THE LEMCOTEC 1½ STAGE FILM-COOLED HP TURBINE: DESIGN, INTEGRATION AND TESTING IN THE OXFORD TURBINE RESEARCH FACILITY

D.N. Cardwell1 - T.Povey1 - K.S. Chana1

1 Oxford Thermofluids Institute, Dept. of Engineering Science, Oxford University, Oxford, UK.
2 Civil Large Engine, Turbine Aerothermal, Rolls Royce PLC, PO Box 31, Derby, UK.
3 Research Center, GKN Aerospace Engine Systems, SE-46181, Trollhättan, Sweden.

ABSTRACT

Under the EU LEMOCTEC programme, the Oxford Turbine Research Facility (OTRF) was upgraded to include a modern 1½ stage, high-pressure turbine with film cooled high-pressure guide vanes (HPVs) and low-turning intermediate pressure vanes (IPVs).

The facility has also been upgraded to include a third-generation engine-representative combustor temperature and swirl simulator at inlet, allowing the study of turbine interactions with inlet conditions representative of a modern lean burn combustor.

This paper presents the aerodynamic and mechanical design of the LEMCOTEC high-pressure turbine and its integration and commissioning in the OTRF. Test data with uniform inlet flow is presented, acting as a baseline to assess the performance benefit of optimising the turbine design for a non-uniform combustor exit flow field. Measurement techniques are discussed, and experimental data is compared to pre-test design CFD results from both the Rolls-Royce HYDRA code and the commercial CFX code.

KEYWORDS: TURBINE, EXPERIMENTAL, CFD, COMPBUSTOR, INTERACTION

NOMENCLATURE

$c_a$ axial chord, m $U$ blade speed, ms⁻¹ $c$ HP vane cooling value
$h$ enthalpy, Wm⁻¹K⁻¹ $V$ absolute flow velocity, ms⁻¹
$m$ mass flow rate, kgs⁻¹ $M$ Mach number
$M$ blowing ratio $=\rho_c u_c/\rho_a u_x$ $0$ absolute or total value
$M$ blowing ratio $=\rho_c u_c/\rho_a u_x$ $1$ HP vane inlet value
$N$ rotational speed, rpm $2$ HP vane exit value
$p$ pressure, Pa $3$ HP rotor exit value
$Re$ Reynolds number $4$ IP vane exit value
$T$ temperature, K $ax$ Axial component

Acronyms

HP high-pressure
IP Intermediate-pressure
PS Pressure surface
RANS Reynolds Averaged Navier Stokes
SS Suction surface
URANS Unsteady RANS

INTRODUCTION

Over the last 50 years, continual advances in high temperature alloys and cooling technologies have enabled increases in gas turbine entry temperature, and hence efficiency, from 900°C to over 1600°C in current in-service aero-engines. As fundamental understanding of turbine flow structures and heat transfer has improved through simplified experiments, the need for experiments with higher complexity achieving more engine-representative boundary conditions (higher Technology Readiness Level or TRL) has been recognised. Such experiments still require precise control of the experiment conditions, and differ from engine bed tests as they are specifically designed for research purposes.
Over many years, the Oxford Turbine Research Facility (OTRF) has provided aero-thermal testing of high pressure turbine stages, achieving engine-representative conditions of Reynolds Number, stage pressure ratio, non-dimensional speed $N/\sqrt{T}$, gas-to-wall temperature ratio, inlet turbulence intensity. Chana et al. (2013) provide a review of the test facility and conducted research. Research has included the evaluation of new concepts (Collins et al. 2017) and high precision measurements (Beard et al. 2013). Research interest in combustor-turbine interaction led to the development of a number of combustor simulators for the facility: inlet temperature distortion (Povey and Qureshi, 2008), inlet swirl (Qureshi and Povey, 2008). Computational models validated against such highly engine-representative test data are more robust in application to engine design.

The EU LEMCOTEC project provided continuation of highly engine-representative turbine research in the OTRF. The main objective of the LEMCOTEC project (von der Bank et al. 2014) was the improvement of core-engine thermal efficiency by increasing overall pressure ratio up to 70 leading to further CO$_2$ reductions whilst managing further NO$_x$ production with the development of advanced lean-burn combustion systems. Research in the OTRF focussed on evaluation of the effect of lean-burn combustor traverse on the aerodynamic and thermal performance of a cooled HP turbine and downstream low-turning IP strut. This paper presents the design and installation of the 1½ stage LEMCOTEC HP turbine and the baseline test results with uniform inlet conditions.

