An investigation of secondary flow features in a low pressure turbine

To cite this article: P. Jonak et al 2018 J. Phys.: Conf. Ser. 1101 012011

View the article online for updates and enhancements.
An investigation of secondary flow features in a low pressure turbine

P. Jonak¹, T. Borzecki², S. Kubacki³

¹Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology, Nowowiejska 24, 00-665, Warsaw, Poland; pjonak@meil.pw.edu.pl
²Avio Aero, Grazynskiego 141, 43-300 Bielsko-Biala, Poland; Tomasz.Borzecki@avioaero.it

Abstract. A flow simulation through a four-stage low pressure turbine (LPT) of aero-engine, using eddy-viscosity based RANS (Reynolds-averaged Navier-Stokes) model, is performed. The numerical results are compared with experimental data obtained in Polonia Aero Lab in Zielonka, Poland. Good agreement between measured and predicted global flow features and the pressure coefficient on surface of inlet strut is reported. The development of vortex structures in inlet duct of the low pressure turbine (LPT), named a turbine central frame (TCF), is analysed. An analysis of secondary flow losses by means of entropy generation rate coefficient is provided.

1. Introduction
A complex three-dimensional flow in endwall region near the base of turbine blades and vanes has an important impact on the level of aerodynamic losses. In the low pressure turbine (LPT) the three main sources of aerodynamic losses might be reported: the profile losses, the flow leakage losses and the endwall losses. The growth of the boundary layers on the airfoil surface is responsible for the profile losses. The flow leakage losses are generated by a leakage of the flow over tips of the rotor blades and stator vanes. The endwall losses appear as result of enhanced mixing caused by the secondary vortex structures developing near the base of the rotor blades or stator vanes. In spite of the fact that all three types of losses have their own origin, they almost never occur separately. For instance, the flow over the blade and vane surfaces is strongly affected by the secondary flow characteristics along the endwall.

The oncoming flow impacts the leading edge of the blade and the horseshoe vortex is formed. Farther downstream, the horseshoe vortex develops in the form of streamwise-oriented vortices called pressure- and suction-side legs of the horseshoe vortex. The pressure-side leg of the horseshoe vortex moves towards the suction-side of adjacent blade, due to strong transverse pressure gradient, and becomes a part of the passage vortex. The suction-side leg of the horseshoe vortex may develop above the passage vortex, it might be entrained into the passage vortex or dissipate rapidly downstream the leading edge depending on the case. We refer to work by Dunham [1], Sieverding [2], Langston [3] and Ligrani et al. [4] for a discussion of secondary flow losses in axial turbines.

Denton [5] provides an extensive analysis of the loss generation mechanism in turbomachines. The physical sources of losses like viscous and turbulence mixing effects in the boundary layers, heat transfer and influence of shock waves were discussed. The formulas for loss coefficient based on entropy generation were defined. The experimental study of the stagnation pressure losses through the passage in a linear turbine blades cascade was made by Harrison [6]. Endwall shear stress, measured by hot film probe, was used to define the loss coefficient. It was shown that the wake mixing is relevant for accurate
prediction of the loss coefficient downstream of the trailing edge of the blade. Experimental study of the leakage losses in a linear turbine cascade with tip clearance was performed by Yamamoto [7]. The clearance gap size and the angle of incidence were claimed to strongly influence the measured leakage and total pressure losses. The interaction between the leakage flow vortices and the passage vortex was reported. Benner et al. [8] [9] performed a two-part study on empirical method for prediction of the secondary flow losses. New loss breakdown scheme with improved correlation for the spanwise penetration depth of the passage vortex was presented in [8]. A novel empirical prediction model for secondary losses was reported in [9]. A considerably large database was used for comparison of predicted and measured cascade data. Analysis revealed that a general scaling factor is required for the new secondary loss correlation in order to give reasonable results for wide range cases. Recently, numerical study of secondary flow in the linear cascade with endwall using LES was made by Cui et al. [10].

