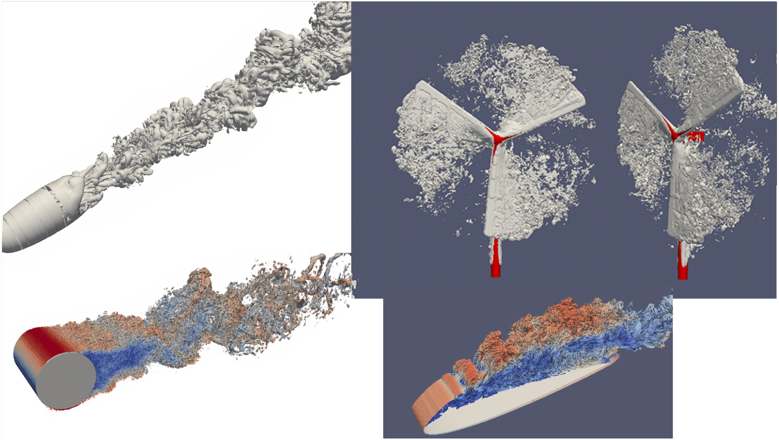
A. Oliva I. Rodríguez O. Lehmkuhl R. Borrell A. Baez
C.D. Pérez-Segarra

Centre Tecnològic de Transferència de Calor
ETSEIAT – UPC

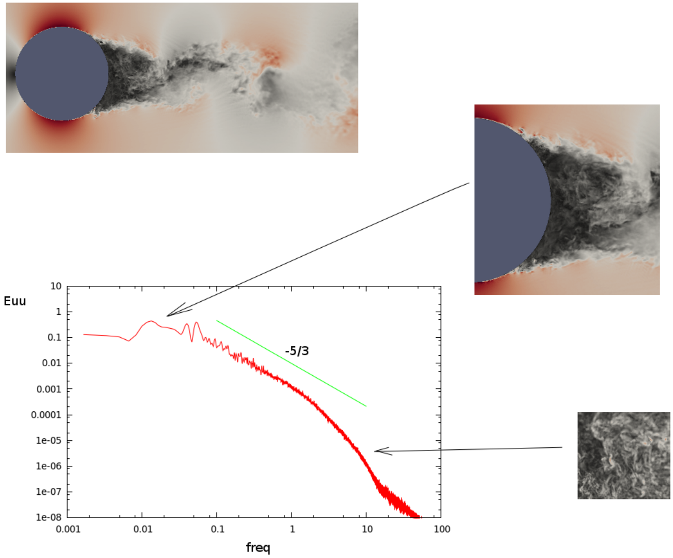
7th RES Users'Conference
Barcelona, September 13th, 2013

Contents

Turbulent flow past bluff bodies



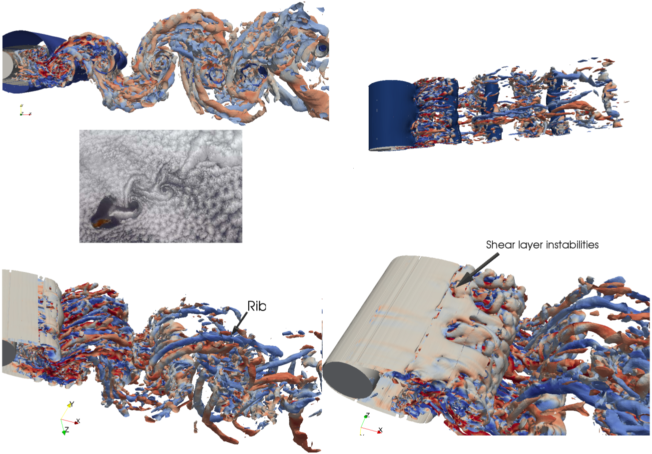
The turbulence problem



Motivations & objectives

- Advance in the understanding of the physics of turbulent flows
- To gain insight in the mechanism of the shear-layer transition and its influence in the wake characteristics and in the unsteady forces on the bluff body surface
- Contribute to the improvement of SGS modelling of complex flows, by the assessment of the performance of models suitable for unstructured grids and complex geometries at high Re, typical encountered in industrial applications.

