CFD Assessment of Original Design of a Single-stage Highly-loaded Axial Turbine

Dr. Justin Jongsik Oh
Motivation & Objectives

- Around 1975 to 1985 there was an interesting national project to develop a 75kW gas turbine to demonstrate the replacement of the internal combustion engines of mid-size passenger cars, led by the Chrysler in the U.S., which final goals were to meet economic and emission requirements set in the year frame.

- Due to design constraints of compactness, less weight, low cost and fast engine response, still enforced to any current business developments, a single-stage axial turbine was designed to meet challenging performance targets, leading to a highly-loaded aerodynamic design (NASA TM X-71717). The followings would be key factors to limit design space.
  - Single stage small axial turbine with a tip radius of 2.2 inch
  - Loading factor of 2.1 with a target isentropic efficiency (T-T) of 86%
  - Large rotor tip clearance of 2.3% span
  - Thicker and fewer blades

- Based on original design geometry provided in the public report, a steady-state CFD analysis has been applied at design point to see how aerodynamic performance predictions would have been if they had been able to use the current level of analysis tools prior to rig tests.

- The efforts would be worthy of learning something about design philosophy as lessons learnt.

- The original design failed to achieve performance targets in the rig test performed later.

- Author’s code of CNSTURBO(*) was used in the study.

(*) Please see previous papers available under the menu of Studies.
Turbine has a single suction flange in the engine layout, and takes full advantage of inlet swirl to reduce stator turning loads by selecting the scroll intake like turbocharger turbines.

Design Specifications

<table>
<thead>
<tr>
<th>Specification</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet total temperature</td>
<td>1325 K</td>
</tr>
<tr>
<td>Inlet total pressure</td>
<td>397500 Pa</td>
</tr>
<tr>
<td>Mass flow</td>
<td>0.598 kg/s</td>
</tr>
<tr>
<td>Speed</td>
<td>58500 rpm</td>
</tr>
<tr>
<td>Power</td>
<td>118.2 kW</td>
</tr>
<tr>
<td>Specific work</td>
<td>198119 J/kg</td>
</tr>
</tbody>
</table>
Inlet Swirl Boundary Conditions

- Because of a single-pitch row CFD setup without the scroll intake, a uniform (or zero swirl) flow direction cannot be applied along the inlet span.
- The dotted curve profile was built close to original design intent as an inlet boundary condition.
- However, in reality the inlet swirls were so non-uniform around the circle, measured in the rig test, which also implies a good match with inlet components must be critical for a turbine core.
Flow Angle at S1-TE and R1-LE

- When S1 inlet swirl is well matched with design intents (shown as a dotted curve), the very first concern would be how well S1 generates an exit swirl profile for R1 along the span, i.e., at least 3 sections of hub, midspan and shroud.

- CFD shows an exit swirl about 5 degree shorter than design goals near the shroud, which will change R1 work distributions.

- A wiggle of flow angles near the hub means a local flow separation along the suction surface was built in S1, caused by an excessive turning loading, as confirmed in Mach contours on 10% span.
Meridional Velocity at S1-TE and R1-LE

- A full profile of meridional velocity at S1 TE, especially near the shroud, supports that the secondary flow migration was well suppressed inside S1 which would be otherwise very easily created throughout flow turning.

- The meridional velocity near the shroud looks so strong that S1 has lost some of its duty of flow turning there.

- Again, the local flow separation near the hub would be one that could have been fixed before a final design.
Blade Loadings of S1

- Original design intent shows a front loading along the span, which should be to avoid unnecessary flow diffusion in the rear side of S1 blade.

- CFD predicts however that max. loadings were built near 60% meridional distance, which misses design goals. Accordingly S1 rear parts get over-loaded resulting in the flow separation near the hub.
Blade Loadings of R1

- CFD warns of some design issues,
  - Incidence across the span
  - Pressure surface curvature near LE to suppress boundary layer growths
  - Smaller hub loading
  - Flow separation near the shroud TE, as also confirmed in Mach contours on 90% span
Summary

- CNSTURBO CFD was applied to the original design geometry of a single-stage small axial turbine at design point which was developed for an automobile power train gas turbine back in 1980’s.

- Through a 3D loading check in each blade row what would be primary reasons why the design had failed to achieve target performance in the rig tests was investigated. Some key findings are,

  - S1 runs short of flow turning near the shroud.
  - S1 gets over-loaded near the hub resulting in a local flow separation.
  - S1 misses the front loading design target.
  - R1 misses the optimum incidence distribution target along the span.
  - R1 needs to revisit the control of pressure surface wall curvatures.
  - R1 misses the blade loading target near the hub.
  - R1 gets over-loaded toward the shroud, resulting in a local flow separation.

- The present case study will provide useful insights in aerodynamic design tasks of a small highly-loaded axial turbine.

- The effects of circumferentially and radially non-uniform flow fields at S1 inlet, coming from upstream intake components, should be considered in separate for a successful aerodynamic design.