HIGH-SPEED AERODYNAMIC INVESTIGATION OF THE MIDSECTION OF A 48” ROTOR BLADE FOR THE LAST STAGE OF STEAM TURBINE

M. Luxa – D. Šimurda
Institute of Thermomechanics AS CR, v.v.i., Prague, Czech Republic
luxa@it.cas.cz, simurda@it.cas.cz

P. Šafařík
Czech Technical University in Prague, Prague, Czech Republic
safarik@fsid.cvut.cz

J. Synáč – B. Rudas
R&D dept., Doosan Skoda Power s.r.o., Pilsen, Czech Republic
jaroslav.synac@doosanskoda.com, bartolomej.rudas@doosanskoda.com

ABSTRACT
A blade cascade representing midspan section of long rotor blading (48”) was investigated experimentally in a high-speed aerodynamic wind tunnel and numerically by means of commercial code. In the experimental part, both optical and pneumatic measurements were performed. Measured data were evaluated and different aerodynamic characteristics were obtained. Velocity characteristics determined inlet parameters namely for subsonic regimes and for aerodynamic choking. Exit angle characteristics provided important information about flow parameters downstream of the blade cascade for comparison of design and off-design conditions. Quality of energy transfer was evaluated from kinetic energy loss coefficient characteristics. Selected results of experiments were compared to 2D numerical simulations. Discussion based on synthesis of relevant results is mainly concerned with influence of high-speed flow field structures and their development on the performance of the blade cascade and study of flow at shock wave/boundary layer interaction on the suction side of the profile.

NOMENCLATURE

\[ \begin{align*}
  a & \quad \text{speed of sound} \quad \text{[m/s]} \\
  a* & \quad \text{speed of sound at crit. condition} \quad \text{[m/s]} \\
  AR=l/b & \quad \text{aspect ratio} \quad [-] \\
  A & \quad \text{area} \quad \text{[m}^2\text{]} \\
  AVDR & \quad \text{axial velocity density ratio} \quad [-] \\
  b & \quad \text{chord} \quad \text{[m]} \\
  c_p & \quad \text{specific heat at const. pressure} \quad \text{[J.kg}^{-1}\cdot\text{K}^{-1}] \\
  F & \quad \text{force} \quad \text{[N]} \\
  l & \quad \text{blade length} \quad \text{[m]} \\
  M & \quad \text{molecular mass} \quad \text{[kg.kmol}^{-1}] \\
  M=v/a & \quad \text{Mach number} \quad [-] \\
  M*=v/a* & \quad \text{dimensionless velocity} \quad [-] \\
  o & \quad \text{throat opening} \quad \text{[m]} \\
  p & \quad \text{pressure} \quad \text{[Pa]} \\
  Re & \quad \text{Reynolds number} \quad [-] \\
  s & \quad \text{pitch} \quad \text{[m]} \\
  S & \quad \text{Sutherlands const.} \quad \text{[K]} \\
  T & \quad \text{Temperature} \quad \text{[K]} \\
  t & \quad \text{thickness, max. th.} \quad \text{[m]} \\
  v & \quad \text{velocity} \quad \text{[m/s]} \\
  \alpha, \beta & \quad \text{flow angles} \quad \text{[deg]} \\
  \gamma & \quad \text{stagger angle} \quad \text{[deg]} \\
  \zeta & \quad \text{loss coefficient} \quad [-] \\
  \eta & \quad \text{dynamic viscosity} \quad \text{[kg.m}^{-1}\cdot\text{s}^{-1}] \\
  \eta & \quad \text{angle of incidence} \quad \text{[deg]} \\
  \kappa & \quad \text{ratio of spec. heats} \quad [-] \\
  \rho & \quad \text{density} \quad \text{[kg.m}^{-3}] \\
  \end{align*} \]
INTRODUCTION

One of the most important energy loss in the steam turbine of large output depends on the magnitude of output kinetic energy downstream the last stage. A simple thermodynamic analysis shows that the optimum value of the output kinetic energy is reached, when the velocity ranges from 150ms\(^{-1}\) to 250ms\(^{-1}\). On the other hand, the steam turbines should be designed in configuration with low number of flows - with respect to compactness and good economical parameters. These requirements result indisputably in increasing of target annulus area downstream the last stage and consequently in increasing of the blades length.

