International Journal of Heat and Fluid Flow 54 (2015) 1–10

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

# Experimental and numerical investigation on steady blowing flow control within a compact inlet duct

John C. Vaccaro [a,*], Yossef Elimelech [b], Yi Chen [c], Onkar Sahni [c], Kenneth E. Jansen [d], Michael Amitay [c]

[a] Hofstra University, Hempstead, NY 11549, United States
[b] Technion – Israel Institute of Technology, Haifa 32000, Israel
[c] Rensselaer Polytechnic Institute, Troy, NY 12180, United States
[d] University of Colorado, Boulder, CO 80309, United States

ARTICLE INFO

*Article history:*
Received 25 September 2014
Received in revised form 23 March 2015
Accepted 18 May 2015

*Keywords:*
S-shape inlet ducts
Aerodynamic interface plane
Active flow control
Steady blowing
Stereoscopic particle image velocimetry
Navier–Stokes numerical simulation

ABSTRACT

A combined experimental and numerical investigation of flow control actuation in a short, rectangular, diffusing S-shape inlet duct using a two-dimensional tangential control jet was conducted. Experimental and numerical techniques were used in conjunction as complementary techniques, which are utilized to better understand the complex flow field. The compact inlet had a length-to-hydraulic diameter ratio of 1.5 and was investigated at a free-stream Mach number of 0.44. In contrast to the baseline flow, where the flow field was fully separated, the two-dimensional control jet was able to eliminate flow separation at the mid-span portion of the duct and changed considerably the three-dimensional flow field, and ultimately, the inlet performance. A comparison between the baseline (no actuation) and forced flow fields showed that secondary flow structures dominated both flow fields, which is inevitably associated with total pressure loss. Contrary to the baseline case, the secondary flow structures in the forced case were established from the core flow stagnating on the lower surface of the duct close to the aerodynamic interface plane. High fidelity spectral analysis of the experimental results at the inlet's exit plane showed that the baseline flow field was dominated by pressure fluctuations corresponding to a Strouhal number based on hydraulic diameter of 0.26. Not only did the two-dimensional tangential control jet improve the time-averaged pressure recovery at the inlet exit plane (13.3% at the lower half of the aerodynamic interface plane), it essentially eliminated the energy content of the distinct unsteady fluctuations which characterized the baseline flow field. This result has several implications for the design of a realistic engine inlet; furthermore, it depicts that a single non-intrusive static pressure measurement at the surface of the duct can detect flow separation.



## 1. Introduction

High performance air-breathing propulsion systems demand high efficiency while minimizing weight. Much research has been focused on the improvement of each component of the propulsion system to achieve this end. Particularly, ongoing research is attempting to reduce the overall length of the inlet. In many applications reducing the streamwise extend of the inlet leads to an increase in centerline curvature resulting in separation and development of secondary flows.

The goal of any inlet design is to decelerate the oncoming flow to speeds which promote high performance of the fan or compressor. At the same time, the inlet should minimize both total pressure loss (have high pressure recovery, *PR*) and flow unsteadiness, and produce a relatively constant pressure profile (low distortion) across the interface plane of the fan/compressor. Development of surge and/or rotating stall may occur if these goals are not achieved (Paduano et al., 2001).

The existence of separations and secondary flow has the adverse effects of increasing total pressure loss, flow unsteadiness, and pressure distortion. For this reason large centerline curvatures (short lengths) are avoided whenever possible. Current unmanned air vehicle designs, however, are calling for such large curvatures because of the weight savings which would be achievable if the propulsion system were shortened. The length of the propulsion system is currently sizing the overall aircraft in such designs. The length of the inlet is typically quantified by establishing the length-to-diameter ratio. A large length-to-diameter ratio corresponds to a less drastic design. Past research focused on length-to-diameter ratios between 2.5 and 3.5 (Bansod and Bradshaw, 1972; Hamstra et al., 2000; Rabe Scribben et al., 2006;

* Corresponding author. Tel.: +1 516 463 7023.
*E-mail address:* John.C.Vaccaro@hofstra.edu (J.C. Vaccaro).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2015.05.011
0142-727X/

**Nomenclature**

*Abbreviations*

| | |
|---|---|
| AIP | aerodynamic interface plane |
| DDES | delayed detached eddy simulation |
| PSD | power spectral density |
| SPIV | stereoscopic particle image velocimetry |