Joslyn and Dring (1992) conducted some of the first experiments to study the effect of inlet temperature distortion on turbine performance, concluding: segregation of hot and cold gas towards the rotor pressure and suction surfaces respectively resulting from differences in relative velocity and incidence angle at rotor inlet; stronger radial migration of hot fluid due to stronger secondary flows. Shang and Epstein (1996) also showed consideration of buoyancy effects is important to predict hot fluid migration. Recent research in rotating turbine facilities include efficiency deficit quantification by Beard et al. (2013), detailed inter-row flow-field measurements by Gaetani and Persico (2017), and heat transfer studies by Qureshi et al. (2012) and Mathison et al. (2010).

Comparatively little research has been conducted on the impact of inlet swirl on turbine performance. Qureshi et al. (2011) presented the first combustor-representative inlet swirl generator, developed for the OTRF. Subsequent research quantified turbine efficiency deficit (Beard et al. 2014) and HP vane and rotor heat transfer (Qureshi et al. 2012). Recently, Jacobi and Rosic (2017) also evaluated the full-stage performance deficit with inlet swirl for an integrated combustor-vane concept for power turbines, as well as showing some this could, in part, be mitigated with a re-design of the HP vane. Research has also been conducted in the Large Scale Turbine Rig at the Technical University in Darmstadt. Werschnik et al. (2017) provides insight into the effect of inlet swirl on hub endwall and surface cooling. Results show increased hub endwall coolant and mainstream mixing, and swirl-vane clocking influence on aerodynamic loss.

To date, no experimental research has been conducted considering the effect of combined temperature non-uniformity and swirl – representative of lean burn combustors – on turbine performance. Khanal et al. (2013) provided a computational study demonstrating significantly different effects are expected than a simple combination of the separate effects.

The LEMCOTEC turbine also features an Intermediate Turbine Duct (IDT) with low-turning IP strut. The IDT transports mainstream flow from the relatively small diameter of the HP turbine to the larger IP turbine diameter in a short axial distance. The flow in the IDT is highly complex, featuring tip leakage flows, secondary flow structures and high vorticity. The IP strut (or vane) also carries structural loads and oil and cooling air feeds to the engine core. A number of recent experimental studies, for example Chana et al. (2004) and Wallin et al. (2011), provide essential validation data for numerical models attempting to accurately predict the complicated flow in the IDT. Innovative designs with small aero-vanes places between large structural splitter-vanes have also been investigated, for example by Spataro et al. (2013). The LEMCOTEC ITD and low-turning strut design targets total pressure loss reduction over the design considered by Johannson et al. (2012).
THE OXFORD TURBINE RESEARCH FACILITY

The LEMCOTEC turbine was designed and tested in the Oxford Turbine Research Facility (OTRF), which is a short duration facility used for aerodynamic and heat transfer investigations on single-stage (HP vane and rotor) or ½ stage (HP stage and IP/LP vane) turbines at matched engine-conditions for a quasi-steady run time of ~0.5s. A schematic of the facility is shown in Figure 1.

Figure 1: Schematic of the Oxford Turbine Research Facility

The fundamental operation of this type of facility was first detailed by Jones et al. (1973). Prior to a test run, the plug valve is closed, and the working section and exhaust tanks are at vacuum. Once the rotating assembly is spun to design speed, the test gas is isentropically compressed by injecting high-pressure air by a light piston, driven by high-pressure air. When the desired test gas pressure is achieved, the fast-acting plug is opened and the test gas passes through the working section. The turbine speed is kept constant by the unique aerodynamic braking system, or turbobrake (Goodisman et al. 1992). With a ½ stage turbine installed, an annular ring of deswirl vanes with a choked throat with adjustable area (Povey et al. 2003) is used to set the turbine pressure ratio and return the flow to axial, meeting the design condition for the turbobrake. Turbine cooling air is delivered from a large reservoir of high-pressure air at ambient temperature. The reservoir pressure is varied to supply the required coolant mass flow which is metered by an ISO calibrated choked venturi. The facility is also capable of testing with inlet profiles of temperature, inlet swirl and combined temperature and swirl representative the flow-field exiting modern combustors.

