Chinese Journal of Aeronautics, (2015), 28(4): 1076–1086

Chinese Society of Aeronautics and Astronautics
& Beihang University

Chinese Journal of Aeronautics

cja@buaa.edu.cn
www.sciencedirect.com

# Optimization of endwall contouring in axial compressor S-shaped ducts

**Jin Donghai** [a,*], **Liu Xiwu** [a], **Zhao Weiguang** [b], **Gui Xingmin** [a]

[a] *School of Energy and Power Engineering, Beihang University, Beijing 100191, China*
[b] *AVIC Aviation Powerplant Research Institute, Hunan 412000, China*

Received 21 October 2014; revised 30 March 2015; accepted 11 May 2015
Available online 20 June 2015

**KEYWORDS**

Adaptive genetic algorithm (AGA);
Artificial neural network (ANN);
Corner separation;
Design of experiments (DOE);
Endwall contouring;
Optimization;
Response surface methodology (RSM);
S-shaped duct

**Abstract** This paper presents a numerical investigation of the potential aerodynamic benefits of using endwall contouring in a fairly aggressive duct with six struts based on the platform for endwall design optimization. The platform is constructed by integrating adaptive genetic algorithm (AGA), design of experiments (DOE), response surface methodology (RSM) based on the artificial neural network (ANN), and a 3D Navier–Stokes solver. The visual analysis method based on DOE is used to define the design space and analyze the impact of the design parameters on the target function (response). Optimization of the axisymmetric and the non-axisymmetric endwall contouring in an S-shaped duct is performed and evaluated to minimize the total pressure loss. The optimal ducts are found to reduce the hub corner separation and suppress the migration of the low momentum fluid. The non-axisymmetric endwall contouring is shown to remove the separation completely and reduce the net duct loss by 32.7%.


## 1. Introduction

An S-shaped duct is used to connect the low pressure and high pressure compressors of aircraft gas turbine engines. Within the duct, flow separation should be avoided to minimize the total pressure loss. In addition, a uniform flow field at the duct exit should also be achieved. However, the demands of modern turbo-fan engines for higher efficiency and lower noise level lead to high by-pass ratio, and result in the engines with large fans and small high pressure compressors, and then bring about a significant radial difference between the low pressure and the high pressure systems. The higher the by-pass ratio is, the more the aggressive S-shaped ducts are needed. This tendency makes duct design increasingly difficult and important. First, transition ducts play a significant role in determining the overall length and weight of the engine. The advantages are obvious if the duct length could be shorter without other penalties. Second, the increase of the thickness of the non-turning struts in the duct would allow improved service access to the core of the engine.

Several researchers have investigated the flows in S-shaped ducts. Bailey et al.[1] investigated the aerodynamic performance of a compressor S-shaped duct with a single strut (the

* Corresponding author. Tel.: +86 10 82339391.
E-mail addresses: jdh@buaa.edu.cn (D. Jin), liuxiwu@buaa.edu.cn (X. Liu).
Peer review under responsibility of Editorial Committee of CJA.

Production and hosting by Elsevier

http://dx.doi.org/10.1016/j.cja.2015.06.011
1000-9361 

thickness-chord ratio is 0.12). The blockage of strut was found to have a significant effect on the pressure field of the duct, which has a direct influence on the turbulent flow field. Dueñas et al.[2] experimentally investigated the effect of reducing the duct length and keeping unchanged the duct inlet height $h_{in}$ and inlet to exit radius change $\Delta R$. It was found that reducing the length of the datum duct without a strut to 74% caused a small increase in loss; however, reducing the length to 64% caused a much greater loss. Walker et al.[3] integrated outlet guide vane (OGV) design for an aggressive S-shaped compressor duct and introduced the effects of compressor generated inlet conditions which shown that system length could be reduced by 21% although the overall system loss nominally remained unchanged.

Studies have shown that the limit of the design space of annular S-shaped duct is set by duct corner separation. Reducing the length or increasing the change in radius or the thickness-chord ratio has a similar effect on the duct performance. The streamlines with the maximum curvature, in other words, the flows with the fastest deceleration, emerge in the hub-strut corner, where flow separation may occur. The separation results in a sharp rise of loss coefficient in the duct, and will create a large-scale blockage in the downstream compressor.