Previous results (1/2)

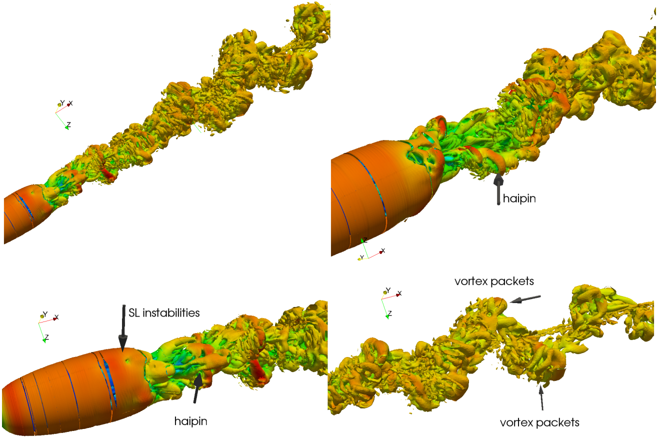


Project: Direct Numerical Simulation of turbulent flows in complex geometries using unstructured meshes. Flow around a circular cylinder. Refs: FI-2008-2-0037 and FI-2008-3-0021.

O. Lehmkuhl, I. Rodríguez, R. Borrell, and A. Oliva. Low-frequency unsteadiness in the vortex formation region of a circular cylinder. Phys. Fluids 25, 085109 (2013)

Image credit: NASA/GSFC/LaRC/JPL, MISR Team

Previous results (2/2)



Project: Direct Numerical Simulation of turbulent flows in bodies of revolution using unstructured meshes. Flow past a sphere at subcritical Reynolds numbers. Refs:FI-2009-3-0011 and FI-2010-2-0018
I. Rodríguez, R.Borrell, O. Lehmkuhl, C.D. Pérez-Segarra, A. Oliva. (2011) Direct numerical simulation of the flow over a sphere at Re=3700. J. Fluid Mech. 679, 263-287

HPC facilities

Curie - TCGG

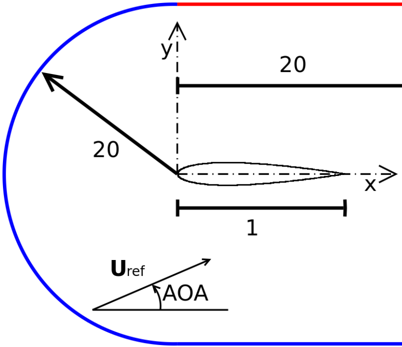
MareNostrum - BSC

Magerit - CeSViMa

Lomonosov - RCC MSU

JFF II - CTTC

Definition of the case



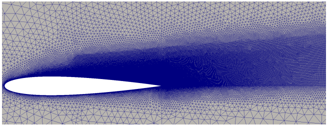
Computational domain

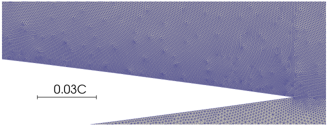
$(x, y, z) \in [-20C, 20C] \times [-20C, 20C] \times [0, 0.2C]$

- $Re = \frac{U_{ref}\ C}{\nu} = 5 \times 10^4$
- $AoA = 5,\ 8,\ 9.25,\ 12^\circ$
- Boundary conditions:
  - Inflow: $(u, v, w) = (U_{ref} \sin(AoA), U_{ref} \cos(AoA), 0)$
  - Outflow: Pressure based
  - Airfoil surface: No-slip conditions
- The flow is fully unsteady and turbulent
- Separations/reattachments are expected depending on AoA

