

Computational Study of Coanda Adhesion Over Curved Surface

Maharshi Subhash and Antonio Dumas
University of Modena and Reggio Emilia

ABSTRACT

This paper presents a set of numerical computations with different turbulence model on an air jet flowing tangentially over the curved surface. It has been realized that jet deflection angle and the corresponding thrust are important parameter to determine with great care. Through the grid independence analysis, it has been found that without resolution of the viscous sub-layer, it is not possible to determine the computationally independent angle of jet deflection and boundary layer thickness. The boundary layer analysis has been performed at different radius of curvature and at jet Reynolds number ranging from approximately about 2400-10,000. The boundary layer thickness has been determined at the verge of separation and found a relation with the radius of curvature and jet Reynolds number. The skin-friction coefficient has been also studied at the verge of separation in relation to the surface radius and jet Reynolds number. In this research work, the inter-relation between flow and geometric parameters has been recognized for the further design of the Coanda nozzle flow. The result presented in this paper can serve as guidelines for the experimental investigation of such type of flow in order to have better insight into the flow.

CITATION: Subhash, M. and Dumas, A., "Computational Study of Coanda Adhesion Over Curved Surface," *SAE Int. J. Aerosp.* 6(1):2013, doi:10.4271/2013-01-2302.

INTRODUCTION

Due to the exponential growth in air-transport in last few decades led to the emphasis on the development of the V/STOL for military mission as well as for humanitarian operations. To meet such requirement many systems have been proposed for V/STOL for air-vehicles[1, 2, 3, 4]. It has been recognized that the most adequate method to develop V/STOL air- vehicles is certainly based on thrust- vectoring. In this line of research Project ACHEON comprises a thrust vectoring propulsive nozzle named HOMER (High-speed Orienting Momentum with Enhanced Reversibility) that is supported in a patent developed at University of Modena and Reggio-Emilia, Italy [5]. The system includes also a plasma actuator that enables to extend the vector range angle of operation of the nozzle as proposed by University of Beira Interior, Portugal. A large bibliographic analysis has been produced by Trancossi [6]. The importance of this research area can be also seen in some literatures [7, 8, 9, 10].

Here an attempt has been made to understand the adhesion of the jet flow on the curved surface, which is also known as Coanda flow [11]. When the jet of air flow tangentially on the curved surface, the jet deviates by the

shear-stresses acting upon contiguous layers of fluid when it adheres to the surface by viscosity. However, the angle of deflection is influenced by the jet velocity. Therefore, the inertia of the fluid (convection) also plays a crucial role. Evidently, wide range of turbulent length scale exist in the Coanda flow. In Navier-Stokes equations, one should consider the effect of inertia and shearstress for the analytical modelling of the phenomena [12].

Due to the inertia of the flow, pressure difference along and cross the flow has been found. As the flow start adhering the curved surface, the boundary layer forms due to the viscous effect near the wall. Therefore, the thickness of boundary layer can be designated as δ and the radius of the surface can be R . An investigation has been started for the ratio of δ/R , in order to see the separation of the flow from the surface.

There are two types of separation induced by the flow, one is pressure induced separation and other is geometrically induced separation. In order to understand the Coanda effect, the numerical study of the flow on the curved surface with radius 0.04m and 0.08m with Reynolds number range from about 2700 to 10000 has been performed with one equation turbulence model i.e., Spalart-Allmaras (SA) [13] in order to

have optimum model and grid independence check. The grid has been resolved until the viscous sub-layer.

The position of the wall shear stress for Spalart-Alamaras and the K- ω SST [14] is same. But, the value of the wall shear stress has shown the difference less than 0.1%. Therefore, in order to resolve the deflection angle of the jet, the one equation model S-A (Spalart-Allmaras) equations serves well. Therefore, it has been decided to use the S-A equations rather than two equations model, i.e., K- ω SST model.

The characteristics of the wall shear-stress has been plotted in terms of the skin-friction co-efficient against the angular position along down-stream of the flow. It has been seen that the $\partial\tau_w/\partial s$ is minimum at the verge of separation. As the flow progresses on the curved surface the flow velocity retards towards the down-flow, therefore, the wall shear stress get decreased and the boundary layer becomes thicker than before.

