SECONDARY FLOW MITIGATION IN TURBINE VANES USING ENDWALL FENCE OPTIMIZATION

Giacomo Mingardo  
Avio Aero  
Giacomo.mingardo@avioaero.it  
Turin, Italy

Kakeru Kusano  
Osaka Institute of Technology  
m1m16406@st.oit.ac.jp  
Osaka, Japan

Matteo Pini  
Delft University of Technology  
M.Pini@tudelft.nl  
Delft, Netherlands

Claudio Lettieri  
Delft University of Technology  
C.Lettieri-1@tudelft.nl  
Delft, Netherlands

Yutaka Kawata  
Osaka Institute of Technology  
yutaka.kawata@oit.ac.jp  
Osaka, Japan

ABSTRACT

Secondary flow is a major source of aerodynamic losses in turbines. The mitigation of these losses is critical to enable efficient turbine designs, but the suppression of secondary flow formation mechanisms is challenging due to the complex three-dimensional nature of the flow field. One solution is the use of three-dimensional endwall fence, to reduce secondary flow losses. On the one hand, fences can mitigate secondary flow penetration, reduce the kinetic energy of the crossflow and decrease the mixing out of secondary vortices downstream of the blade passage. On the other hand, the fence addition leads to an increment of wetted area and viscous losses. The design of such devices must be tailored to the specific turbine and is usually obtained via lengthy and time-consuming experimental campaigns. Shape optimization can greatly enhance the design process of these components as it provides a way to investigate the fluid dynamic performance of a multitude of fence geometries at affordable computational cost. This paper presents a shape optimization framework for endwall fences in steam turbines. A surrogate-based optimization is applied to minimize the total pressure losses of a steam turbine cascade. The results of the numerical analysis indicate a reduction of 48% of the secondary kinetic energy leading to a decrease of 1.96% in total pressure loss of the optimized design with respect to the empty cascade geometry. The improvement is due to the partition of the passage vorticity; the passage crossflow is reduced and the interaction between the passage vortex and the counter vortex on the blade suction surface is weakened.

INTRODUCTION

Secondary flows account for nearly 30-50% of the overall fluid-dynamic losses in low aspect ratio turbine cascades [1-6]. The mitigation of these losses is crucial to achieve significant increments in turbine efficiency. The physical mechanism of formation of secondary flows has been extensively studied through linear cascade models, as the one shown in Figure 1 [7]. The vortex lines of the inlet boundary layer wrap around the blade leading edge, forming two vortex structures that are convected downstream the flow passage. The left vortex develops in the so-called passage vortex and is pushed towards the suction surface by the pitchwise pressure gradient. The right vortex, usually referred to as counter-vortex, remains attached to the suction surface. The interaction between these two vortices causes boundary layer separation.
as well as the creation of a large region of low momentum flow that gradually mixes with the mainstream, leading to increased mixing losses. Corner vortices have been also observed forming at the junctions between the blade surfaces and the endwall [8], as result of cross-flow separation. The resulting vortex system furthermore interacts with the shed vorticity arising from the blade trailing edge [9].

The mitigation of performance penalties induced by secondary flow is key to improve turbine efficiency. Several mitigation strategies have been proposed to meet this objective, including endwall contouring [10] and unload endwall blade sections via 3D blade design [11]. Endwall contouring consists in modifying the shape of the endwalls or adding 3D structures called fences positioned inside the flow passage [2], as shown in Figure 2. The physical principle of the fence is to partition the streamwise vorticity, thus reducing the associated mixing losses by reducing the amount of secondary kinetic energy generated at the exit of the cascade. This comes at the expense of higher viscous losses and flow blockage that can overcome the reduction of mixing losses.