Mesh design

- Meshes are adapted to follow the turbulent zones in the suction side and in the near wake
- The unstructured grid has allowed to cluster more CVs in the suction side and near wake.
- Mesh generation is done by means of a constant-step extrusion of a 2D unstructured grid
- The span-wise direction is divided into N_{planes} identical planes





Meshes solved

- Meshes adapted to follow the turbulent zones in the suction side and the near wake
- Meshes build to resolve well all the relevant scales of the flow, thus have been verified with *a-posteriori* methodology

AoA	$N_t \times 10^{-6}$ CVs	$N_{CV\ plane}$	N_{planes}	**NCPU**
5°	25.3	263,522	96	240
8°	27	280,876	96	240
9.25°	43.6	340,526	128	320
12°	48.9	381,762	128	320

Numerical method

- Discretisation of the GE by means of a second-order conservative schemes on a collocated unstructured grid arrangement.
- Temporal discretisation based on a second-order explicit scheme on a fractional-step method
- Poisson equation solved by means of a FFT method with an explicit calculation and direct solution of a Schur Complement system for the independent 2D systems
- This methodology has been previously used successfully in other similar flows [1]

[1] I. Rodríguez et al. JFM 679 (2011)
I. Rodríguez et al. Comput Fluids. 80 (2013)
O. Lehmkuhl et al. Phys. Fluids. 25, 085109 (2013)

All computations were carried out on different clusters

- **JFF cluster** at CTTC. 76 nodes in-house cluster, each node has 2 AMD Opteron 2350 Quad Core processors linked with an infiniband DDR4 network
- **MareNostrum supercomputer II** at the Barcelona Supercomputing Center (BSC). IBM BladeCenter JS21 Cluster with 10 240 PowerPC 970MP processors at 2.3 GHz with 1 MB cache per processor. Quad-core nodes with 8 GB RAM were coupled by means of a high-performance Myrinet network.
- **Magerit**, CeSViMa Supercomputer at UPM. 260 computer nodes, of which 245 nodes are eServer BladeCenter PS702 with 16 Power7 processors 3'3 GHz (26.4 GFlops) and 32 GB de RAM, and the rest are 15 nodes eServer BladeCenter HS22 with eight Intel Xeon 2'5GHz (10.2 GFlops) processors with 96 GB RAM, implying 4,160 CPUs and 9.2TB RAM

Pre-processing tasks

- Mesh extrusion and partitioning
- Partitioner optimisation
- Building mesh topology (50M CVs and 512 CPUs more than 48h $\rightarrow > 1h$!
- Solver pre-processing (50M CVs and 512 CPUs more than 17h $\rightarrow > 10min!$)

Partitioner optimisation

- Use of Metis for mesh partition
- Save/load data is more efficient when using HDF5 (vs. ASCII)
- Mesh information during partition is now saved in RAM
  1. All the process is done by 1 CPU
  2. (if not enough RAM) All the process is done by 1 CPU, but 1 by 1
  3. The process is parallelised but the number of partitions and CPU must be the same

Structured binaries

Use of HDF5 for saving data.

- Data is organized as in a file system, i.e. directories(=groups); files(=datasets)
- Data from all processors are saved in 1 file (lustre as distributed file system)
- All the information of the case is saved/loaded from the file (HDF5)
- Endianness: Now data is accessible with independence of the architecture (x86-64, PowerPC)
- Binary of 50M CV $\Longrightarrow$ 18G; mesh $\Longrightarrow$ 16G

Optimisation of the code

At each iteration (time-step):

- Evaluation of gradients, coefficients, etc.
- Solving the Poisson equation
- Other tasks i.e. saving instantaneous data, eval. averaged quantities, saving numerical probes, etc.

Coef, grads, ...	24.4%
Solver	51%
After Δt	24.6 %

38 M CVs $\rightarrow$ 384CPUs$\rightarrow$ 0.52s/ite

Coef, grads, ...	61%
Solver	35.4%
After Δt	3.6 %

38 M CVs $\rightarrow$ 384CPUs$\rightarrow$ 0.38s/ite (about 1.4 times faster!)

Optimisation of the code

At each iteration (time-step):

- Evaluation of gradients, coefficients, etc.
- Solving the Poisson equation
- Other tasks i.e. saving instantaneous data, eval. averaged quantities, saving numerical probes, etc.