Considering great changes of velocity triangles along the blade height, the large drop of the steam enthalpy, as well as the complicated thermodynamic condition of steam in the last stage of low-pressure cylinder et c., the long rotor blades are typical example of extremely loaded blades. The flow in the last stages rotor row can be classified as a flow of compressible and viscous fluid, which complies with the conception of complex flow field (Bradshaw, 1983).

Investigated profile cascade, which represents midsection of the 48˝(1220mm) long rotor blade, is located on diameter of 3100mm right above the tie-boss (a special type of blade vibrations damping device) – see Fig. 1. Nominal exit Mach number is \(M_{2\text{nom}} = 1.323\), which means the cascade is operated under supercritical flow conditions and the inter-blade channels are aerodynamically choked during the operation. Exit part of the inter-blade channel is relatively large (it results from the cascade geometry) hence the supersonic flow field covers one third of inter-blade channel area approximately.

The data obtained from detailed experiments (including off-design regimes), using mutually independent methods, forms an important basis for designing, studying and operating turbo-machines. They are also essential for verification of various codes of Computational Fluid Dynamic (CFD). The analyses of evaluated experimental data provide important information about the structures of flow field and their origin that have a strong influence on the loss of kinetic energy and on other flow field parameters.

![Figure 1: Rotor blading 48˝](image1.png)

![Figure 2: Scheme of the profile cascade geometry](image2.png)
The experimental and numerical investigation described in this paper follows up the former study of the same cascade (Šimurda et al., 2008), but under better condition of exit flow field periodicity - the adjustable perforated tailboard was used at the exit of the cascade.

**EXPERIMENT**

**Model of profile cascade**

The profile cascade, which represents the mid-span section of the 48´´ long rotor blade, consists of 6 prismatic blades only. The profile chord of the model was chosen \( b = 0.1 \text{m} \), i.e. the reduction scale is 1:2 approximately considering the real blade dimensions. The blade height is \( l = 0.16 \text{m} \), thus the aspect ratio is \( AR = 1.6 \). The model dimensions result from many requirements and restrictions (e.g.: corresponding values of Reynolds number, solidity of blades, dimension of test section, et c.). Two static pressure taps (diameter 0.8mm) are situated in the middle of blade height and profile chord on the blades which form the boundary of the cascade middle channel (one tap “A” is on the pressure side, the other “B” is on the suction side of the neighbouring profile – see Fig. 3).

The basic geometry and characteristic dimensions of the profile cascade are shown in Figure 2 and Table 1, respectively.

**Table 1: Characteristic dimensions of the blade cascade**

<table>
<thead>
<tr>
<th>Pitch/Chord</th>
<th>( s/b )</th>
<th>0.718</th>
<th>Stagger Angle</th>
<th>( \gamma )</th>
<th>54.8°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max. Thickness/Chord</td>
<td>( t/b )</td>
<td>0.13</td>
<td>Trailing Edge Thickness/Chord</td>
<td>( t_{re/b} )</td>
<td>0.0073</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>( AR )</td>
<td>1.6</td>
<td>Design Inlet Flow Angle</td>
<td>( \beta_{1des} )</td>
<td>-29.1°</td>
</tr>
<tr>
<td>Throat Opening/Chord</td>
<td>( o/b )</td>
<td>0.28</td>
<td>Design Incidence</td>
<td>( t_{des} )</td>
<td>0°</td>
</tr>
</tbody>
</table>

