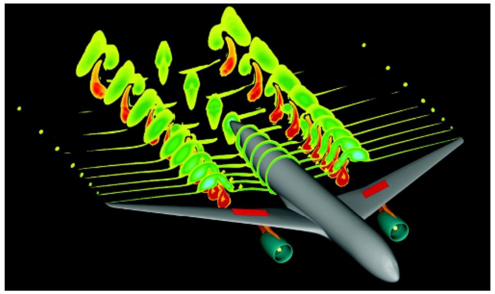
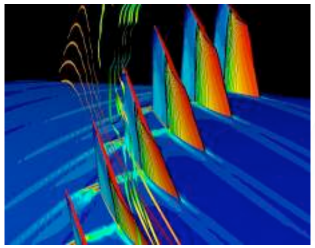
Complex Fluid Dynamics, KAUST, March 22-25, 2010

High Performance Computing of Industrial Flows: Application to Aeronautic and Propulsion Challenges





L.Y.M. Gicquel[1]

B. Cuenot[1], G. Staffelbach[1], O. Vermorel[1], E. Riber[1], A. Dauptain[1]

N. Gourdain[1], M. Montagnac[1], J-F. Boussuge[1], M. Gazaix[2]

T. Poinsot[3]

[1] CERFACS - CFD Team, Toulouse

[2] ONERA - DSNA Dpt., Chatillon

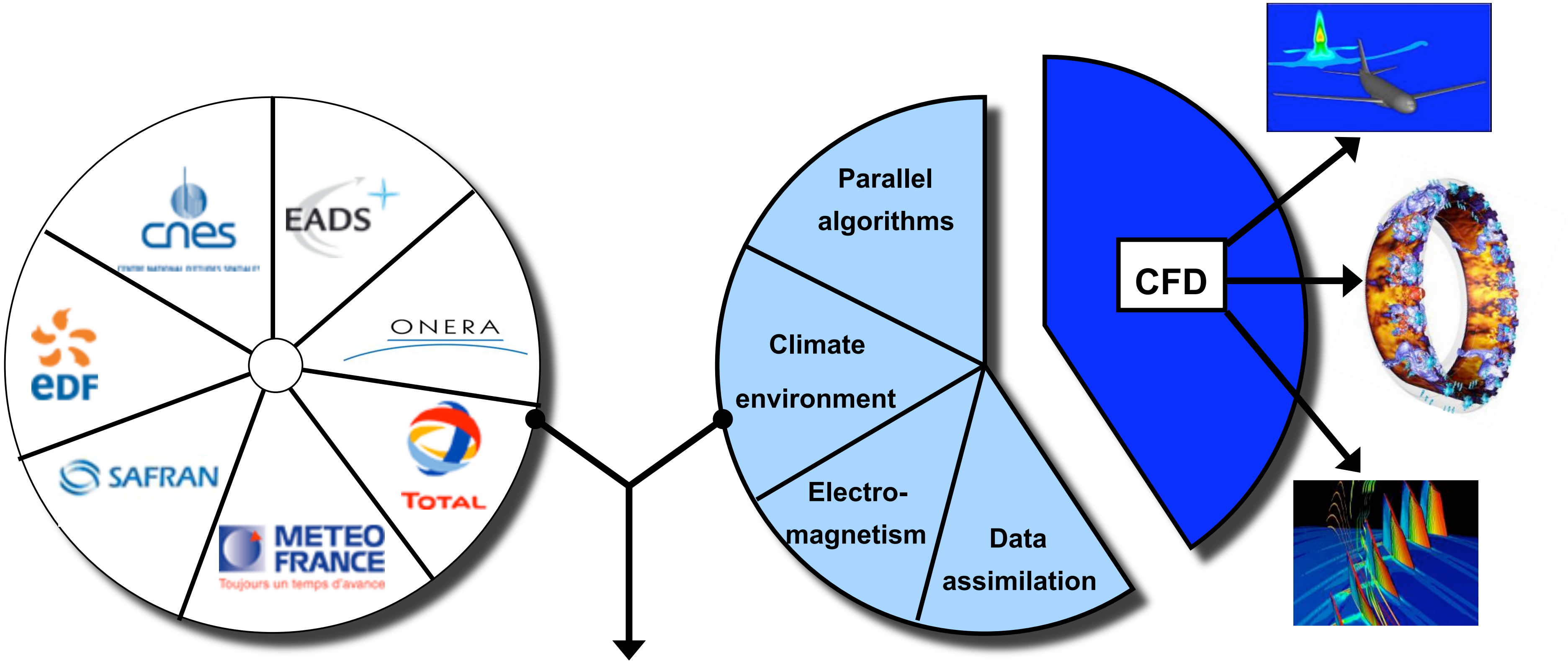
[3] IMFT, Toulouse

http://www.cerfacs.fr

Laurent.Gicquel@cerfacs.fr

What's CERFACS?

CERFACS has seven shareholders | **One hundred people in 5 teams**



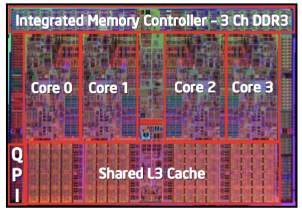
- Expertise in scientific computation
- Access to large computational resources

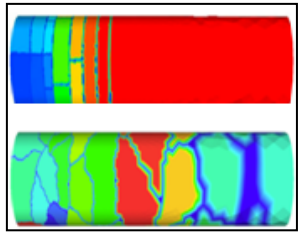
1 Introduction
- Context

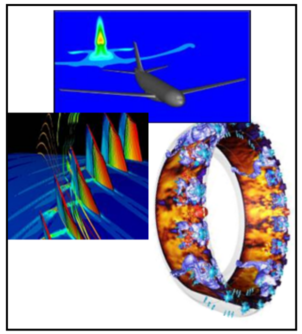
2 Numerical developments for HPC
- Flow solver examples
- Speed-up and Mesh-partitioning
- Communication, Impact on numerical solutions

3 Application to aeronautic challenges
- Performance indicators
- Civil aircrafts
- Compressor
- Combustion chambers
- Turbines

4 Conclusion and perspectives





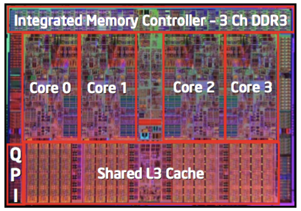


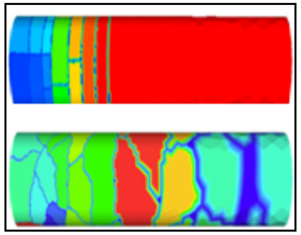
1 Introduction
- **Context**

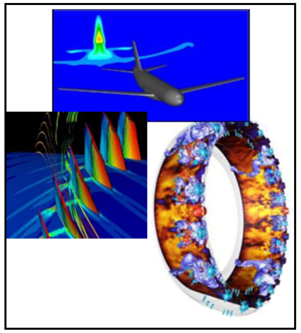
2 Numerical developments for HPC
- Flow solver examples
- Speed-up and Mesh-partitioning
- Communication, Impact on numerical solutions

3 Application to aeronautic challenges
- Performance indicators
- Civil aircrafts
- Compressor
- Combustion chambers
- Turbines

4 Conclusion and perspectives

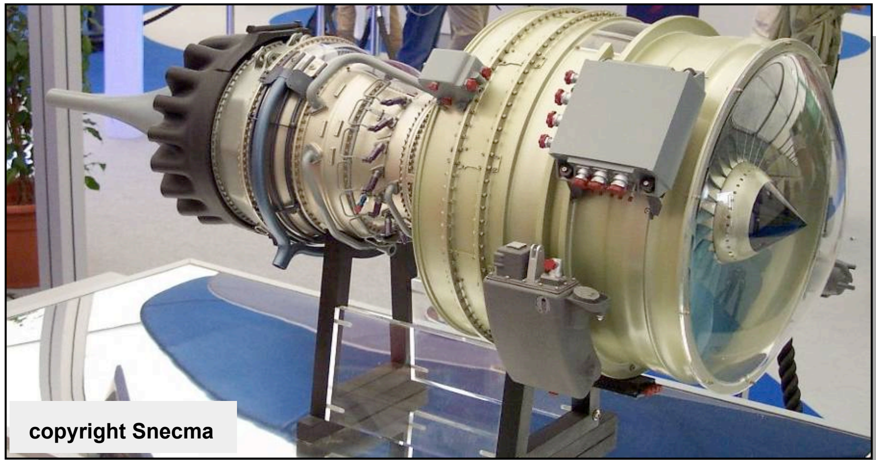






The aeronautic context

- CO_2 emissions from 1990 to 2025[1]: **+100-600%** (2008: 2.2% of the total).
- **European objectives for 2020[2]:**
  - reduce pollutant emissions (NO_x: -80%, CO_2: -50%),
  - reduce the noise emissions (-10dB).
- **Economical constraints:**
  - cut the engine cost (today it represents 30% of the aircraft cost).



Constraints are not just technical but also economical and environmental!

(1) INRETS, 2004

(2) ACARE recommendations

What is the status of CFD today?

• Computational Fluid Dynamics (CFD) describe the flow behavior, usually based on the Navier-Stokes equations,

• CFD is now an essential tool in industry for design and development,

• Strong industrial demands to tackle more and more complex flow phenomena.

On the one hand:

• CFD investigates modeled physical flows at a lower cost than "pure" experimental methods and can thus help complementing fundamental and industrial developments.

On the other hand:

• CFD is not yet always predictive for most industrial applications (complex geometries, high Reynolds numbers…).

Numerical codes require high-end computing platforms.

The term High Performance Computing (HPC) usually refers to (massively) parallel processing (also used as a synonym for supercomputing).

The physical limit of CFD: turbulence and the large range of flow scales

Aeronautical flows have a very high Reynolds number: $Re = \frac{\rho U L}{\mu} \Rightarrow N \propto (0{,}1\, Re)^{9/4}$

- Aircraft at cruise conditions:
- Compre
- Combu

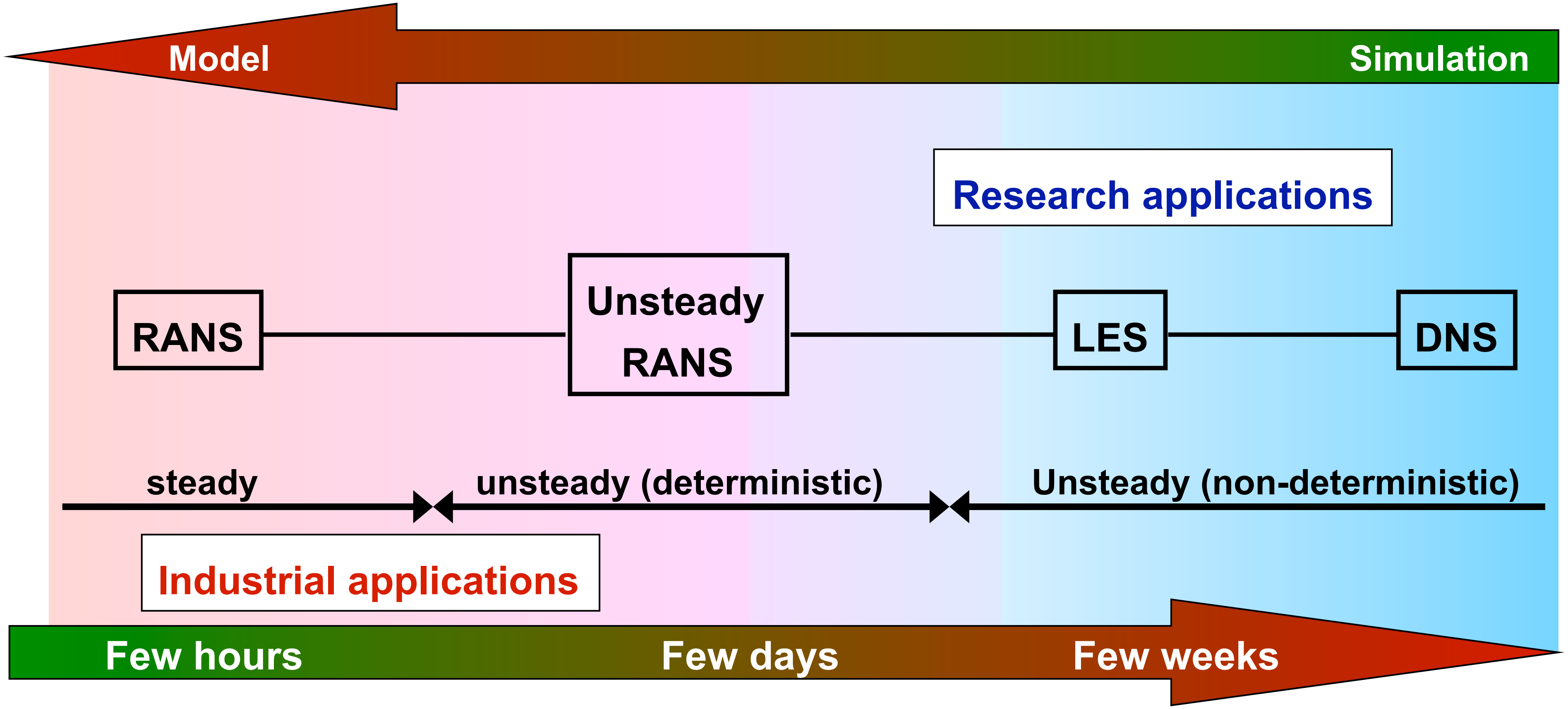
- Turbine at operating conditions:

  Re ~ 1 10^6 => N ~ 1 $10^{11.25}$

You need to do something to your set of governing equations to allow descent computing effort and take care of turbulence

Flow / Turbulence modeling

Overview of the computational methods

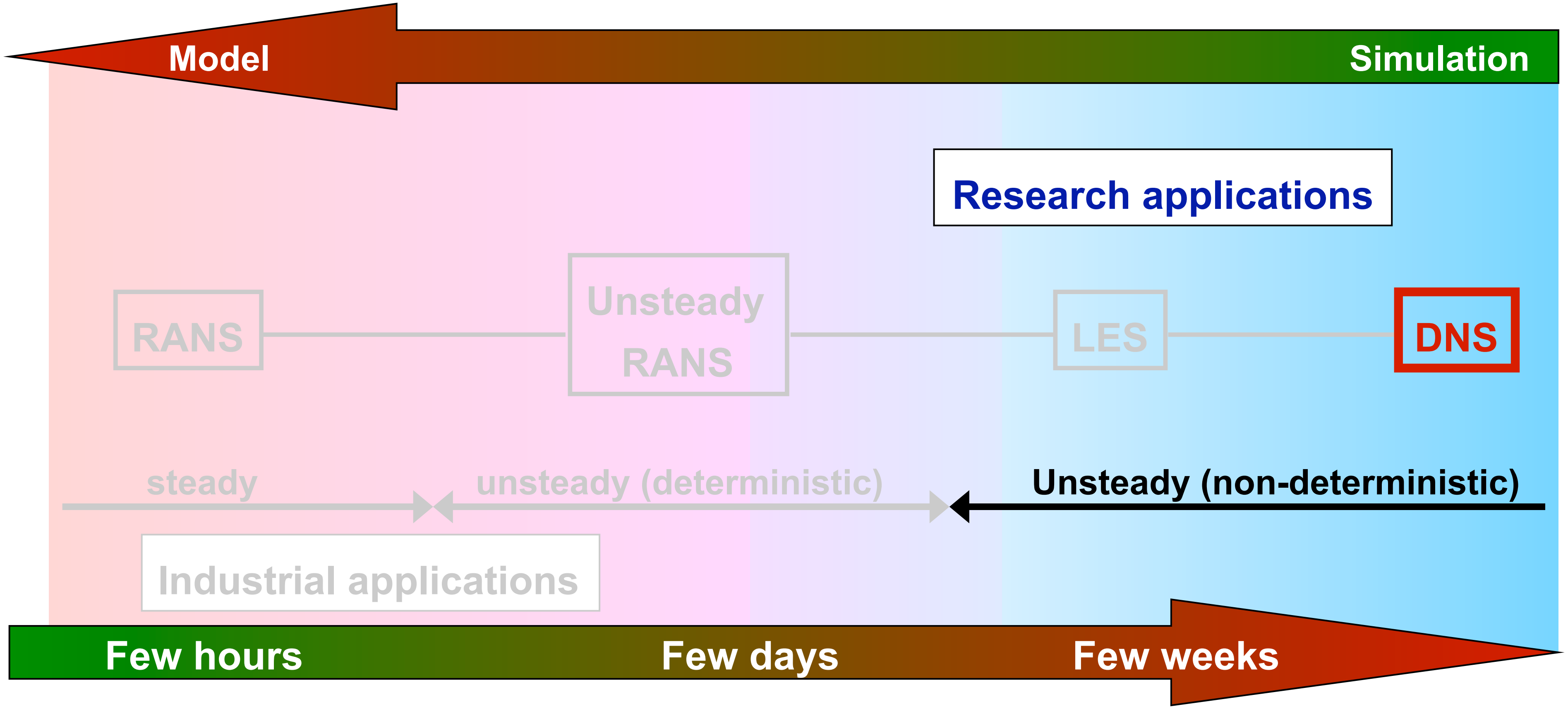


RANS: Reynolds-Averaged Navier Stokes

LES: Large Eddy Simulation

DNS: Direct Numerical Simulation

Overview of the computational methods

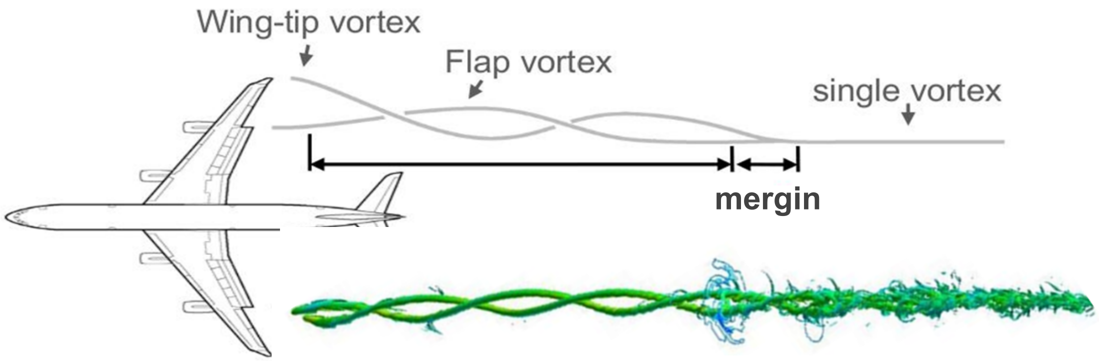


RANS: Reynolds-Averaged Navier Stokes

LES: Large Eddy Simulation

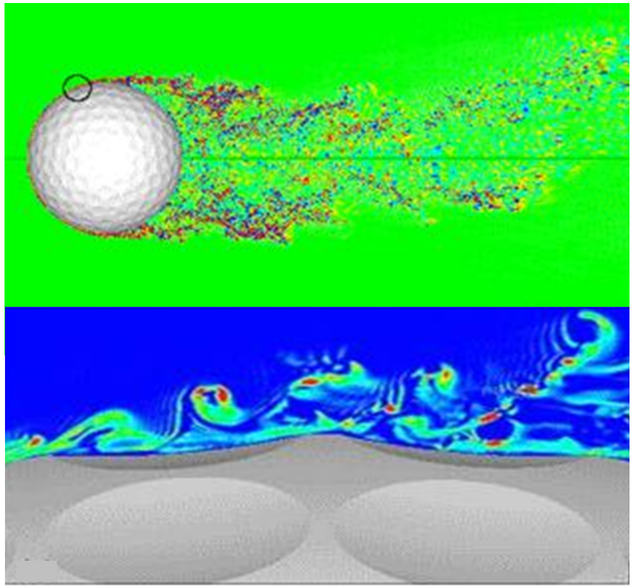
DNS: Direct Numerical Simulation

Examples of recent complex flow simulations: DNS



Simulation of wake vortex instabilities behind aircrafts (Nybelen et al., 2008):

