

DATABASE MECA (in progress)

- You find here some information about databases related to several numerical studies in fluid mechanics performed at LIMSIS.
- The use of databases is currently reserved to the members of LIMSIS but conditional access rights can be considered for any collaborator.
- Some databases are in open access while other ones are in restricted access. A simple e-mail demand is enough to obtain access to a database (yann.fraigneau-toremove@limsi.fr).
- Most numerical simulations have been computed with the code Sunfluidh.



Information about the code Sunfluidh, files provided by the code and tools for reading these files can be found [in the wiki sunfluidh](#).

DATABASE ORGANIZATION

The full database is stored on the server "DATAMECA". Each dataset is generally organized as follows :

The path to access the root directory of the database (**DATABASE_"name"**) can be defined in two ways:

- **/vol/DATABASE_MECA/DATABASE_"name"** for databases in free access
- **/vol/DATABASE_MECA/RESTRICTED_ACCESS/DATABASE_"name"** for databases in restricted access

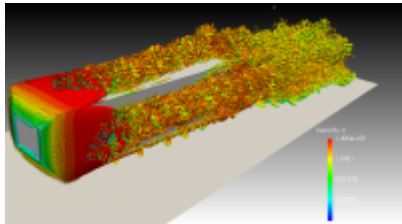
In the root directory are present folders in which are sorted the different kinds of data files :

- **/GRID** : grid files used by the CFD code to perform the simulation as well as data files used to create the grid (with 'meshgen') and report files containing some information about the grid features.
- **/DATASETUP** : input data files used by the CFD code (except the grid files) to compute the simulation.
- **/SNAPSHOTS** : instantaneous fields of physical quantities (also called snapshots)
- **/STATISTICS** : space and time averaged data
- **/TIMESERIES** : time series provided from probes placed in the flow. Sometimes are present time series about residual quantities or other data which are described according to the studied case.
- **/SLICES** : 2D instantaneous fields recorded over specific plans (slices in 3D computational domain).
- **/RESTART_AR** : any useful files in order to resume the simulation from a specific time.

- **/FROM_COMPUTATION** : Some potentially useful files resulting from the simulation used to check the computation behaviour during the run.

Some data are sometimes unavailable (this depends on the aim of the numerical study).

CONTENTS



1. 3D Ahmed body flow - $Re_H=10000$

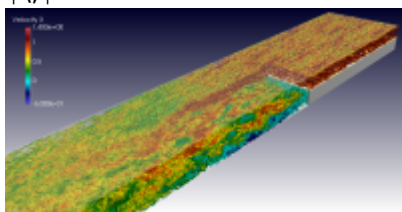
A direct numerical simulation using $512 \times 256 \times 256$ cells over a time range of 638 time units.

The data size is about 609 Gb.

\$\backslash\$, \$

\$\backslash\$, \$

\$\backslash\$, \$



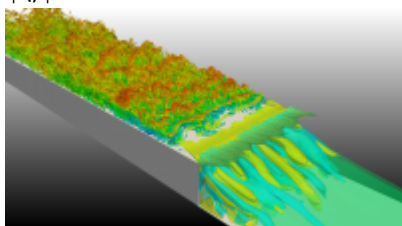
2. 3D backward-facing step flow - $Re_h = 6000$

A direct numerical simulation using 83,886,080 cells split into 160 subdomains (with $128 \times 64 \times 64$ cells per subdomain) over a time range of 510 time units.

The data size is about 4.4 Tb.

\$\backslash\$, \$

\$\backslash\$, \$



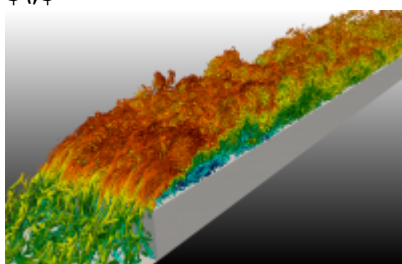
3. 3D forward-facing step flow - $Re_h = 8000$ (laminar inflow)

A direct numerical simulation using 471,859,200 cells split into 200 subdomains (with $128 \times 192 \times 96$ cells per subdomain) over a time range of 397 time units.