Coef, grads, ...	24.4%
Solver	51%
After Δt	24.6 %

38 M CVs → 384CPUs→ 0.52s/ite

Coef, grads, ...	61%
Solver	35.4%
After Δt	3.6 %

38 M CVs → 384CPUs→ 0.38s/ite (about 1.4 times faster!)

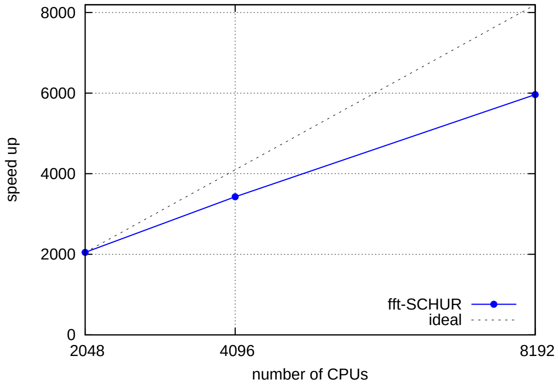
Poisson Solver (1/2)

The Poisson system can be written as:

$$\mathsf{L}_c = \underbrace{(\Omega_{2d} \otimes \mathsf{L}_{c_{per}})}_{\text{periodic couplings}} + \underbrace{\Delta_{per}(\mathsf{L}_{c_{2d}} \otimes I_{N_{per}})}_{\text{2D couplings}}$$

- $\Omega_{2d} = diag(\mathcal{A}_1, ..., \mathcal{A}_{2d})$ areas of 2D mesh.
- $\mathsf{L}_{c_{per}}$ Poisson system discretized in $\mathcal{M}_{per}$.
- $\mathsf{L}_{c_{2d}}$ Poisson system discretized in $\mathcal{M}_{2d}$.
- $$\Omega_{2d} \otimes \mathsf{L}_{c_{per}} = \begin{bmatrix} \mathcal{A}_1 \mathsf{L}_{c_{per}} & & \\ & \ddots & \\ & & \mathcal{A}_{N_{2d}} \mathsf{L}_{c_{per}} \end{bmatrix}$$
- $\mathsf{L}_{c_{per}} = circ(2, -1, 0, ..., 0, -1)$.
- $\mathsf{W}^* \mathsf{L}_{c_z} \mathsf{W} = diag(\lambda_1, ..., \lambda_{N_{per}})$ Fourier diagonalizable.

Poisson Solver (2/2)



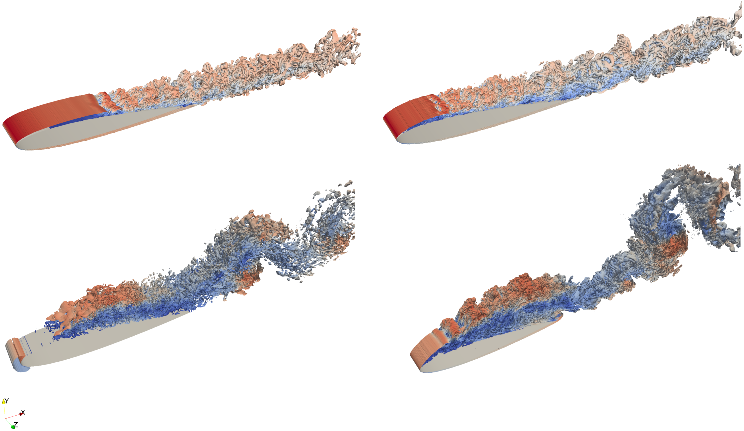
- $2 \cdot 10^6 x256$ planes→ $512 \cdot 10^6$ CV
- CPU is increased 4 times (2048 → 8192) Solve time decreases 2.9 times (0.69 s → 0.24 s)
- 73% parallel efficiency

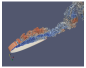
R. Borrell et al., Parallel direct Poisson solver for discretisations with one Fourier diagonalisable direction, *Computational Physics*, 230(12):4723–4741, 2011.