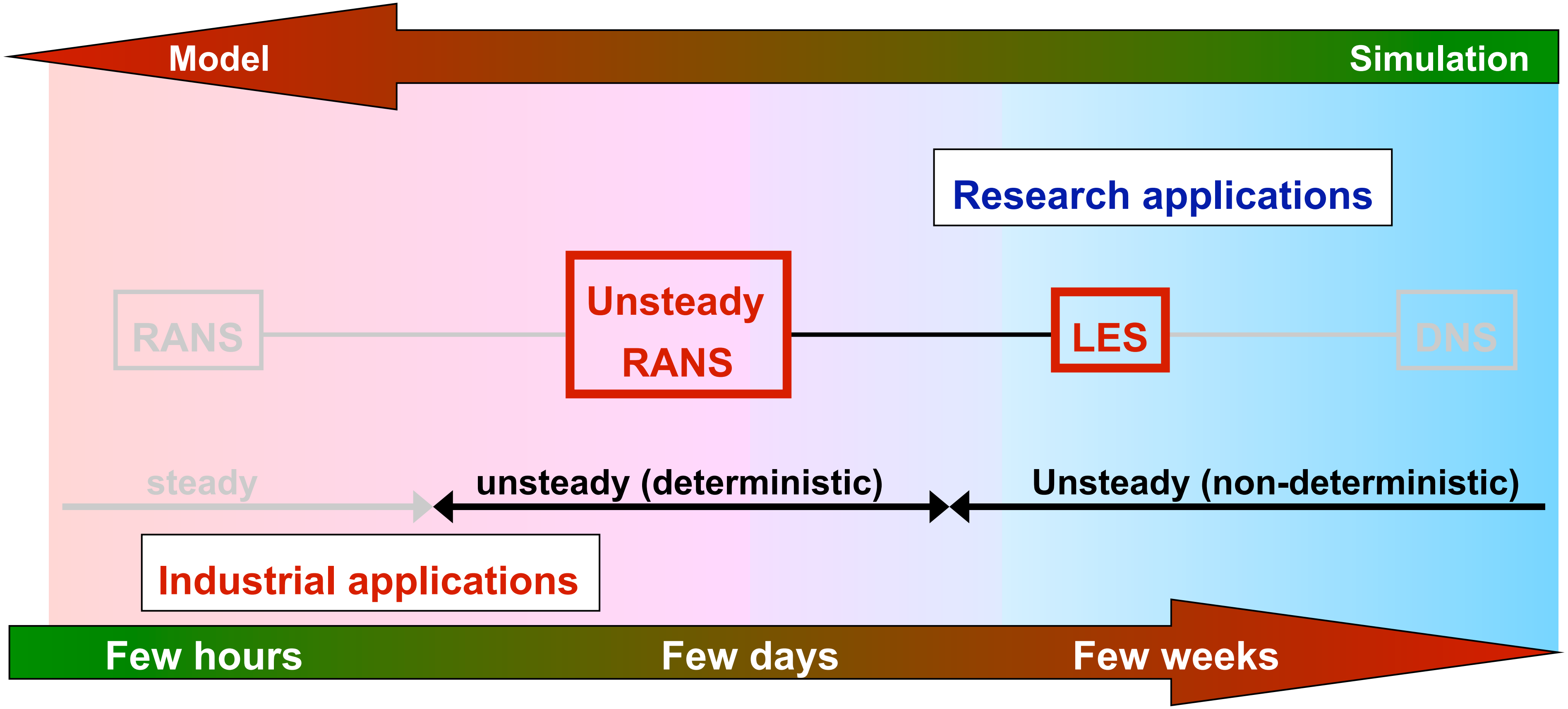
- DNS method (Re=10^4) with NTMIX,
- 110M cells (structured),
- **350 hours with 1024 computing cores (Blue Gene /L).**



Flow simulation around a dimpled sphere (Smith et al., 2008):

- DNS method (Re=10^5),
- 61M - 1200M cells,
- **300 hours with 500 computing cores.**

Overview of the computational methods



RANS: Reynolds-Averaged Navier Stokes

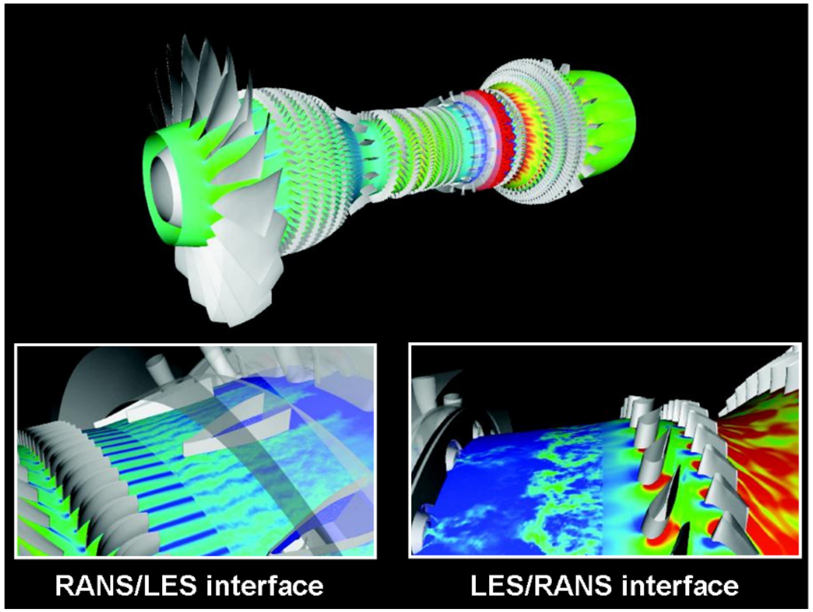
LES: Large Eddy Simulation

DNS: Direct Numerical Simulation

Examples of recent complex flow simulations: URANS/LES

Tip vortex noise simulation in a wind turbine (Arakawa et al., 2005):

- LES method,
- 320M cells (structured),
- 300 hours with 112 vector cores.



Whole gas turbine flow simulation (van der Weide, 2008):

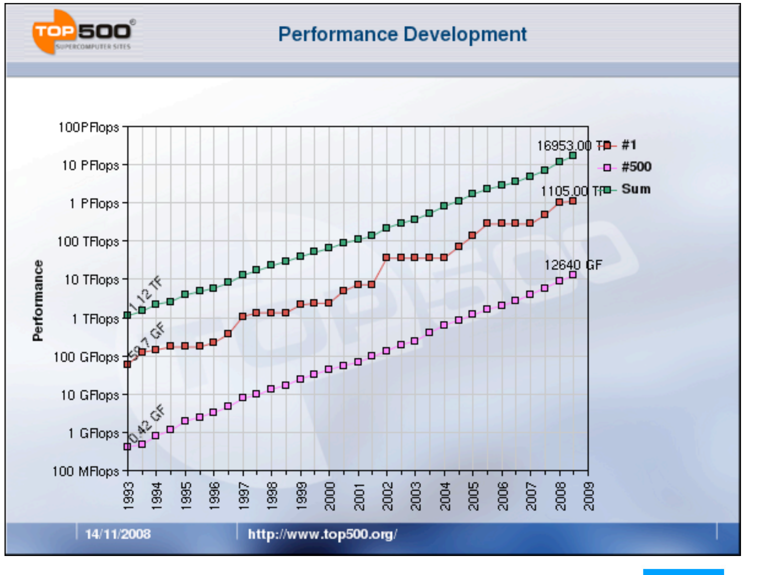
- RANS/LES coupling method,
- 350M cells (unstructured/structured),
- 2600 hours with 1024 computing cores.

Overview of the most powerful computers in the world

- **Numerical methods, such as DNS/LES/URANS are known since a long time,**
- **But enough computing power is available since only few years to apply them for industrial configurations.**

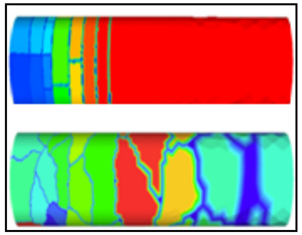
Ranking based on the top500, 11/2008 (www.top500.org)

Rank	System	Cores	R_{max} (Tflops)	Power (kW)
1	USA, Roadrunner - IBM BladeCenter, 2008	129600	1105.00	2483.47
2	USA, Jaguar - Cray XT5, 2008	150152	1059.00	6950.60
3	USA, Pleiades - SGI Altix 2008	51200	487.01	2090.00
4	USA, IBM Blue Gene/L, 2007	212992	478.20	2329.60
5	USA, IBM Blue Gene/P, 2007	163840	450.30	1260.00
6	USA, Ranger - SunBlade, 2008	62976	433.20	2000.00
7	USA, Franklin - Cray XT4, 2008	38642	266.30	1150.00
8	USA, Jaguar - Cray XT4, 2008	30976	205.00	1580.71
9	USA, Red Storm - Cray XT3/4, 2008	38208	204.20	2506.00
10	China, Dawning 5000A, 2008	30720	180.60	
11	Germany, JUGENE - IBM Blue Gene/P, 2007	65536	180.00	504.00
...				
14	France, Jade - SGI Altix, 2008	12288	128.40	608.18
...				
73	Japan, NEC Earth-Simulator, 2002	5120	35.86	3200.00



- Best-ranked systems are massively scalar platforms,
- First vector supercomputer is 73^{th} (Earth simulator).

Integrated Memory Controller - 3 Ch DDR3
Core 0
Core 1
Core 2
Core 3
Shared L3 Cache
Q
P
I

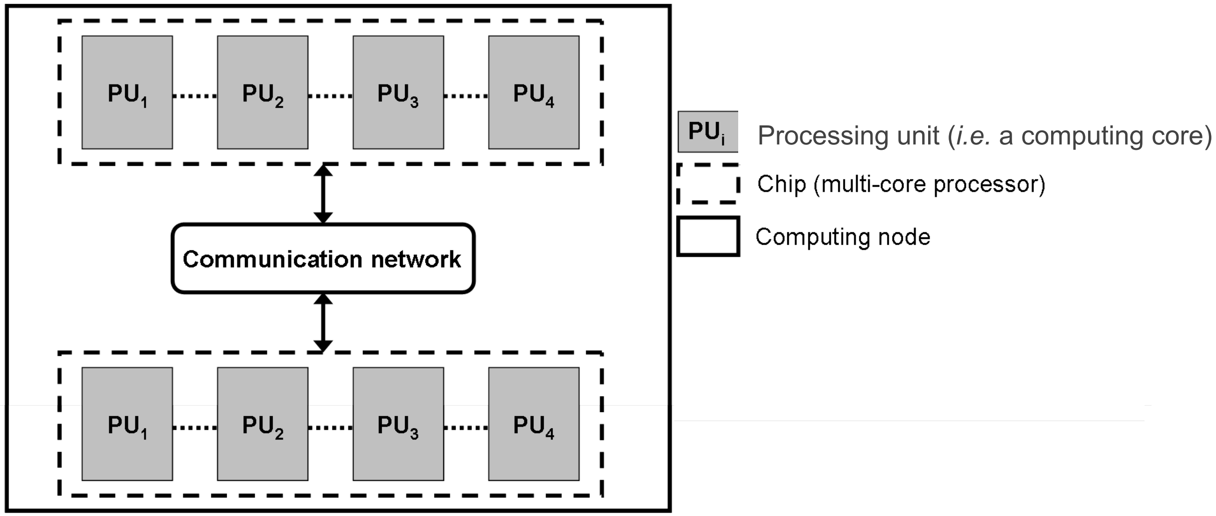


Few definitions

About core, processor, thread…

(Massively) **parallel platforms** are composed of one or more **computing node**(s),

- a computing node includes one or more **processor**(s),
  - a processor (chip) includes one or more **computing core**(s),
    - a computing core is dedicated to one **process/thread***,
      - a process is assimilated to a counter program and an address space.

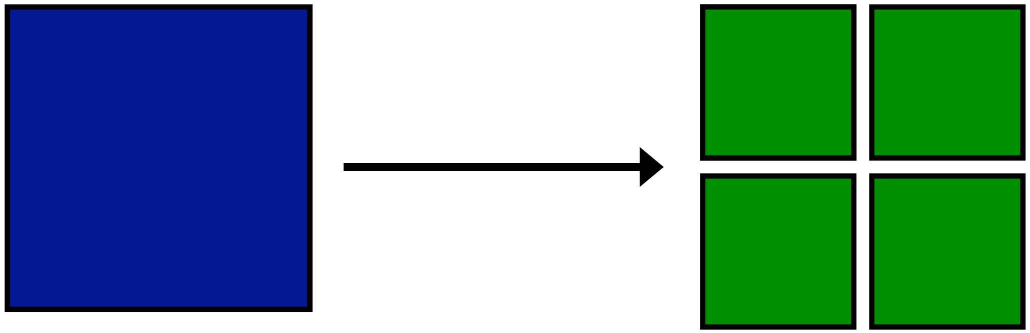


Load balancing is the task sharing between computing cores

*not always true, but here we will set PU=CC

What is the impact of massively parallel platforms?

- HPC based on (massively) parallel is a new challenge for CFD flow solvers,
- Problem related to an efficient use of a large number of computing cores:
  - ➢ **Mesh partitioning, load balancing, communication**?
  - ➢ Impact on flow solvers **implementation**, numerical **solutions**?



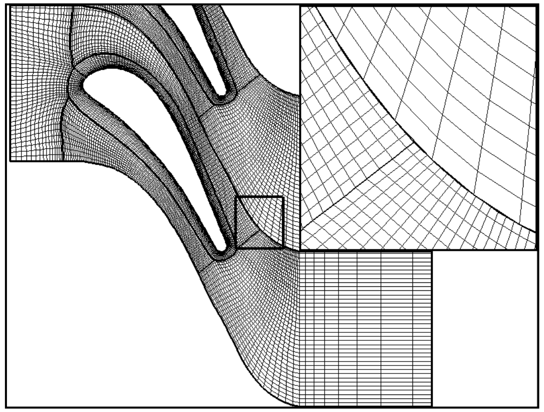
Original problem

New (more) adapted problem

SPMD model:
Single Program /
Multiple Data

- Multidisciplinary team required for adapting flow solvers to HPC platforms,
- Work performed by scientists and computer experts with background on physics modelling, programming, hardware, HPC… and engineers for providing industrial configurations.

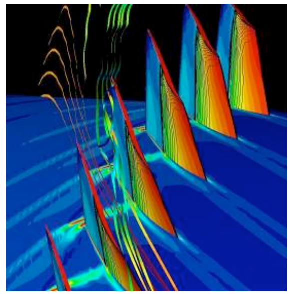
A structured multi-block flow solver: *elsA*

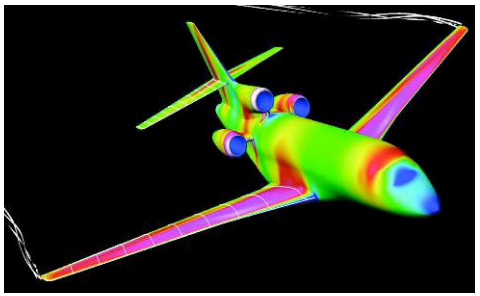


Multi-block structured grid

(Coincident/non-coincident interfaces)

- developed by ONERA[1,2] and CERFACS,
- vector and (massively) parallel capacities,
- ***cell-centered approach, implicit in time,***
- Compressible finite volume formulation,
- ***External/internal flow simulations and multi-disciplinary applications, (Aerodynamics, aero-elasticity, aero-thermal, aero-acoustics…),***
- ***(U)RANS/LES and intermediate methods (TSM/DES),***
- ***Mono-species (perfect gas or equilibrium real gas)**,*
- Languages: C++/Fortran/Python,
- SPMD approach.





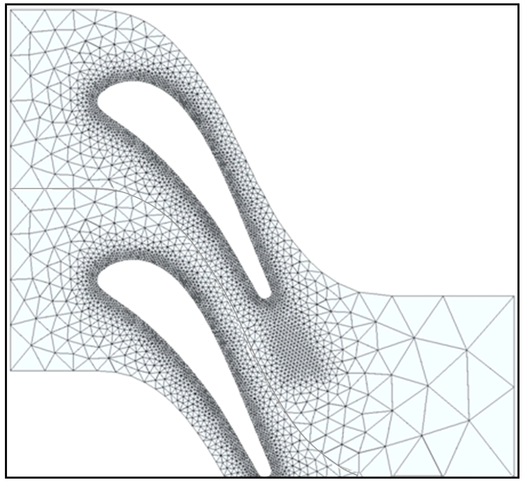
(1) Cambier, 2002

(2) Cambier, 2008

Flow solver examples

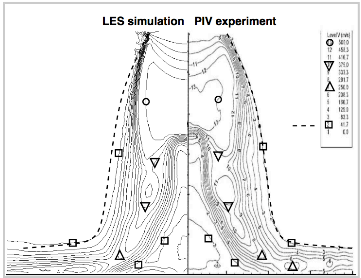
An unstructured flow solver: AVBP

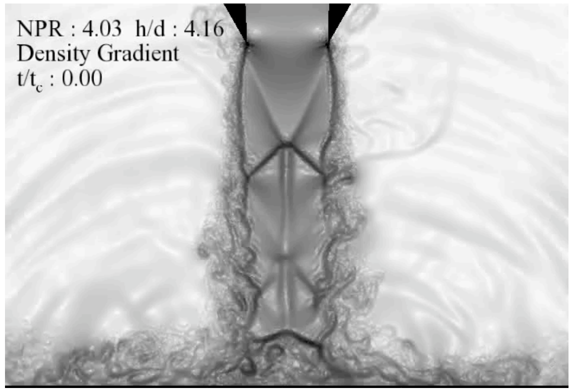
- Developed by CERFACS and IFP,
- External/internal flows,
- ***Fully compressible turbulent reacting flows,***
- ***DNS/LES approaches,***
- ***Unstructured hexaedral, tetraedral, prisms & hybrid meshes,***
- Massively parallel,
- C/Fortran languages,
- SPMD approach.

