Numerical Investigation of the Effect of Geometric Design Parameters on Swirl Brake Performance

Neil R. Matula* and Paul G. A. Cizmas†

Texas A&M University, College Station, TX, 77843-3141, United States

This paper presents a numerical investigation into the three-dimensional, compressible, turbulent flow around a swirl vane, as well as the effects of vane design on seal inlet swirl. Computations were carried out using an in-house Reynolds-averaged Navier–Stokes solver with a Shear Stress Transport turbulence model. The effects of vane count, stagger angle, chord length, and thickness on seal inlet swirl were investigated. Finally, the results are used to provide recommendations on swirl vane design improvement.

Nomenclature

\begin{align*}
E & \quad \text{Specific energy} \\
F_c & \quad \text{Convective flux tensor} \\
F_v & \quad \text{Viscous flux tensor} \\
G & \quad \text{Vector of source terms} \\
H & \quad \text{Total specific enthalpy} \\
I & \quad \text{Identity matrix} \\
k & \quad \text{Thermal conductivity} \\
n & \quad \text{Normal vector} \\
N_e & \quad \text{Number of edges} \\
p & \quad \text{Pressure} \\
Q & \quad \text{Vector of state variables} \\
Q_T & \quad \text{Vector of turbulent state variables} \\
R & \quad \text{Hub radius} \\
S & \quad \text{Area} \\
t & \quad \text{Time} \\
V & \quad \text{Volume} \\
V_\theta & \quad \text{Tangential velocity} \\
v & \quad \text{Velocity} \\
\gamma & \quad \text{Stagger angle} \\
\zeta & \quad \text{Swirl} \\
\kappa & \quad \text{Turbulent kinetic energy} \\
\mu & \quad \text{Dynamic viscosity} \\
\rho & \quad \text{Density} \\
\tau & \quad \text{Viscous stress tensor} \\
\Omega & \quad \text{Control volume} \\
\omega & \quad \text{Turbulent specific dissipation rate} \\
\omega & \quad \text{Wheel speed}
\end{align*}

Subscripts

\begin{align*}
i & \quad \text{Node number} \\
j & \quad \text{Edge number}
\end{align*}

*Graduate Researcher, Aerospace Engineering, College Station
†Professor, Aerospace Engineering, College Station, Associate Fellow
I. Introduction

Studies conducted on labyrinth seals have suggested that the circumferential velocity component of the flow entering the seal has a substantial effect on rotodynamic stability.\(^1\) In particular, swirl vanes that attempt to turn the flow against the direction of shaft rotation have been demonstrated to reduce the cross-coupled stiffness coefficients, thereby improving stability. Much interest has been shown in determining swirl brake configurations that function well over a variety of operating conditions.\(^2\) However, the nature of the swirl vane’s effect on the flow is not well understood. Nielsen et al.\(^3\) simulated the flow around several swirl vane configurations, and found that the core mechanism for their operation was the large region of separated flow between the vanes. However, the simulations in that work used wall functions to model the boundary layer, which may have adverse effects on accuracy for heavily separated flows. Baldassarre et al.\(^4\) examined the effects of several geometric design parameters on swirl brake performance, including vane chord length, pitch, and span. However, the effects of vane stagger angle and thickness were not evaluated.

This paper presents a numerical investigation into the effects of vane count, stagger angle, thickness, and chord length on seal inlet swirl. The flow is treated as three-dimensional, compressible, and turbulent. Computations are carried out using an in-house Reynolds-averaged Navier–Stokes solver with a Shear Stress Transport turbulence model. Each design parameter is varied separately, and its effects on the exit swirl and the flowfield are discussed. The layout of this paper is as follows: Section II presents the governing equations including the turbulence model. Section III discusses the method of grid generation and the discretization of the governing equations. Section IV discusses the effect of each design variable on seal inlet swirl, and Section V summarizes the results and offers recommendations for swirl vane design improvement.

II. Flow Model

The flow through the swirl vanes was assumed to be steady, viscous, and compressible. It was modeled using the conservation of mass, momentum, and energy, which we refer to collectively as the Navier–Stokes equations. These equations were written for a control volume \(\Omega\) as

\[
\frac{\partial}{\partial t} \int_{\Omega} Q \, dV + \oint_{\partial \Omega} (F_c - F_v) \cdot \mathbf{n} \, dS = \int_{\Omega} G \, dV
\]  

where \(Q\) is the vector of conservative variables, \(F_c\) is the convective flux tensor, \(F_v\) is the viscous flux tensor, and \(G\) is the vector of source terms. These are given by

\[
Q = \begin{bmatrix} \rho \\ \rho v \\ \rho E \end{bmatrix}, \quad F_c = \begin{bmatrix} \rho v^T \\ \rho vv^T + \rho I \\ \rho Hv^T \end{bmatrix}, \quad F_v = \begin{bmatrix} 0^T \\ \tau \\ (\tau \cdot \mathbf{v} + kN T)^T \end{bmatrix}
\]

where \(\rho\) is the density, \(E\) is the mass-specific total energy, \(p\) is the pressure, \(T\) is the temperature, and \(H\) is the mass-specific total enthalpy of the fluid. Here, the vector of source terms is the zero vector, as there are no body forces or internal sources. The fluid velocity, \(\mathbf{v}\), and viscous stress tensor, \(\tau\), are given by

\[
\mathbf{v} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}, \quad \tau = \mu \left( \nabla \mathbf{v} + \nabla \mathbf{v}^T - \frac{2}{3} (\nabla \cdot \mathbf{v}) I \right)
\]

where \(u, v,\) and \(w\) are the cartesian components of velocity, and \(\mu\) is the dynamic viscosity of the fluid.