All traversing measurements were performed only once, and the repeatability of the measurements was not evaluated. The accuracy of the measuring equipment (traversing) enabled us to measure the kinetic energy loss coefficient \( \zeta \) and exit flow angle \( \beta_2 \) with absolute uncertainties less than 0.0025 and 2°, respectively. The uncertainty of inlet Mach number \( M_1 \) reaches maximum value 0.002. The exit Mach number \( M_2 \) was evaluated with maximum of the uncertainty 0.02. The Reynolds number in real steam turbine is at nominal condition \( Re=5.10^5 \). The value in aerodynamic wind tunnel ranges within interval \( Re \in (1.45.10^6;1.6.10^6) \). The Reynolds number is related to the profile chord \( b \) and the isentropic exit Mach number \( M_{2ls} \).

**Measurement setup**

The experiments were carried out at the Institute of Thermomechanics ASCR, v.v.i. in Prague (Aerodynamic Laboratory in Nový Knín). The wind tunnel for 2D cascade measurement is of a suction type.

The test section is supplemented by adjustable perforated tailboard, that helps to keep the periodicity of exit flow field (Luxa et al., 2012). The open-area ratio is 0.5. The perforated tailboard was mounted next to the lateral profile whose suction side is outer suction side.
of the whole cascade, see Fig. 3. This way of tailboard design prevents the origin of the free jet boundary and at the same time prevents the reflection of the system of exit shock waves back to the exit flow field. The value of angle between perforated tailboard and the plane of trailing edges was set according to the simple “empirical cosine rule”, used for predicting of fluid outlet angle in design of turbomachines (e.g. Dixon, 1998). Application of this method is possible only in the case of exit flow field conditions under the limit load regime (Šafařík et al., 2012).

Optical and pneumatic measurements
The mutually independent methods (optical and pneumatic) were used in described investigation.

Optical measurements (interferometry – method of infinite fringe, schlieren method in Toepler configuration, colour schlieren method) were performed in the range of isoentropic exit Mach numbers of \(0.897 \leq M_{2is} \leq 1.535\) and the incidence angles \(-30^\circ \leq \iota \leq +30^\circ\). The isoentropic Mach number distribution \(M_{is}\) along the profile was evaluated from interferograms. The schlieren pictures together with interferograms provide excellent information about the flow field structures, particularly in transonic and supersonic area.

Pneumatic measurement included mainly 2D traversing measurement downstream the cascade. The conical five-hole probe moves continuously along the straight path in the middle of blade height 22mm distant from the trailing edges. The traversing probe is automatically positioned into the direction of exit velocity vector. The length of traversing trajectory was in these experiments 160\% of the cascade pitch. Traversing was carried out practically in the same range of velocities and incidence angles as in the case of optical measurements, i.e. \(0.698 \leq M_{2is} \leq 1.515\) and \(30^\circ \leq \iota \leq +30^\circ\). The loss coefficient \(\zeta\) defined by relation (1) and exit flow reference values were evaluated using the data reduction method, which is based on solving all the conservation laws (mass, momentum, energy), the equation of state for an ideal gas and the condition of adiabatic flow (Amecke, Šafařík, 1995).

\[
\zeta = 1 - \frac{M_{2is}^2}{M_{2is}^*}
\]  

(1)

NUMERICAL SIMULATION
The CFD calculations of selected regimes were performed using commercial code Ansys Fluent, version 13.0. The Fluent code solves the system of governing equations numerically using the finite volume method.