GOVERNING EQUATIONS

The mass and momentum conservation equations are given in the Reynolds Averaging form as follows:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 U_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} \overline{u_i u_j} \quad (2)$$

Spalart-Allmaras Model

The Spalart-Allmaras equations are given below:

$$\begin{aligned} \frac{\partial \hat{v}}{\partial t} + u_j \frac{\partial \hat{v}}{\partial x_j} = c_{bl} (1 - f_{t2}) \hat{S} \hat{v} - \left[c_{wl} f_w - \frac{c_{bl}}{\kappa^2} f_{t2} \right] \left(\frac{\hat{v}}{d} \right)^2 + \\ + \frac{1}{\sigma} \left[\frac{\partial}{\partial x_j} \left((\nu + \hat{v}) \frac{\partial \hat{v}}{\partial x_j} \right) + c_{b2} \frac{\partial \hat{v}}{\partial x_i} \frac{\partial \hat{v}}{\partial x_i} \right] \end{aligned} \quad (3)$$

and the turbulent eddy viscosity is computed from

$$\mu_t = \rho \cdot \hat{v} \cdot f_{vl} \quad (4)$$

where,

$$f_{vl} = \frac{\chi^3}{\chi^3 + c_{vl}^3} \quad (5)$$

$$\chi = \frac{\hat{v}}{\nu} \quad (6)$$

and ρ is the density, $\nu = \mu/\rho$ is the molecular kinematic viscosity, and μ is the molecular dynamic viscosity. Additional definitions are given by the following equations:

$$\hat{S} = \Omega + \frac{\nu}{\kappa^2 d^2} f_{v2} \quad (7)$$

where $\Omega = \sqrt{2W_{ij}W_{ij}}$ is the magnitude of the vorticity, d is the distance from the field point to the nearest wall, and

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{vl}} \quad (8)$$

$$f_w = g \left[\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right]^{1/6} \quad (9)$$

$$g = r + c_{w2} (r^6 - r) \quad (10)$$

$$r = \min \left[\frac{\hat{v}}{S \kappa^2 d^2}, 10 \right] \quad (11)$$

$$f_{t2} = c_{t3} \exp(-c_{t4} \chi^2) \quad (12)$$

$$W_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \quad (13)$$

The constants are:

$$\begin{aligned} c_{bl} = 0.1355, \quad \sigma = 2/3, \quad c_{b2} = 0.622, \\ \kappa = 0.41, \quad c_{w2} = 0.3, \quad c_{w3} = 2, \\ c_{vl} = 7.1, \quad c_{t3} = 1.2; \quad c_{t4} = 0.5, \\ c_{wl} = \frac{c_{bl}}{\kappa^2} + \frac{1 + c_{b2}}{\sigma} \end{aligned}$$

k-epsilon Model [15]

The equation are presented below:

$$\rho \frac{\partial k}{\partial t} + \rho u_j \nabla k = \nabla \left[\left(\mu + \frac{\mu_T}{\sigma_k} \right) \nabla k \right] + \mu_T P - \rho \varepsilon \quad (14)$$

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho u_j \nabla \varepsilon = \nabla \left[\left(\mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} \mu_T P - \rho C_{\varepsilon 2} \varepsilon) \quad (15)$$

where,

$$P = \nabla u \times [\nabla u + (\nabla u)^T] \quad (16)$$

$$\mu_T = \rho C_\mu \frac{k^2}{\varepsilon} \quad (17)$$

The model constants are given below:

$$C_\mu = 0.09, C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, \\ \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