The present work reports the simulation results of main flow path (flow without taking into account the flow through the blade tip and inter-stage labyrinth seals) through the four-stage low pressure turbine (LPT). Numerical results are compared with the experimental data from the test rig. The origin of the vortex structures which develop in the vicinity of the shroud and hub in the entrance duct, called a turbine central frame (TCF), is discussed. The coefficient based on entropy generation is used to describe the aerodynamic losses in the passage. In our earlier work [11], the three different turbulence models were tested for simulation of flow through linear cascade and the \( k-\omega \) SST was found to give the best correspondence with experiment. The present simulations are performed with the \( k-\omega \) SST model.

2. Test rig
The low pressure turbine consists of 11 conceptual parts: swirler, turbine central frame (TCF), four turbine stages (each consists of vane and blade rows) and exhaust vane turbine rear frame (TRF). The main flow inside the LPT was measured in Polonia Aero Lab in Zielonka (Poland). The measurements of the static pressure on the surface of inlet strut and the global flow characteristics were performed.

![Figure 1. Schematic of LPT and localisation of measuring traverse towers.](image)

Figure 1 shows the cross-section through the LPT and positioning of measuring traverse towers. The three traverse systems, spaced 120°, were installed at the inlet to the LPT. The five-hole probe that was installed in the traverse tower can rotate ±15° relative to its central axis. The measurement of temperature, pressure and fluid flow angles were possible by the radial movement of the probe along the span. One traverse passes the total cycle in about 2 hours. The same test system was installed at the
outlet from LPT. The pressure taps, were placed at 20, 50 and 80% of a radial span on the TCF vane. The mean distance between the taps was about 9% of the TCF axial chord length.

3. Computational details

The Moving Reference Frame (MRF) method was used to take into account for the rotating flow dynamics. The air was treated as an ideal gas. The energy equation was used with the viscous work term active. High resolution scheme of ANSYS CFX solver was used for discretization of momentum, continuity and transport equations. Normalized residuals drop down to $10^{-4}$ for all equations. Constant values of the total pressure, total temperature and turbulence intensity were defined at the inlet to the LPT placed 2.6$c$, upstream of the leading edges of the swirler vanes ($c$, refers to the swirler vane axial chord). The mass-averaged inlet values of total pressure and total temperature were set according to experiment. The flow velocity vectors were set normal to the inlet plane. The value of the static pressure at the outlet from the LPT was set according to experiments. The periodic boundary conditions were applied on side boundaries. The shroud, hub, vane and blade surfaces were treated as no-slip, adiabatic walls. The wall function technique was employed near to walls. The Stage and Frozen Rotor approaches (see description below) of the ANSYS CFX solver were used at the interfaces between consecutive vane- and blade-rows in different simulations. Only connection between the swirler and TCF domain, located in the front part of the turbine, was defined without the use of any interface technique. The pitch of the swirler domain was adjusted somewhat, in order to obtain 1:1 area contact between the swirler and the TCF zone.

The Stage technique employs the circumferential averaging of fluxes in bands at the outlet from a given part (vane and blade domains) and later transmits the averaged fluxes to the downstream part. The model is useful for large pitch ratios between the consecutive domains (vane and blade domains). This way, a single blade domain is used for simplified representation of the whole blade-row and a single vane domain is used for representation of whole vane-row. This reduces significantly the computational costs related to simulation of flow through the whole LPT. In present work, the Stage interface model has been applied for the grid sensitivity study related to the TCF region (see discussion below).

The Frozen Rotor technique produces a steady state solution to the local frame of reference on each side of the interface. The two frames of reference are connected in such a way that they have a fixed relative position during the calculation. If the frame changes the appropriate transformation of the equations is introduced. With the Frozen Rotor technique the pitch change between the consecutive domains has to be avoided. But if the pitch changes, the fluxes are scaled by the pitch change coefficient. In current work, the largest pitch was reported for the TCF domain. The single vane and blade domains downstream of TCF domain were multiplied several (few) times to match the pitch of the TCF domain. The computational grid for simulation with the Frozen Rotor technique is therefore much larger than for simulation with the Stage model. An analysis of secondary flow details in TCF region and partly also an experimental validation of the LPT results was performed using the Frozen Rotor interface model.