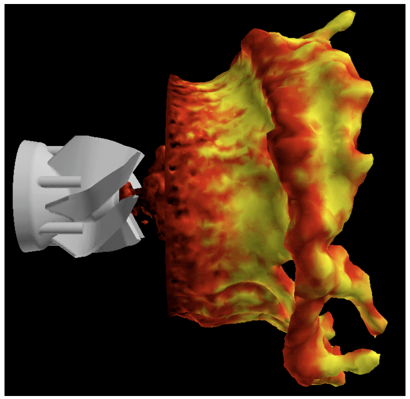


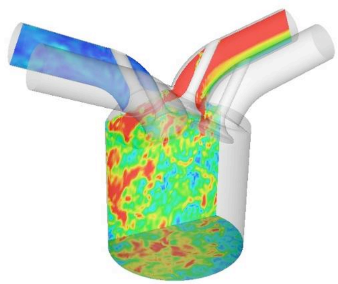
Unstructured grid

(Coincident interfaces)







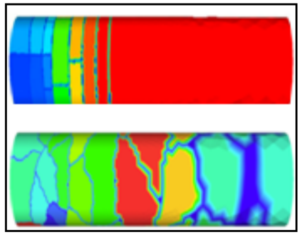


1 Introduction
- Context

2 Numerical developments for HPC
- Flow solver examples
- **Speed-up and Mesh-partitioning**
- Communication, Impact on numerical solutions

3 Application to aeronautic challenges
- Performance indicators
- Civil aircrafts
- Compressor
- Combustion chambers
- Turbines

4 Conclusion and perspectives



Estimation of the theoretical computing efficiency

- **Computing efficiency** is related to the computational time/communication ratio, load balancing…

- **The speed-up** is used to quantify the time reduction related to parallel computing (S=1 is the sequential time, S=N corresponds to a reduction by N…)

- **Predict the computational time** associated to a (massively) parallel simulation is essential for:
  - ➢ estimating the computational cost,
  - ➢ managing task scheduling.

- **Different methods** can be used:
  - ➢ ideal efficiency,
  - ➢ Amdahl's law,
  - ➢ Extended Amdahl's law.

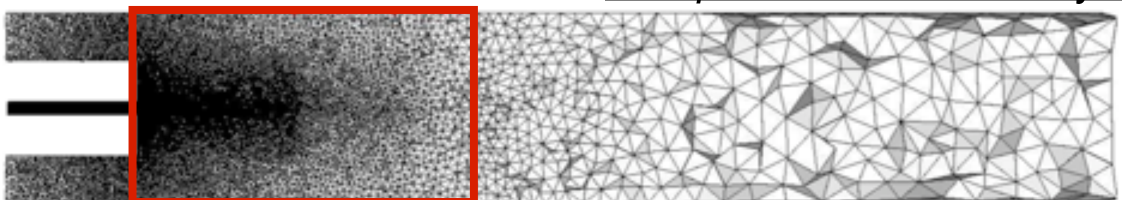
Mesh partitioning for unstructured mesh : splitting algorithms

Different partitioning algorithms (AVBP):

(1) Karypis et al., 1998

- RCB / RIB: geometric based algorithms,
- RGB: graph theory based algorithm,
- METIS[1]: multi-constraint multilevel graph partitioning.

Two-phase flow bluff body configuration



RIB

(32 cores)

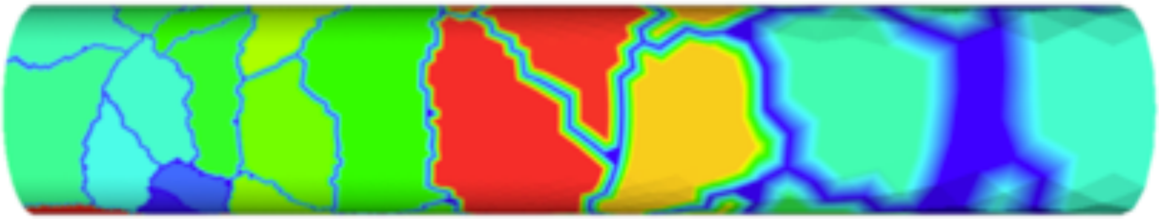
One constraint

(geometry)

METIS

K Way

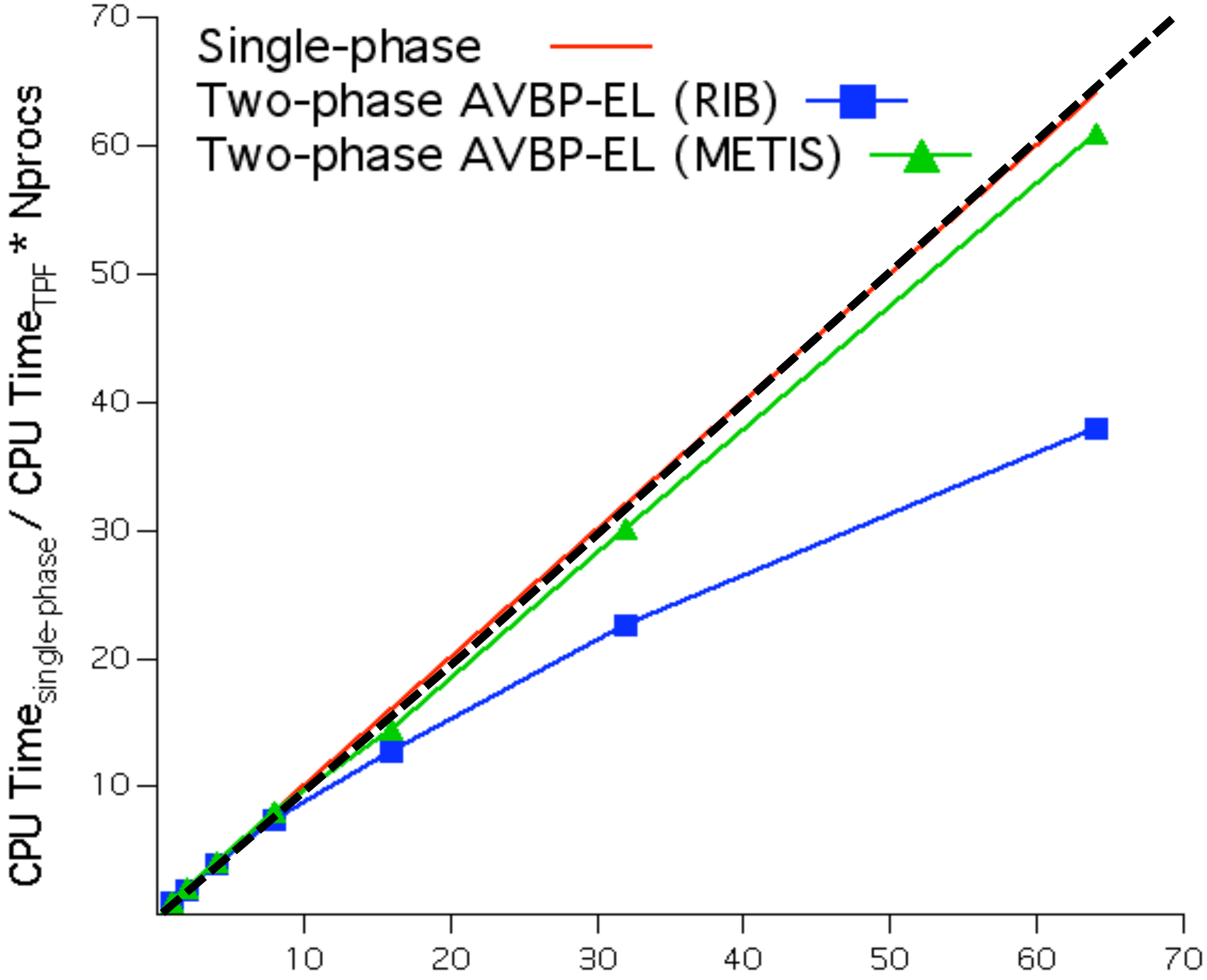
(32 cores)



Two constraints

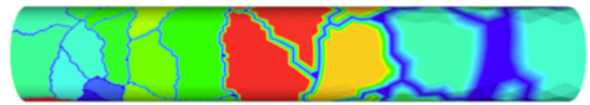
(graph + particles)

Mesh partitioning for unstructured mesh : load balancing



RIB

Partitioning on 32 cores

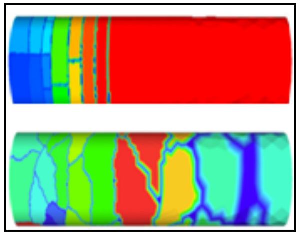


METIS

Garcia, PhD, 2009

1 Introduction
- Context

2 Numerical developments for HPC
- Flow solver examples
- Speed-up and Mesh-partitioning
- **Communication, Impact on numerical solutions**

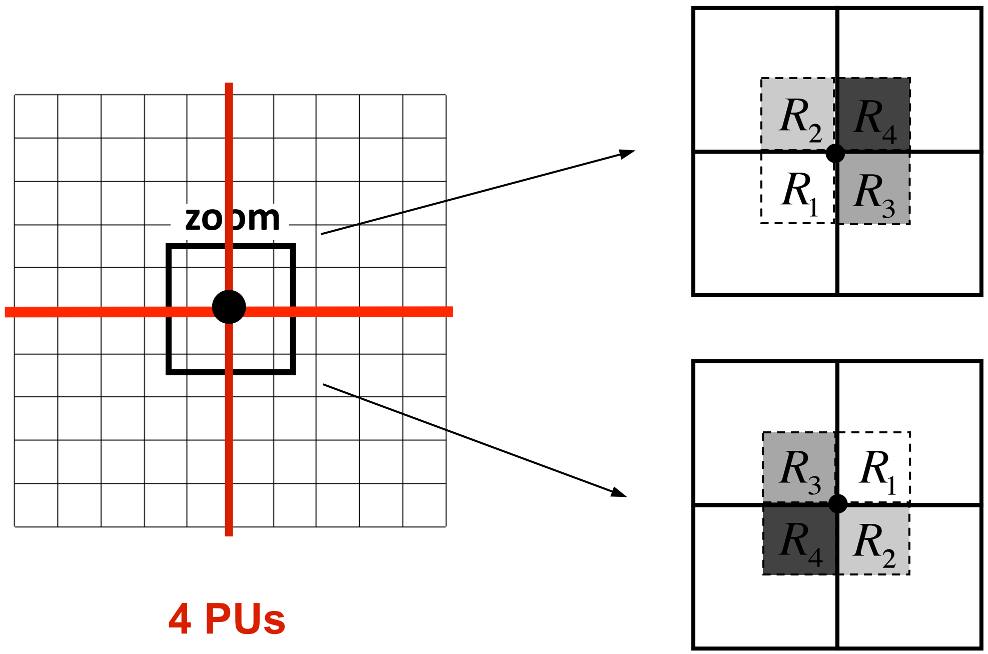


3 Application to aeronautic challenges
- Performance indicators
- Civil aircrafts
- Compressor
- Combustion chambers
- Turbines

4 Conclusion and perspectives

Communication strategy: MPI non-blocking calls

How to compute a residual at partition interfaces?



Like this…

$$\frac{1}{4}\left\{R_4 + \left[R_3 + \left(R_2 + R_1\right)\right]\right\}$$

D C B A

≠

$$\frac{1}{4}\left\{R_1 + \left[R_2 + \left(R_3 + R_4\right)\right]\right\}$$

A B C D

…or like this…

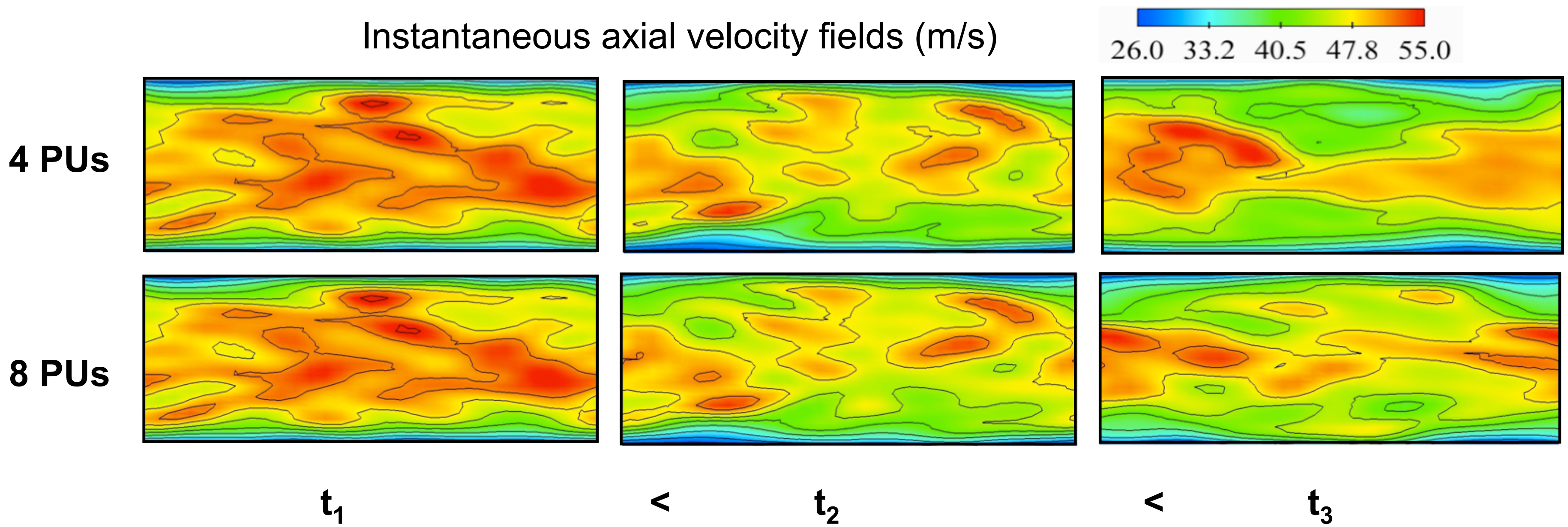
Problem: non-blocking communications induce a non-deterministic behavior

Impact of rounding errors on LES

Consequence of the lack of associativity property (Floating point arithmetic):

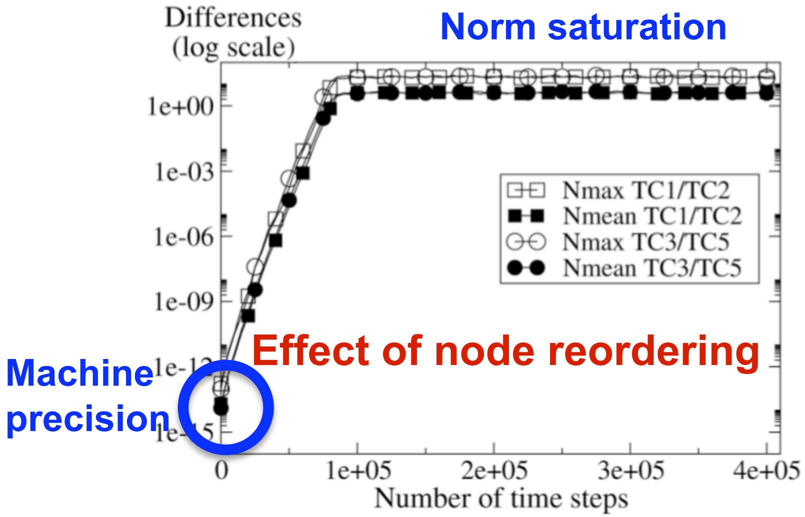
- illustration on a temporally evolving turbulent channel (AVBP).

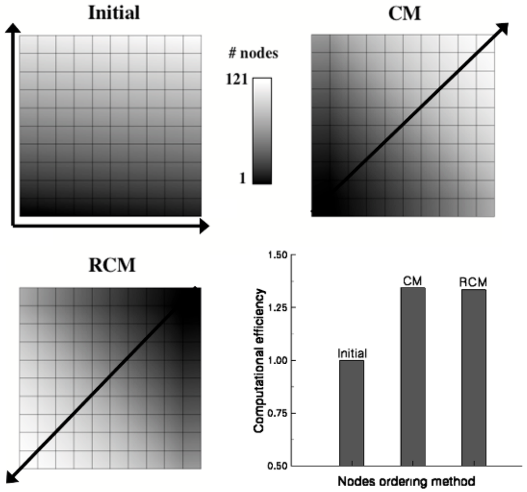
Instantaneous axial velocity fields (m/s)

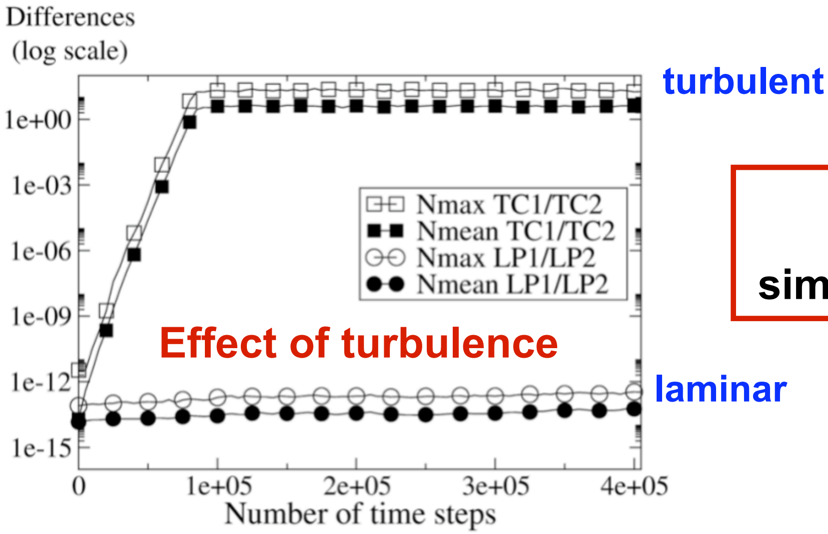


Senoner et al., 2008

Impact of rounding errors on LES







These results are not induced by parallel simulations (calc. performed with a single core!)

Senoner et al., 2008

Impact of mesh-partitioning

- Non-blocking communications participate to rounding errors (non-deterministic behavior),
- Blocking communications are good for deterministic behavior,
- Any sufficiently turbulent flow computed in LES exhibits significant sensitivity to small perturbations, leading to instantaneous solutions which can be totally different,
- The divergence of solutions is explained by 2 combined factors:
  - exponential separation of trajectories in turbulent flows,
  - propagation of rounding errors induced by domain partitioning and scheduling operations that can be different.
- Implicit stages done on a block basis can result in different convergence/instantaneous solutions:
  - in practice, this also impacts RANS convergence history… However since the solution is unique (?) and stationnary there should be no degradation of solution observed.

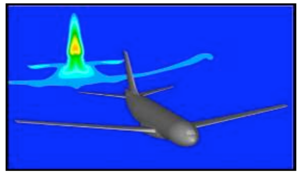
Validation of LES code after modifications may only be based on statistical fields!