The Reynolds number in the investigated flows was on the order of \(10^6\), so resolving all scales of motion was prohibitive. Equation (1) was Reynolds- and Favre-averaged to obtain equations for the mean flow variables [5, p. 232]. The Reynolds stress was modeled using Menter’s two-equation shear stress transport model.\(^6,7\) These equations were expressed in a vectorial integro-differential form analogous to Eq. (1). The vector of additional conserved quantities is \(Q_T = (\kappa k, \kappa \omega)^T\), where \(\kappa\) is the mass-specific turbulent kinetic energy and \(\omega\) is the mass-specific turbulent dissipation rate.
III. Numerical Method

A. Grid Generation

The computational grids were generated using in-house software based on the NASA code TCGrid.\textsuperscript{8} An overview of the grid is shown in Fig. 1. The flow was assumed to be periodic in space, with period equal to the pitch. Consequently, it was only necessary to generate the grid around one vane. The topology of the grid is shown in Fig. 2. Blocks II, III, and IV were generated algebraically, while Block I was partially smoothed, using an algorithm due to Steger and Sorenson.\textsuperscript{9}
B. Solution of the Governing Equations

The governing equations were discretized using an unstructured finite-volume approach.\(^{10}\) The flow variables were stored at the grid vertices, and the control volumes were chosen to be the mesh duals. Since no higher than second-order spatial accuracy was used, the flux through all dual-faces attached to an edge was replaced with a single edge-based flux calculation, using an effective normal vector and face area. An edge-based data structure was then used to efficiently sum the fluxes through all dual-faces.

A separate discretization for time and space was employed for Eq. (1). The volume integral was approximated using a single-point quadrature at the dual-cell centroid, and since grid deformation was not considered, the cell volume was independent of time. The surface integral of the flux through all faces attached to an edge was replaced by a single-point quadrature located at the centroid of that edge. This allows Eq. (1), together with the turbulence model equations, to be cast in semi-discrete form as

\[
\frac{dQ_i}{dt} \Omega_i = - \sum_{j=1}^{N_e} \left[ F_c - F_v \right]_{ij} \cdot n_{ij} S_{ij}
\]

where \(N_e\) is the number of edges attached to node \(i\), \(S\) is the effective face area, and the subscript \(ij\) indicates the value at the midpoint of edge \(j\) attached to node \(i\).

The convective flux \(F_c\) was evaluated using Roe’s approximate Riemann solver\(^{11,12}\) with Harten’s entropy fix.\(^{13}\) Second-order spatial accuracy was obtained through a piecewise linear reconstruction.\(^{14}\) The gradients at mesh vertices were found using a least squares approach.\(^{15}\) For the viscous flux, the gradients at edge centroids were evaluated with an arithmetic average, and improved using directional derivatives.\(^{16}\)

The ordinary differential equation Eq. (2) was integrated using a four-stage Runge-Kutta algorithm [5, p. 183]. The stability region of this scheme was expanded using implicit residual smoothing,\(^{17}\) and convergence was accelerated using local time-stepping.\(^{18}\)

IV. Results

This section presents the results of a suite of simulations designed to test the effect of various swirl vane design parameters on seal inlet swirl. The geometry and flow conditions are described in Subsection A. Subsection B presents the results of a grid resolution study. The remaining subsections discuss the effect of each design parameter.

The key output for the parametric studies was the swirl at the seal inlet, defined as \(\zeta = V_\theta / (R \omega)\), where \(V_\theta\) is the tangential velocity component, \(R\) is the hub radius, and \(\omega\) is the hub angular velocity. Swirl may be negative if \(V_\theta\) is against the direction of rotation. To obtain a single output parameter for comparison, the swirl was averaged with respect to mass flow rate over a planar slice of the domain normal to the axial direction and located at the seal inlet. The goal of the studies presented herein was to obtain negative exit swirl that was as large as possible in magnitude.

A. Geometry and Flow Conditions

The simulations presented herein are based on experiments performed by Childs et al.\(^2\) The swirl vane geometry is illustrated in Fig. 3. The flow enters from the left with a large positive inlet swirl, passes around the vane, and exits through the seal. The flow domain extends downstream into a plain annular approximation of the seal. This was done to place the domain exit far from the region of interest to avoid the deleterious effects of the boundaries. The shaft radius was 57.15 mm, and the casing radius was 59.4 mm. The clearance for the vane and seal was 0.2032 mm (8 mil). The baseline vane was 5.2 mm in length, and 1.1 mm in thickness. The vane pitch, stagger angle, chord, and thickness were design variables.