SST k- ω Model

The SST k- ω model is given by

$$v_T = \frac{a_1 \cdot k}{\max(a_1 \omega, SF_2)} \quad (18)$$

$$S^2 = \left| \frac{1}{2} (\partial_j u_i + \partial_i u_j) \right|^2 \quad (19)$$

$$F_2 = \tanh \left[\left[\min \left(\max \left(\frac{2\sqrt{k}}{\beta \omega y}, \frac{500v}{y^2 \omega} \right), 100 \right) \right]^2 \right] \quad (20)$$

$$\rho \frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta k \omega + \left[(v + \sigma_k v_T) \frac{\partial k}{\partial x_j} \right] \quad (21)$$

$$P_k = \min(G, 10\beta k \omega) \quad (22)$$

$$G = v_T \frac{\partial U_i}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (23)$$

$$CD_{k\omega} = 2\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial k}{\partial x_j} \\ \left(\frac{\sqrt{k}}{\beta \omega y}, \frac{500v}{y^2 \omega} \right), \frac{4\sigma_{\omega 2} k}{CD_{k\omega}^{+y^2}} \quad (24)$$

$$\max \\ \left\{ \min(\min[, 10]) \right\}^4 \\ F_1 = \tanh \quad (25)$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(v + \sigma_\omega v_T) \frac{\partial \omega}{\partial x_j} \right] + \\ + 2(1 - F_1 \sigma_{\omega 2}) \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (26)$$

SYSTEM MODEL

In this analysis a very simple geometrical model has been adopted and has been reported in Fig.1.

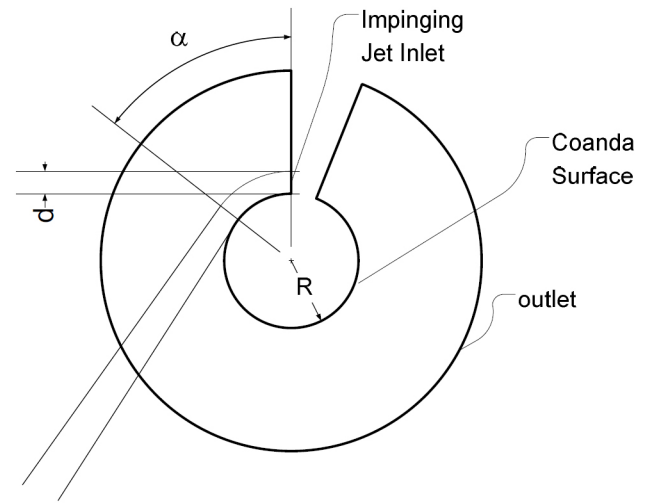


Figure 1. Geometry of the problem

The system can be then be geometrically described by d_{jet}/R , where, d_{jet} is the thickness of the impinging jet inlet, by Re_d which describes the jet speed and physical properties.

$$Re_d = \frac{\rho \cdot u \cdot d_{jet}}{\mu} = \frac{u \cdot d_{jet}}{\nu} \quad (27)$$

SOLUTION METHODOLOGY AND GRID INDEPENDENCE CHECK

The grid has been generated using the commercial software Gambit [16] and the boundary layer has been resolved appropriately as shown in Fig.2-3. Computation has been performed using the commercial code Fluent 6.3.16 [16].

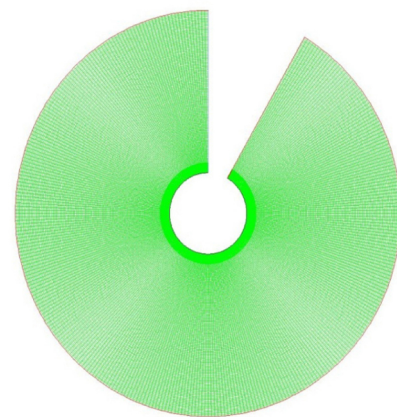


Figure 2. Domain grid

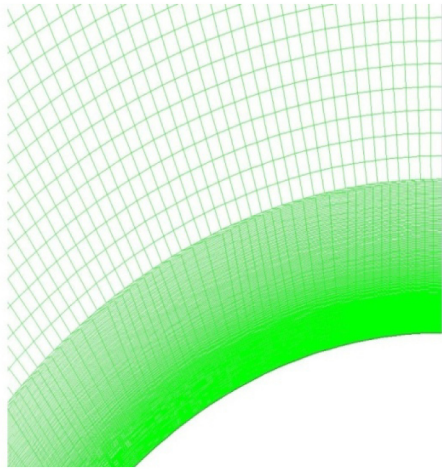


Figure 3. Detail of computational grid near the high speed inlet

The Fig.3 describes the grid generation in detail. Grids are sufficiently fine at the inlet of the jet and other part of the jet which is ambient fluid at rest have larger grid size. The grid independence check has been performed according to the ERCOFTAC [17] guidelines and as depicted in the following papers[18-19]. In this present paper the spatial discretization error has been minimized using the second order upwind scheme for the momentum equation and for modified turbulent viscosity in Spalart-Allmaras Model (Eq. 3). The pressure and velocity has been coupled through the SIMPLE (Semi-Implicit Method for Pressure Linked Equation) algorithm [20].