1 Introduction
- Context

2 Numerical developments for HPC
- Flow solver examples
- Speed-up and Mesh-partitioning
- Communication, Impact on numerical solutions

3 Application to aeronautic challenges
- **Civil aircrafts**
- Compressor
- Combustion chambers
- Turbines

4 Conclusion and perspectives



Objectives

• Aircraft design requires interdisciplinary and multi-physics numerical methods,

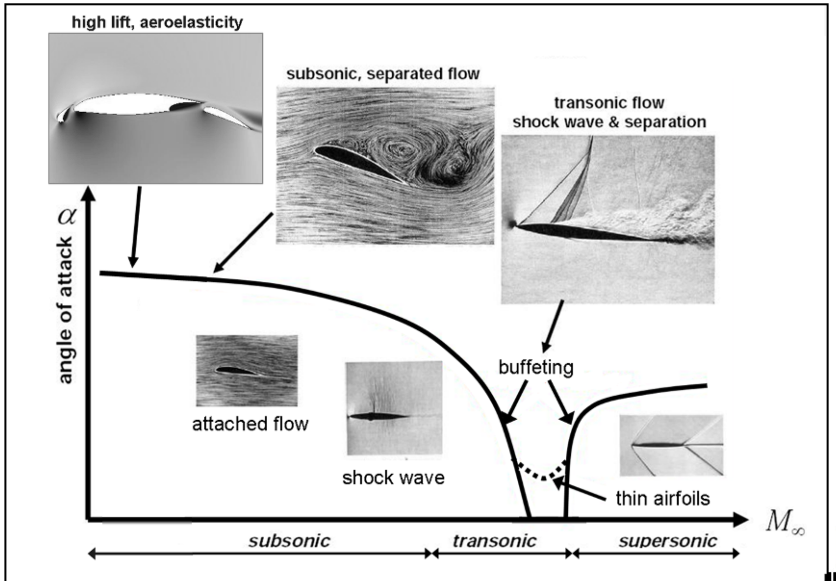
• HPC can be useful at the design stage:

  ➢ by reducing the time required to obtain the solution,

  ➢ for optimization,

  ➢ by helping to manage the evaluation of the complete flight domain.

• Design of greener and quieter aircrafts has to tackle with complex physics such as shock/boundary layer interaction (buffet phenomenon), massively separated flows, aero-elastic instabilities…



Rossow, 2008

Application to aero-elasticity

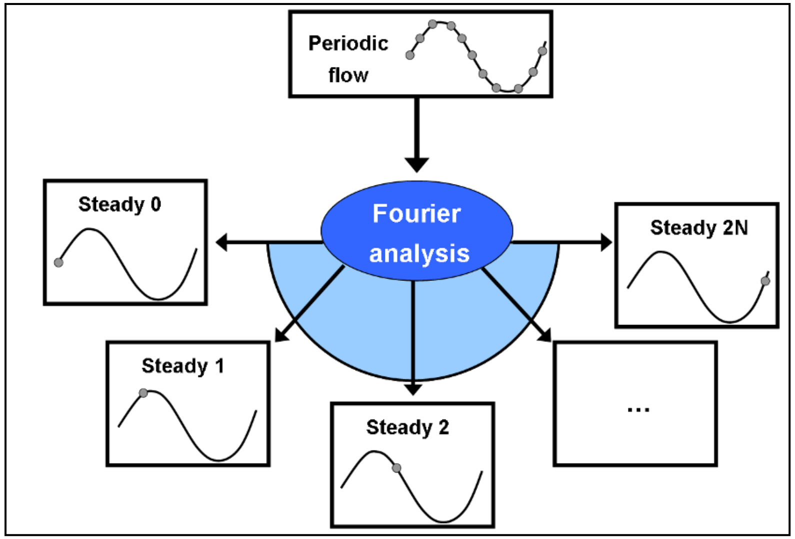
• A direct fluid/structure coupling is beyond industrial resources,

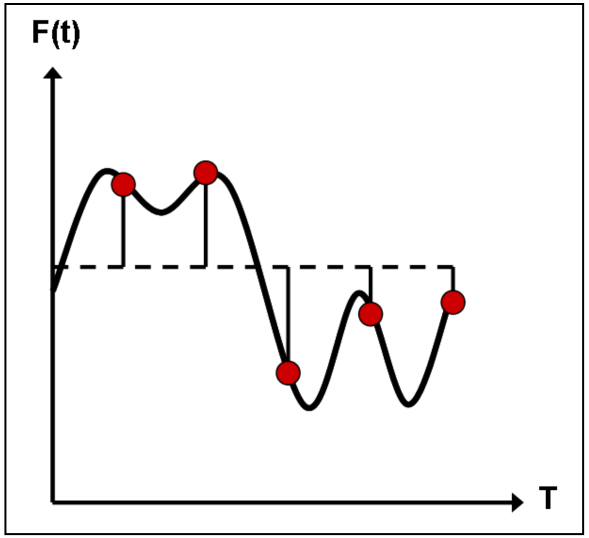
• A common practice is thus to perform forced motion simulations to obtain unsteady loads on the wings of aircraft configurations,

• Simulation of aero-elastic effects, coupling with flaps/slaps/spoilers ,… need very large grids due to complexity (30-100M cells),

• Classical unsteady methods (URANS) still very costly for industrial applications, even when considering massively parallel platforms → Time Spectral Method (TSM).

Hall, 2003
Gopinath, 2007
Sicot, 2008

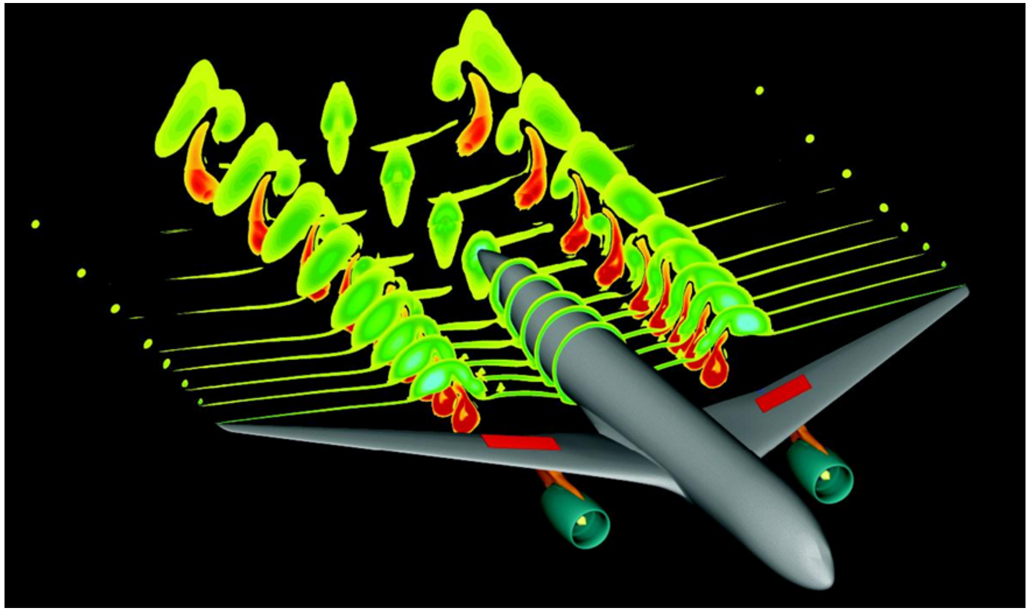




Application to aero-elasticity

Simulation with TSM: reduce computational time (<10), but increase memory too (<7).

Computing method	computational time	memory consumption
Standard aeroelastic solver	1	1
TSM-aerolasticity, 1 harmonic	0.09	3
TSM-aerolasticity, 2 harmonic	0.16	5
TSM-aerolasticity, 3 harmonic	0.24	7

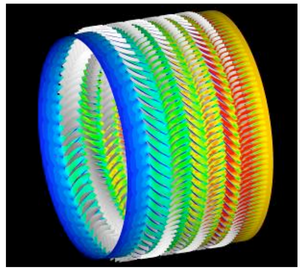


Aeroelastic effects computed with the TSM (including spoilers) in a whole generic long range aircraft performed with *elsA*

1 Introduction
- Context

2 Numerical developments for HPC
- Flow solver examples
- Speed-up and Mesh-partitioning
- Communication, Impact on numerical solutions

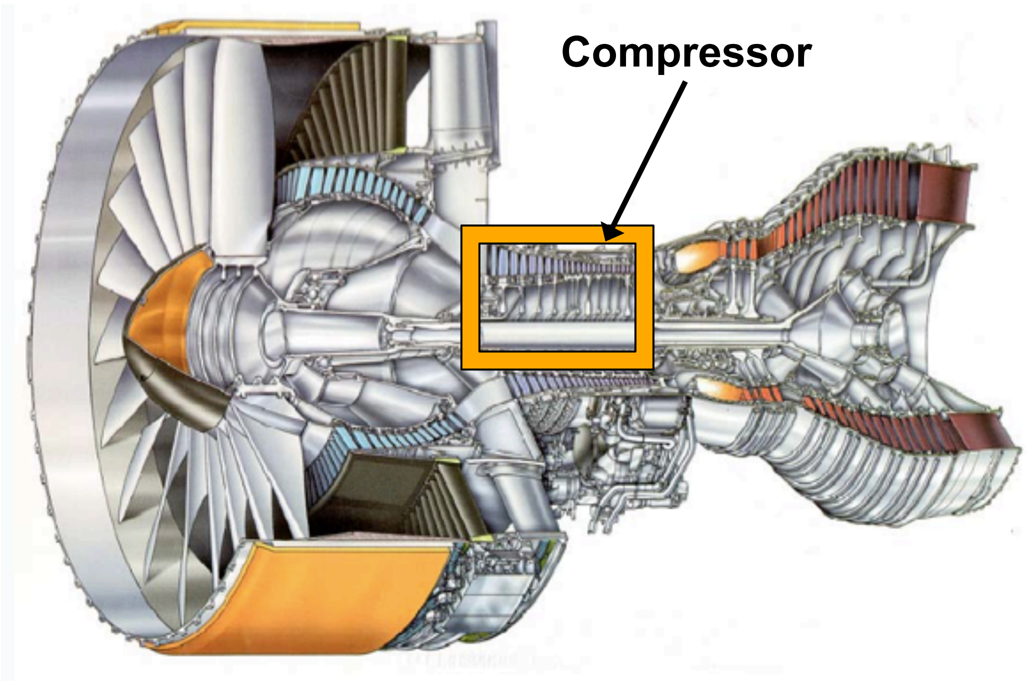
3 Application to aeronautic challenges
- Civil aircrafts
- **Compressor**
- Combustion chambers
- Turbines



4 Conclusion and perspectives

Objectives

- Rotating machines are involved in most of the energy conversion processes,
- Unsteady flows are still not well understood, especially in multistage turbomachines,
  - ➢ aerodynamic instabilities are penalizing for efficiency (design margins).
- Main reasons are computational cost, size and complexity of the configurations:
  - ➢ most of the industrial simulations focus on limited parts of the system (such as isolated blades) that are solved with a steady RANS approach.

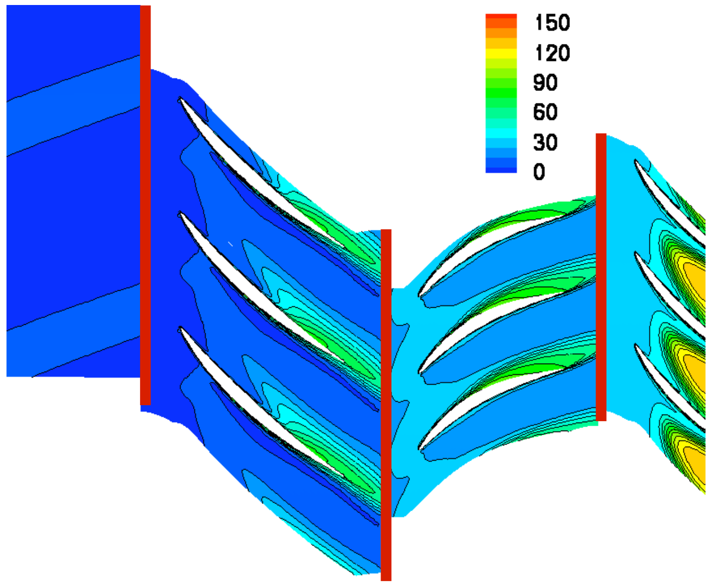


Industrial approach: steady mixing plane method

Mixing plane proposed by Denton (1979):

- Steady calculation considering only one passage for each row,
- Unsteady flows at interface are filtered,
- Whole mesh is around 1 – 10M cells for a 3 stage compressor.

- **low cost,**
- **unlimited number of rows,**
- **no unsteady interactions,**
- **not adapted to all configurations.**

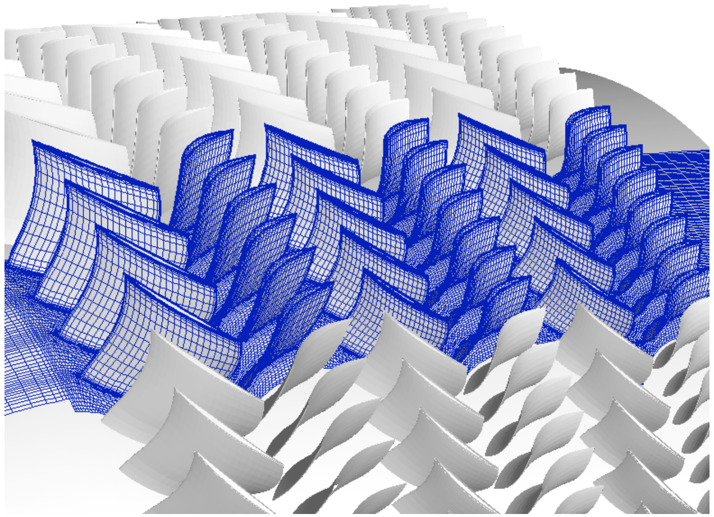


Steady mixing plane solution (entropy flow field)

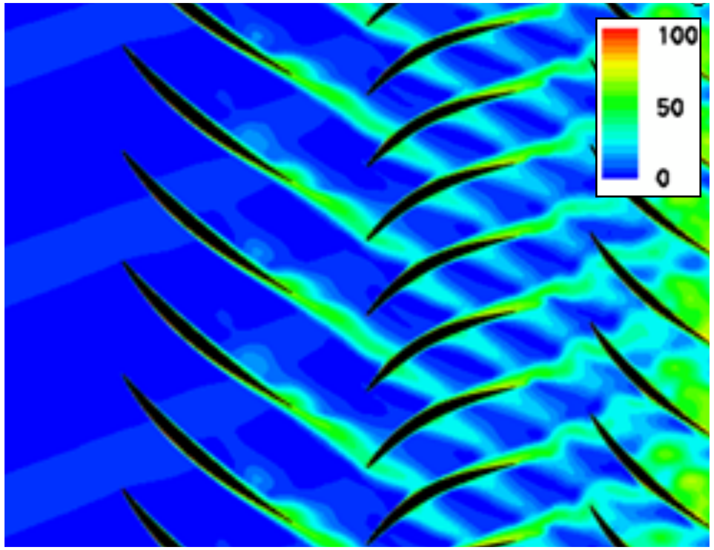
Research approach: unsteady whole configuration

Sliding mesh method (non-coincident interface):

- Unsteady RANS calculation considering the whole geometry,
- All unsteady interaction at interface are simulated,
- Whole mesh is around 100M – 1000M cells for a 3 stage compressor.



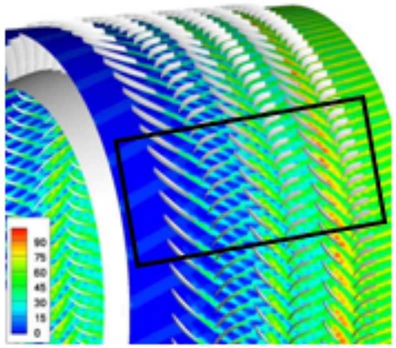
- unlimited number of rows,
- unsteady flow interactions,
- adapted to all configurations,
- important cost.

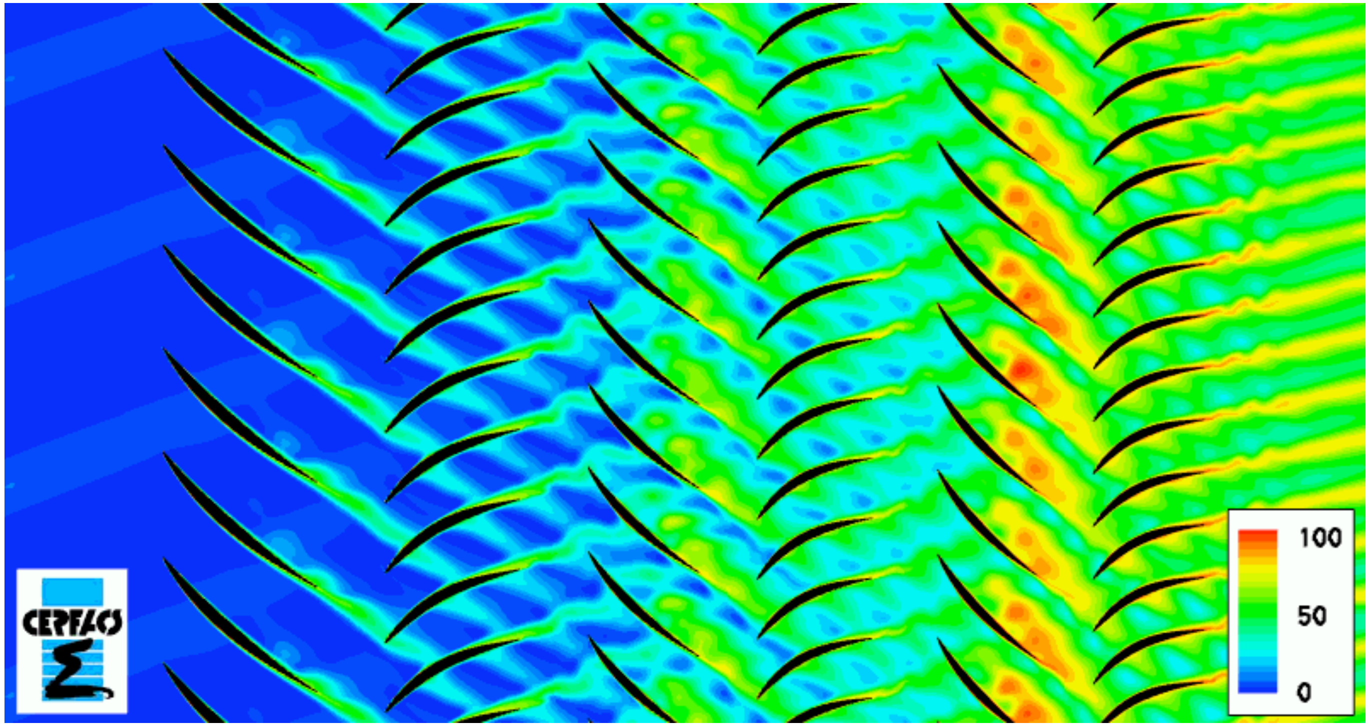


Unsteady whole configuration solution (entropy flow field)

Simulation at design operating point

- **512 processors (Blue Gene/L),**
- **24 days of computation (one rotation), *i.e.* 300,000 CPU hours.**