*List of symbols*

| | |
|---|---|
| $C_p$ | coefficient of pressure |
| $D$ | hydraulic diameter at AIP |
| $f$ | frequency |
| $H$ | distance normal to local lower surface; height |
| $h$ | dimensionless distance normal (height) to local lower surface, $H/D$ |
| $L$ | inlet duct axial length |
| $M$ | Mach number |
| $M_\infty$ | Mach number at inlet entrance, measured 0.5 exit hydraulic diameters upstream of inlet |
| $\widehat{m}$ | mass flow ratio, $\dot{m}_j/\dot{m}_{in}$ |
| $\dot{m}_{in}$ | mass flow through inlet duct |
| $p$ | static pressure |
| $p_\infty$ | static pressure at inlet entrance, measured 0.5 exit hydraulic diameters upstream of inlet |
| $P_o$ | total pressure |
| $P_{o,\infty}$ | total pressure at inlet entrance, measured 0.5 exit hydraulic diameters upstream of inlet |
| $P_{P_oP_o}$ | power spectral density of the total pressure signal |
| $PR$ | pressure recovery, $P_o/P_{o,\infty}$ |
| $R$ | radius of curvature |
| $r$ | dimensionless radius of curvature, $R/D$ |
| $\mathrm{Re}_D$ | hydraulic diameter Reynolds number, $\rho_\infty U_\infty D/\mu_\infty$ |
| St | Strouhal number, $fD/U_\infty$ |
| $U$ | velocity component tangential to local lower surface |
| $U_\infty$ | velocity component tangential to local lower surface at inlet entrance, measured 0.5 exit hydraulic diameters upstream of inlet |
| $u$ | dimensionless velocity tangential to local lower surface, $U/U_\infty$ |
| $u_*$ | friction velocity, $\sqrt{\tau_w/\rho}$ |
| $\bar{u}$ | time-averaged dimensionless velocity tangential to local lower surface |
| $V$ | velocity component normal to local lower surface |
| $v$ | dimensionless velocity normal to the local lower surface, $V/U_\infty$ |
| $W$ | velocity component in the spanwise direction |
| $w$ | dimensionless velocity in the spanwise direction, $W/U_\infty$ |
| $\bar{w}$ | time-averaged dimensionless velocity in the spanwise direction |
| $X$ | streamwise (axial) coordinate |
| $x$ | dimensionless streamwise coordinate, $X/D$ |
| $Y$ | cross-stream coordinate in the vertical direction |
| $y$ | dimensionless cross-stream coordinate, $Y/D$ |
| $y^+$ | dimensionless wall distance, $u_*H/\upsilon$ |
| $Z$ | spanwise coordinate, measured from the duct's centerline |
| $z$ | dimensionless spanwise coordinate, $Z/D$ |
| $\gamma$ | ratio of specific heats |
| $\mu_\infty$ | fluid's dynamic viscosity at inlet entrance, measured 0.5 exit hydraulic diameters upstream of inlet |
| $\upsilon$ | fluid's kinematic viscosity |
| $\rho$ | fluid density |
| $\rho_\infty$ | fluid density at inlet entrance, measured 0.5 exit hydraulic diameters upstream of inlet |
| $\sigma$ | standard deviation (of pressure signal) |
| $\tau_w$ | shear stress at the wall |

Kirk et al., 2007). A more thorough review of such flow fields can be found in Vaccaro et al. (2013).

Compact inlet designs must suppress separation and development of secondary flows if they are to be effective. This can be achieved through the use of passive or active flow control strategies. Most passive flow control studies on compact inlets have focused on the utilization of vortex generators (Anderson and Levy, 1991; Povinelli and Towne, 1986; Vakili et al., 1985, 1987; Reichert and Wendt, 1994; Anabtawi et al., 1999; Anderson et al., 2004; Allan et al., 2006; Jirasek, 2006; Lee and Liou, 2010). Typical vortex generators are either micro-vanes or micro-ramps, which generate a streamwise vortex when immersed in a cross-flow. The size of these vortex generators are determined by the boundary layer thickness. The tips of the generators are usually located just outside the edge of the boundary layer, which allows for the greatest interaction of the shed vortex and the low momentum fluid. The vortex shed from the vortex generators introduces mixing and brings high momentum fluid into the boundary layer which helps to delay, or even suppress, separation. The downside of utilizing passive vortex generators is that they have an inherent parasitic drag associated with them; furthermore, they cannot be changed for off-design conditions.

In order to improve efficiency of flow control devices throughout a larger flight envelop, active flow control has also been researched heavily in the field of inlet design. Active flow control differs from passive flow control in that there is a net energy input. The added benefit is twofold. First, an active device could be employed only when needed which would eliminate any drag associated with them when they are not required. Second, an active device could be implemented in a closed-loop control scheme where the effect of the device could be dynamically adjusted based on information of the current flow field. Most active flow control research has focused on blowing or suction from the inlet's surface. Blowing devices are found in many forms: Coanda-type steady or unsteady ejectors (McElwain, 2002; Luers, 2003; Tournier et al., 2006; Vaccaro et al., 2009), microjets (Hamstra et al., 2000; Kumar and Alvi, 2006; Reynolds and Reeder, 2009), vortex generator jets (Pradeep, 2004; Owens et al., 2006; Sullerey et al., 2006; Rabe Scribben et al., 2006), and synthetic jet actuators (Amitay and Glezer, 2002).

The present study expands on a previous investigation by the authors (Vaccaro et al., 2013) to include the effects of blowing on a short, diffusing, rectangular cross-sectional S-shaped duct in the compressible flow regime. A highly aggressive length-to-hydraulic diameter ratio of 1.5 has been studied at a Mach number of 0.44, which corresponds to a hydraulic diameter based Reynolds number of $7.5 \times 10^5$. The present analysis employs pressure measurements, stereoscopic particle image velocimetry, and an extensive three-dimensional unsteady flow simulation.

## 2. Experimental and numerical setups and methods

### 2.1. Wind tunnel facility

The experiment was conducted in an open return blowdown inlet duct facility. The facility was capable of reaching a Mach number, $M$, of 0.5 corresponding to mass flow rates of 2.3 kg/s. This speed and duct geometry produced a maximum achievable Reynolds number based on exit hydraulic diameter, $D$, of $1.17 \times 10^6$. A detailed description of the test facility can be found in Vaccaro et al. (2013).

Please cite this article in press as: Vaccaro, J.C., et al. Experimental and numerical investigation on steady blowing flow control within a compact inlet duct. Int. J. Heat Fluid Flow (2015), http://dx.doi.org/10.1016/j.ijheatfluidflow.2015.05.011