Several authors reported about the fluid-dynamic benefits of fences. Kawai et al. [3] provided the first experimental results, showing a loss reduction of 25% as compared to a baseline configuration without fence. The optimized fenced cascade was located in the middle of the bladed channel and featured a height of 1/3 of the inlet boundary layer thickness \( \delta_{BL} \), while the camber line retained the same shape of the blade camber. By using a fence to block the vortex migration to the suction side, Chung and Simon [11] observed a reduction of the separated area on the blade suction surface which led to about 17% reduction in total pressure loss coefficient [12]. In this case, the optimal fence shape featured a triangular cross-section with a sharp leading edge and was positioned along the centerline of the channel. Aunapu et al. [13] continued the work of Chung and Simon by investigating the impact of shortening the optimal fence shape. Despite the shortened fence reduced the secondary kinetic energy by 30%, it also increased viscous losses, resulting in 30% increment of total pressure loss coefficient. Kumar and Govardhan [14] adapted the fence design through systematic CFD calculations, finding that a linear fence height distribution is convenient to minimize the extra viscous losses. Previous works demonstrated the effectiveness of fences in mitigating secondary flow loss, but the analysis focused on a limited number of fence configurations. There is no clear guideline on how to design effective endwall contouring. Clark et al. [5] applied automatic design optimization to the design of splitter blades in low aspect ratio, identifying the key physical mechanisms for loss and indicating strategies to improve turbine performance.

**SCOPE OF THE PAPER**

From the large body of work on fence design is apparent that the detailed shape of these devices must to be tailored to the target application. The shape of the fence is arguably function of the inlet boundary layer thickness \( \delta_{BL} \), the boundary layer state, the loading, and the aspect ratio the turbine cascade. Moreover, for a given application, one can consider to modify not only the pitchwise position, height and length, as was investigated in previous work, but also to change the camber line shape, the streamwise position, and the thickness distribution from leading to trailing edge. The optimal design of the fence is therefore a complex endeavour difficult to accomplish experimentally or through parametric CFD calculations. In this paper, we resort to CFD-based optimization to address this problem. More in particular, we devise a numerical methodology to automate endwall fence design and to assess the performance through CFD. In this study, our objectives are i) to demonstrate the feasibility of a robust and flexible optimization tool for fence automated design ii) to assess the potential offered by the design methodology by performing a fence design optimization for an exemplary turbine vane, and iii) to gain insight of the physical reasons at the basis of the reduction of fluid-dynamic penalties due to secondary flows in turbine vanes.

![Figure 2. Passage crossing of the horseshoe vortex with and without fence, from [12].](image)

![Figure 3. Block scheme of the optimization process.](image)
NUMERICAL METHODOLOGY

The automated design methodology has been developed by resorting to surrogate-based CFD optimization [15]. The surrogate model is constructed by fitting the total pressure loss of the turbine cascade as function of fence design parameters. To guarantee a wide design space, the fence geometry is modelled with 13 parameters: the pitchwise and streamwise position, the camber line shape, the thickness and the height distributions of the fence. The fitting surface is created by training a support-vector machine using a database of 100 samples. The objective function of each sample point is evaluated through a turbulent CFD simulation, where turbulence effects are modelled via the SST k-ω turbulence closure. All simulations were run in Ansys CFX 17.1 [16] using a high resolution advection scheme [16]). The simulation were considered converged when the RMS values of the residuals reached values below 10⁻⁶. The numerical methodology proposed is represented in the block diagram displayed in Figure 3.

The entire optimization process can be summarized as follows: based on the input design parameters and imposed bounds, the design of experiment creates a set of blade cascade configurations. To ensure optimal coverage of the design space, the Latin Hypercube sampling method is used [16]. The fluid-dynamic performance of the different cascade geometries is evaluated by means of CFD calculation. The response surface is then generated by fitting the computed cost functions. Finally, the optimization is performed. Once the optimal point is identified on the response surface, it is verified through CFD simulations: if the value of the objective function provided by the surrogate model is within a given accuracy, the design process is terminated. More precisely, if the value of the loss coefficient differs by more than 0.5 percentage points as compared to the one given by the original CFD model, a new response surface is generated by including the evaluation point into the training dataset. This process is repeated until the objective function value predicted by the surrogate model satisfies the accuracy criterion.

