CERFACS State-of-the-art and recent investigations for temperature predictions in Turbo-machineries

L.Y.M. Gicquel\textsuperscript{1}

B. Cuenot\textsuperscript{1}, G. Staffelbach\textsuperscript{1}, O. Vermorel\textsuperscript{1}, E. Riber\textsuperscript{1}, A. Dauptain\textsuperscript{1}

F. Duchaine\textsuperscript{1}, N. Gourdain\textsuperscript{1}, F. Sicot\textsuperscript{1}

T. Poinsot\textsuperscript{2}

\textsuperscript{1} CERFACS - CFD Team, Toulouse

\textsuperscript{2} IMFT, Toulouse

http://www.cerfacs.fr

Laurent.Gicquel@cerfacs.fr
Need to ensure scalability/portability and ‘engineering’ use of the tools:

AVBP – strong scaling  GUI of AVBP

Produce state-of-the-art codes, methods and expertise for HPC while maintaining scientific excellence.

CFD team in 2011:

- 9 PhD dissertations – including 2 priced documents
- 35 Conference presentations – including invited conference talks
- 25 Journal publications

- Many EU research projects
- Many National research projects
- Many Bilateral research contract
Where is CERFACS?

42 avenue G. Coriolis
31057 Toulouse Cedex 1, FRANCE

MUSAF II Colloquium – 18th-20th September 2013 – Toulouse (CIC)

Multiphysics, Unsteady Simulations, Control and Optimization Around aircraft and within engines

<table>
<thead>
<tr>
<th></th>
<th>Day 1</th>
<th>Day 2</th>
<th>Day 3</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Morning</strong></td>
<td>Acoustic &amp; Noise Predictions</td>
<td>Reacting Flows</td>
<td>Optimization</td>
</tr>
<tr>
<td><strong>Afternoon</strong></td>
<td>Heat transfer, structures &amp; loads</td>
<td>Rotating Flows</td>
<td>UQ-Control</td>
</tr>
</tbody>
</table>
I ] State-of-the-art of unsteady simulations in combustors:

=> Massively parallel LES of combustors
=> Trends and potential orientations for LES in industrial burners

II ] State-of-the-art massively parallel CFD for rotating and blade flows:

=> Massively parallel RANS/URANS of compressors
=> LES for turbine flows and aero thermal environment predictions
   - High fidelity flow simulations (modeling issues)
   - Wall heat transfer predictions and LES

III ] Towards multi-physics CFD based on LES:

IV ] Conclusions:
I ] State-of-the-art of unsteady simulations in combustors:

=> Massively parallel LES of combustors
=> Trends and potential orientations for LES in industrial burners

II ] State-of-the-art massively parallel CFD for rotating and blade flows:

=> Massively parallel RANS/URANS of compressors
=> LES for turbine flows and aero thermal environment predictions
   - High fidelity flow simulations (modeling issues)
   - Wall heat transfer predictions and LES

III ] Towards multi-physics CFD based on LES:

IV ] Conclusions:

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 publications demonstrate the superiority of LES [1]: i.e. captures the strong coupling between turbulence/mixing/combustion

**Target configuration:** Single sector gaseous (partially premixed) LES [1]


KEY ISSUE: Scalability of the codes

Scalability/portability of LES codes open new perspectives:

- Full azimuthal chamber LES
  => azimuthal thermo-acoustic instabilities [1]

- Increase model accuracy of single-sector LES
  => grid resolution sensitivity [2]
  => pseudo-detailed chemistry [3]
  => multi-phase flow models
    - Euler / Euler approach
    - Euler / Lagrange approach

- Extended single-sector LES (whenever possible)
  => cover the elements after / before the combustor

Extending in the downstream direction seems feasible and there exists multiple justifications for such computations:

- Better acoustic boundary conditions => thermo-acoustic instabilities
- Potential effects of the NGV on FRT => thermal predictions
- Noise predictions => Direct and Indirect noise issues

However, one enters the realm of wall bounded flows which is a known weakness of LES.