2D model of the investigated geometry was generated in the preprocessor GridPro, where the calculation domain was meshed with quadrilateral cells. The computational domain included areas of flow field one pitch upstream and downstream the cascade approximately. The domain is shown in the Fig. 4 (the evaluation plane is marked downstream the cascade). Grid quality was measured by equiangle skew. The 2D grid consisted of 113 850 cells. Only 0.53\% cells exhibited skew value over 0.5, while the maximal value of skew was 0.68. The mesh was adapted three times during calculations in the regions of shock waves and wake downstream the trailing edges. Detail of the cells around the trailing edge after the third adaptation is shown in Fig. 5. The grid adaption was based on pressure gradient. In the near wall region, the original mesh was find enough \((y^+ < 2)\) to allow use of a low-Reynolds turbulence model and grid adaption was not necessary. The number of elements after adaption was 466773. If the number of cells increases about 119\%, the loss coefficient value will change by about 0.34\%. The numerical results show, that the changes of this coefficient are much more sensitive to the position of evaluation plane downstream the cascade than to the number of cells. The shift of evaluation plane by a half of pitch causes the change of the loss coefficient by about 22.3\%. The sensitivity of the value of the exit angle in relation to the mesh resolution is significantly lower (by about 0.07\% of the exit angle value).

The problem was solved using the density based implicit solver with Menter’s \(k-\omega\) SST model of turbulence. The flowing medium was dry air, which was considered as an ideal gas, and its
characteristics are presented in the Table 2.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$</td>
<td>kg.m$^{-3}$</td>
<td>from ideal gas equation</td>
</tr>
<tr>
<td>$\varepsilon_p$</td>
<td>J.kg$^{-1}$.K$^{-1}$</td>
<td>1006.43</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>W.m$^{-1}$.K$^{-1}$</td>
<td>0.0242</td>
</tr>
<tr>
<td>$\eta$</td>
<td>kg.m$^{-1}$.s$^{-1}$</td>
<td>1.716*10$^{-5}$</td>
</tr>
<tr>
<td>$\theta$</td>
<td>[K]</td>
<td>273.11</td>
</tr>
<tr>
<td>$\ell$</td>
<td>[K]</td>
<td>110.56</td>
</tr>
<tr>
<td>$M$</td>
<td>kg.kmol$^{-1}$</td>
<td>28.966</td>
</tr>
<tr>
<td>$Re$</td>
<td>[-]</td>
<td>(1.46 ÷1.51).10$^b$</td>
</tr>
</tbody>
</table>

The inlet boundary conditions were determined by constant total pressure, total temperature and inlet flow angle. The outlet boundary condition was defined as a constant static pressure. Periodicity conditions were applied on corresponding free side boundaries of the computation domain. Non-reflecting boundary condition was applied where appropriate.

RESULTS AND DISCUSSION

Aerodynamic characteristics of the cascade

Aerodynamic characteristics of the tested cascade, i.e. velocity characteristics, and loss coefficient $\zeta$ characteristics are shown in Fig. 6 and 7, respectively. It can be seen in Fig. 6 that the flow past the cascade experiences aerodynamic choking at isentropic exit Mach numbers $M_{2is}>0.9$. Beyond this point, the mass flow through the blade cascade remains constant despite further decrease of the backpressure. At design incidence and exit flow velocity, the inlet Mach number $M_1$ reaches value of $M_1 = 0.29$. Due to development of the wind tunnel sidewall boundary layer and corresponding change of the cross-section ($AVDR = 0.954$), $M_1$ is somewhat higher than the value according to the 1D theory, i.e. $M_{1\text{teor}} = 0.27$ (see Fig. 8).
Specific slight variation of the inlet Mach number beyond $M_{2is} = 0.9$ can be observed mainly in the case of incidence angle $\iota = -30^\circ$. The reason of these variations consists in modifications of the effective interblade channel as a result of the flow separation on the pressure side of the blade (Fig. 9). The extent of the separation region changes under different conditions concerning the outlet/inlet pressure ratio.

Aerodynamically smooth profiles of the midsection form nearly straight interblade channels, therefore velocity gradient across the channel at the exit is small and losses of the kinetic energy represented by the loss coefficient $\zeta$ are very low around 0.03 in wide ranges of both exit velocities and angles of incidence (Fig. 7). The blade cascade is not sensitive to the angle of incidence. The only exception are regimes at incidence angle $\iota = -30^\circ$. This is due to local flow separation on the pressure side (mentioned above) at all investigated isentropic exit Mach numbers. At nominal regime the blade cascade exhibit loss coefficient $\zeta = 0.03$.