The supplied stagnation pressure and static temperature were 7.31 MPa and 288 K, respectively. The mass flow rate through the entire annulus was 0.626 kg/s. The wheel speed was 10,200 RPM, and the vane inlet swirl was 1.3. The inlet turbulence intensity was 5%, and the ratio of eddy viscosity to molecular viscosity at the inlet was approximately 1.0.
Figure 3: Illustration of the simulation geometry. Dashed lines indicate the boundaries of the computational domain.

B. Grid Independence Test

A grid convergence study was performed to investigate the effects of grid resolution on exit swirl, and to determine a good compromise between accuracy and computational cost. The results displayed in Fig. 4 indicated oscillatory convergence, but the spread was small enough that the medium grid was deemed sufficient.

C. Effect of Vane Stagger Angle

The first parameter investigated was the stagger angle of the vane. Figure 5a shows the swirl at the seal inlet as a function of stagger angle. The best angle of those tested was 5 degrees, although it was only marginally better than the baseline of 0 degrees. The effectiveness of the vane decreases sharply below 0 degrees and above 5 degrees. The reason that the swirl improves as the stagger angle increases toward 5 degrees is that, much like an airfoil at a positive angle of attack, the turning angle improves as angle of attack increases. However, because the flow is entering with a large preswirl, a stagger angle of 10 degrees corresponds to an attempted turning angle of 90 degrees. Therefore, the reason the swirl begins to decrease as the stagger angle is increased beyond 5 degrees is that the flow has become so heavily separated that it cannot be turned further. Figures 5b-5d show solution contours of swirl for a slice of constant radius located at the tip of the vane.
D. Effect of Number of Vanes

The second parameter investigated was the number of vanes. Figure 6a shows the swirl at the seal inlet as a function of the number of vanes. The baseline value of 72 vanes performed the best, with 36 vanes not far behind. The poor performance of 144 vanes is interesting, as intuition would suggest that a larger number of vanes should result in greater control over the flow. Baldassarre et al. also found that too many vanes was ineffective. One possible explanation is that the core vortex responsible for creating negative swirl requires a certain amount of space to develop. Figures 6b-6d show solution contours of swirl for 36, 72, and 144 vanes. One can clearly see the lack of negative swirl at the seal entrance in the 144 vane case.

E. Effect of Vane Chord Length

The next parameter investigated was the chord length of the vane. Figure 7a shows the swirl at the seal inlet as a function of the vane chord length. The largest tested chord length of 6.675 mm was the most effective, although it is only marginally better than the baseline value of 4.62 mm. The fact that the smallest chord length performed poorly is consistent with our aforementioned assertion that the core vortex needs space to develop. Figures 7b-7d show solution contours of swirl at the vane tip for the tested chord lengths.

F. Effect of Vane Thickness

The final parameter investigated was the thickness of the vane. Figure 8a shows the swirl at the seal inlet as a function of the vane thickness. The second smallest thickness fared the best. The fact that the swirl degrades as thickness increases can be explained, once again, by the assertion that adequate space is needed for the core vortex to develop. However, the swirl also degrades for the thinnest vane. This can be explained by noting that thinner airfoils tend to result in larger amounts of separation. There is, therefore, a trade off between the need for adequate space for the vortex to develop and the excessive separation of very thin vanes. The second thinnest vane was found to be the best compromise between these competing effects. Figures 8b-8d show solution contours of swirl at the tested thicknesses. It is interesting to note that the second thinnest vane, which performed the best, appears to have the most coherent vortex.
V. Conclusions

A Reynolds-averaged Navier–Stokes solver was used to examine the effects of several design parameters on the performance of a swirl brake. For the tested flow conditions, the results indicated the following:

1. The exit swirl is moderately sensitive to the vane stagger angle. The peak performance occurs at a stagger angle of 5 degrees, although it is only marginally better than the baseline value of 0 degrees. Thus, the simpler 0 degree configuration is probably preferable for manufacturing reasons.

2. The vane pitch and chord must be made large enough to give the blade-to-blade vortex adequate space to form.

3. The vane thickness should be small enough for the vortex to properly develop, but not so small that there is excessive separation and the vortex is not coherent. Structural concerns would likely enforce a stricter lower limit than separation, however.
(a) Variation of swirl with number of vanes. 

(b) Swirl contours, 36 vanes. 

(c) Swirl contours, 72 vanes. 

(d) Swirl contours, 144 vanes. 

Figure 6: Simulation results for vane count tests. 

Acknowledgments

This work was funded by the Turbomachinery Research Consortium. The authors would like to thank Dr. Dara Childs for the experimental data and insight he provided. The authors would also like to thank Dr. Forrest Carpenter for his suggestions. Finally, the authors would like to thank the Texas A&M High Performance Research Computing Center for making their resources available. 

References


Figure 7: Simulation results for chord length tests.


Figure 8: Simulation results for thickness tests.