!!! MORE IS NEEDED BEFORE JUMPING THE STEP !!!
I ] State-of-the-art of unsteady simulations in combustors:

=> Massively parallel LES of combustors
=> Trends and potential orientations for LES in industrial burners

II ] State-of-the-art massively parallel CFD for rotating and blade flows:

=> Massively parallel RANS/URANS of compressors
=> LES for turbine flows and aero thermal environment predictions
   - High fidelity flow simulations (modeling issues)
   - Wall heat transfer predictions and LES

III ] Towards multi-physics CFD based on LES:

IV ] Conclusions:
• 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).

• Problems to simulate these devices are the **size**, the **complexity**, the **Re number** $\Rightarrow$ **CPU costs**:
  ➢ most of the industrial simulations focus on limited parts of the system
    (such as isolated blades) that are solved with a steady RANS approach.
**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 [1].

Simulation at design operating point

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

Large multistage effects (blade rows interactions):
- flow in the 3rd rotor is partially driven by wakes of the 2nd stator.

Compressors: existing CFD methods

Comparisons of experimental/research results


1: stator wakes
2: rotor potential effects
3: rotor-stator interaction modes
I ] State-of-the-art of unsteady simulations in combustors:

=> Massively parallel LES of combustors
=> Trends and potential orientations for LES in industrial burners

II ] State-of-the-art massively parallel CFD for rotating and blade flows:

=> Massively parallel RANS/URANS of compressors
=> LES for turbine flows and aero thermal environment predictions
   - High fidelity flow simulations (modeling issues)
   - Wall heat transfer predictions and LES

III ] Towards multi-physics CFD based on LES:

IV ] Conclusions:
• As for compressors, unsteady flows are still not well understood,
• Problems are the size, the complexity, the Re number (although lower than for compressors)
  => CPU costs

• Challenges today for turbine designers is the prediction of heat transfer:
  ➢ a 15 K difference on the temperature prediction leads to a reduction of the engine life duration by a factor 2,
  ➢ (U)RANS methods may not be adapted to complex flows: transition, heat transfer…

Two leverages to release or anticipate better the aero thermal constraints of this device

Flow around the blade

Improved cooling systems
Impact on unsteady aerodynamic performance

- 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.


N. Gourdain et al., in ASME Turbo-Expo, Vancouver, 2011.
Comparisons with experiments

• RANS predicts a non-physical shock on suction-side,
• URANS/LES correctly predict global values,
• LES estimates correctly the experimental Strouhal number.
The heat transfer coefficient is defined as 
($\Delta H$~5%):

$$H = \frac{Q_{wall}}{T_\infty - T_{wall}}$$

The heat transfer is driven by the *freestream turbulent* intensity 
(i.e. the turbulence at the inlet)

---

Basic questions about LES around blades:

- what numerical scheme (explicit vs implicit), what mesh topology / resolution
- what SGS model (wall model or wall resolved)
- what computational domain extent
Turbines: LES vs RANS

Tu0 = 0%

Tu0 = 6%
Turbines: LES vs RANS

Tu0=0%
Structured

Tu0=6%
Transition

Hybrid


One key element for LES to reproduce such behaviors is the introduction of an unsteady turbulent field at the inlet of the vane \[1, 2\]

**Wall-resolved LES** of the flow in the vane *seems possible* and *does improve reliability* of the thermal predictions (aerodynamic response of the flow).

**HOWEVER:**

- Very large grids (structured or unstructured)
- Massively parallel machines and code scalability are pre-requisite

- **Alternatives** => wall models (DES, DDES, wall laws…)
  => need for reliability studies of such solutions

!!! What is really needed in terms of design for these flows !!!
As for compressors, unsteady flows are still not well understood,

Problems are the size, the **complexity**, the Re number (although lower than for compressors)

=> CPU costs

Challenges today for turbine designers is the prediction of heat transfer:

- a **15 K difference** on the temperature prediction leads to a **reduction of the engine life duration by a factor 2**,
- (U)RANS methods may not be adapted to complex flows: transition, heat transfer…

---

Two leverages to release or anticipate better the aero thermal constraints of this device