**Interblade channel**

Typical structures of interblade flow field are shown in interferogram taken at $M_{2is} = 1.325$ and $\iota = -10^\circ$ (Fig. 10). The shapes of the fringes clearly show local region of a decelerated flow nearby the profile’s suction side in the supersonic part of the flow field. This region of recompression (or “supersonic compression at transonic expansion”) (Šťastný, Šafařík, 1990) takes place close to the sonic line where the curved surface of the suction side changes into straight surface. A sudden change of the surface curvature results in a reflection of the compression characteristics. The expansion characteristics coming from the surface convex part reflect from the sonic line as compression characteristic. These reflected characteristics

**Figure 7:** Dependence of the loss coefficient $\zeta$ on the isoentropic exit Mach number $M_{2is}$ and the angle of incidence $\iota$.

**Figure 8:** Comparison of experimental and theoretical dependences of the inlet Mach number $M_1$ on the angle of incidence $\iota$. The profile cascade is choked.

**Figure 9:** Interferogram taken at $M_{2is} = 1.325$ and $\iota = -30^\circ$, flow separation on the pressure side.
reflect again, this time from the straight surface, keeping their compression character. The region of compression occurs between the point of change in the curvature and the first neutral characteristic. Hence, the compression

region extent depends on the sonic line shape. Origin of this phenomenon is intentional and its function is to promote transition of the suction surface boundary layer, which in turn should be able to withstand pressure jump imposed by the internal branch of exit shock wave. This shock wave first appears at \( M_{2is} \sim 0.9 \) as a normal shock wave. With the increasing exit Mach number the shock wave develops into an oblique shock and it reflects from the suction side of the neighbouring blade. As the exit Mach number \( M_{2is} \) continues to increase, the exit shocks obliqueness grows and the point of its reflection shifts downstream. Interaction of the boundary layer with the internal branch of the exit shock wave is also apparent in the Fig. 10. It is a strong interaction since a local flow separation takes place. Judging by the extent of the region of separated flow (e.g. Délyery, 1985; Dvořák, 1987), i.e. distance of the two reflected shocks (A and C in Fig. 10), the boundary layer is laminar thus the recompression does not promote transition as intended. The boundary layer transition prior to the shock wave/boundary layer interaction was observed at incidence angles \( t > +15^\circ \) (see Fig. 11).

**Exit flow angle**

In general, the values of exit flow angle correspond to the cascade geometry till the isentropic exit Mach number \( M_{2is} \sim 1.0 \). Above this value supersonic deflection takes place and the exit flow angle depends

---

**Figure 10:** Interferogram taken at \( M_{2is}=1.317 \) and \( t = -10^\circ \) showing the recompression and laminar shock wave/boundary layer interaction (IS - incident shock, A – 1. reflected shock, B – expansion, C – 2. reflected shock)

**Figure 11:** Interferogram taken at \( M_{2is}=1.297 \) and \( t = +30^\circ \) showing the recompression and turbulent shock wave/boundary layer interaction (A – reflected shock)

**Figure 12:** Dependence of the exit flow angle \( \beta_2 \) on the isentropic exit Mach number \( M_{2is} \) and the angle of incidence \( t \)
on the velocity conditions on both the pressure and suction sides near the trailing edge. The values of exit flow angle obtained from experimental data (Fig. 12) confirm these assumptions. The small unusual increase of the exit flow angle at the exit flow conditions near the $M_{2is} \sim 1.3$ is noteworthy.