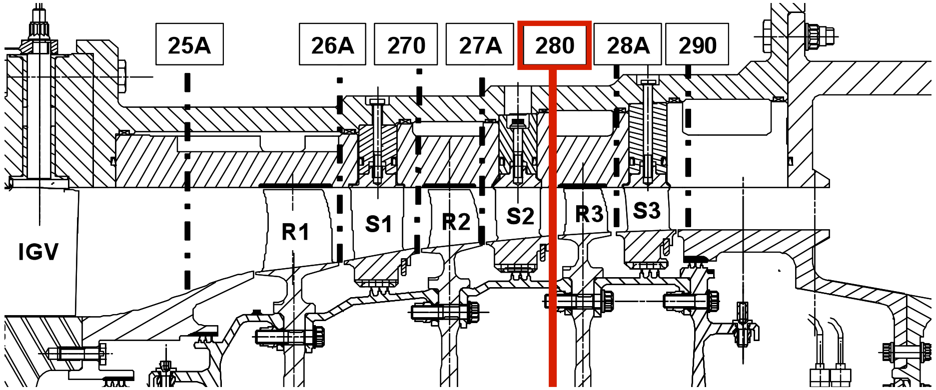


Entropy flow field (h/H=83%)

Large multistage effects (blade rows interactions):

➢ **flow in the 3rd rotor is partially driven by wakes of the 2nd stator.**

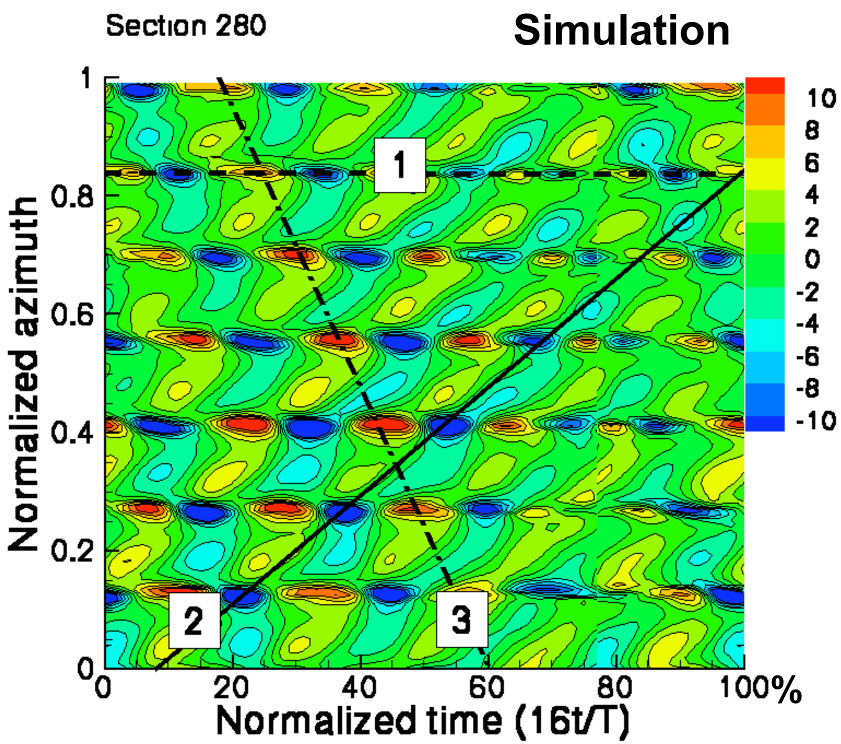
Comparisons of experimental/research results

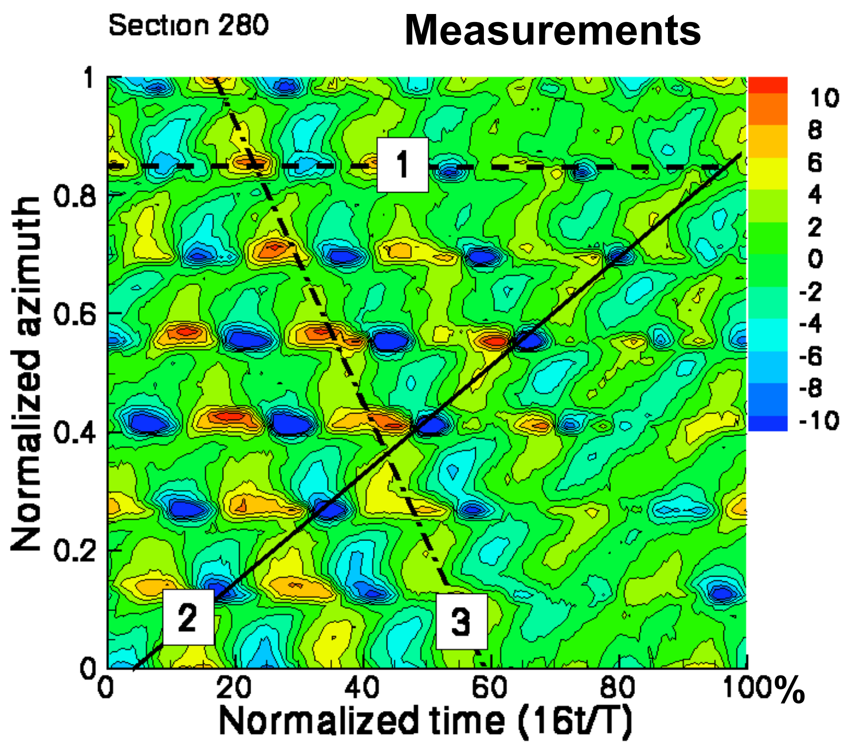


1: stator wakes

2: rotor potential effects

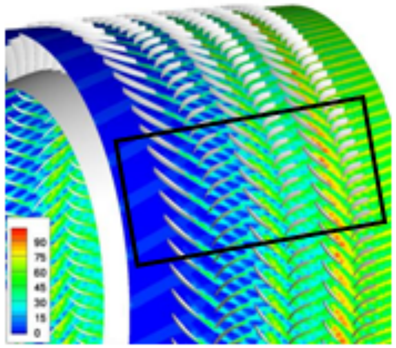
3: rotor-stator interaction modes

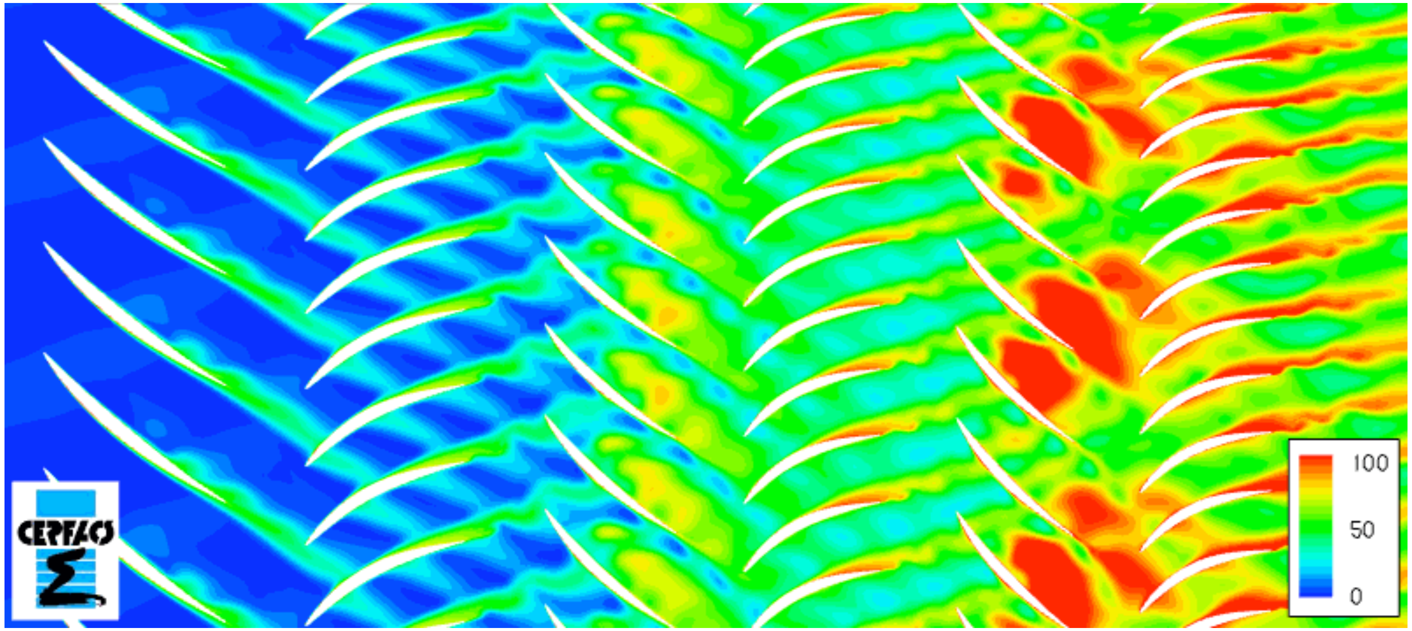




Simulation at off-design conditions

- **4096 PUs (Blue Gene/P, EDF),**
- **40 days of computation (one rotation), *i.e.* 4,000,000 CPU hours.**



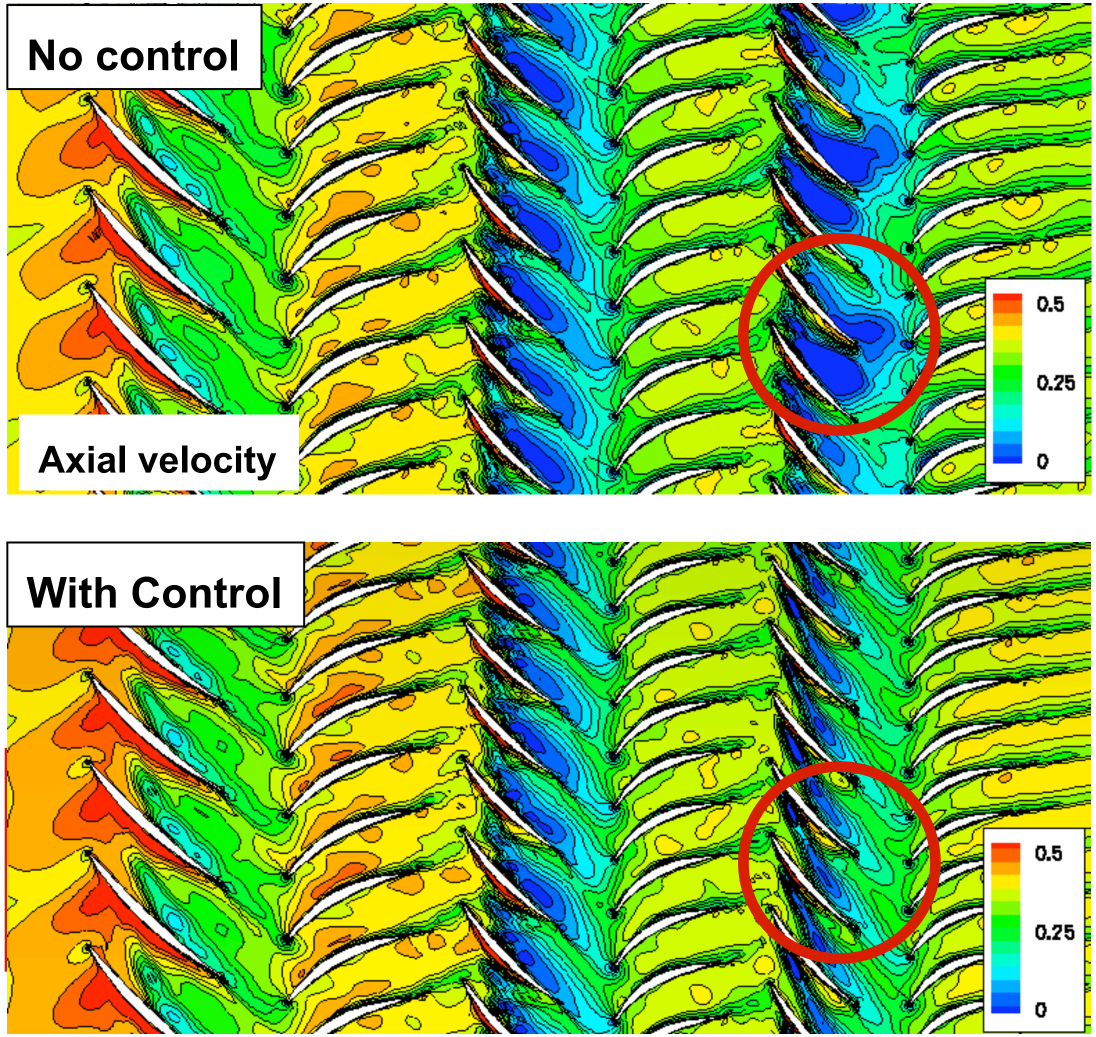


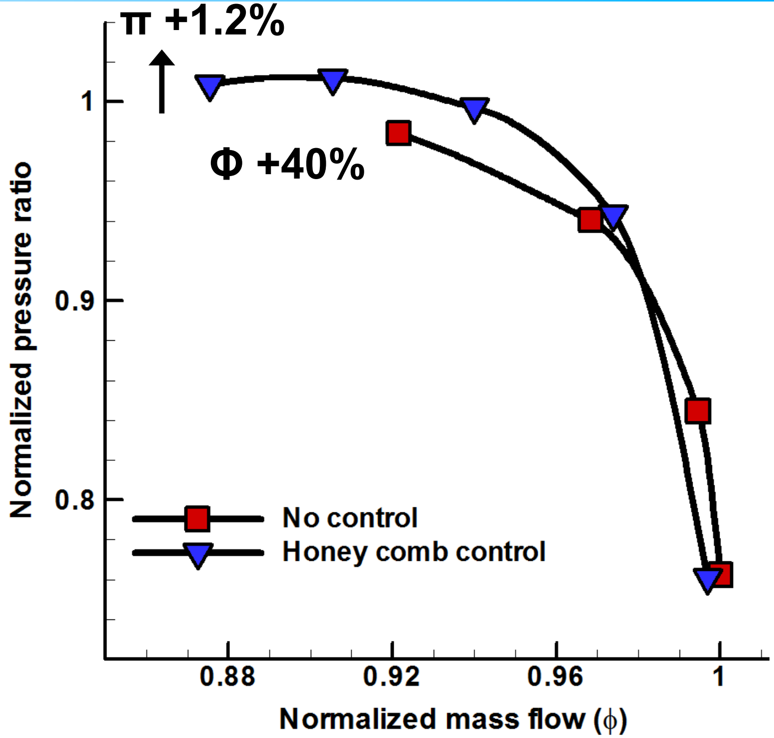
Entropy flow field

Large multistage effects (blade rows interactions):

➢ **aerodynamic instabilities develop in the 3rd rotor.**

Simulation of a control solution





- Suppression of rotating stall,
- Increase of performance.

Overview

1 Introduction
- Context

2 Numerical developments for HPC
- Flow solver examples
- Speed-up and Mesh-partitioning
- Communication, Impact on numerical solutions

3 Application to aeronautic challenges
- Civil aircrafts
- Compressor
- **Combustion chambers**
- Turbines

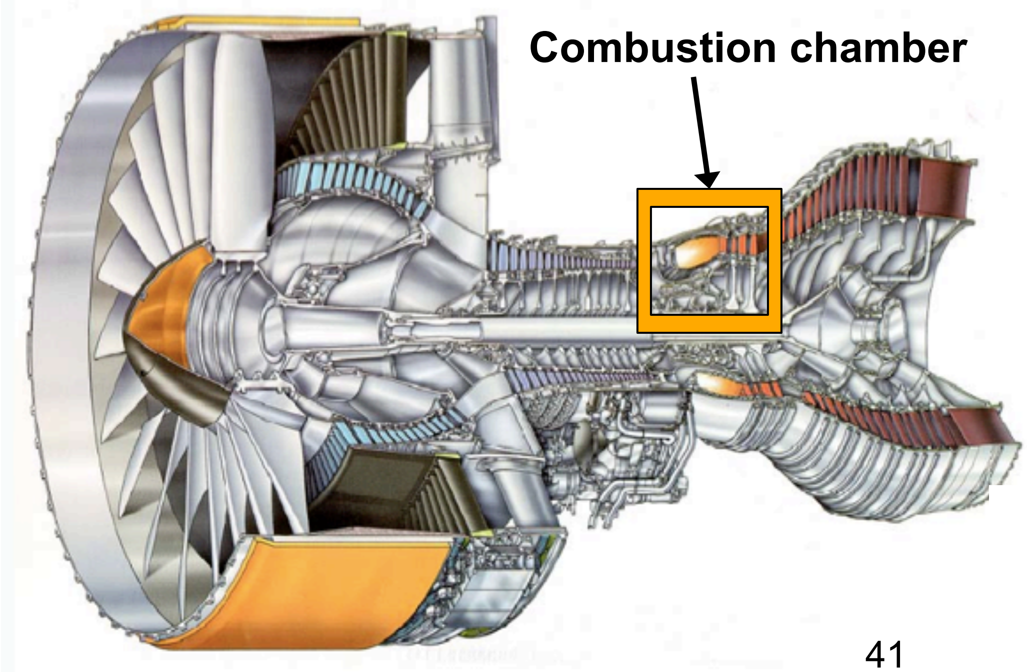
4 Conclusion and perspectives

Objectives

Flow acceleration due to gas expansion/combustion:

- Subject to thermo-acoustic oscillations (highly destructive and quasi unpredictable),
- Locus of pollutant formation,
- Strong thermal constraints…

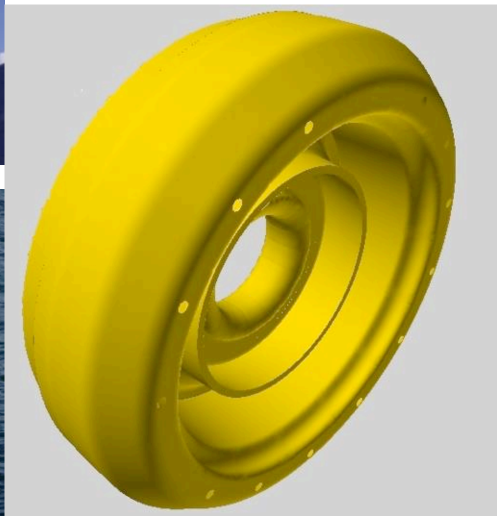
=> Most recent pubications demonstrate the superiority of LES: i.e. captures the strong coupling between turbulence/mixing/combustion

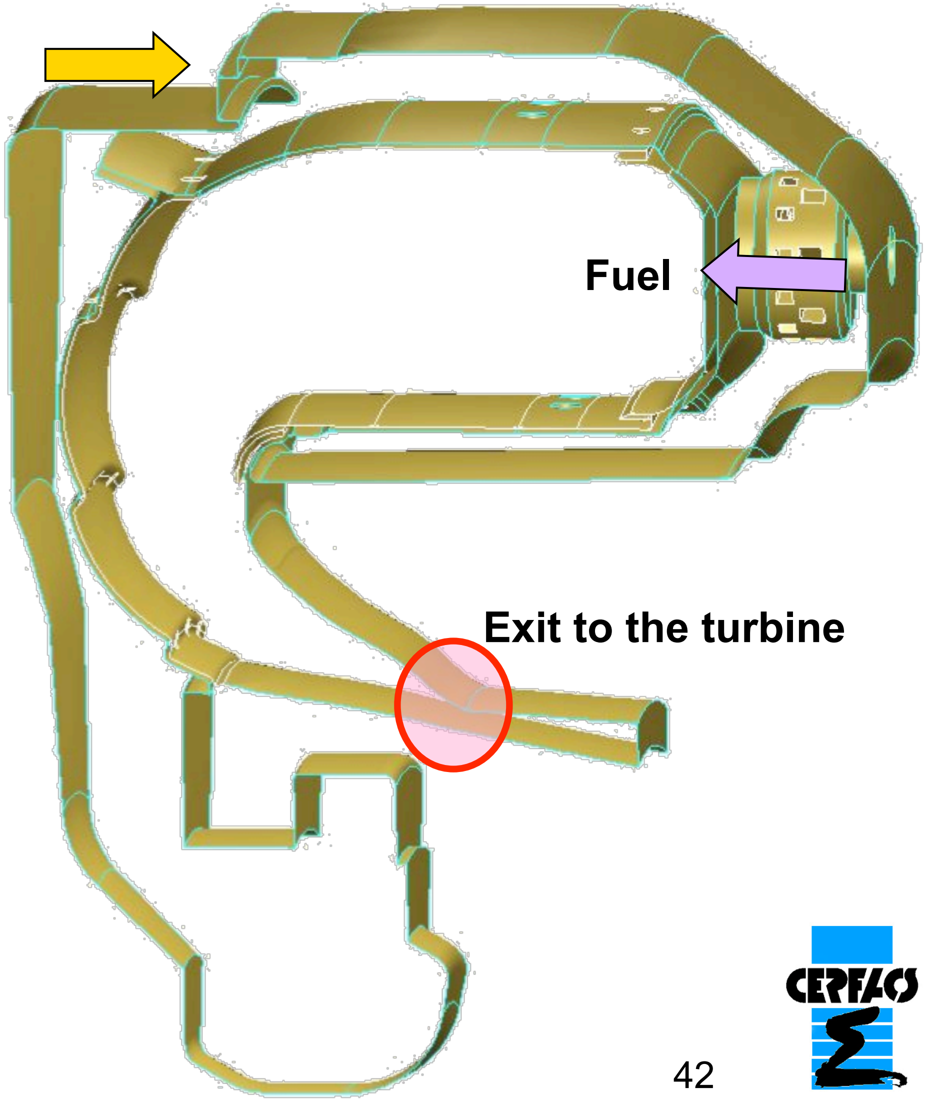


Context: higly complex geometry

Target configuration: a helicopter combustion chamber at cruise conditions.