This experimental program includes testing of a new film-cooled HP turbine and downstream low-turning strut, focusing on the impact of engine-representative lean-burn combustor exit profiles on turbine aero-thermal performance.

LEMCOTEC ½ STAGE TURBINE DESIGN

The LEMCOTEC ½ stage turbine provides the OTRF with an engine realistic, modern, HP turbine stage and downstream low-turning strut configuration. The turbine included a film-cooled HP vane and was designed for two inlet conditions: uniform conditions of pressure and temperature, and an inlet with a strong radial total temperature profile and swirling velocity profile representative of that exiting a modern lean-burn combustor. The combustor burner : HP vane : HP blade : IP strut count ratio was 1:2:3:1 to facilitate unsteady CFD validation with minimum mesh size. The HP blade included an unshrouded design with a flat tip and nominal clearance of 1% radial span. The standard non-dimensional parameters for the LEMCOTEC turbine are tabulated in Table 1, and the corresponding operating point in the OTRF is also summarised.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>$\Delta h/U^2$</th>
<th>$V_a/U$</th>
<th>$U/\sqrt{T}$</th>
<th>$\Delta h/T$</th>
<th>$\Lambda$</th>
<th>$T_{gas}/T_{wall}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>LEMCOTEC</td>
<td>1.64</td>
<td>0.42</td>
<td>11.35</td>
<td>211.51</td>
<td>50.71</td>
<td>1.64</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Parameter</th>
<th>$Re_{Cax}$</th>
<th>$M_2$</th>
<th>$p_{01}/p_3$</th>
<th>$p_{03}/p_4$</th>
<th>$p_{01}$</th>
<th>$T_{01}$</th>
<th>$N$</th>
<th>$\dot{m}_1$</th>
<th>$\dot{m}_c/\dot{m}_3$</th>
<th>$T_{01}/T_c$</th>
<th>$T_{01}/T_{wall}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Units</td>
<td>$\times10^6$</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>K</td>
<td>Rpm</td>
<td>kg s$^{-1}$</td>
<td>%</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>OTRF value</td>
<td>2.13</td>
<td>0.917</td>
<td>2.86</td>
<td>1.37</td>
<td>8.5</td>
<td>475</td>
<td>8394</td>
<td>23.00</td>
<td>6.85</td>
<td>1.64</td>
<td>1.64</td>
</tr>
</tbody>
</table>

Table 1: LEMCOTEC turbine non-dimensional parameters and OTRF operating point
The Rolls-Royce through-flow software (Q263) – a streamline curvature and radial equilibrium solver – was used to initially simulate the gas path. At this early design stage, the focus was to predict whirl angles at the inlet and exit of each blade row and lift coefficients necessary to achieve these, while keeping to reasonable values within current, or future, design and manufacturing capability. The through-flow code results are compared to experimental results later in this paper.

**High Pressure Turbine Aerodynamic Design**

The LEMCOTEC HP turbine was designed following industry design practise, applying a hierarchy of tools. As previously mentioned, through-flow (streamline curvature) models were created using the Rolls-Royce in-house solver Q263 for vortex design. Subsequently, the Rolls-Royce 2D blade-to-blade solver FINSUP was used to create suitable base section designs at five radial spans (12%, 35%, 56%, 76%, and 89%). The definitions of these base sections were then iterated between 2D blade-to-blade and 3D single row CFD RANS calculations. The HP vane was specifically designed with a high incidence tolerance to minimise the performance deficit with the presence of engine-representative swirled inlet conditions.

Finally, the performance of the entire HP turbine stage was further matched to the design intent using multi-row 3D CFD RANS models. Simulations were performed using the Roll-Royce in-house HYDRA solver and utilised mixing planes between the vane and rotor meshes. The modelled turbine design closely matched the design operating point listed in Table 1. The full details of the computational setup are provided later in this paper and results are compared to experiment.

**Cooling and 2D Thermal Model**

The HPV cooling system was designed using a 2D thermal model incorporating industry standard correlations to calculate heat transfer, pressure loss, discharge coefficients and film effectiveness. The HP vane included two simple coolant feed cavities (front LE duct and rear) without internal cooling features as the focus of the study was external heat transfer. The geometry of the feed cavities (visible in Figure 3) was implemented into the 2D model as a function of radius. The cooling scheme included 10 rows of cylindrical cooling holes, five connected to the LE duct and five connected with the rear passage. The first PS and SS film cooling rows were positioned such that no film row crossed the predicted stagnation line with either inlet condition. Such behaviour would add unnecessary complexity during post-analysis of a test campaign primarily focussed upon CFD validation for new combustor and turbine technologies.