Block-structured mesh was generated for simulation of the flow through the low pressure turbine (LPT). The grid in boundary layer region in every part of the LPT contained 25 nodes and was characterized by 1.15 cell growth ratio. The thickness of the boundary layer mesh zone (O-grid block) was defined to be about 20% larger than the turbulent boundary layer thickness estimated from the correlation. The vane and blade grids consisted of about 1 million nodes. The swirler, TCF and TRF meshes had about 1.3, 2 and 3 million nodes, respectively. The mesh refinement was introduced in the TCF region only due to the fact that the mesh resolution in this part of the LPT domain was found to have a strong influence on the numerical results. About 25, 35 and 45 nodes were assigned inside the boundary layer for coarse, medium and fine grids in TCF region, respectively. The thickness of O-grid mesh and the growth ratio was kept the same in all three cases. The maximal values of $y^+$ on the strut surface in TCF region were equal to 21, 11 and 6 on coarse, medium and fine meshes, respectively. The maximal value of $y^+$ at other walls was less than 24. Table 1 summarizes the total number of points applied in the present work for simulations using the Stage (basic, medium and fine grids applied in TCF region) and the Frozen Rotor techniques.
In present work the reduced-order approach, namely the eddy-viscosity RANS model, was employed for prediction of the mean flow characteristics. Note that the grids applied by Cui et al [10] for LES of flow through a single blade cascade with endwall are about 40 times denser than the current single blade grid applied with the Stage model. In present work, the whole grid for simulation of flow through the LPT using the Frozen Rotor technique consists of about 100 million nodal points (Table 1). An application of LES will require much denser grid (refined more than two order of magnitude with respect to current one) and integration of governing equations in time using a low value of the CFL number. We cannot afford such prohibitive costs of application of LES in the present study.

The grid sensitivity study was performed by means of the pressure coefficient

\[ C_p = \frac{p - p_1}{0.5 \rho_1 U_1^2}, \]  

where \( p \) is the static pressure on the blade surface and \( p_1, \rho_1, \) and \( U_1 \) are the mean static pressure, density and freestream velocity at midspan at the inlet to LPT, respectively. The second quantity was the normalized turbulent kinetic energy

\[ q = \frac{k}{(U_1^2)}, \]  

where \( k \) denotes the turbulent kinetic energy. The \( C_p \) distribution along 50% of span of TCF vane on different density grids is presented in figure 2(a). Hardly any difference between coarse, medium and fine grids is visible. Figure 2(b) shows the circumferentially averaged profiles of normalized turbulent kinetic energy, \( q \), at the outlet from TCF. Despite some difference are reported in the predicted turbulent kinetic energy profiles, between 5 and 80% of span, they had almost no influence on the mean flow characteristics at the outlet from TCF region (results not shown). As shown in Table 1 there are also only small differences reported between predicted mean flow characteristics on different density meshes.

**Figure 2.** Pressure coefficient along TCF vane at 50% of span (a) and profiles of circumferentially averaged normalized turbulence kinetic energy at outlet from TCF (b) for different density grids.

### 4. Results

#### 4.1. Comparison of numerical results with experiment

Table 1 shows a comparison between measured and predicted global flow characteristics on different density meshes, using the two different approaches at interfaces between domains (Stage vs. Frozen Rotor technique). The comparison is made by means of the relative difference
\[ D = \frac{\varphi_{\text{num}} - \varphi_{\text{exp}}}{\varphi_{\text{exp}}} \times 100\% , \]  

where \( \varphi_{\text{num}} \) and \( \varphi_{\text{exp}} \) are the numerical and experimental values of given variable. The mass flow rate \( (mf) \), total pressure \( (p_t) \) and total temperature \( (T_t) \) are analyzed. The global flow parameters were measured in two sections one at the outlet from TCF (Section 48) and one after the Blade04 (Section 55). Their locations are indicated in figure 1. The lack of tip blade and inter-stage sealing cavities in numerical model might be responsible for small differences between the predicted and measured mass flow rate (difference less than 1.5%). But the differences between the measured and predicted average values of the total pressures and total temperatures are small. They do not exceed 0.4% in all analyzed cases. So the agreement between measured and predicted global flow quantities is encouraging.

