International Journal of Heat and Fluid Flow 50 (2014) 467–478

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

journal homepage: www.elsevier.com/locate/ijhff

Effect of boundary layer thickness on secondary structures in a short inlet curved duct

Jeremy Gartner[1], Michael Amitay[2,*]

Rensselaer Polytechnic Institute, Troy, NY 12180, USA

ARTICLE INFO

Article history:
Received 4 February 2014
Received in revised form 15 October 2014
Accepted 17 October 2014
Available online 13 November 2014

Keywords:
S-shape inlet
Secondary flows
Three dimensional separation
Secondary flow structures

ABSTRACT

The flow pattern in short inlet ducts with aggressive curvature has been shown to lead, in some cases, to an asymmetric flow field at the aerodynamic interface plane. In the present work, a two-dimensional honeycomb mesh was added upstream of the curved duct to create a pressure drop across it, and therefore to an increased velocity deficit in the boundary layer. This velocity deficit led to a stronger streamwise separation, overcoming the instability that can result in an asymmetric flow field at the aerodynamic interface plane. Experiments were conducted at Mach numbers of $M = 0.2$, 0.44 and 0.58 in an expanding aggressive duct with rectangle to a square cross section with area ratio of 1.27. Steady and unsteady pressure measurements, together with Particle Image Velocimetry (PIV), were used to explore the effect of the honeycomb on the symmetry of the flow field. The effect of inserting a honeycomb was tested by increasing its height from 0 to 2.2 times the boundary layer thickness of the baseline flow upstream of the curve. Using the honeycomb, flow symmetry was achieved for the specific geometrical configuration tested with a negligible decrease of the pressure recovery.

1. Introduction

Heightened interest in short and curved inlet ducts for aircraft has led us to further explore the flow field existing in such devices as well as to a better knowledge of the issues associated with their design and methods of mitigating such issues. Several factors related to engine and aircraft performance and operation drive the use of short inlet duct designs, such as the overall airframe length reduction enabled by a shorter duct and reduction of frontal planform by burying the engine into the airframe. These factors led to a reduction in weight and fuel consumption, and allowed for innovative external and integrated aerodynamics, such as Blended Wing Bodies (Dagget et al., 2003). Another factor that must be taken into consideration is the stability margin for operation of a jet engine following the duct, where uneven pressure distribution and secondary flow structures can lead to engine stall at the fan/compressor stages (surge stall) (Scribben et al., 2006; Mattingly et al., 2002).

A considerable body of work is available in the literature concerning the analysis of the flow field in short inlet ducts (Bansod and Bradshaw, 1972; Launder and Ying, 1972; Enayet et al., 1982; Wellborn et al., 1992, 1993; Whitelaw and Yu, 1993a,b; Ng et al., 2008, 2006). These previous research efforts have shed light on the main features of the flow field existing in aggressively curved ducts, where the rapid curvature in the duct results in pressure gradients in the direction normal to the turn, leading to the onset of secondary flow structures in the form of two counter rotating vortices. Other structures were also noticed to co-exist with these counter-rotating vortices, such as cross stream flow at the internal surfaces, which invade the local boundary layer leading to further flow detachment (Wellborn et al., 1992, 1993; Ng et al., 2008, 2006; Chen, 2012) disrupting the flow and creating recirculation zones in the duct. The symmetric counter-rotating vortices can be described by inviscid flow equations, caused solely by the turning of the flow. These pressure driven counter-rotating vortices convect the low momentum fluid of the boundary layer towards the center of the duct impacting flow uniformity and pressure recovery at the face of the engine located downstream, at the aerodynamics interface plane.

Implementation of passive and active flow control techniques in short inlet ducts has been an active field of research (Scribben et al., 2006; Ng et al., 2008, 2006; Chen, 2012; Vaccaro, 2011; Debronsky, 2012; Amitay et al., 2002; Gissen et al., 2011; Jirasek,

---

* Corresponding author.
E-mail address: amitam@rpi.edu (M. Amitay).

[1] Department of Mechanical, Aerospace & Nuclear Engineering, 110 8th Street, Troy, NY, USA.

[2] James L. Decker'45 Endowed Chair of Aerospace Engineering, and Director of the Center for Flow Physics and Control (CeFPaC), 110 8th Street, Troy, NY, USA.

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.10.016
0142-727X/© 2014 Elsevier Inc. All rights reserved.