In the OTRF, the HPV coolant is supplied to the feed cavities from a common plenum, thereby setting a common feed pressure and temperature ($T_{0c} = 290$ K). To match the momentum flux ratio to first order, the overall coolant-to-mainstream total pressure ratio was set representative of engine conditions, $p_{0c}/p_{01} = 1.025$. With these conditions, the model predicted a coolant-to-mainstream mass flow ratio of $\dot{m}_{c}/\dot{m}_{1} = 6.78\%$, -0.07\% from design intent, for the final cooling scheme.

![Figure 2: Blowing ratio and pressure ratio predicted by 2D thermal model for all films](image-url)

The coolant-to-mainstream static pressure ratio, $p_{0c}/p$, and the blowing ratio, BR, predicted for each film are plotted in Figure 2. The predicted pressure ratios were between 1.02\% and 3\% for PS
film rows 1-3 and SS film row 1. At mid-height, all pressure ratios are above 2%, meeting the design intent to prevent ingestion during operation. The blowing ratio is below 2.5 for most films, which indicates the film rows are unlikely to lift-off from the vane surface. Blowing ratios above 2.5 are predicted for PS film row 1, SS film row 1 and the lower half of PS film row 2. This results from low mainstream Mach number at the exit to these film cooling rows, driving the blowing ratio high. It is very common to witness such high blowing ratios for the first leading edge rows.

**Low-Turning Intermediate Duct and Vane Design**

The intermediate turbine duct (ITD) and low-turning IP vane were designed using GKN methods based on boundary conditions exported from the exit plane of the multi-row 3D CFD of the HP turbine design. The design criteria for the ITD and IP vane were geometric and aerodynamic and representative of a modern large civil engine with counter rotating HP and IP turbines. The ITD and IP vane were designed by GKN. Single row 3D CFD was conducted using the commercial ANSYS CFX code to iterate the design to meet the targets and check for boundary layer separation. Full details of the computational setup are provided later in this paper and the results compared to experiment.

**INTEGRATION INTO OXFORD TURBINE RESEARCH FACILITY**

A sectional view of the OTRF LEMCOTEC working section and photographs of various manufactured components are shown in Figure 3. Installation of the ½ stage LEMCOTEC turbine into the OTRF required manufacture of the coloured (non-grey) components in the CAD model. The manufacture of the HP vane was particularly complicated, requiring a three stage process: (1) CNC machining of exterior form and front coolant cavity, (2) constant radius electro-discharge machining of rear coolant cavity, (3) CNC drilling of film-cooling holes. A number of HP vanes were sectioned along the radial span to verify wall thicknesses were within design limits. The HP turbine outer case included four removable HP vane cassettes, each containing five HP vanes, allowing fast change-over of HP vane instrumentation without removing the working section from the test facility.

**Figure 3: LEMCOTEC working section CAD and various manufactured components**

A mechanical assessment of the rotor blades and disc was performed using in-house Rolls-Royce stress analysis software. Maximum nominal stress values of 26% and 69% of material yield were calculated at 120% facility design speed in the blade root and disc post respectively. The blade design accounted for the predicted radial growth of the blade tip at design speed. The as-build tip clearance was evaluated from static measurement and corrected using the predicted radial growth.

Assessment of terminal speed indicated blade failure at 28,000 rpm (before disc failure). The life of the blades and disc was estimated at $10^5$ cycles and $5 \times 10^4$ cycles respectively. Finally, a vibrational analysis indicated the lowest margin between an engine-order excitation and resonance was 8.9%.

**COMPUTATIONAL SETUP**

The commercially available ANSYS CFX solver and Rolls-Royce in-house HYDRA suite (Lapworth, 2004) were used to perform CFD simulations during the design and post-test analysis. The simulation domains consisted of 2 HP vanes, 3 HP rotor blades and a single IP vane reflecting the minimum count ratio for the turbine. The single passage meshes used in the simulations are
summarised in Table 2. All meshes were structured grids capable of resolving the viscous sublayer ($y^+ < 5$) and included endwall fillets. PADRAM (Milli and Shahpar, 2012) and G3DMESH are in-house meshing tools developed at Rolls-Royce and GKN Aerospace, respectively. A formal grid independence study was not conducted during the aerodynamic design phase, rather a fine mesh resolution was targeted for future heat transfer studies beyond normal industry design standard.