Numerical results of the pressure coefficient on the TCF vane at two spanwise locations, namely 20% and 80% from the hub surface, obtained using the Frozen Rotor technique (Case 4 in Table 1) are compared with experimental data in figure 3. Measurement uncertainty of the pressure sensors is ±0.05%. One can notice some under-prediction of \( C_p \) value at \( x/c = 0.2 \) on the suction and pressure side of the vane in figure 3(b). The satisfactory level of agreement between measured and predicted pressure coefficient at TCF vane is obtained.

### Table 1. Total number of nodes is simulation using the Stage and Frozen Rotor techniques and relative differences between measured and predicted global flow parameters at sections 48 and 55.

| Case   | TCF grid | Interface model | LPT grid size [nodes] | Section 48 | Section 55 |
|--------|----------|-----------------|-----------------------|------------|------------|
|        |          |                 | \( mf [%] \)           | \( p_t [%] \) | \( T_t [%] \) | \( p_t [%] \) | \( T_t [%] \) |
|        |          |                 |                       | ref        | ref        | ref        | ref        |
| Experiment | –        | –                | ref                   | ref        | ref        | ref        | ref        |
| Case 1  | coarse   | Stage           | 20.8 M                | 1.2431     | -0.2173    | 0.2195     | -0.2240    | -0.2949    |
| Case 2  | medium   | Stage           | 22.7 M                | 1.3812     | -0.1284    | 0.2195     | -0.2489    | -0.3042    |
| Case 3  | fine     | Stage           | 26.8 M                | 1.3352     | -0.1581    | 0.2195     | -0.2489    | -0.2949    |
| Case 4  | fine     | Frozen Rotor    | 98.7 M                | 1.4733     | -0.1680    | 0.2170     | -0.3733    | -0.3840    |

**Figure 3.** Pressure coefficient distribution on the TCF vane at 20% (a) and 80% (b) of span.

#### 4.2. Secondary flow losses

In present section, the secondary flow structures in TCF region are analyzed. Figure 4 presents the visualization of the vortex structures by means of the q-criterion (level 0.001) and the mean flow pathlines in vicinity of the shroud and hub. The blue pathlines, which have been released in the vicinity
of the shroud at the leading edges of swirler vanes, are denoted by A – E symbols (figure 4(a)). The blue pathlines, which have been released near the hub at the leading edges of the swirler vanes, are denoted by H – L symbols (figure 4(b)). Figure 4(a) shows also the pressure- and suction-side legs of the horseshoe vortex at the leading edge of the strut. They are denoted by G and F symbols, respectively. The interaction of the vortex structures, originating from the endwalls of the swirler passages, with the endwall boundary layers in TCF region results in a formation of the streamwise-oriented vortex structures (figure 4). Some vortices impact onto the axially-oriented grooves on the shroud and hub in TCF region (notice that the axially-oriented groves are present due to the technological and assembly reasons). A red circle in figure 4(a) indicates a perturbation of the vortex structure nearby TCF shroud by one of the grooves. The small separation flow region is formed near the shroud as result of this interaction. The suction-side leg of the horseshoe vortex (F symbol) and the streamwise-oriented vortex structures denoted by A and B symbols interact with each other at the outlet from TCF. Somewhat weaker interaction is reported between the pressure-side leg of the horseshoe vortex (denoted by G symbol) at the other streamwise-oriented vortices.