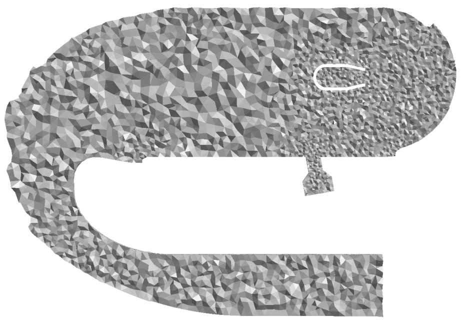
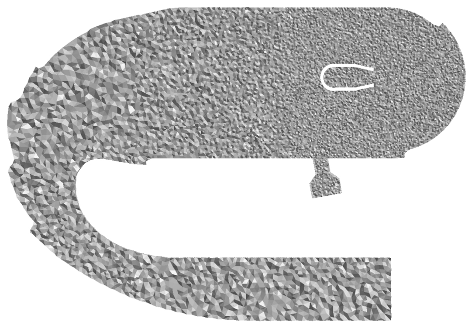
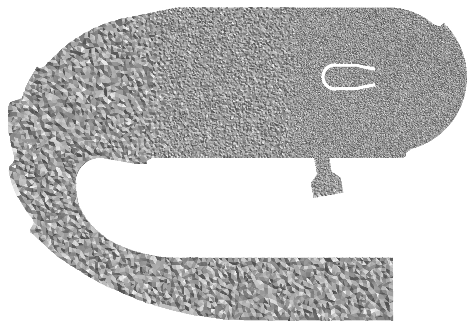
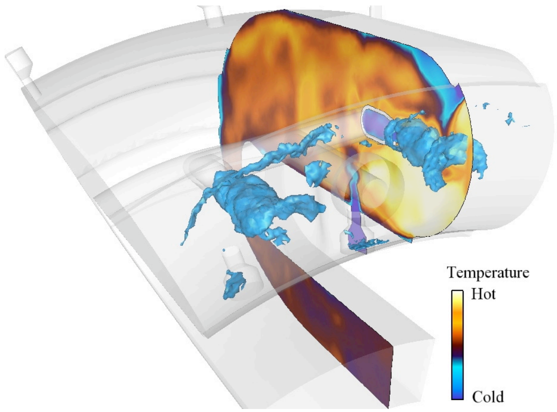
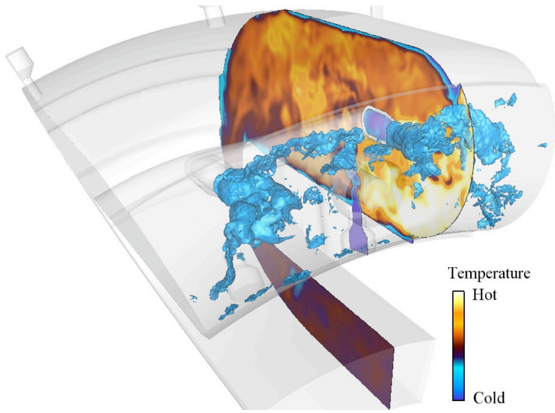
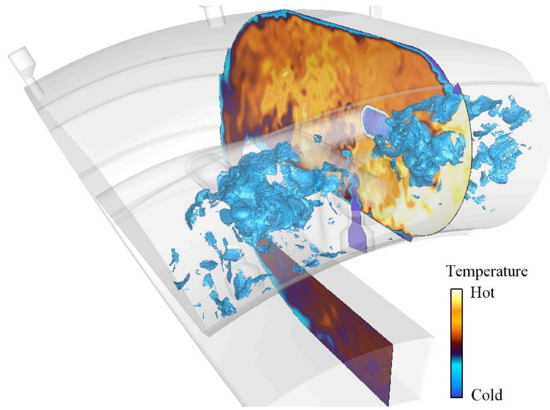
Annular burner





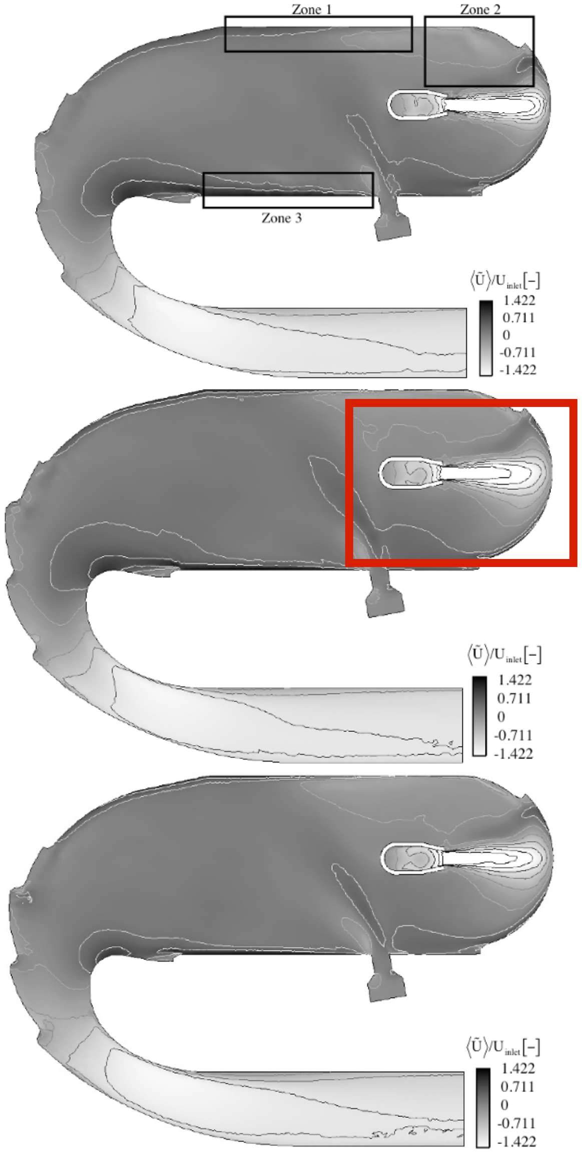
Effect of grid resolution: overview

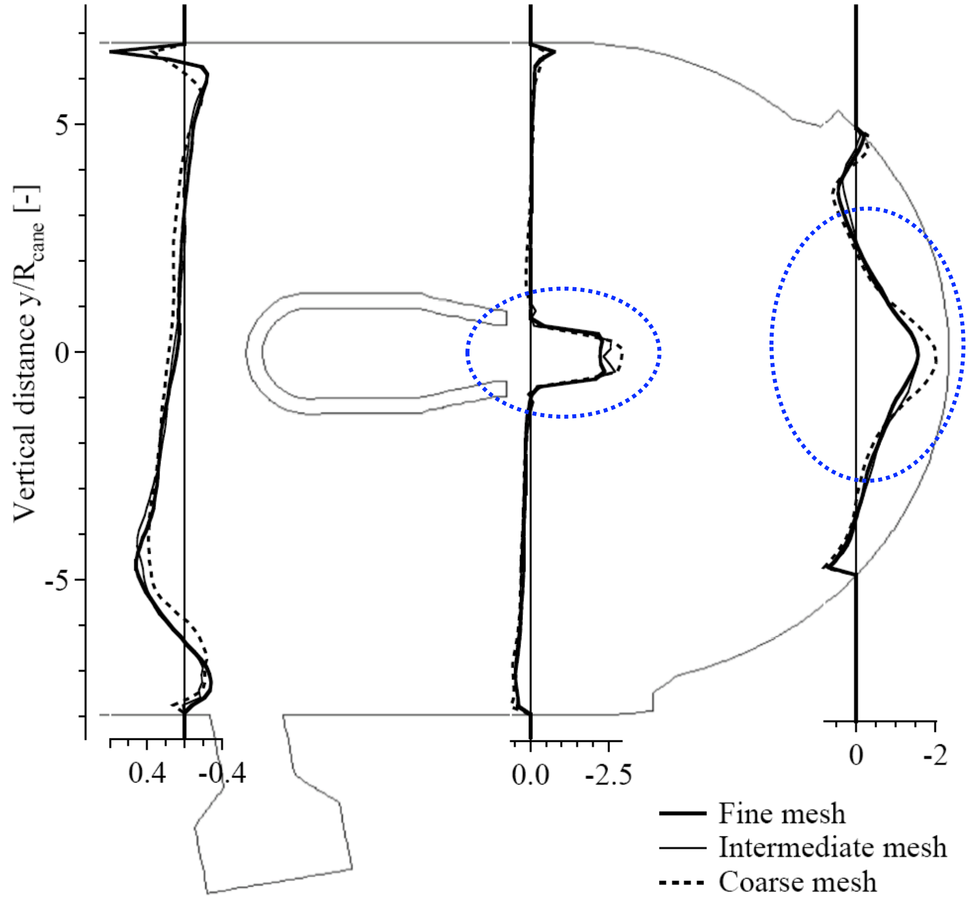
LES in a single sector burner

Coarse mesh	Intermediate mesh	Fine mesh
		
		
315 CPU hours	**4,550 CPU hours**	**30,200 CPU hours**

Effect of grid resolution: mean quantities

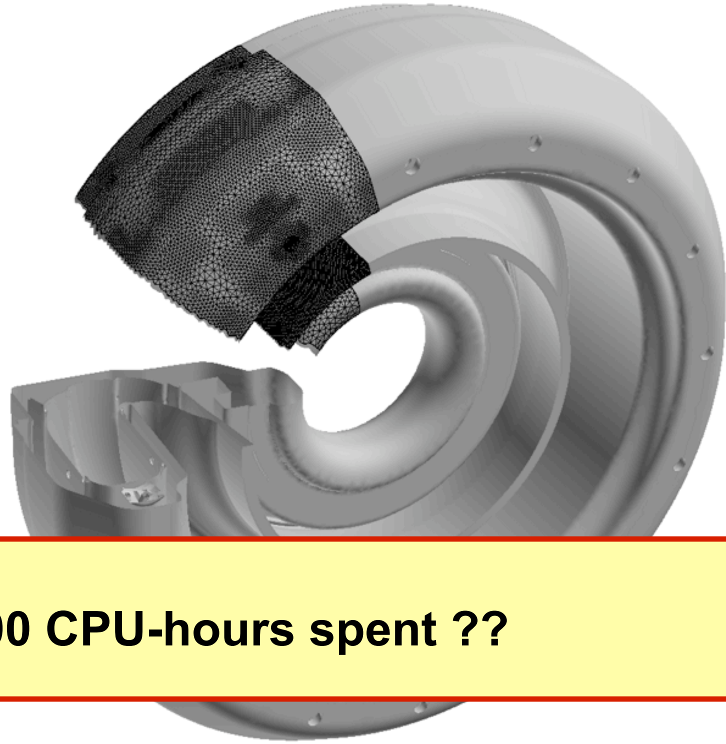
Increasing mesh resolution





Boudier et al., 2008

Application to full annular burner: overview



• **Numerical aspects:**

- 3D compressible LES (AVBP),
- reactive Navier-Stokes solver,
- TTGC convective scheme (3rd order),
- Smagorinsky model[1],
- NSCBC boundary conditions[2],
- Initial conditions from statistically converged mono-sector results.

What do you get out of the 1,000,000 CPU-hours spent ??

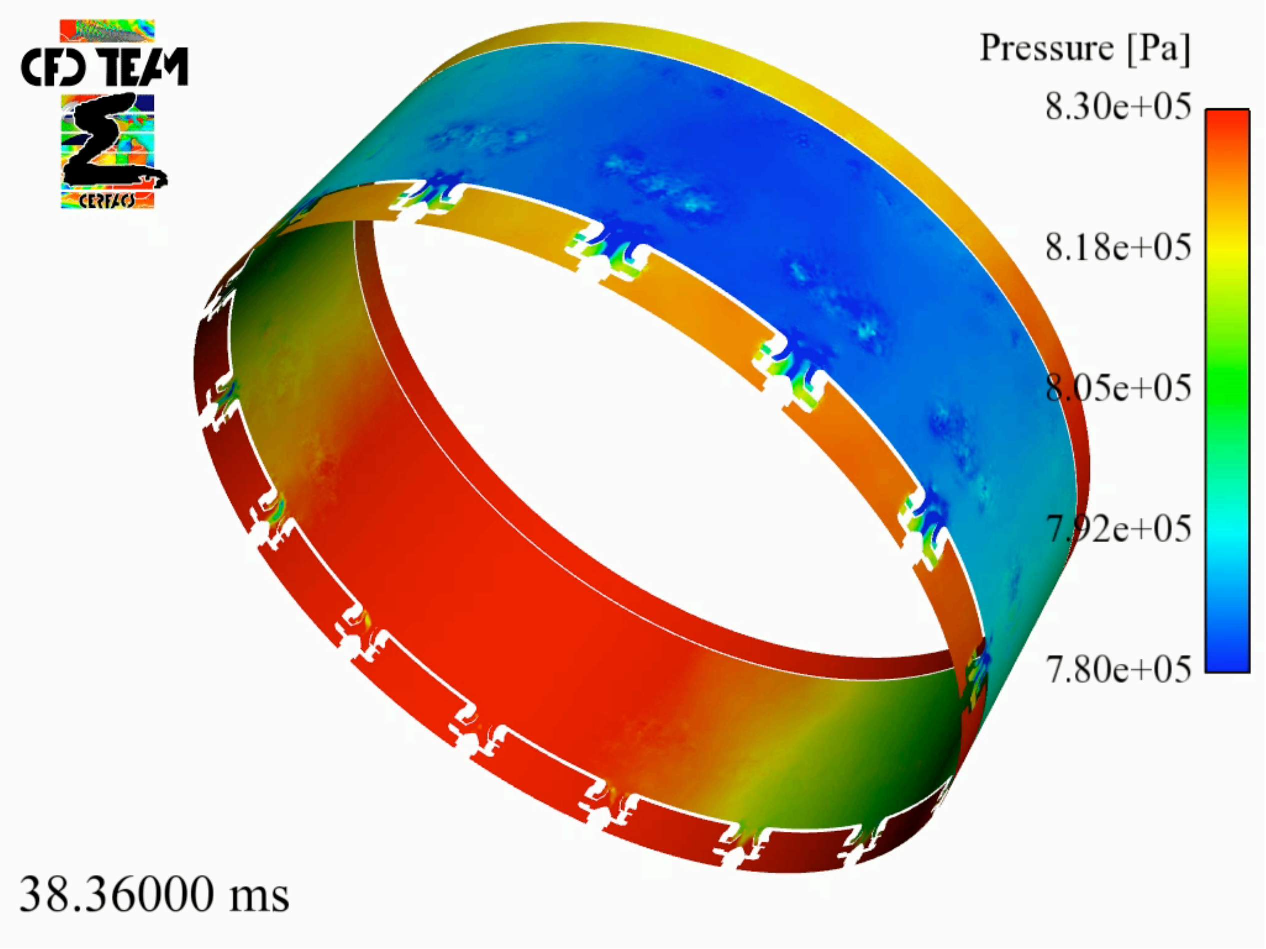
• **Chemical aspects:**

- JP10 1-step fitted mechanism (surrogate for kerosen[3])
- Dynamic Flame Thickening[4].