Data post-processing

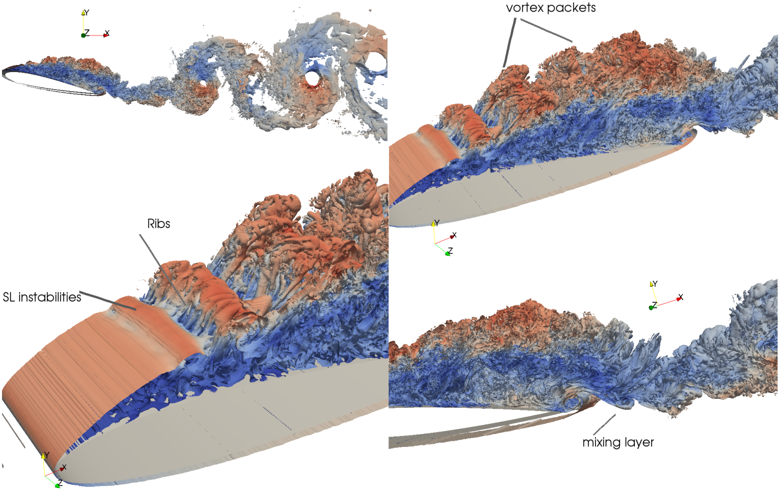
- Data and mesh information are on separated files but visualization through xdmf file
- Parallel visualization in paraview with HDF5 files
- Parallel visualization can be done without downloading data from cluster
- Parallel visualization + pvbatch scripts $\longrightarrow$ data visualization (and videos) is faster
- Processing a video of 75 frames from a 50M CV dataset ($\sim$ 3.75h) with 64 CPUs

Flow structures (1/2)



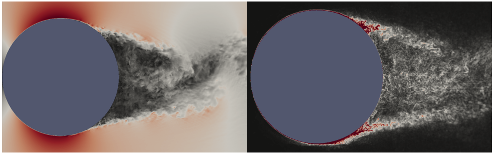


Flow structures (2/2)



What's next?

- High Performance Computing of the flow past a spinning cylinder. Application to flow control. Refs. FI-2013-2-0009
- Moving on from Tier-1 to Tier-0: DRAGON - Understanding the DRAG crisis: ON the flow past a circular cylinder from critical to trans-critical Reynolds numbers



Up-until-now: $Re = 2 \times 10^6 \sim$ 200M CVs $\rightarrow$ 2560 CPUs

step forward: $Re = 4 \times 10^6 \sim$ 400M CVs $\rightarrow$ 4098 CPUs

- I. Rodriguez, O. Lehmkuhl, R. Borrell and A. Oliva. (2013). Direct numerical simulation of a NACA0012 in full stall. International J. of Heat and Fluid Flow. 2013. http://dx.doi.org/10.1016/j.ijheatfluidflow.2013.05.002
- O. Lehmkuhl, I. Rodriguez, A. Baez, A. Oliva, C.D. Perez-Segarra. (2013). On the Large-Eddy Simulations for the flow around aerodynamic profiles using unstructured grids. (2013) Computers&Fluids. http://dx.doi.org/10.1016/j.compfluid.2013.06.002
- G. Colomer, R. Borrell, F.X. Trias, I. Rodriguez. (2013) Parallel algorithms for Sn transport sweeps on unstructured meshes. Journal of Computational Physics 232 118135. http://dx.doi.org/10.1016/j.jcp.2012.07.009
- R.Borrell et al., Parallel direct Poisson solver for discretisations with one Fourier diagonalisable direction, *Computational Physics*, 230(12):4723–4741, 2011.

Thank you for your attention!