The respective computation details for each solver are summarised in Table 2. Simulations were conducted with both turbine inlet conditions, and with and without HP vane cooling. The HP vane inlet boundary conditions defined 2D profiles of: $p_0$, $T_0$, yaw angle and pitch angle, as well as 10% turbulence intensity. A radial profile of static pressure was defined at the exit of the IP vane domain. All computations were performed with isothermal wall conditions to best model the experiment.

<table>
<thead>
<tr>
<th>Solver</th>
<th>HP vane</th>
<th>HP rotor</th>
<th>IP vane</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mesh tool</td>
<td>Size</td>
<td>y+ max</td>
</tr>
<tr>
<td>HYDRA</td>
<td>PADRAM</td>
<td>7.8M</td>
<td>1</td>
</tr>
<tr>
<td>CFX</td>
<td>G3DMESH</td>
<td>4.9M</td>
<td>1</td>
</tr>
<tr>
<td>Solver</td>
<td>Multigrid levels</td>
<td>Turbulence model</td>
<td>Mixing planes for steady computation</td>
</tr>
<tr>
<td>HYDRA</td>
<td>4</td>
<td>$k-\omega$ SST</td>
<td>HP rotor inlet &amp; exit</td>
</tr>
<tr>
<td>CFX</td>
<td>5</td>
<td>$k-\omega$ SST</td>
<td>HP rotor inlet &amp; exit</td>
</tr>
</tbody>
</table>

Table 2: CFD mesh and computation details

The two solvers used different methods for introducing the HP vane cooling. The HYDRA solver used a strip source model for each film row. The ANSYS CFX solver used a source term method described by Nagawakar (2016) that added coolant at individual hole locations. This method was validated against CFD simulations with fully resolved coolant feed plenums, holes and mainstream.

A further subtle difference between the models is that the HYDRA results are design (true predictions) whereas the CFX results are post-dictions, in that the measured exit boundary conditions have been applied, which may account for small differences. Also the HYDRA model applied an adiabatic condition at the walls, commonly adopted in turbine design to evaluate aerodynamic performance, whereas the CFX model applied an isothermal condition as in the experiment. This subtlety also presents interesting differences between the design and post-diction cases as seen later.

**EXPERIMENTAL RESULTS AND CFD COMPARISON**

**Measurement of Inlet Boundary Conditions**

Inlet area survey measurements were conducted for both stage inlet conditions: uniform inlet and non-uniform (radial $T_0$ and inlet swirl profiles representative of a modern lean-burn combustor). The measured turbine inlet boundary conditions for the two configurations are shown as radial profiles of $p_0$, $T_0$ and yaw angle in Figure 4 below.

With uniform inlet conditions, area survey measurements of $p_0$ and $T_0$ were conducted in the OTRF. Due to temporal variations in the turbine inlet flow caused by small piston oscillations, inlet area surveys in the OTRF are acquired by traversing three stationary radial rakes over a number of tests, between which the radial and circumferential positions of the rakes are altered. A full survey comprises 27 circumferential by 18 radial measurements distributed over two NGV pitches, equivalent to one combustor simulator pitch. Total pressure measurements were conducted (at half radial resolution) using Kiel head pitot tubes with 1mm diameter connected to Sensor Technics CTE8000 series pressure transducers with a full-scale traceable uncertainty of $\pm 0.07\%$ (or 0.7 kPa). Total temperature measurements were conducted (also at half radial resolution) using k-type 25.4μm bare bead thermocouples with an absolute accuracy of $\pm 0.32\%$ (or $\pm 1.5 \text{ K}$). The radial locations for each measurement type in indicated by blue circles in Figure 4.
Figure 4: Stage inlet conditions showing radial profiles (circumferential average of area survey data) of: (a) \( p_0 \), (b) \( T_0 \), (c) yaw angle; and area survey of \( T_0 \) with non-uniform inlet.

