Comparison of steady and unsteady RANS CFD simulation of a supersonic ORC turbine

Benoit OBERT a*, Paola CINELLA b

a Enertime, b Arts et Métiers ParisTech

ORC 2017, 4th International Seminar on ORC Power Systems
Context

- ORC systems need high efficiency and cost-effective expanders

- **Multi-stage axial turbines** are the most commonly used solution for applications above 1 MW

- For high temperature applications ($T_{HS} > 250/300^\circ C$), highly loaded **transonic to supersonic stages** are used to keep their number low

- Accurate design and performance estimation through CFD must be used to ensure high turbine efficiency
Objective

• Design of a highly loaded turbine stage

• Simulations using unsteady Reynolds Averaged Navier-Stokes (RANS) calculations to capture transient nature of the flow inside of an axial turbine stage

• Analyse flow structure including shock interactions and blade loading

• Performance assessment (entropy creation and isentropic efficiency)

• Comparison with steady state mixing plane RANS simulations
Turbine Characteristics

- Working fluid: siloxane MM (hexamethyldisiloxane)
- **3-stage 2.5 MW axial turbine** running at 3000 rpm
- 85 overall pressure ratio
- Inlet total temperature: 233°C
- Inlet total pressure: 14.5 bar

In this work we focus on the **first stage of the turbine**

![T-s Diagram](image-url)
First Stage Characteristics

- Impulse stage (low reaction degree)
- Converging diverging nozzle
- Number of blades determined using Zweifel optimal loading coefficient

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value</th>
<th>Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure ratio</td>
<td>7.5</td>
<td>Blade height</td>
<td>20 mm</td>
</tr>
<tr>
<td>Specific speed</td>
<td>0.2</td>
<td>Rotor inlet Mach number</td>
<td>0.8</td>
</tr>
<tr>
<td>Specific Diameter</td>
<td>6.2</td>
<td>Rotor outlet Mach number</td>
<td>1.2</td>
</tr>
<tr>
<td>Nozzle outlet Mach number</td>
<td><strong>1.84</strong></td>
<td>Rotor inlet blade angle</td>
<td>62°</td>
</tr>
<tr>
<td>Nozzle outlet blade angle</td>
<td>76°</td>
<td>Rotor outlet blade angle</td>
<td>64°</td>
</tr>
<tr>
<td>Nozzle blade number</td>
<td>47</td>
<td>Rotor blade number</td>
<td><strong>142</strong></td>
</tr>
</tbody>
</table>

Changed to **141** to reduce computational domain.
Dense gas behavior

- MM properties from multi-parameter Equation Of State (EOS) based on Helmoltz free energy [Colonna et al, 2006]

- Fundamental derivative of gas dynamics [Thompson, 1971]:

\[ \Gamma = 1 + \frac{\rho}{a} \left( \frac{\partial a}{\partial \rho} \right)_s \]

- \( \Gamma \in [0.25, 1.0] \) along first stage expansion
  - Classical behavior when \( \Gamma > 1.0 \)
  - Non classical behavior \( \Gamma < 1.0 \)

- Dense gas effects expected

\( \Gamma \) evolution along expansion
Blade design

- **Nozzle divergent** part designed using Method Of Characteristics (MOC) extended to real gases

- **Nozzle convergent** part designed using simple geometrical shapes

- **Rotor blades** designed using
  - Circular arc for pressure side
  - Circular arc and splines for suction side
  - Ellipses for leading and trailing edges
CFD simulation setup

- Commercial software: **ANSYS CFX 17.2**
- **Unsteady RANS 2-D, k-ω SST** for turbulence closure
- Real gas properties: look up tables generated from **NIST REFPROP**
- Numerical schemes
  - Advection scheme: implicit 2\textsuperscript{nd} order bounded scheme
  - Turbulence scheme: implicit 2\textsuperscript{nd} order bounded scheme
  - Transient scheme: implicit second order Euler (60 steps per period)

- Boundary conditions:
  - Total inlet pressure and temperature
  - Static outlet pressure
  - No slip blade wall

- Mesh:
  - 350,000 elements
  - Structured grid
  - $y+\sim1$ at walls
  - Grid independence study

Simulation mesh close up
Flow structure

I: Series of weak oblique shocks

II: Fish tail shock

III: Reflexion

IV: Fish tail shock Reflexion

V: Bow shock

Pressure gradient
Nozzle blade loading

- The flow acceleration in the nozzle is essentially stationary
- Fish tail shock impingement thickens boundary layer at the suction side
- Small fluctuation near the trailing where bow shock impinges
- Second boundary layer thickening at this impingement

Stator blade loading

Mach number in stationary frame
**Rotor blade loading**

- Front part of the rotor blade sees important blade loading fluctuations due to bow shock interacting with shocks and wake coming from the nozzle row.
- Rear part has a more steady behavior.

---

**Blade loading comparison**

- **PS Time Averaged**
- **SS Time Averaged**
- **PS Unsteady**
- **SS Unsteady**

**Mach number in stationary frame**
• The torque on one blade varies by more than 40% and the average torque on the three rotor blades of the domain varies by about 20%
Losses

- Entropy creation dominated by nozzle turbulent wake advected through the rotor blade row
- Rotor turbulent wake
- Small contribution of shocks to entropy creation

\[ \eta_{tt} = \frac{H_{in} - H_{out}}{H_{in} - H_{out,isentropic}} \]

Total to total isentropic efficiency

Isentropic efficiency time evolution

Entropy field
Comparison with steady results

Setup:
- Stator/rotor interface: mixing plane
- Same boundary conditions
- Same advection and turbulence schemes

Results:

Mach number fields
Comparison with steady results

- Nozzle flows are very similar
- Small differences in front part of the rotor blade where stator-rotor interaction is important

<table>
<thead>
<tr>
<th>Quantities</th>
<th>Steady</th>
<th>Unsteady</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stator total pressure loss coefficient</td>
<td>0.1000</td>
<td>0.1022</td>
</tr>
<tr>
<td>Rotor blade torque (N.m/m)</td>
<td>2.6299</td>
<td>2.6255</td>
</tr>
<tr>
<td>Total to total isentropic efficiency</td>
<td>0.9193</td>
<td>0.9179</td>
</tr>
</tbody>
</table>

![Blade loading comparison](image-url)

![Rotor blade loading](image-url)
Conclusion and Perspectives

Conclusions
• Expected flow structure
• High variation of rotor load but lower than in similar work [Rinaldi, 2015]
  → Larger gap (0.5 chord vs 0.25 chord)
  → Lower Mach number (1.8 vs 2.8)
• Good prediction with mixing plane steady simulations

Perspectives
• Reduced stator-rotor gap would increase stator-rotor interaction effects
• 3D unsteady simulations:
  → Low h/D ratio for the first stage
  → Important secondary flow contribution expected
• Comparison with time/harmonic transformation methods available in ANSYS CFX
• Simulation of transonic and higher reaction degree stages
Thank you for your attention