The global optimum is obtained through a hybrid method: initially, the response surface is randomly screened [18], to identify the region where the global optimum is located. A gradient-based optimization based on the NLPQL algorithm [19] is eventually used to reach the optimum.

Mesh adaptation algorithm

Rapid and robust 3D structured mesh generation of the turbine flow passage including the fence is obtained automatically using Ansys ICEM [17]. The mesh is adapted to the geometry modifications by associating the mesh topology to the blade parametric curves. In this process, the mesh density is maintained constant by means of automatic local refinement. The method yield high quality meshes even in presence of large geometry deformations, as illustrated in Figure 4.

CASE STUDY

Computational setting and definition of the design problem

The cascade considered in work is a prismatic linear cascade representative of a steam turbine vane. The detailed geometry can be found in [2]. The relevant geometrical features are reported in Tab. 1. Simulations are conducted on a single passage.

![Figure 4 Example of mesh deformation in presence of large displacements of the fence geometry](image)

<table>
<thead>
<tr>
<th>Inlet blade angle</th>
<th>21.4°</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet blade angle</td>
<td>-68.2°</td>
</tr>
<tr>
<td>Thickness-to-chord ratio</td>
<td>0.171</td>
</tr>
<tr>
<td>Aspect ratio</td>
<td>2.6</td>
</tr>
<tr>
<td>Solidity</td>
<td>0.637</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>7*10⁴</td>
</tr>
</tbody>
</table>

Table 1 - Design parameters of the empty cascade

The fluid-dynamic performance of the empty cascade has been measured in the low-speed air wind-tunnel at the Osaka Institute of Technology. Although compressible flow effects are known to affect the magnitude of secondary flows and mixing losses in actual steam turbine cascades [1], the physical mechanisms responsible of their generation and mitigation through fences are deemed to be same [19]. Dynamic similarity with the experimental cascade is maintained by carrying out simulations at the same the characteristic Reynolds number. The inlet spanwise velocity distribution has been extracted from the wind tunnel experiments at a distance of one blade chord upstream of the cascade and imposed as inlet boundary condition for the CFD simulations. The turbulence level has been set to 10% to simulate the flow conditions in real cascade applications [12]. The computational domain consists of the flow passage in

1 *Reynolds number is based on the inlet free-stream velocity and blade chord.
which the baseline fence is positioned along the centreline of the channel, as shown in Figure 8. The baseline fence has been developed using best practices at the Osaka Institute of Technology [4]. At the outlet boundary, it is imposed an average static atmospheric back pressure. The outlet plane is located at a distance of six blade pitches downstream of the trailing edge to ensure complete mixing. The objective function to be minimized is the total pressure loss coefficient of the cascade, defined as:

\[ C_{\text{loss}} = \frac{\bar{P}_{T,\text{inlet}} - \bar{P}_{T,\text{outlet}}}{\bar{P}_{T,\text{inlet}} - \bar{P}_{S,\text{outlet}}} \]  

(1)

where the averaged flow quantities are determined through mass-flow averaging at the inlet and the outlet boundary plane.

Figure 6 shows the trend of the \( C_{\text{loss}} \) as function of the mesh elements. A mesh size of 3.5 million elements per passage provided grid-independent results and was then retained for the optimization. For the whole set of generated grids, the mesh elements have been clustered to guarantee \( y^+ \approx 1 \).

**Fence parameterization**

The 13 design parameters of the fence are shown in Figure 7.

The fence pitchwise \((x_1)\) and streamwise \((x_2)\) positions refer to the coordinates of the leading-edge point, which lays in the plane of the endwall. The camber line has been modelled as a quadratic Bézier curve, parametrizing the relative position of the control points with respect to the position of the fence leading-edge. The parameters \(x_3\) and \(x_9\) enable changes of the camberline shape, while \(x_4\) allows to move the trailing edge in pitchwise direction. The parameter controlling the length \((x_6)\) moves both the points in such a way to maintain the camber line shape when the fence is lengthened. The thickness distribution is determined by interpolating splines imposing the fence thickness in three different locations: leading edge \((x_7)\), trailing edge \((x_8)\) and half camber line \((x_9)\). Finally, the height distribution is constructed by means of splines, in turn controlled by the vertical positions \((x_{10} - x_{13})\) of four points, as shown in Fig. 4 (d).