The design of a combined hot streak and swirl generator is detailed by Hall et al. (2014). The installation and testing of the combined hot streak and swirl generator in the OTRF has been more recently described by Adams et al. (2019). In the OTRF, pressure loss and flow angle measurements downstream of the combustor simulator are very challenging due to limited access for rakes of multiple hole probes and the short test duration. Instead, a bespoke experimental facility – see Hall et al. (2014), was used to characterise the combustor simulator, permitting high-resolution area survey measurements of the simulator exit flow at a location corresponding to the stage inlet plane. The radial profiles of total pressure and yaw angle from these experiments are also plotted in Figure 4 as dashed red lines without markers. The total pressure profile shows a well-defined vortex loss core at mid-span and elevated total pressure near the endwalls. The radial profile of yaw angle also indicates a well-defined vortex core. The peak yaw angles measured across the area survey were +30.9° and -28.4° respectively (clockwise vortex viewed from upstream). A full area survey of \( T_0 \) was measured in the OTRF with the non-uniform inlet condition as described above for the uniform inlet case. The measured area survey data and radial profile are both presented in Figure 4. The data shows a distinct radial variation with a \( T_0, max/T_0, min = 1.54 \). This collection of data illustrates the successful simulation of an inlet boundary condition representative of that exiting a modern lean burn combustor, exhibiting a distinct vortex structures and little circumferential variation compared to a rich bean combustor as a result of increased mixing between the fuel and air.

Inter-row Endwall Static Pressures

Static pressure measurements were conducted on the hub and casing endwalls at six inter-row axial planes throughout the turbine stage: HP vane exit, HP rotor exit, IP vane inlet, IP vane exit near plane, IP vane exit traverse plane and IP vane exit far plane (boundary for CFD domains). At each axial plane, measurements were positioned circumferentially across a single combustor profile pitch (equivalent to two HP vane passages and a single IP vane passage). A minimum of 10 measurements were conducted at each location, with 122 measurements in total.

The bar chart in Figure 5 compares the measured inter-row static pressures to those predicted by the CFX steady CFD model. The percentage differences are labelled above each group of bars. The test facility is operated to match the 1.5 stage total-to-static pressure ratio between the stage inlet and IP vane exit far plane. It can be seen this boundary condition is achieved to within +0.08% (facility run-to-run tolerance is ±2%). Although the CFD model omits to rotor hub seal flows, mismatch between experiment and CFD could be explained by small differences in component capacity from design intent. Significant difference is observed at the HP vane exit. With a CNC machining tolerance of ±0.05mm a difference in throat area of order ±0.6% could be expected. Considering the predicted HP vane capacity curve presented in Figure 5, a reduction in HV vane pressure ratio by 5.35% could be explained by a reduction in throat area by 0.62%.
Figure 5: Comparison of measured and predicted inter-row static pressures; and predicted HP vane capacity as a function of vane pressure ratio

HP Vane Loading Results

Static pressure measurements were conducted on the HP vane aerofoil surfaces at discrete locations along 10%, 50% and 90% radial span with and without vane cooling. Measurements were conducted using pneumatic tapping lines (CNC drilled with Ø1mm) connected to Sensor Technics CTE8000 series transducers with a traceable full-scale accuracy of ±0.07%. On uncooled components, 16-19 measurements were acquired at each span, whereas on cooled components, 8 measurements were acquired due to the space restrictions imposed by the 10 rows of film cooling holes. Measurements at 10% radial span on cooled components were also unfeasible for this reason.

Experimental data is compared to RANS computational results in Figure 6 for the uniform inlet condition. The data indicates an aft-loaded HP vane with a small increase in loading with decreasing radial span. Generally, the agreement between experiment and CFD is good. On the pressure surfaces (PS), experiment and all computations agree well, with little effect observed with the introduction of cooling and little difference between solver performance. On the suction surfaces (SS), the agreement between experiment and computation is also good over the front section (0-65% axial chord). Aft of this region, all computations over-predict the surface static pressure, which is consistent with the over-predicted HP vane exit static pressure discussed above and shown in Figure 5. With the introduction of cooling, little change is observed at 10% radial span between the CFX CFD results, or at 50% span between experimental or CFD data. Interestingly, at 90% span a reduction in front vane loading is measured and predicted by CFD between 25-60% axial span. Some differences between the CFD results from the two solvers is observed between 50-100% axial span. In this region, the flow is transonic with the presence of shock waves making this a more challenging flow-field to predict. Whether the discrepancy between the solvers is grid or solver related requires confirmation.