(1) Smagorinsky et al., 1963
(2) Poinsot et al., 1992
(3) Légier et al., 2001
(4) Colin et al., 2000

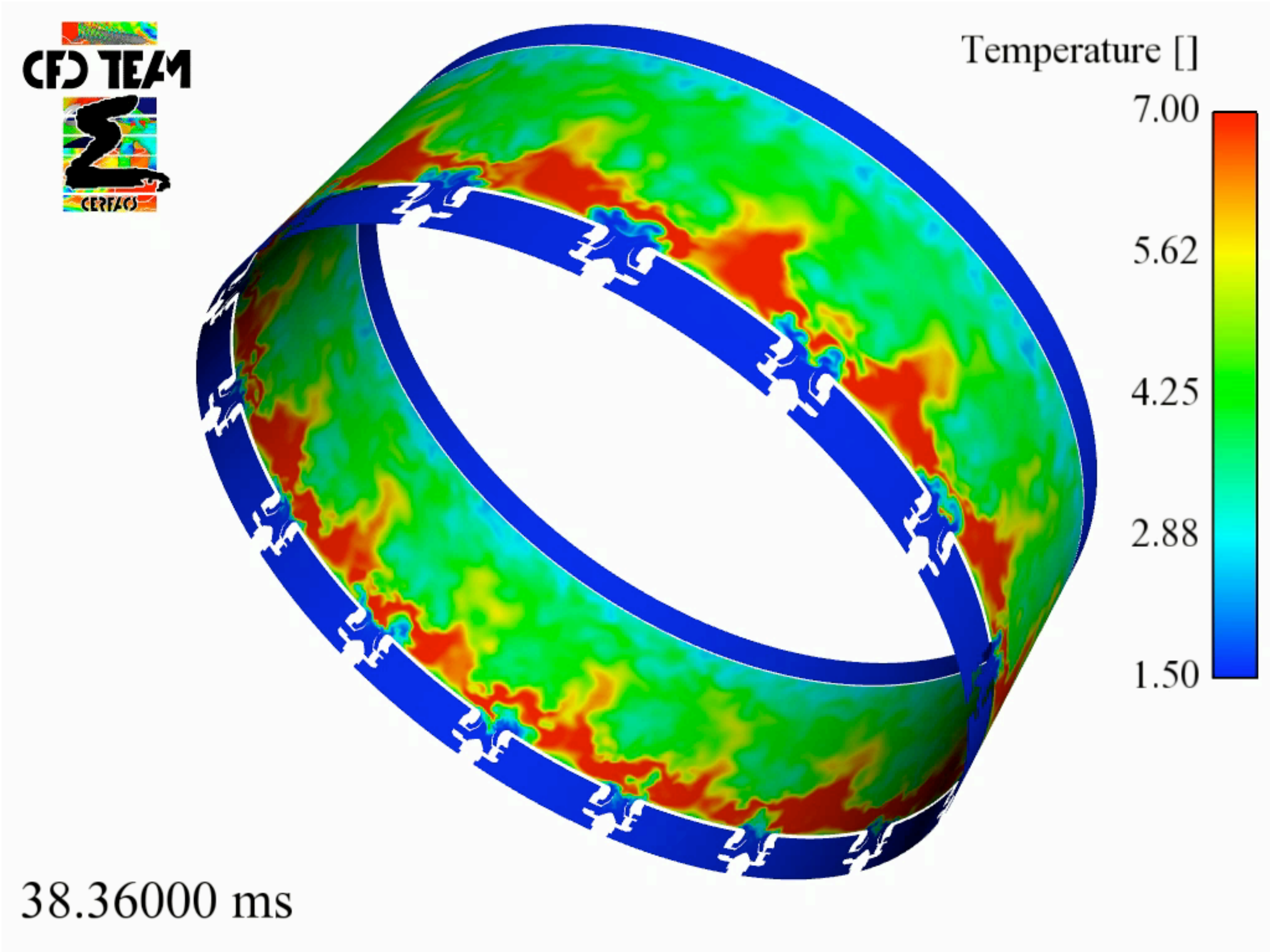
G. Staffelbach et al., 2008
G. Boufier et al., IJ Aeroacoustic, 2007

Application to full annular burner: impact on pressure



- **Temporal evolution of pressure typical of the expression of two counter-rotating pressure waves: self-sustained azimuthal thermo-acoustic instability.**

Application to full annular burner: impact on temperature



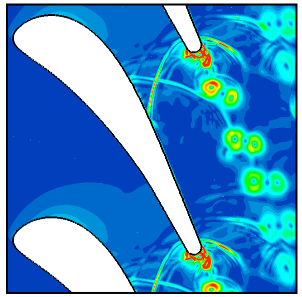
- **Unexpected implication of the instability: azimuthal oscillation of combustion and the temperature field.**

1 Introduction
- Context

2 Numerical developments for HPC
- Flow solver examples
- Speed-up and Mesh-partitioning
- Communication, Impact on numerical solutions

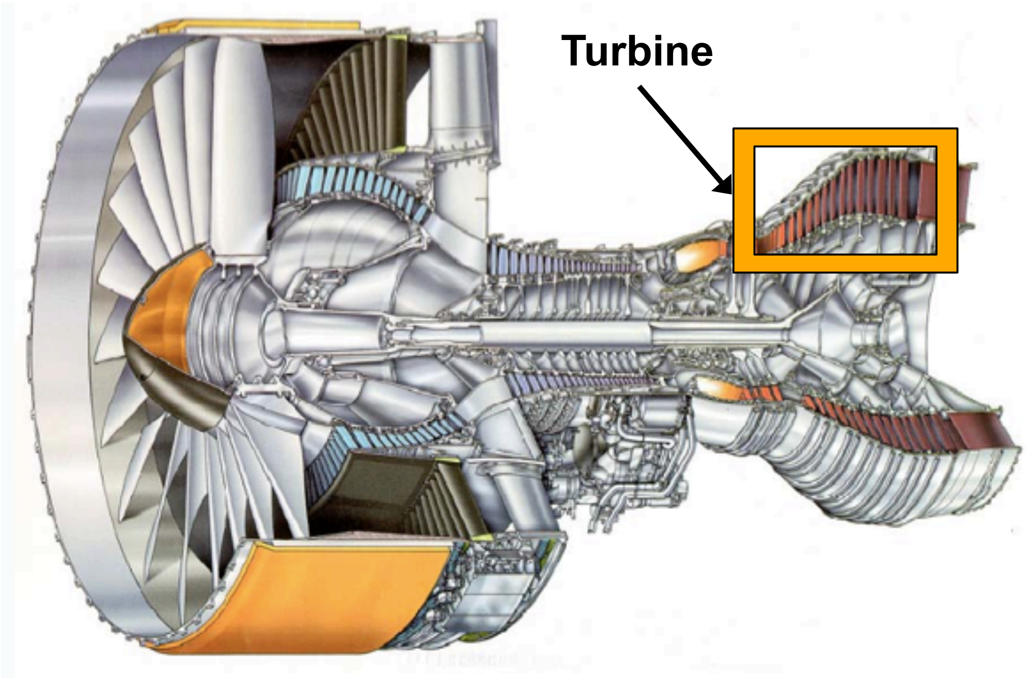
3 Application to aeronautic challenges
- Civil aircrafts
- Compressor
- Combustion chambers
- **Turbines**

4 Conclusion and perspectives

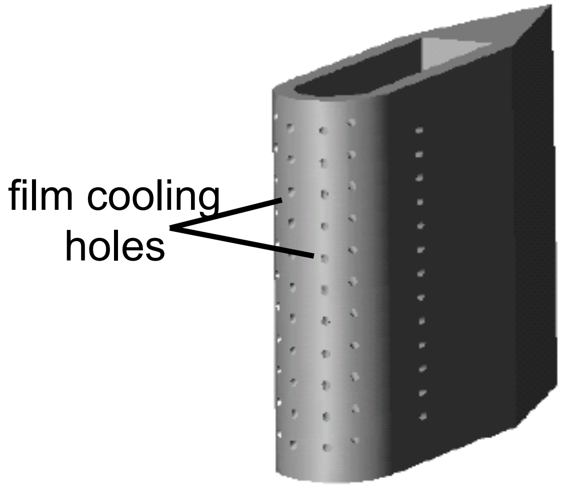


Objectives

- As for compressors, unsteady flows are still not well understood,
- Main reasons are also computational cost, size and complexity of the configurations,
- Challenges today for turbine designers is the prediction of heat transfer:
  - a 15 K difference on the temperature prediction leads to a reduction of its life duration by a factor 2,
  - (U)RANS methods are not adapted to complex flows.

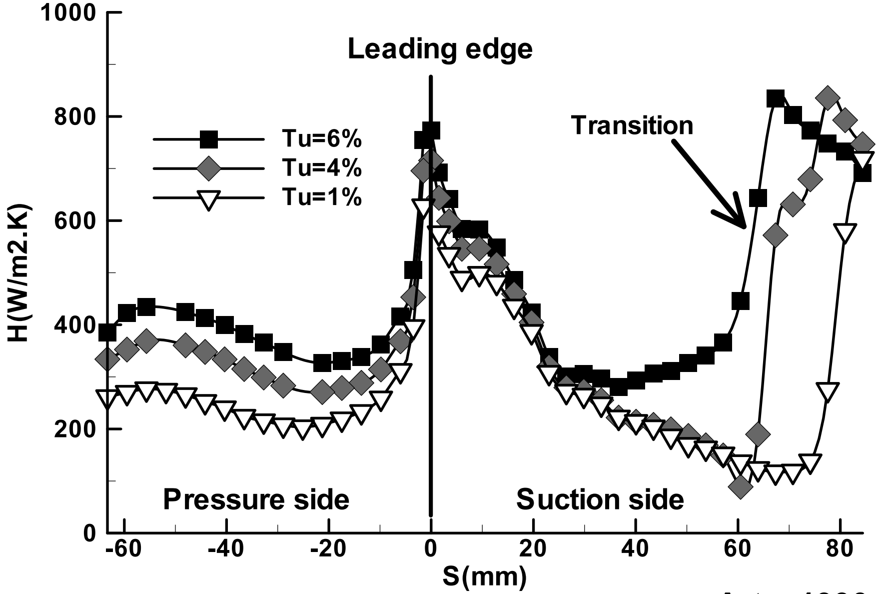


Aerothermal in turbine blades: overview

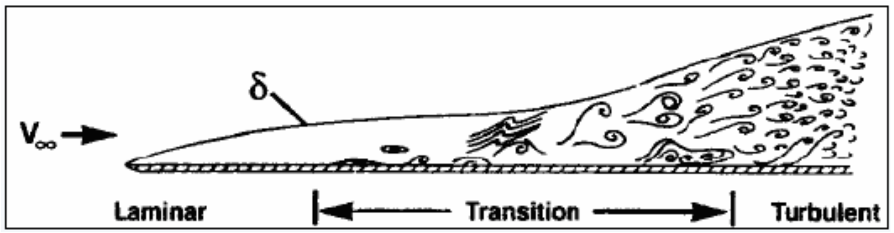


Complex flows that can not be efficiently computed with a (U)RANS method:

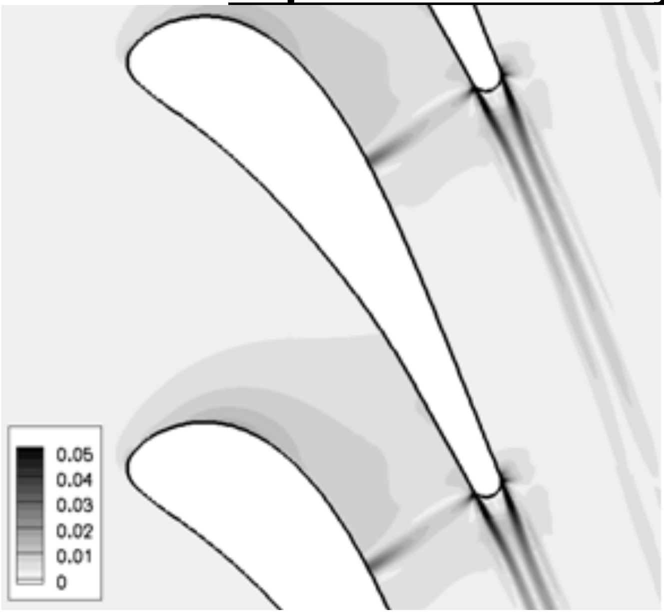
- laminar to turbulent transition,
- hot spot incoming from the combustion chamber,
- aero-thermal interactions (adiabatic is not true).

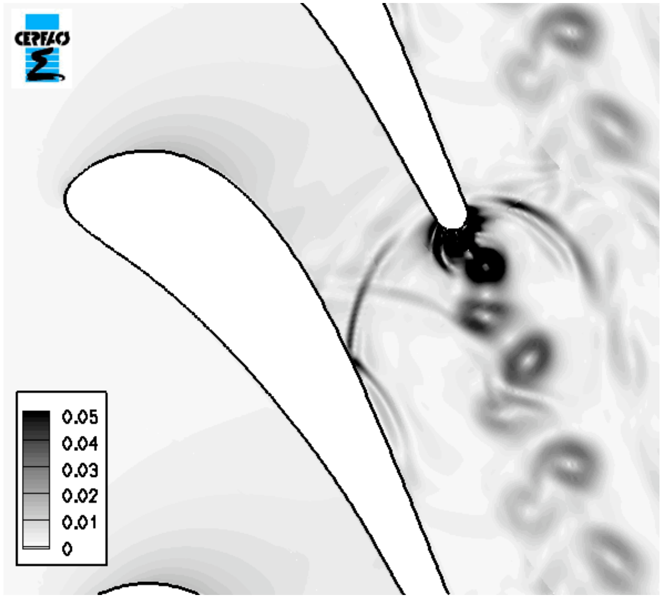


Arts, 1990

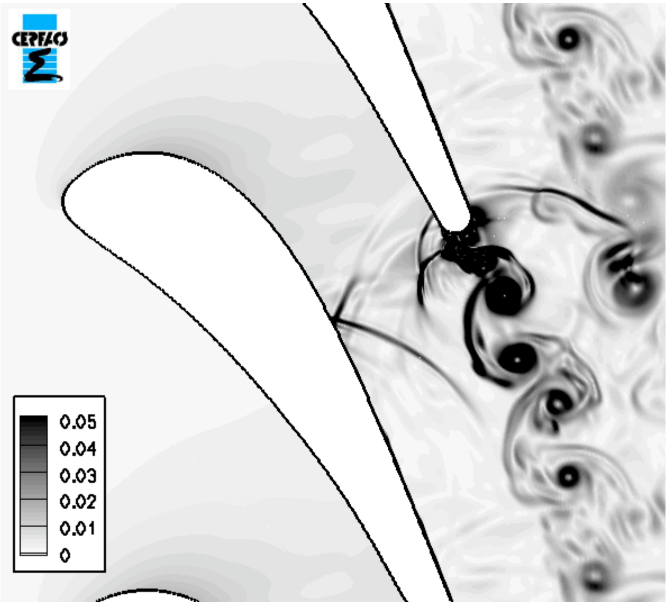


Impact on unsteady aerodynamic performance





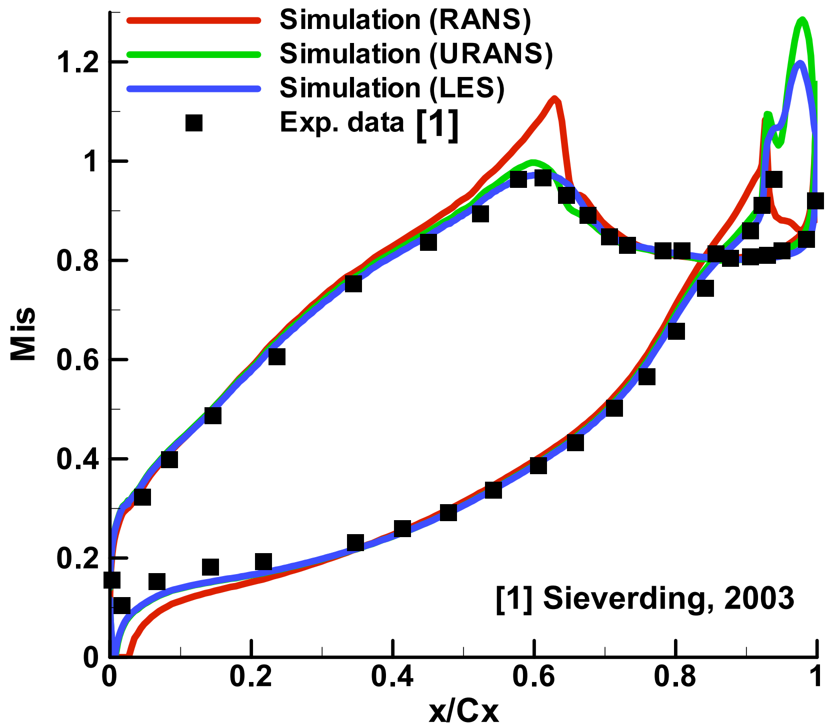
T. Léonard et al., in ASME Turbo-Expo, Glasgow, 2010.



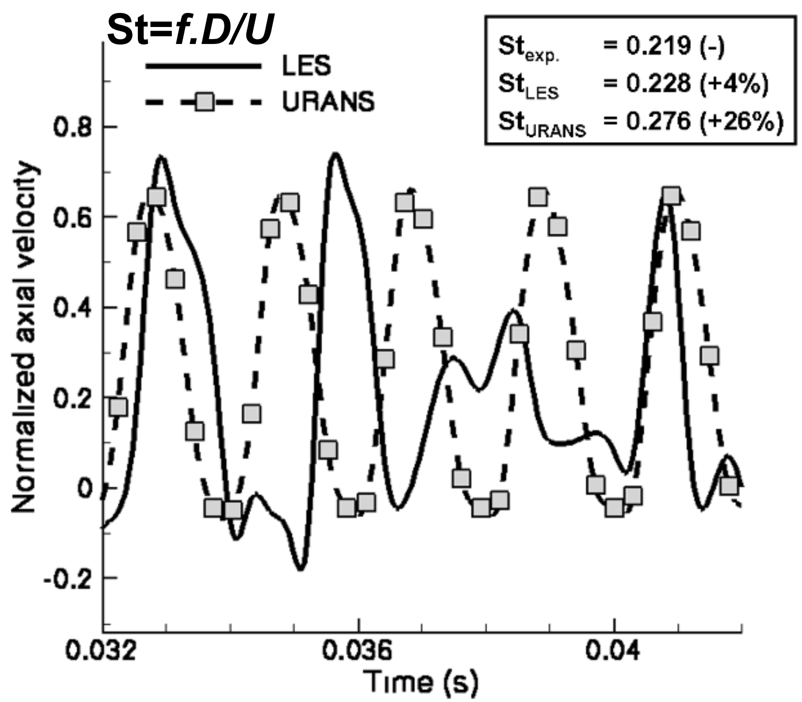
Instantaneous gradp flow field (*elsA*)

- RANS predicts a non-physical shock-wave,
- URANS predicts the vortex shedding but flow features are damped by artificial viscosity,
- LES demonstrates its capacity to transport flow vortices and acoustic waves.