- Improved cooling systems
- Flow around the blade
Comparison of CFD methods to predict blade internal cooling channel flow:

Flow in U-bend [1]

Flow in ribbed channel [2]


→ Context of this study:

U-bend, Ribbed channel

Thermal study

Rotating study

Industrial configuration with conjugated heat transfer

**Geometrical parameters**

- Hydraulic $D_h = 0.075$ m
- External radius $= 1.26D_h$

**Re = 40000**

**Grid**: full-tetra 6M cells

- Inlet profile obtained from RANS predictions
- Pressure outlet BC
- No-slip adiabatic walls – (wall-resolved LES)
Recirculation bubble

Flow reorientation

PIV

LES

RANS

No inlet turbulence

1%

10%

PIV

(Ux^2+Uy^2)/Ub: 0.2 0.4 0.6 0.8 1 1.2 1.4 1.6 1.8
Interactions between turbulence and the recirculation bubble flow strongly impact the minimum velocity peak value and its positioning within the vein.
• Geometrical parameters
  ‣ Hydraulic diameter = 0.1 m
  ‣ Rib spacing = 10 x $h_{\text{rib}}$
  ‣ Blockage ratio = 30%

• Ref. data:
Ribbed channel

Time = 3.751 s

Complex unsteady separated flow between the ribs
Ribbed channel

PIV

RANS (EARSM)
Struct. mesh / 4.5M cel. / $y^* \sim 1$

LES
full tetra / 7M cel. / $y^* \sim [5, 10]$

LES
tetra+prisms / 1.4M cel. / $y^* \sim [2.5, 5]$

Same difficulties – same conclusions…

Axial velocity mean profiles in the symmetry plane

Whatever grid topology (provided that you can do a wall-resolved LES)
=> you will capture the first moments
Higher order moments (turbulence)

\[ TI = \sqrt[3]{\frac{1}{3} (u_{x,\text{rms}}^2 + u_{y,\text{rms}}^2 + u_{z,\text{rms}}^2) / U_b} \]

High Turbulence Intensity zone after the rib
(maximum and 50% of maximum lines)

Value of Turbulence Intensity on the maximum line


LES predicts correctly the unsteady features (intensity and extend)
(lack of 10/20% being in modeled kinetic energy [1])
• The main outcome is a better estimation of the wall shear stress map:

- PIV: 3.6 x/h (error 6%)
- Hybrid LES: 3.8 x/h (error 6%)
- RANS: 5.1 x/h (error 42%)

• One has to keep in mind that LES cost is still much higher than RANS…
As well as the wall heat flux: here expressed in terms of an Enhancement Factor [1]

Outline of the talk

I] State-of-the-art of unsteady simulations in combustors:

=> Massively parallel LES of combustors
=> Trends and potential orientations for LES in industrial burners

II] State-of-the-art massively parallel CFD for rotating and blade flows:

=> Massively parallel RANS/URANS of compressors
=> LES for turbine flows and aero thermal environment predictions
    - High fidelity flow simulations (modeling issues)
    - Wall heat transfer predictions and LES

III] Towards multi-physics CFD based on LES:

IV] Conclusions:
LES in the vane & the cooling channels: conjugate heat transfer problem…

Multiple difficulties appear covering physical and HPC issues:

1/ All-in-one or *multiple dedicated* solvers

2/ How to *couple efficiently two partitioned non coincident domains*?
   - data distribution versus centralization
   - interpolation, conservation

3/ What *quantities / fields* to exchange and at *what rate*?

4/ How to converge two fields dictated by very *different time scales*?

