An Experimental Investigation of the Flow Over and Through an Airfoil Shaped Strut in a Confined Duct

James Crawford
Department of Mechanical and Materials Engineering, Queen’s University, Kingston, Canada

Synopsis
The flow in and around an airfoil shaped strut, typical of those found in advanced gas turbine exhaust systems, was investigated experimentally with ambient temperature and pressure air. Assessing and characterizing the cold flow performance of the test rig was a necessary first step towards the ultimate goal of investigating the heat transfer through the strut both computationally and experimentally. In terms of the flow around the airfoil, the influence of Reynolds number, airfoil angle of attack, and of downstream wake confinement were investigated in terms of their effects on overall losses, local momentum deficits, turbulent wake spread rate, and vortex generation. In terms of the flow inside the airfoil strut, it was discovered that although the Reynolds number of the mean flow was fully turbulent, because the airfoil is quite short, the effects of the laminar boundary layer development were significant for the range of flow rates tested.

Introduction
Advanced gas turbine exhaust systems, such as the generic system shown in Figure 1, often make use of a shaped centrebody to provide optical blockage of the last turbine stage. This significantly reduces the infrared (IR) signature of the engine, which is beneficial in many applications. A well designed exhaust system is a net diffuser, and so the pressure along the centrebody is typically subatmospheric, which provides an opportunity to realize passive cooling of the centrebody surface. In order to do this, however, the airfoil shaped struts that provide structural support to the exhaust system must be hollow. The cool air flowing through the inside of these hot struts creates significant temperature gradients, leading to significant thermal stresses, and potential structural failure. There is a desire to understand whether it is possible to model the flow and heat transfer around these struts using modest computational grids and commercial CFD codes. To this end, the experiments described in the present work will be used both to provide fundamental physical insight into the flows, and as a tool to validate future CFD simulations.

Methods and Results
The flow around an airfoil strut was tested by constructing a stainless steel test rig with a rectangular cross section with a hollow NACA 0020 airfoil placed in the centre of the cross section. The airfoil was rotated in 10° increments, and 3 different lengths of ducting were installed downstream of the airfoil to investigate the influence of confining the separated wake at high angles of attack. The test rig is shown in Figure 2. Three dimensional time-averaged velocity fields were measured in the airfoil wake using seven hole probes for each case, and for selected tests, a uniaxial hot wire anemometer was used to characterize the instantaneous velocity fields. The velocity field measured in the near wake of the airfoil at a 20° AOA is shown in Figure 3 (a). Contours of axial velocity are plotted with vectors of in plane velocity components. Qualitatively, the momentum deficit in the airfoil wake appears as the axial velocity deficit in the middle of the duct, while the secondary flow vectors show the flow on the suction (right) side of the airfoil has been directed to the left, as expected. The in plane vorticity, which is a measure of rotation, was derived from the velocity field and plotted in Figure 3 (b). Four wall-airfoil interface vortices, which are not apparent from the secondary flow vectors, are clearly visible in this plot. The results show that with increasing angle of attack, the interface vortices on the pressure (left) side of the airfoil are simply transported further to the left, without increasing in size. The vortices on the right side of the airfoil are transported in a similar fashion, although the negative pressure gradient on the suction side causes these vortices to stretch in the width-wise direction with increasing angle of attack.

String tufts were also attached to the rig to characterize the width of the turbulent wake at various downstream locations, as well as to identify the point of first stall on the airfoil surface. The onset of stall was observed to be somewhere between 10° and 20° AOA, which is consistent with the literature. Limitations of the test rig made it impossible to improve the resolution of that observation. In terms of the turbulent wake, it was shown using the hot wire anemometer that the string tufts were an excellent indicator of the turbulent wake width. It was then shown using several rows of tufts that the wake on the suction side of the airfoil was unaffected by the addition of additional downstream ducting to confine the wake, while the spread rate of the wake on the suction side was increased when additional ducting was added.

In terms of the flow through the airfoil, the performance was characterized in two ways. An auxiliary blower was connected to the airfoil strut, and configured to blow air through the strut. The time-averaged velocity profile was then measured using a seven hole probe traverse. The blower connection was then reversed, and the blower was configured to suck air through the airfoil. A pitot-static tube was mounted in the ducting connecting the strut to the blower, allowing for direct measurement of the total pressure drop through the airfoil. Both sets of results suggest a significant Reynolds number effect that is the result of laminar to turbulent boundary layer transition. Although the flow through the airfoil was fully turbulent (Re_D ~ 10^5), the struts are quite short, and the Reynolds number based on the length of the strut ranged from 7.5x10^5 to 5x10^5. This is quite close to the typical transition point for a flat plate boundary layer (Re ~ 4x10^5), suggesting that at the lowest flow rates tested, the transition point may have been as much as 50% of the way along the flow length. This is a potentially extremely significant effect, because heat transfer rates in laminar and turbulent boundary layers are drastically different. Boundary layer transition is also a notoriously difficult problem to model with RANS based CFD, and so this effect may also complicate the computational investigation.

Conclusions and Future Work
Some experimental data characterizing the cold flow through and around a typical airfoil strut have been presented and discussed. The influence of several parameters, including Reynolds number, airfoil angle of attack, and downstream wake confinement has been investigated. A similar experimental programme will be repeated using hot flow. A comparison of the differences between the hot flow results and these results will allow the effects of heat transfer on the mean flow to be isolated and discussed in greater detail. As a parallel activity, the present experiments can be modeled using RANS based CFD, and the ability of RANS based CFD to model this type of flow can be investigated.