Nomenclature

AIP	aerodynamic interface plane
D	width of the duct (mm)
h	honeycomb height (mm)
δ	boundary layer thickness (mm)
h/δ	relative height of the honeycomb
θ	momentum thickness (mm)
L	length of the duct (mm)
M_{inlet}	Mach number at the inlet
PR	pressure recovery
P_{inlet}	static pressure at the inlet (atm)
γ	specific heat ratio of air
C_p	pressure coefficient
P_0	total pressure (atm)
P	static pressure (atm)

2006; Reichert and Wendt, 1994). The predominant forms of actuation are vortex generators, steady and unsteady jet blowing tangent to the surface, synthetic jet actuators, and many more. Recent work has studied multiple actuation devices (Vaccaro, 2011; Gissen et al., 2011) including combination of flow control techniques. Although most of the previous work was focused on circular cross section ducts (Wellborn et al., 1992, 1993; Whitelaw and Yu, 1993a,b; Gissen et al., 2011; Jirasek, 2006), emphasis was also given to rectangular cross section ducts (Launder and Ying, 1972; Ng et al., 2008, 2006; Chen, 2012; Vaccaro, 2011; Debronsky, 2012; Amitay et al., 2002). All of the work performed with flow control had the objective of improving the pressure recovery and pressure distribution at the exit of the duct.

Recent experiments (Vaccaro, 2011; Debronsky, 2012) and numerical simulations (Chen, 2012) have shown that, under some geometrical conditions, the flow can become asymmetric, where one of the counter-rotating vortical structure supersedes the other. In the case of the rectangular ducts, the secondary flow structures were shown to move towards one of the corners of the duct (Chen, 2012; Vaccaro, 2011; Debronsky, 2012).

As noted by Chen (2012), the secondary flow phenomenon (i.e., a turbulent flow with mean streamwise vorticity) is attributed to two mechanisms: (i) the skew induced, inviscid mechanism, which is caused by any bend in the flow path of ducts with any cross sectional shape as shown by Miller (1991) (and Fig. 1 below), and (ii) a stress-induced mechanism occurring in any non-circular ducts, straight or not, due to anisotropy of the Reynolds stresses. A more in-depth description of secondary flow can be found in Perkins (1970) and Bradshaw (1987). Also note that further complexity in the flow structures is due to swirl development in the second bend of the s-duct. This reverse in the curvature is accredited with the crossover of the transverse velocity component near the side walls, an essentially inviscid process. Another feature of short inlet ducts is the adverse pressure gradient caused by the opposite curvature of the second bend. Therefore, the secondary flow generated by the first bend is attenuated, being reversed depending on the aggressiveness of the turn (i.e., the aspect ratio L/D, the offset and area ratio between the inlet and the exit sections).

Fig. 2a (taken from Wellborn et al., 1993) shows a three-dimensional perspective of the owl face separation topology with a counter-rotating pair of vortices orientated with upwelling along the centerline. The skeleton drawing of Fig. 2b shows the schematic of a symmetric but unstable owl face separation of the first kind. Perry and Hornung (1984) suggested that this unstable symmetric distribution could exist due to slight variations in the flow field. However, they stated that it was a special condition and that any asymmetry in the flow would cause the streakline pattern to shift to one side as shown in Fig. 2c.

Based on previous experiments (Ng et al., 2008, 2006; Vaccaro, 2011) and numerical analysis (Chen, 2012), it was shown that a critical length to diameter ratio exists in a rectangular cross section compact inlet duct, controlling the asymmetry of the secondary structures. For ducts longer than this critical length the flow patterns are asymptotically stable. With the reduction of duct length below this critical value, the streamwise pressure gradient increases and interacts with the transverse invasion. Simultaneously, the forward moving main flow confronts the backflow to the streamwise separation. The symmetric pattern becomes unstable due to the saddle–saddle connections existing in the topology of the flow, leading to a flow bifurcation (Tobak and Peak, 1982).

This asymmetric configuration is the starting point for the current work, which also derives from the observation (Enayet et al., 1982; Chen, 2012) that the secondary flow structures forming on the inside of duct bends have their strength dependent on inlet flow conditions, specifically the momentum thickness of the

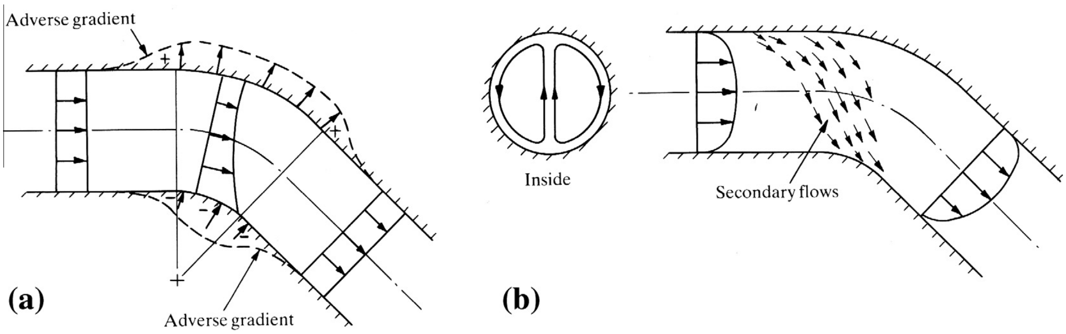

Fig. 1. Development of secondary flows in a pipe bend showing the (a) presence of an adverse pressure gradient, and (b) direction and orientation of the secondary flow (from Miller, 1991).