Figure 6: HP vane surface static pressure measurements and CFD RANS results at 10%, 50% and 90% radial span with and without film cooling
Computational results with non-uniform inlet conditions from the CFX solver are also plotted without cooling. The observed increase and decrease in loading at 10% and 90% radial spans respectively are consistent with that expected from negative and positive local incidence changes with the introduction of the generated swirled profile.

IP Vane Loading Results

The blade count ratio of 2:3:1 results in each IP vane experiencing an identical flow-field, in particular with respect to the incoming HP vanes wakes. Static pressure measurements were conducted on the aerofoil surfaces of three IP vanes at discrete locations along 10%, 50% and 90% radial span with a total of 107 measurements, using pneumatic tappings (CNC drilled with Ø1mm) connected to Sensor Technics CTE8000 series transducers with a full-scale accuracy of ±0.07%.

Experimental data is compared to RANS and URANS computational results in Figure 7 for the uniform inlet condition. In contrast to the data comparison for the HP vane, greater variation is observed on both aerofoil surfaces between experiment and computation. Even so, all computation results of static pressure are within 3% of the experimental values. At 10% span, agreement on the SS is good, but all CFD models under-predict the static pressure on the PS in a similar way. At 50% span, the experimental data also shows higher loading than that predicted by either solver. At this span, some difference between the solvers is seen but neither performs better, with CFX and HYDRA matching better with experiment on PS and SS respectively. At 90% span, the degree of loading is well matched, although all computations under-predict the static pressures on both PS and SS.

Little difference is observed between steady and time-averaged unsteady computational results for both CFD solvers. Differences between solvers are also small compared to the differences between experiment and computation. Therefore, neither solver can be alleged to perform better in predicting the experiment.

![Figure 7: IP vane surface static pressure measurements with CFD comparisons at 10%, 50% and 90% radial span (uniform inlet)](image)

HP Rotor Casing Pressure and Exit Total Temperature

The static pressure on the casing wall above the rotor blade tip was measured over a region extending approximately 2 HP vane passages in the circumferential direction and 1.25 times rotor blade axial chord in the axial direction. Measurements were performed using 96 pneumatic tappings distributed as 8 axially spaced rows each with 12 circumferentially spaced tappings. The third row is aligned with the rotor blade leading edge plane. The pneumatic lines were connected to Sensor Technics CTE8000 series transducers with a traceable full-scale accuracy of ±0.07%.

The measured area map of rotor over-tip static pressure acquired with uniform inlet conditions is shown in Figure 8 below. The tapping locations are marked by ‘o’ symbols and HPR axial chord location 0 refers to the HPR leading edge. The distribution is largely axial (also plotted), with very small circumferential variations caused by HP vane wakes. As these measurements are acquired in the stationary frame, a reduction in static pressure in the axial direction results from work extraction.
by the rotor blades. Steady and time-averaged unsteady computational results from the CFX model are compared to the measured axial profile of static pressure in Figure 8 showing excellent agreement.

The drop in total temperature through the turbine stage is also a measure of the work extraction by the HP rotor. Total temperature measurements at the HP rotor exit were acquired by traversing a radial rake, containing ten aspirated k-type 25.4μm bare bead thermocouples with an absolute accuracy of ±0.42% (or ±1.5 K), circumferentially across a single HP vane pitch during a test run. Traverses were conducted with the rakes at two radial locations to result in an area survey with a resolution of 20 radial measurements in the radial direction.

![Figure 8: Experimental and computational results of rotor over-tip static pressure: area map (experimental only) and axial profile; HP rotor exit total temperature (uniform inlet).](image)

The measured radial profile (circumferential average of area traverse data) of HP rotor exit total temperature with uniform inlet conditions is plotted in Figure 8. Steady and time-averaged computational data for both solvers are also plotted. The CFX RANS and URANS solutions show reasonably good agreement with the experiment data, albeit mismatches in the highly sensitive tip gap region, and at mid-span where a higher total temperature is measured – the origin of which is currently unresolved. The HYDRA solutions generally under predict the total temperature drop through the turbine across most of the radial span.

**IP Vane Exit Traverse Data**

Area traverse data was also acquired at the IP vane exit using a circumferential traverse system. Data was taken using a combined three-hole pneumatic probe with two nose mounted aspirated k-type 25.4μm bare bead thermocouples providing measurements of yaw angle, $p_0$, $M$ and $T_0$. To obtain a full area survey across one IP vane pitch, the probe was traversed dynamically across circumferential arcs of constant radial span over multiple test runs. Measurements at a total of 21 radial spans were acquired ranging from approximately 6% to 95% radial span.