**Figure 4.** Vortex structures in the TCF region near shroud with the main flow directed outward the paper (a) and near hub with the main flow directed towards the paper (b).

Figure 4(b) shows the vortex structures near to the hub. The oncoming boundary layer impacts the leading edge of the TCF vane and starts to roll-up. The pressure- (N symbol) and suction-side (M symbol) legs of the horseshoe vortex are formed. The suction-side leg of the horseshoe vortex (M symbol) is pushed away from the TCF surface. Subsequently, it interacts with the one of the swirler wakes denoted by J symbol. A flow perturbation in the vicinity of one of the grooves on TCF hub, connected with the local flow separation is denoted by orange circle in figure 4(b). Finally, the J and M structures merge and form a single vortex denoted by M+J symbol. The pressure-side leg of the TCF horseshoe vortex (symbol N) interacts with the vortex structure generated behind one of the swirler vane (symbol K). The structures move nearby the pressure side of the strut. After reaching the trailing edge they merge into (N+K) structure which can be observed at the outlet form TCF region. The formation of a strong counter vortex (O symbol) is observed on the suction side of the TCF vane. The structure, aligned with the red pathlines, develops in the TCF wake region. The swirler wakes denoted by L and H symbols travel through the TCF passage without interaction with the strut boundary layer.

The secondary flow losses in the LPT passage is evaluated by means of the loss coefficient expressed by:

$$Y_e' = \frac{2h_{in}}{\rho_{in}U_{in}^3} TP_S,$$  \hspace{1cm} (4)
where $h$ is span, $\rho$ is mean density, $U$ is mean velocity, $T$ is temperature and $P_s$ denotes to the rate of entropy generation per volume unit. Subscript $i_{n}$ corresponds to the values at the inlet to TCF domain. The entropy generation rate in equation (4) is defined by Adeyinka and Naterer [12]:

$$TP_s = \left(\frac{k + k_t}{T} \frac{\partial T}{\partial x_i}\right)^2 + \left(1 + \frac{\mu}{\mu_c}\right) \tau_{ij} \frac{\partial u_i}{\partial x_j},$$

where $k$ and $k_t$ are thermal molecular and turbulent conductivities, $\mu$ and $\mu_c$ are dynamic molecular and turbulent viscosities and $\tau_{ij}$ is the viscous stress tensor.

Figure 5 presents the contour plots of loss coefficient (normalized entropy generation rate) at outlet from TCF (a) and at outlet from the first vane-row (b), denoted by V1 in figure 1. The lines at constant radial distances correspond to 25 and 75% of the span. The tilting of wakes coming from the swirler vanes can be observed in central part of the figure 5 (a). The wake structure which is formed behind the middle swirler vane at the shroud (C symbol in figure 4(b)) is visible on the pressure side of the TCF vane (same C symbol in figure 5(a)). In contrast, the wake structure which is formed behind the same middle swirler vane at the hub (J symbol in figure 4(b)) can be found on the suction side of the TCF vane (M+J symbol in figure 5(a)). A redistribution of the middle swirler wake in circumferential direction is caused by the backward skewed tips of the swirler vanes. An interaction of the suction-side leg of the horseshoe vortex (F symbol) and two vortex structures generated behind the swirler wakes (A and B symbols) results in a high loss production at the trailing edge of the strut, visible in central part of the figure 5(a). The increased value of the loss coefficient which results from interaction of the pressure-side leg of the horseshoe vortex (N symbol) and one of the swirl vortices (K symbol) and the counter vortex (O symbol) can also be noticed at about 25% of span. Note that the high entropy losses occur mostly near to the endwalls.

The secondary flow losses distribution at the outlet from first vane-row (downstream of the TCF outlet) are presented in figure 5(b). The increased losses production is observed in the bended wakes behind the vanes of the first stage. A large patch of high entropy production can also be observed in the central part of the figure between the third and fifth wake from the right. The origin of the patch should be searched in the downstream development of the wake behind the strut.