The data size is about 3.2 Tb (simulation in progress).

\$\backslash\$, \$

\$\backslash\$, \$



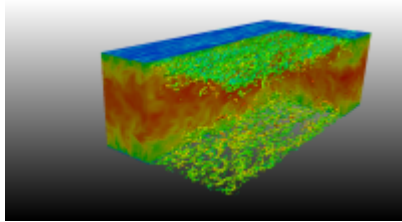
4. 3D forward-facing step flow - $Re_h = 8000$ (turbulent inflow)

A direct numerical simulation using $471,859,200$ cells split into 200 subdomains (with $128 \times 192 \times 96$ cells per subdomain) over a time range of 397 time units.

The data size is about 3.2 Tb (simulation in progress).

$\$, \$$

$\$, \$$



5. 3D turbulent channel flow - $Re_{\tau}=590$

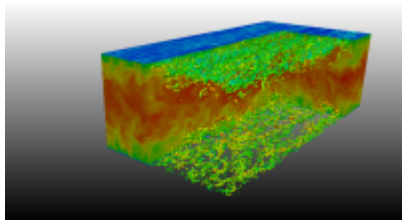
A large eddy simulation using 256^3 cells split into 128 subdomains over a time range of 200 time units.

The data size is about 5.1 Tb.

$\$, \$$

$\$, \$$

$\$, \$$



6. 3D turbulent channel flow - $Re_{\tau}=180$

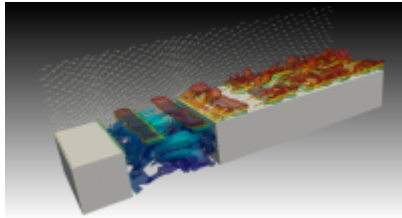
A direct numerical simulation using $96 \times 128 \times 96$ cells over a time range of 202 time units.

The data size is about 13 Gb.

$\$, \$$

$\$, \$$

$\$, \$$



7. 3D shear-layer driven cavity flow - $Re_{L_c}=7826$

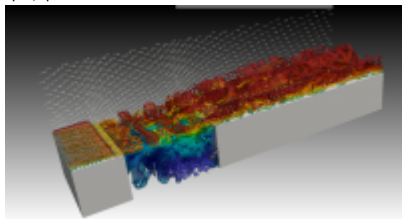
A direct numerical simulation using $192 \times 128 \times 64$ cells over a time range of 260 time units.

The data size is about 9.3 Gb.

$\$, \$$

$\$, \$$

$\$, \$$



8. 3D shear-layer driven cavity flow with a constant forcing source term - $Re_{L_c}=7826$

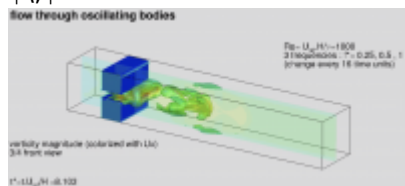
A direct numerical simulation using $192 \times 128 \times 64$ cells over a time range of 510 time units.

The data size is about 8.6 Gb.

$\$, \$$

\$\\\$,

\$\\\$,

9. 3D planar jet through oscillating bodies - $Re_{L_y}=1000$

A direct numerical simulation using $180 \times 80 \times 40$ cells over a time range of 100 time units on which three oscillation frequencies are considered.

The data size is about 2.3 Gb.

\$\\\$,

10. 2D flow around an oscillating cylinder - $Re_D=185$

Four direct numerical simulation using 480×480 cells over a time range of 600 time units. Four different configurations have been considered (2 oscillation magnitudes and 2 frequencies).

The data size is about 7 Gb.

\$\\\$,

\$\\\$,

\$\\\$,



Videos about some simulations can be watched [here](#)

From:

<https://datasetmeca.lisn.upsaclay.fr/> - **CFD DATABASE**

Permanent link:

<https://datasetmeca.lisn.upsaclay.fr/doku.php?id=start&rev=1625662367>

Last update: **2021/07/07 14:52**