Figure 9 shows the trend of the \( C_{\text{loss}} \) as function of the mesh elements. A mesh size of 3.5 million elements per passage provided grid-independent results and was then retained for the optimization. For the whole set of generated grids, the mesh elements have been clustered to guarantee \( y^+ \approx 1 \).

The use of thirteen parameters gives, on the one hand, a high design freedom, but it requires a large number of samples to construct the response surface. This work aims also at identifying the most influencing parameters.
OPTIMIZATION RESULTS

The optimization was run in parallel on 16 Intel-Xeon E7 cores and lasted about 20 hours. Figures 8-9 show the shape of the baseline and the optimal fence configuration. The optimal fence leads to a reduction of $C_{\text{loss}}$ of about 1.96% as compared to the fluid-dynamic performance of both the empty and the baseline cascade with fence. By comparing the two fence configurations, it can be noted that both the fence geometry and its position in the bladed channel are significantly different. Experiments demonstrated that the baseline fence yields loss identical to that of the empty cascade, which is taken as comparison term in the present analysis. The main geometrical dimensions of the optimal fence are listed in Table 2.

![Figure 8 Pitchwise and streamwise position of the baseline and optimal fence.](image)

| Pitchwise position, $x_1$ [% of $x$] | 35% |
| Streamwise position, $x_2$ [% of $a$] | 33% |
| Chord length, $x_4$ and $x_6$ [% of $c_b$] | 52% |
| LE thickness, $x_7$ [mm] | 0.2 |
| Half-chord thickness, $x_9$ [mm] | 1 |
| TE thickness, $x_0$ [mm] | 0.75 |
| 1st height, $x_{10}$ [% of $h_{BL}$] | 60% |
| 2nd height, $x_{11}$ [% of $h_{BL}$] | 21% |
| 3rd height, $x_{12}$ [% of $h_{BL}$] | 19% |
| 4th height, $x_{13}$ [% of $h_{BL}$] | 0% |

Table 2 Main geometrical parameters of the optimal fence configuration.

As opposed to the initial fence design, which is placed in the middle of the flow passage, the optimal pitch-wise position of the optimal fence configuration is closer to the pressure side of the adjacent blade, along the trajectory of the passage vortex. This suggests the need of partitioning the total streamwise vorticity in smaller vortex structures, each one containing a fraction of the global vorticity, in the attempt to reduce the amount of secondary kinetic energy downstream of the cascade. These results are consistent with the work of Clark et al. [3]. The optimal height distribution reveals that major fluid-dynamic benefits are achieved by adopting a sharp leading edge and by placing the maximum height of the fence at approximately 20% of the camberline length. Moreover, the maximum fence height is found to be about 60% of the inlet boundary layer thickness, higher than the value of one third indicated in previous studies.

Figure 10 shows the streamwise evolution of the total pressure loss coefficient from the blade TE to the domain outlet for the cascade without fence and for the cascade equipped with the optimal fence. At $Y/s = 0$, namely at a section corresponding to the blade trailing-edge, the loss coefficient is the same for both configurations. This indicates that the additional profile losses introduced by the fence are negligible for the case at hand. The reduction of total pressure losses becomes apparent far downstream of the trailing-edge, at the section corresponding to $Y/s = 6$, where the flow undergoes complete mixing. The reason thereof is arguably the decreased secondary kinetic energy downstream of the vane, which reduces the entropy generation occurring during the dissipation of the vortex structures with the bulk flow. The associated streamwise distribution of the secondary kinetic energy coefficient $C_{\text{SKE}}$ is reported in Figure 11 between a section upstream the fence leading edge, $Y/s = -2.1$, and the outlet domain, $Y/s = 6$. The coefficient is defined as
\[ C_{SKE} = \frac{1}{2} \rho \bar{V}_{sec}^2 \frac{\bar{T}_s}{\bar{T}_{inlet} - \bar{T}_{outlet}}. \] (2)