Comparisons with experiments



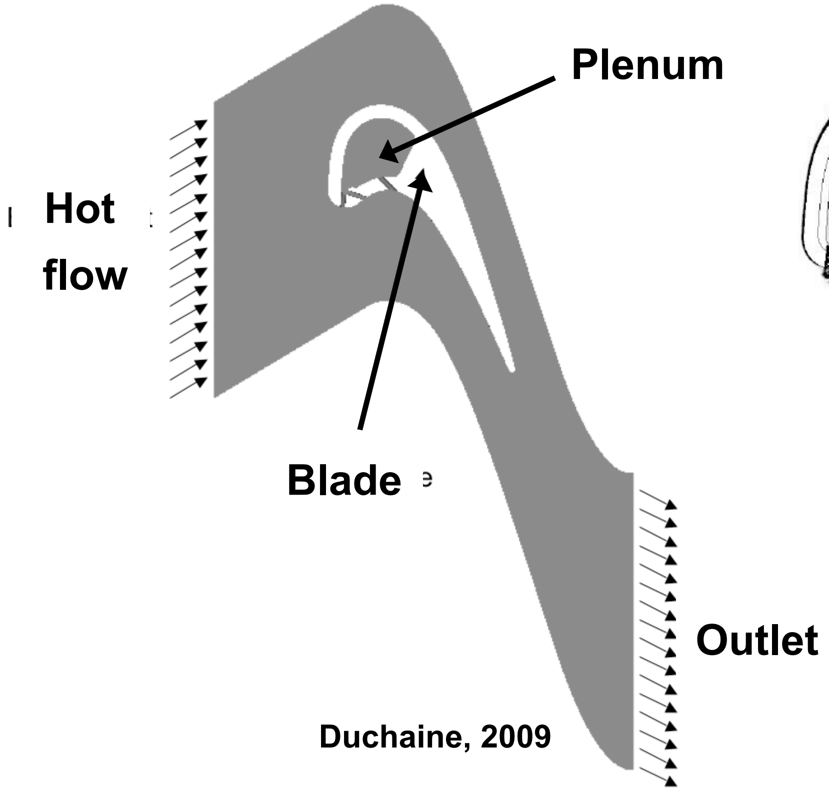
Axial velocity registered behind the trailing edge

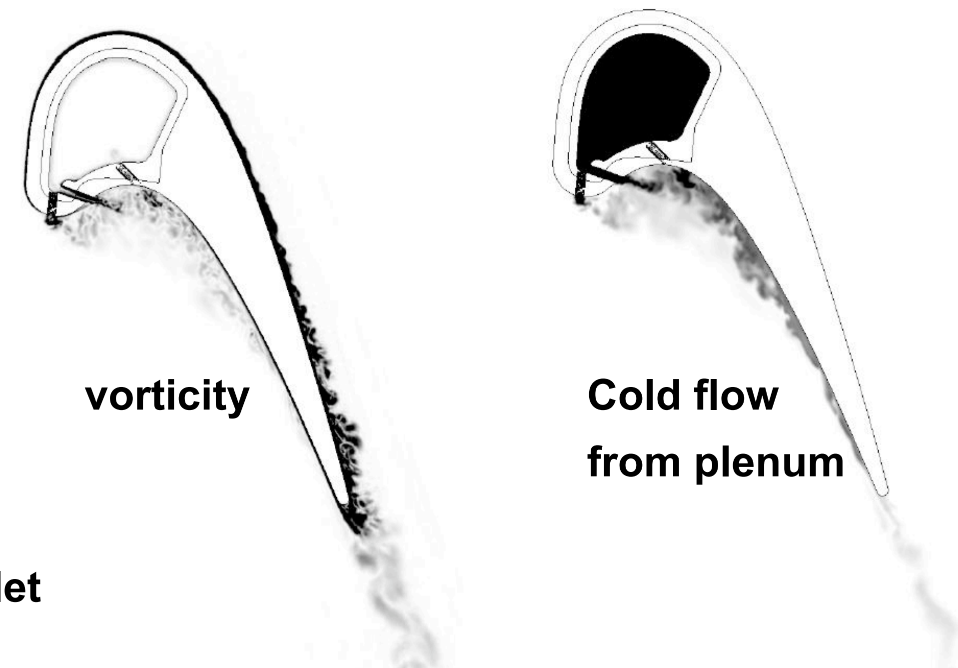


- RANS predicts a non-physical shock on suction-side,
- URANS/LES correctly predict global values,
- LES estimates correctly the experimental Strouhal number.

Aerothermal coupling simulation

- T120D configuration (VKI),
- **AVBP/AVTP coupling simulation**,
- Mesh is 6,5 millions of cells.
- Outlet Mach number: 0.87,
- Re = 4.0 10^5,
- scheme: TTGC (3^{rd} order),
- WALE turbulence model.

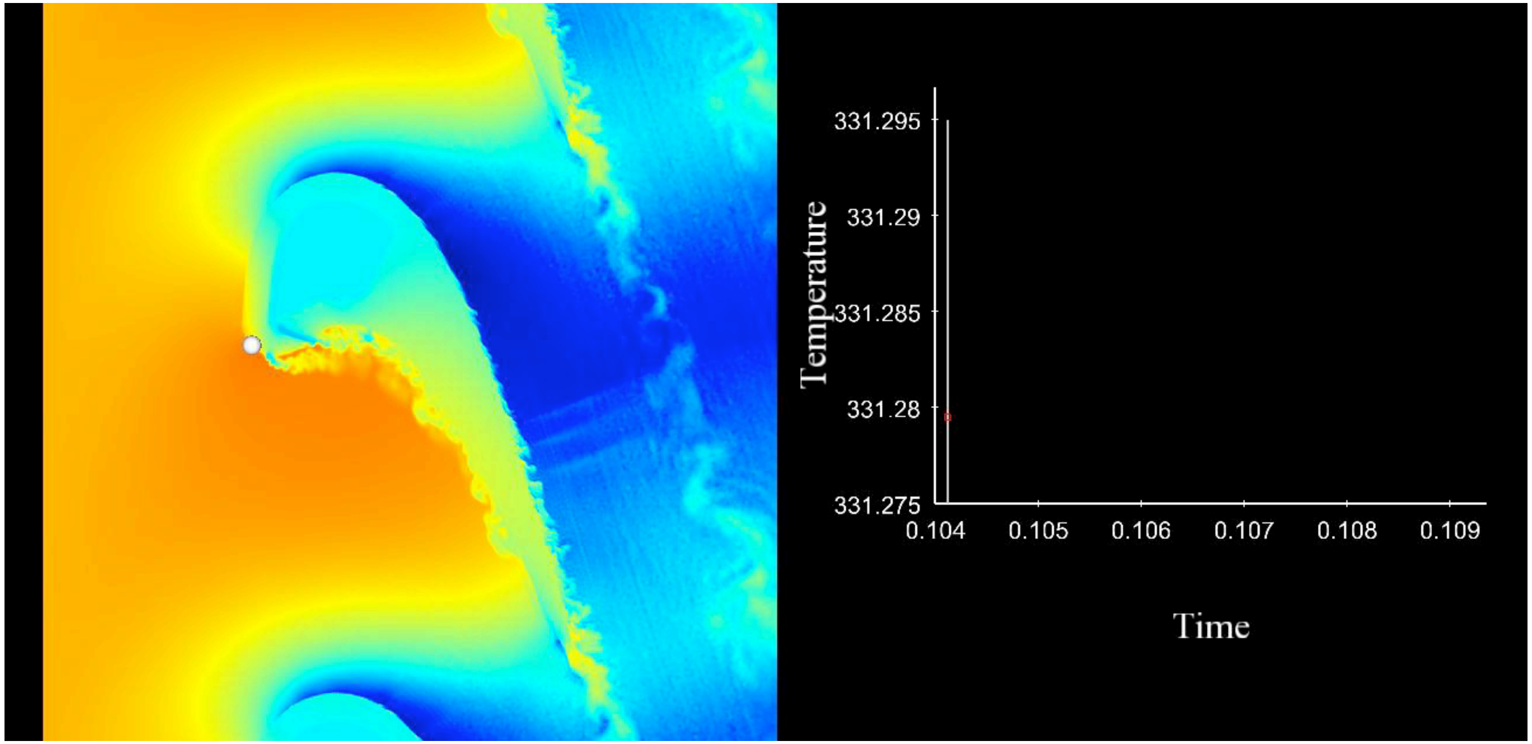




Aerothermal LES including plenum effects

Aerothermal coupling simulation

An unsteady entropic wave interacts with the IGV
***i.e.* a hot flow region impacts the turbine blade.**



Evolution of flow temperature in the turbine passage

1 Introduction
- Context

2 Numerical developments for HPC
- Computing platforms
- Flow solver examples
- From sequential to massively parallel
- Communication
- Impact on numerical solutions

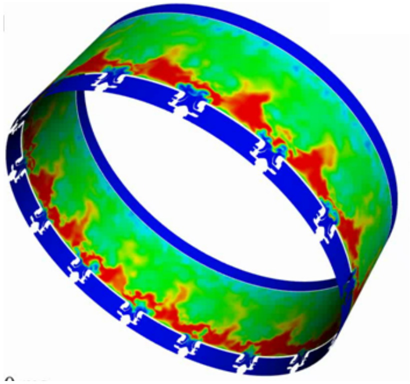
3 Application to aeronautic challenges
- Performance indicators
- Civil aircrafts
- Compressors
- Combustion chambers
- Turbines

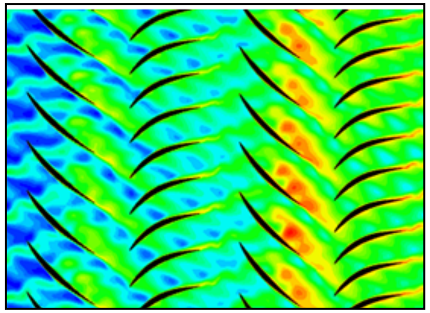
4 Conclusion and perspectives

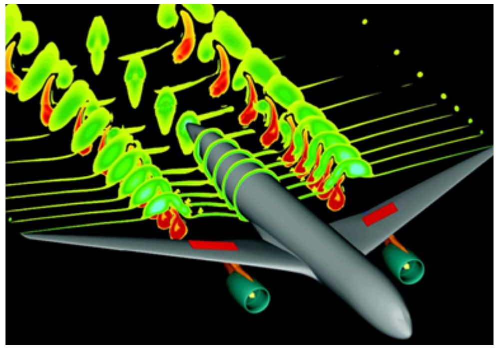
• Examples of applications have been presented for aeronautic and propulsion domains,

• The estimation of the parallel efficiency is complex in industrial context:

- **the most relevant indicator is the time needed to obtain the solution… but this goes though the proper understanding of speed-ups and parallel coding**







• High-fidelity simulations allowed by HPC improve the numerical solution reliability

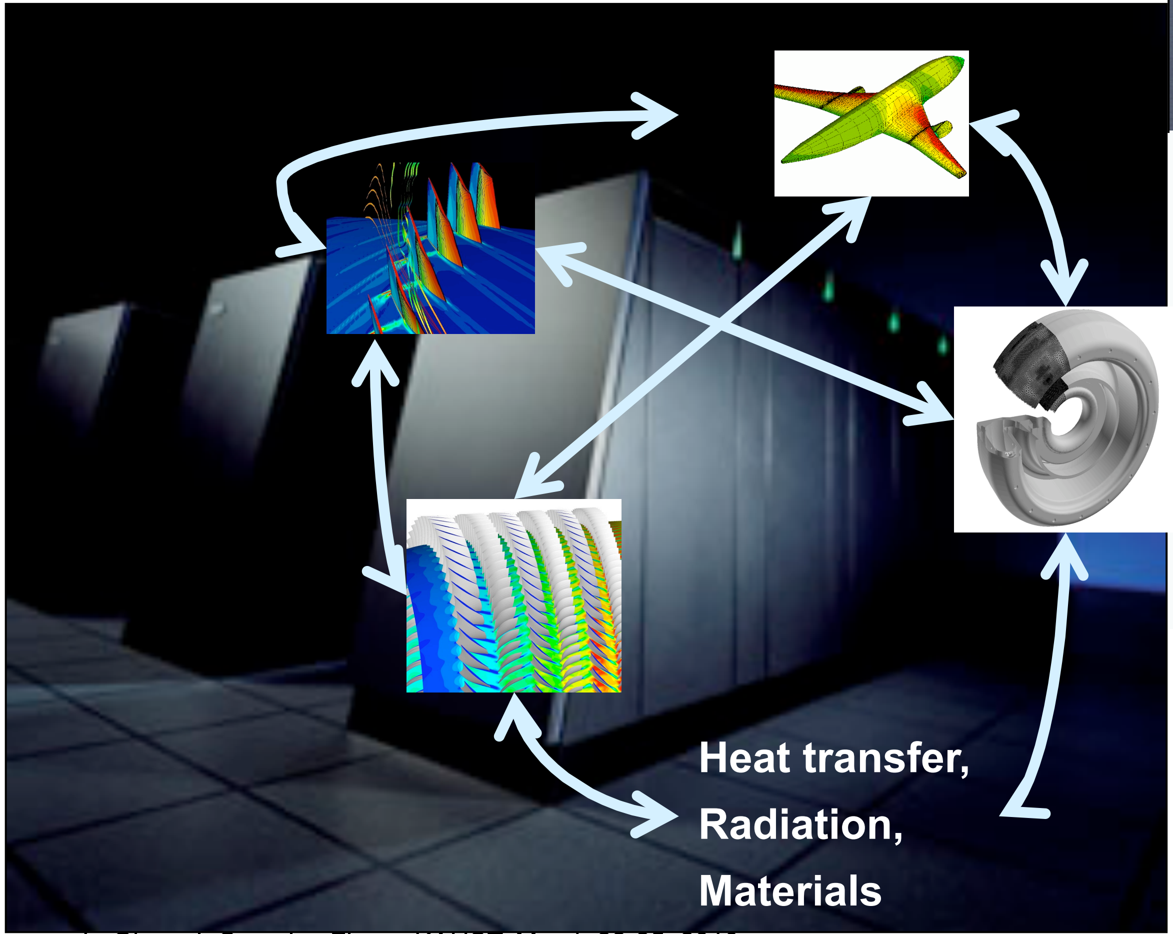
- clear impact on industrial application
- clear impact for fundamental research

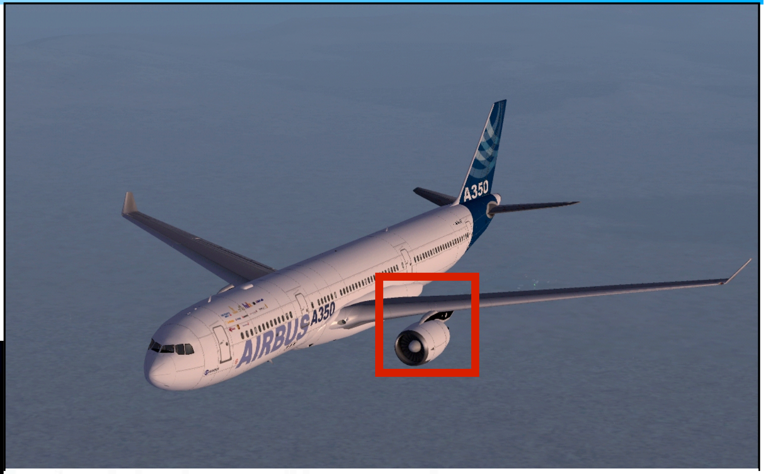
• For aeronautic industry, CFD is a key technology for design, time and cost developments,

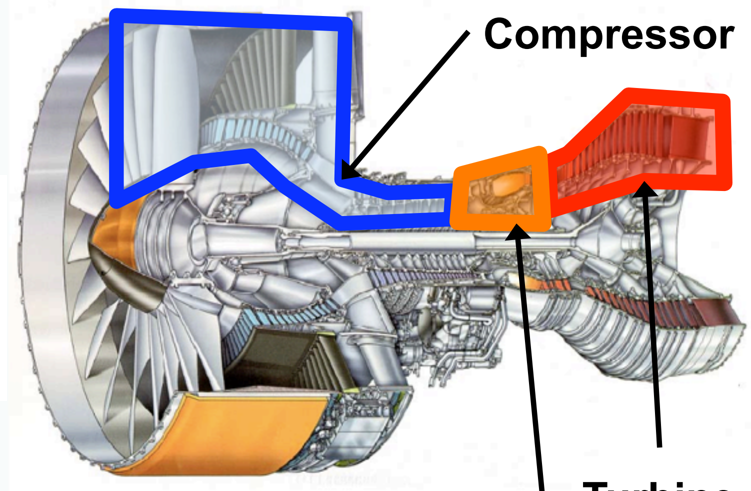
• It is also a very effective tool for investigating complex flow phenomenae,

- **need to go for fully unsteady flow simulations**

The perspectives?



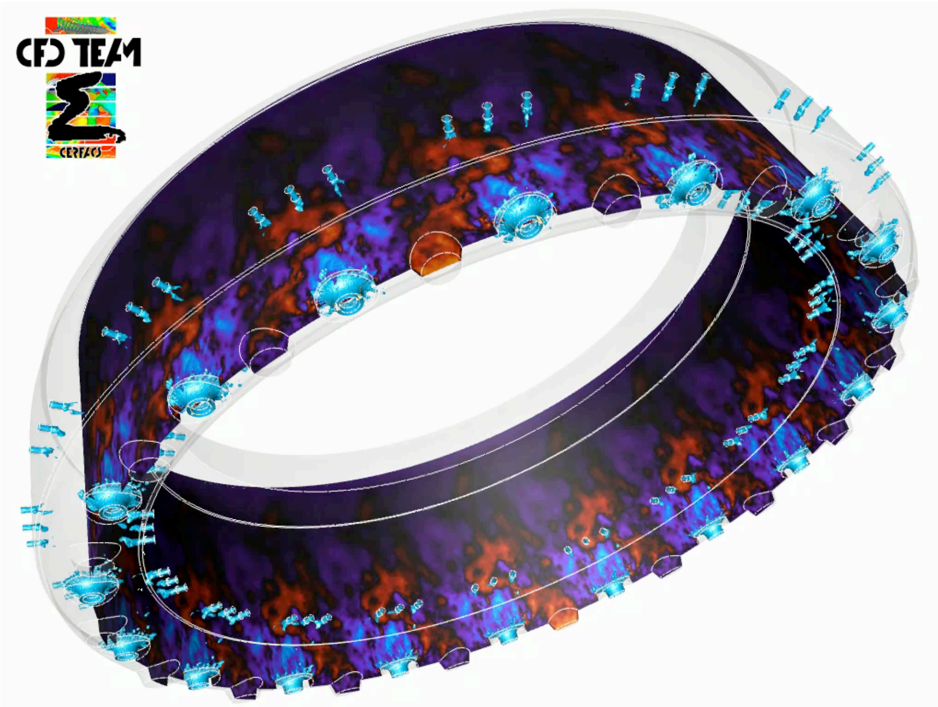




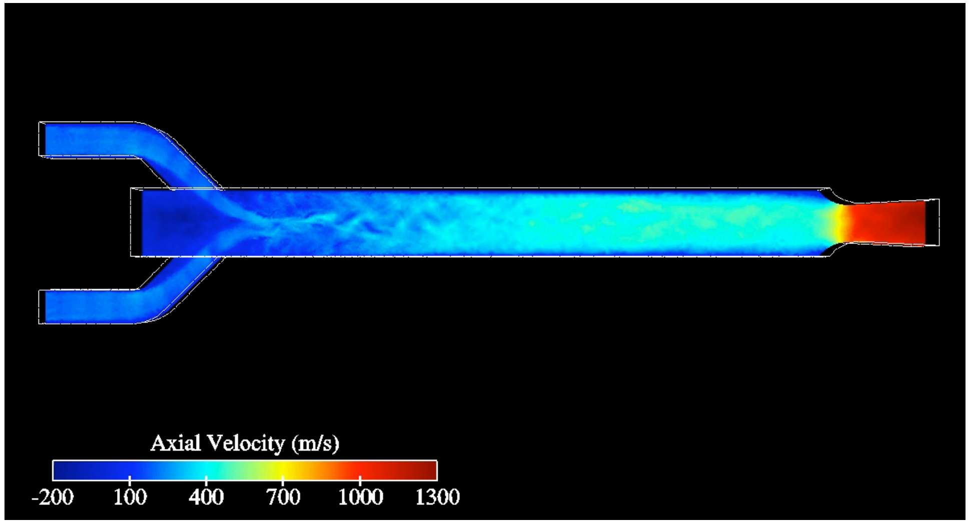
Thanks for your attention

Any questions?

Helicopter engine ignition sequence



Thermo-acoustic oscillation in ramjet burner



Thanks to the CERFACS-CFD team and our partners!

http://www.cerfacs.fr
Laurent.Gicquel@cerfacs.fr