The Coanda Effect anomaly present in 2D CFD simulations of installed rectangular jets

Katherine Meaden, Mohammed Afsar, Rohella Muhel, Ioannis Kokkinakis
Department of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow.

Abstract

The Coanda Effect describes the attachment of a jet flow to a nearby surface. In this paper we show that a Computational Fluid Dynamics (CFD) solution of an installed rectangular jet introduces an anomalous Coanda Effect (i.e. bending of the jet towards the trailing edge of the flat plate positioned adjacent to the nozzle lower lip line) when the simulation is performed in 2D. The latter are often used for fast estimation of a full 3D RANS calculation (that can take up to 6 days for the residuals to converge to $1\text{x}10^{-4}$ on a grid of 7 million cells using 16 cores). Our results show that for a 3D simulation, the jet flow passed smoothly over the surface without producing significant bending.

To fix ideas and prove that a 2D numerical simulation results in such anomalous deviation, we examine a rectangular nozzle of $8:1$ aspect ratio (AR) with a flat plate positioned parallel to the level curves of the jet at a transverse (stand-off) distance of $y_2/D_J = 1.9$" and trailing edge length, 12". Bridges (2014, cited in Afsar *et al.*[1]) found that this particular configuration results in an increase in sound of approximately 10 dB at low frequencies. This was also analysed in Afsar *et al.*[1] who used Rapid-Distortion Theory to model the observed noise amplification.

We compare a number of 2D simulations under a variety of conditions including: the effect of using an alternative CFD solver (Fluent (a) vs. STARCCM+ (b)), using different turbulence models ($k-\varepsilon$ Realizable vs. $k-\omega$ SST (c)), varying the upstream far field Mach number (d) and, finally, modifying the upstream location of the flat plate (while keeping the downstream location and vertical stand off distance fixed, (e)). In all the above numerical experiments, the bending of the jet flow persists indicating that a Coanda Effect is taking place. In Figure 1 we show contours of a typical simulation and in Table 1 the so-called 'bend angle', ψ_b, of the jet relative to the centreline is quantified for the above cases.

Case	Bend Angle, ψ_b (°)
a (Fluent, $k-\varepsilon$)	12.079
b (STARCCM+, $k-\varepsilon$)	12.110
c (Fluent, $k-\omega$)	12.881
d (M(upstream)=0.05)	11.516
e (Plate Length 7")	7.874

Table 1: Bend angle for Cases a-e

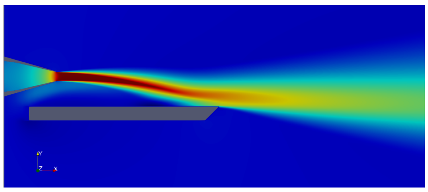

Figure 1: U/U_J for Case a

Although a slight attraction of the jet towards the surface takes place for the 3D simulation, having the same configuration as the simulations in Table 1 (trailing edge location: $(x,y)/D_J = (5.7, 1.1)$), and in which ψ_b is estimated at $< 2°$, this is much less than any of the $2D$ cases. In the accompanying talk, we discuss these results and explain how the apparent role of the spanwise shear, $\partial U/\partial z$, that exists in the 3D case, is likely to be the reason a 3D simulation prevents the jet from bending to a noticeable degree.

[1] Afsar,M.Z., Leib,S.J. and Bozak, R.F.Y *J. Sound & Vib*, Vol. 386, pp. 177-207, 2017.