The value of \( C_{SKE} \) in the two cases is equal upstream of the fence, but the subsequent production and peak value are then strongly attenuated by the fence. As can be observed, in this case up to 50% reduction of secondary kinetic energy at the blade trailing-edge. This seems to suggest that the best improvement in fluid-dynamic performance can be obtained by positioning the fence leading edge prior the peak value of SKE, while having the remaining body of the fence that extends in the region where SKE reaches the highest values for the cascade without fence. The physical reason leading to a reduction of the secondary kinetic energy can be better elucidated by looking at trends in Figures 12, 13, and 14.

Figure 11 Streamwise evolution of the secondary kinetic energy for the empty and the optimized fenced cascade.

Figure 12 Spanwise distribution of the total pressure loss coefficient for three different streamwise sections. Empty cascade (red line), optimized fenced cascade (blue line).

induced by the mixing of the fence wake close to the endwall. The dampening of the passage vortex intensity becomes beneficial at higher spanwise locations to attenuate the peak of total pressure losses. This is shown in Figure 13, which shows the planar distribution of \( C_{SKE} \) at \( Y/s = 0 \).

Figure 13 Comparison of \( C_{SKE} \) distribution at the blade TE plane between the empty and the optimized fenced cascade.

Figure 14 Vorticity field at the hub endwall hub for both the empty and the optimized fenced cascade.
suppression of the separation bubble occurring on the blade suction side, which can be also detected by the contour of the wall shear distribution over the blade suction surface reported in Figure 15.

![Wall Shear Comparison](image)

*Figure 15 Comparison of wall shear stress distribution along the blade suction side for the empty and the optimized fenced cascade.*

**CONCLUSIONS**

A fully automated CFD-based methodology for secondary flow mitigation through fence design optimization is documented in this paper. The method was successfully applied to the reduce the total pressure losses of a linear steam turbine cascades. The final fence shape is achieved by scanning a wide design space through surrogate-based CFD optimization using 13 fence design parameters.

Based on the results of this study, the following conclusions can be drawn:

- The optimized fence meets the objective to mitigate secondary flows, while keeping the additional profile losses at minimum.
- The performance of the cascade equipped with the optimized fence has been improved by 1.96% as compared to both the empty and the baseline fenced cascade.
- For the case study, the secondary kinetic energy coefficient has been reduced by 48%.
- The fence splits the inlet boundary layer vorticity mitigating the growth of the passage vortex. The net result is the reduction of dissipation caused by the mixing process downstream of the cascade.

The numerical framework developed in this study will be instrumental to develop new fence concepts for targeted turbine applications and to reveal trends via systematic design optimizations.

**NOMENCLATURE**

- \(a\) = blade axial length
- \(CFD\) = computational fluid dynamics
- \(ch\) = blade chord
- \(C_{loss}\) = total pressure loss coefficient
- \(C_{SKE}\) = secondary kinetic energy coefficient
- \(DOE\) = design of experiments
- \(DP\) = design point
- \(H\) = blade span
- \(LHS\) = Latin hypercube sampling
- \(LE\) = leading edge
- \(NLPQL\) = Non-Linear Programming by Quadratic Lagrangian
- \(\overline{p}_{T,\text{inlet}}\) = mass flow average of inlet total pressure
- \(\overline{p}_{S,\text{outlet}}\) = mass flow average of outlet static pressure
- \(\overline{p}_{T,\text{outlet}}\) = mass flow average of outlet static pressure
- \(OSF\) = optimal space filling
- \(RANS\) = Reynolds averaged Navier – Stokes
- \(RMS\) = root mean square
- \(s\) = blade pitch
- \(SST\) = shear stress transport
- \(SKE\) = secondary kinetic energy
- \(TE\) = trailing edge
- \(V_{sec}\) = secondary component of velocity
- \(Y\) = streamwise coordinate
- \(Z\) = spanwise coordinate
- \(\delta_{BL}\) = boundary layer thickness
- \(\rho\) = density

**REFERENCES**