**Aerodynamic forces**

The dependencies of non-dimensional aerodynamic forces on isoentropic exit Mach number $M_{2is}$ for constant incidences $\iota$ are shown in Fig. 13. Forces are evaluated from interferograms. The methods is based on the principle of the interferometric methods (e.g. Šafařík, 2000), the fringes in the flow field represent lines of constant index of refraction and on assumption of isoentropic changes in the main flow and constant static pressure across the boundary layer. Aerodynamic force is solved by integration of a distribution of static pressure along pressure and suction sides in peripheral direction. Obtained data are related to the possible maximum of forces $A_{p0l}$.

The diagram shows clearly the force dependence on incidence angle. The force is reduced at negative incidences and – contrarily - the profile is overloaded at positive angles. Considering the range of measured isoentropic exit Mach number $M_{2is}$, the changes of non-dimensional force are not very significant in relation to the changes of Mach number downstream the cascade once the interblade channel is aerodynamically choked. Further, the dependence is constant when the limit load regime is reached, i.e. approximately $M_{2lim} = 1.8$ in this profile cascade. (Šafařík et al., 2012).

**Comparison of experimental data and numerical simulations**

Figures 14 and 15 show comparison of distribution of $M_{is}$ along profiles at design incidence and two supersonic exit velocities. It can be seen that results of CFD correspond reasonably well to the data obtained experimentally. The departure of the two distributions behind the shock/suction side interaction ($x/c_{is} = 0.75$ and 0.8 respectively) is probably caused by the fact that during process of evaluating interferograms, it is difficult to accurately count fringes through the shock wave. Also values of $M_{is}$ behind the shock wave are evaluated with respect to the inlet total pressure. Position of the interaction is predicted well. Comparison of reference values of the loss coefficient $\zeta$ (Fig. 16) and the exit flow angle $\beta_2$ (Fig. 17) evaluated from distributions $p(y)$, $p_0(y)$, and $\beta_2(y)$ at the traversing plane also exhibit good agreement between CFD and experimental data.

An attempt was made to numerically simulate the regime at design exit Mach number and incidence $\iota = -30^\circ$. It was already mentioned that at this incidence, flow separation on the pressure side took place at all investigated exit velocities. However, at standard setup of the solver (with $sst k-\omega$ turbulence model) described in the paragraph 3, simulations predicted no flow separation at all.
Therefore, other models of turbulence were applied in order to catch the flow separation. Transition \( k-\omega \) SST model predicted no separation either (Fig. 18). Transition \( k-kl-\omega \) model predicted small region of separated flow (Fig. 19), whose extent was not comparable to that in Fig. 9. RSM model also predicted flow separation at some stage of the calculation but it was difficult to get converged solution. This was probably due to unsteadiness in the separated flow region. Therefore, to successfully apply RSM model, an unsteady solution should be considered.

**CONCLUSIONS**

Application of mutually independent methods (optical methods, pneumatic methods and CFD methods) is important for complex research on compressible fluid flow past blade cascades. The experimental and numerical investigation of a blade cascade representing midspan section of long rotor blading (48") provided information about important cascade aerodynamic characteristics in wide range of flow regimes including off design conditions.

Aerodynamically smooth profiles of the midsection form nearly straight interblade channels, therefore values of the loss coefficient \( \zeta \) are very low around 0.03 in wide ranges of both exit velocities and angles of incidence. Flow structures that significantly damage the quality of the flow did not appear until incidence \( \iota = -30^\circ \), which is far off design.

The results of CFD calculations exhibit a good agreement with experimental data under nearly
design conditions. However, turbulence models available in the used commercial code are not capable to capture much more complex flow field at regimes with flow separation.

**Figure 18:** Distribution of refraction index (transition \( k-\omega \) SST turbulence model)

**Figure 19:** Distribution of refraction index (transition \( k-kl-\omega \) turbulence model)

ACKNOWLEDGEMENTS

Financial support from the Czech Science Foundation, Grant No. GAP 101/08/1329, is gratefully acknowledged.

REFERENCES