The optimum number of grid (numerically stable grid) has been determined through the numerical computation of the grid at different refinement level of the grid at the curved surface (first grid from the wall at 80 μm , 40 μm and 20 μm). It has been found that, when the grid resolved the viscous sub-layer until y^+ value less than two (first grid from the wall at 20 μm), then one can get the jet deflection angle independent of the grid. Also, the x^+ value is less than 60 for the stable solution of the flow along the downstream.

First order implicit method has been used to discretize unsteady term. The advantage of the fully implicit scheme is that it is unconditionally stable with respect to time step size. However, the time steps has been taken $\Delta t = 1 \times 10^{-3}$ s. The residual RMS plot (Fig.4, 5, 6) of all the three models has been given, it has been found that for the SA model the error are in the range of 10^{-9} - 10^{-10} , that is lowest in comparison to the other turbulence model.

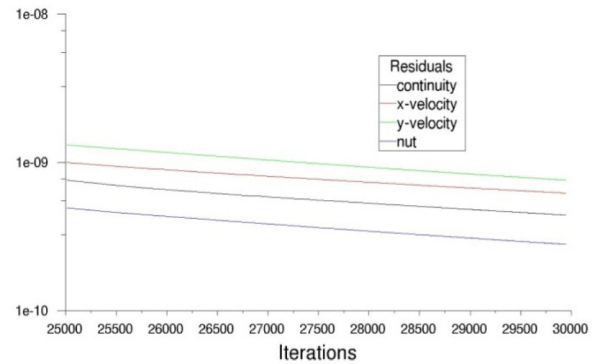


Figure 4. Residual plot for SA turbulence model

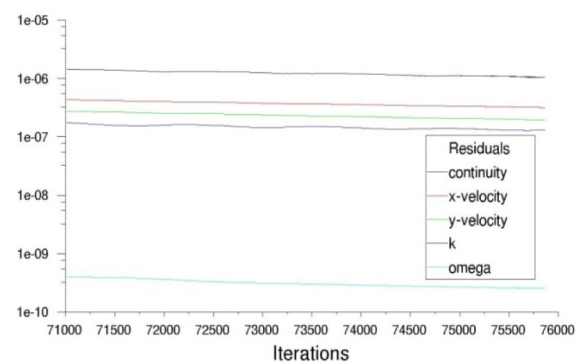


Figure 5. Residual plot for SST k- ω turbulence

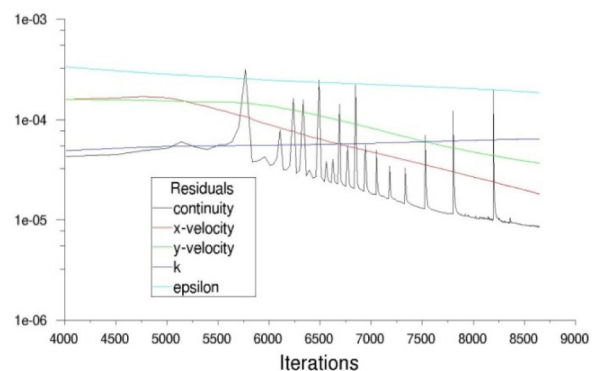


Figure 6. Residual plot for k- ϵ different turbulence models

The jet deflection angle and the boundary layer thickness at the verge of separation have been reported in Table 1 and contours have been shown in Fig.7, 8, 9, for different turbulence model. It has been seen that for SST k- ω and SA model has minimum difference. However, the residual RMS error in the SA model (Fig.4) is least. Therefore, for computational study SA model has been employed. This fact has been also reported by the other researchers [21, 22, 23].

The reason for the high deflection angle for the k- ϵ model is that, this model take the isotropic nature of turbulence, but