Experimental data and computational results are compared in Figure 9. All computations agree with the experimental data of yaw angle to within 2.5° below 60% radial span. Above 60% span, the flow-field is dominated by the impact of the HP rotor tip leakage that has been transported through the IP vane vastly complicating the predictive task. Apart from the unsteady CFX results, the computations have generally predicted the profile shape above 60% span, but all include maximum discrepancies of approximately 8-10°.

The radial profile of total pressure is predicted reasonably well by all computations with generally good agreement between the solvers. The maximum discrepancy is 3.3% at 70% radial span. The time-averaged CFX results shows closer agreement than the RANS result, although this model performed worst in predicting the yaw angle variation. The HYDRA models show little variation.

The radial profile of Mach number is also predicted reasonably well by all computations. Comparison with the total pressure data shows an under-predicted static pressure below 40% radial span. Between 50-80% span the over-predicted Mach number is driven by the over-predicted total pressure in this region. Again, the URANS CFX result compares better than the RANS and the HYDRA data shows little difference between models.
Finally, the radial profile of total temperature is presented and compared to computational results. The profile shape is well captured by the CFX models, but the magnitude is under predicted by approximately 9K. The HYDRA models show a better agreement from 10-70% radial span, but diverge from the experimental data near the endwalls – in particular over predicting the near casing total temperature by almost 40K.

CONCLUSIONS

The LEMCOTEC 1½ cooled HP turbine with low-turbine IP strut has been successfully designed, installed and tested in the Oxford Turbine Research Facility. The turbine design was a collaborative effort with the HP turbine stage and low turning IP strut being designed by Rolls-Royce UK and GKN Aerospace respectively. The HP turbine was designed for two inlet conditions: uniform inlet and an inlet profile representative of that exiting a modern lean burn combustor (strong radial $T_0$ profile and swirling flow), requiring a HP vane with high incidence tolerance and consideration of the HP vane stagnation line movement between the two inlet conditions. The detailed design provided a HP vane cooling scheme with representative blowing and pressure ratios without ingestion.

A new non-reacting combustor simulator capable of generating profiles representative of a lean burn combustor has also been successfully installed and commissioned in the OTRF as part of this project. Area traverse measurements acquired at the turbine inlet plane demonstrated a strong radial temperature variation ($T_{0,max}/T_{0,min} = 1.54$) and a stable vortex structure with peak yaw angles of $+30.9^\circ$ and $-28.4^\circ$. To the authors’ knowledge, this is the first time such a system has been successfully installed in a turbine research facility and measurement made.

Experimental data have been acquired throughout the 1½ stage LEMCOTEC turbine and compared to design steady and time-resolved CFD results from two solvers (ANSYS CFX and the Rolls-Royce in-house HYDRA solver). Annulus line static pressure measurements agreed with CFD predictions to within ±1.6% of design, except at the HP vane exit. The comparatively large discrepancy of -5.63% at this location was explained by possible manufacturing tolerances and HP vane capacity sensitivity with pressure ratio. 3D laser light scanning of the turbine components will be conducted as part of post-test CFD investigations. HP and IP vane surface static pressure measurements were conducted at 3 radial spans (10%, 50% and 90%). All computational data showed reasonably good agreement, with little difference observed between steady and time-averaged predictions. Finally, IP vane exit traverse
measurement data were compared to the computational models. Particularly good agreement was achieved between total pressure measurements and predictions for all models. Discrepancies of up to 10° between measured and predicted yaw angle in the near casing region were attributed to difficulties in predicting the HP rotor tip leakage flow and its transport through the IP vane.

All of the data presented in this paper provide a baseline for future experimental and computational investigations on the LEMCOTEC under test in the Oxford Turbine Research Facility.

ACKNOWLEDGEMENTS
The authors would like to gratefully acknowledge the financial support provided by the European Commission and its partners, especially Rolls-Royce and GKN Aerospace, through the LEMCOTEC project within the 7th Framework Programme. The authors would also like to thank T. Godfrey for his dedication to the project instrumentation needs and S. Chana for his assistance in running test facility.

REFERENCES