In order to reduce the large extent hub-strut corner separation and avoid higher loss coefficient in ducts, the technology of endwall contouring is used in this paper. Endwall profiling has been widely investigated in other turbomachinery components. The successful profiled endwall design begins with Rose[4], who demonstrated the fundamentals of controlling the endwall static pressure field by means of endwall contouring. That is, convex wall curvature locally accelerates the flow and thus reduces the static pressure, while concave curvature causes diffusion, raising the static pressure. This phenomenon was confirmed in the experiments by Hartland et al.[5] in the Durham linear cascade. Harvey et al.[6], Torre et al.[7], and Sonoda et al.[8] all had achieved improvements in turbine aerodynamic performance by using endwall contouring through measurement or prediction. The researches of Mahmood[9] and Gustafson[10] et al. showed that turbine passage endwall heat transfer rates can also be reduced by using endwall profiling. For compressor rotors, Hoeger et al.[11] discovered a positive effect of endwall contouring in terms of influencing the shock position. For compressor stator application, Harvey[12] discussed several non-axisymmetric endwall configurations in a linear cascade, and showed that non-axisymmetric endwall, though not optimum, has effects on the crossflow, and the corner stall can be suppressed. Later on, Harvey and Offord[13] investigated the non-axisymmetric endwall in multi-stage high pressure compressor through computational fluid dynamics (CFD) study, and found that the corner stall can be suppressed either by endwall contouring or 3D-blading. In regard to turbine duct application, Wallin and Eriksson[14] presented studies on CFD-based non-axisymmetric hub endwall shape optimization for an intermediate turbine duct, and showed benefits for duct performance as well as weight reduction. However, the authors found that there were few studies on the use of endwall contouring in compressor ducts. Wallin and Eriksson[15] presented studies on optimization of a 2D axisymmetric turbine duct and a 3D axisymmetric compressor duct by using response surface methodology (RSM). It was found that axisymmetric endwall optimization can reduce duct losses significantly. Naylor et al.[16] used endwall contouring in duct, and showed that the non-axisymmetric endwall completely removed strut-hub corner separation. However, the duct investigated was two-dimensional.

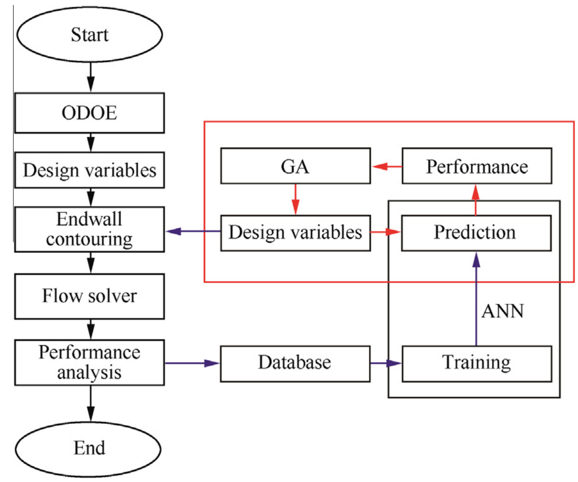


**Fig. 1** Flowchart for the algorithm of the endwall design optimization system.

In this paper, the focus lies on intermediate S-shaped duct endwall profiling and its influence on the flow field in the 3D annular duct. A numerical optimization coupling with adaptive genetic algorithm (AGA) and RSM is undertaken to design the axisymmetric and non-axisymmetric endwall profiling. Finally, the performance of the optimal endwall profiling is compared with the datum S-shaped duct.

## 2. Endwall design optimization platform

The use of design optimization in turbomachinery is possible today thanks to the CFD analysis. Fig. 1 shows the flowchart for the algorithm of the endwall design optimization system. One of its advantages is the use of a response surface model based on an artificial neural network (ANN) as an approximate substitute for the goal-function. The tremendous computational cost of evaluating the endwall performance by 3D CFD can thus be reduced.

The optimization system consists of three steps. First is the training of ANN based on the database provided by the orthogonal design of experiment (ODOE). The second step is the prediction of the optimal aerodynamic performance of endwall contouring by the combination of AGA and ANN, as shown in Fig. 1 with red arrows. Finally, a comparison of the performance obtained by CFD with that of the one predicted by the ANN is executed. If the design requirements are not achieved, the evaluations computed by CFD are added to the database, and the loop is repeated until an optimal geometry is obtained. A more detailed description of the optimization method can refer to Jin[17–19] and Ning and Liu.[20] The following subsections summarize some components and their applications.

### 2.1. Endwall parameterization

Fig. 2 presents the parameterization of axisymmetric endwall contouring. The parameterization is performed adopting a B-