![Contour plots of the loss coefficient](image)

**Figure 5.** Contour plots of the loss coefficient (eq. 4) at outlet from turbine central frame, TCF (a) and outlet from first vane-row (b).

5. **Conclusions**

The flow through the four-stage low pressure turbine was simulated using the RANS technique with the moving reference frame approach. Satisfactory agreement between experimental and numerical data in terms of the pressure coefficient on surface of the strut at entrance to LPT was reported. Good agreement...
between measured and predicted global flow characteristics (1.5% difference in the mass flow rate) was obtained.

The identification of the vortex structures in the entrance duct (turbine central frame, TCF) to the low pressure turbine was made. The interaction of the streamwise-oriented vortex structures, forming at the endwalls of the swirler passages, with the endwall boundary layers in the TCF region was reported. Analysis revealed that the streamwise-oriented vortex structures interacted strongly with the suction- and pressure-side legs of the horseshoe vortex developing around the inlet strut. The local separation of the endwall boundary layer, caused by impact of the streamwise-oriented vortex structures onto the axially-oriented grooves on hub and shroud in TCF region, was also reported.

The secondary flow losses were evaluated by means of the entropy generation rate coefficient. Spots with high loss production were identified at the outlet from turbine central frame (TCF) and also farther downstream at outlet from first vane-row. A great part of losses were generated in the vicinity of the endwalls. The losses were later redistributed and dissipated across the passage due to interaction of the streamwise-oriented vortex structures with the endwall boundary layers. The interaction of the streamwise-oriented vortices, originating from the swirler passages, with the vortex system along TCF vane resulted in formation of the complex vortex pattern in the wake region behind the TCF vane.

Acknowledgments
Authors acknowledge a support from the research project COOPERNIK financed partly by the Polish National Centre for Research and Development (INNOLOT/I/11/NCBR/2014) and partly by Avio Polska Sp. z o.o. The work was partly supported by the Interdisciplinary Centre for Mathematical and Computational Modelling (ICM) of the University of Warsaw (grant ID GA73-24).

References
[1] Dunham J 1970 A review of cascade data on secondary losses in turbines Journal of Mechanical Engineering Science 12 48–58
[2] Sieverding C H 1985 Recent progress in the understanding of basic aspects of secondary flows in turbine blade passages, Journal of Engineering for Gas Turbines and Power 107 248–257
[3] Langston L S 2001 Secondary flows in axial turbines – a review, Annals of the New York Academy of Sciences 934 11–26
[4] Ligrani P, Potts G and Fatemi A 2017 Endwall aerodynamic losses from turbine components within gas turbine engines, Propulsion and Power Research 6 1–14
[5] Denton J D 1993 Loss mechanisms in turbomachines Journal of Turbomachinery 115 621–656
[6] Harrison S 1990 Secondary loss generation in a linear cascade of high-turning turbine blades Journal of Turbomachinery 112 618–624
[7] Yamamoto A 1989 Endwall flow/loss mechanisms in a linear turbine cascade with blade tip clearance Journal of Turbomachinery 111 264–275
[8] Benner M W, Sjolander S A and Moustapha S H 2005 An empirical prediction method for secondary losses in turbines – part I: A new loss breakdown scheme and Journal of Turbomachinery 128 273–280
[9] Benner M W, Sjolander S A and Moustapha S H 2005 An empirical prediction method for secondary losses in turbines – part II: A new secondary loss correlation Journal of Turbomachinery 128 281–291
[10] Cui J, Nagabhushana Rao V and Tucker P 2016 Numerical investigation of secondary flows in a high-lift low pressure turbine International Journal of Heat and Fluid Flow 63 149–157
[11] Jonak P, Borzęcki T, Konopa M and Kubacki S 2018 Prediction of secondary flow features in a low pressure turbine GT2018-75673 Proc. of ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition GT2018 June 11-15 Oslo Norway 12 pages
[12] Adeyinka O B and Naterer G F 2004 Modeling of entropy production in turbulent flows Journal of Fluids Engineering 126 893–899