5/ Whatever the method retained is the *aggregated numerical solver stable*?
Typical investigation for couple LES / conduction: courtesy of F. Duchaine

- Size of the span = 10 mm
- 5 prim layers in the fluid boundary layer
- Convergence of fluid to track heat transfer coefficient with iso-thermal wall
- Influence of turbulence injection (not done)
- Coupled aero thermal simulation

Conjugate heat transfer based on LES

Instantaneous flow field: Iso Q-criterion colored by velocity

Transitioning boundary layer

Recirculation bubble

Instantaneous view on heat transfer: Q-criterion and wall heat flux

Pressure side

Suction side

Conjugate heat transfer problem based on LES:

=> the tools exist although the most efficient way to use them is not yet clear

=> the potential is clearly present

- LES capacity to represent fine scale / transitions / bifurcations of the flow to upstream and near wall event
- Brute force and HPC seem a pre-requisite at least for the scientific context (produce understanding and modeling databases)

Alternatives for industrial use of such solutions:

=> wall models (DES, DDES, wall laws…) versus brute force wall resolved LES

!!! You also need to consider other aspects !!!
• LES in rotating channels is needed:
  - Wall normal rotation
  - LES simulations of stabilizing and destabilizing effect of Coriolis and centrifugal forces.

• Experimental data from VKI [1]:
  - full rotating test bench
  - time resolved PIV

[1] Coletti et al., Flow field investigation in rotating rib-roughened channel by means of particle image velocimetry, Exp. in Fluids (2011)

Courtesy of R. Fransen: CERFACS PhD student

• LES in rotating vanes is also needed:
  
  ‣ Strategies need to be evaluated
    – Single passage
    – Multiple passages
    – Interface treatment
  
  ‣ Gain in flow physics needs to be confirmed
  
  ‣ CPU cost of such tools ???

• Experimental data??

CREATE Compressor

Outline of the talk

I ] State-of-the-art of unsteady simulations in combustors:

=> Massively parallel LES of combustors
=> Trends and potential orientations for LES in industrial burners

II ] State-of-the-art massively parallel CFD for rotating and blade flows:

=> Massively parallel RANS/URANS of compressors
=> LES for turbine flows and aero thermal environment predictions
   - High fidelity flow simulations (modeling issues)
   - Wall heat transfer predictions and LES

III ] Towards multi-physics CFD based on LES:

IV ] Conclusions:
**Conclusions**

**LES for combustion chamber is quasi-acknowledged as being a mandatory tool:**

- Can serve as multiple purpose tool: design, advance diagnostics…
- How to use it as an efficient complement to RANS

**LES around blades:**

- There is a need for such a tool: especially if aero thermal quantities are to be accurately estimated
- Wall-resolved LES will be expensive (other alternatives?)

**LES in the cooling channels of the NGV or blades:**

- Clearly accessible today provided that you have CPU time and a massively parallel code

**Future:** fully coupled LES and conduction solver…
Gas turbine flows have a very high Reynolds number:

\[ \text{Re} = \frac{\rho U L}{\mu} \implies N \propto (0.1 \text{Re})^{9/4} \]

- Compressor at operating conditions:
  \( \text{Re} \sim 5 \times 10^6 \implies N \sim 37 \times 10^{11.25} \)

- Combustor at operating conditions:
  \( \text{Re} \sim 5 \times 10^5 \implies N \sim 37 \times 10^9 \)

- Turbine at operating conditions:
  \( \text{Re} \sim 1 \times 10^6 \implies N \sim 1 \times 10^{11.25} \)

PROPER HPC DESIGN OF CODES AND MACHINES WILL MAKE THE DIFFERENCE IF LES IS TO BE USED BY INDUSTRY

What’s CERFACS?

CERFACS has seven shareholders

- CNES
- EADS
- EDF
- ONERA
- METEO FRANCE
- TOTAL
- SAFRAN

One hundred permanent people in 5 teams

- Parallel algorithms
- Climate environment
- Electromagnetism
- Data assimilation
- CFD

- Expertise in scientific computation
- Access to large computational resources

http://www.cerfacs.fr

RANS versus LES: Impact on a design criterion (i.e. RDTF) [1-4]

\[
RTDF(r) = \frac{\langle T(r,\theta) \rangle_\theta - \langle T(r,\theta) \rangle_{\theta r}}{\langle T(r,\theta) \rangle_\theta - T_{inlet}}
\]

RTDF(r) profile measures the radial temperature heterogeneities through the exit plane of the chamber ⇒ controls the turbine lifetime!!
