11TH INTERNATIONAL SCIENTIFIC CONFERENCE
ON DEFENSIVE TECHNOLOGIES
OTEH 2024
Serbia, Tara, 9-11 October 2024

COMPUTING AERODYNAMIC DAMPING IN ROLL AND PITCH OF A SUPERSONIC FINNER MODEL

JELENA SVORCAN
University of Belgrade, Faculty of Mechanical Engineering, Belgrade, jsvorcan@mas.bg.ac.rs

TONI IVANOV
University of Belgrade, Faculty of Mechanical Engineering, Belgrade, tivanov@mas.bg.ac.rs

OGNJEN PEKOVIĆ
University of Belgrade, Faculty of Mechanical Engineering, Belgrade, opekovic@mas.bg.ac.rs

ALEKSANDAR SIMONOVIĆ
University of Belgrade, Faculty of Mechanical Engineering, Belgrade, asimonovic@mas.bg.ac.rs

Abstract: *Accurate aerodynamic coefficients and their derivatives are crucial for flight dynamics analysis and appropriate control definition. Even when the investigated geometry is simple (e.g. a revolution body, with simple control or stabilizing surfaces), determination of its aerodynamic derivatives, particularly at supersonic regimes, is rather challenging, and usually includes extensive computational and experimental campaigns. This paper computationally investigates quasi-steady or unsteady supersonic flows that include rigid body motion effects around the Army-Navy Finner geometry (a basic, revolution body with L/D = 10 fineness, slender, conical nose section, and wedge-section fins in × configuration). To elucidate the flow fields, the Reynolds-averaged Navier-Stokes equations are closed by k-ω SST turbulence model. Roll and pitch damping are estimated by different approaches, and the obtained values are compared mutually, but also to the available experimental data. In addition, some representative flow visualizations are included.*

Keywords: *roll damping, pitch damping, supersonic finner model, CFD.*

1. INTRODUCTION

The correct estimation of aerodynamic coefficients (both numerical and experimental) of objects flying at supersonic speeds is a prerequisite for their subsequent flight dynamics analyses and appropriate control definition, as well as their overall performance. For that reason, comparative numerical and experimental studies have been conducted for decades to ensure greater levels of detail, acquired knowledge and useful data [1-8]. While static aerodynamic coefficients can be computed and measured with acceptable accuracy, dynamic coefficients (assumed in the form of Taylor series) and their corresponding derivatives still pose quite a challenge to both experimental and computational aerodynamics (due to the increased complexity, expenses and sensitivity to outer instigations).

There are several different standard models (geometries), designed for higher velocities, that are typically used for validation of experimental set-ups and installations, but also numerical codes. One of them is Army-Navy Finner geometry, a simple revolution body with four aft stabilizing/control surfaces [1-5,8]. Here, it is chosen for its geometric simplicity and great amount of data available for comparison, and used to check the ability of flow simulations performed in ANSYS FLUENT to accurately estimate aerodynamic damping, while keeping the numerical set-up as simple as possible, as will be demonstrated in the following sections.

2. MODEL DESCRIPTION

The Army-Navy Finner model comprises an axisymmetric body with four rear wedge-shaped stabilizing/control surfaces arranged in × configuration, as illustrated in Fig. 1. Body-fixed coordinate system is used, attached to the model center-of-gravity (CG), where x-axis points forward, y-axis to the right side, and z-axis downward. Here, body length is L = 0.5 m, and fineness is L/D = 10. More geometric details are available in [5].

Similarly to [5], experimentally-determined aerodynamic roll and pitch damping data from the Arnold Engineering Development Center (AEDC) Supersonic Wind Tunnel facility are used to validate the current numerical results.

Corresponding author: J. Svorcan;
E-mail: jsvorcan@mas.bg.ac.rs
DOI: 10.5937/OTEH24005S

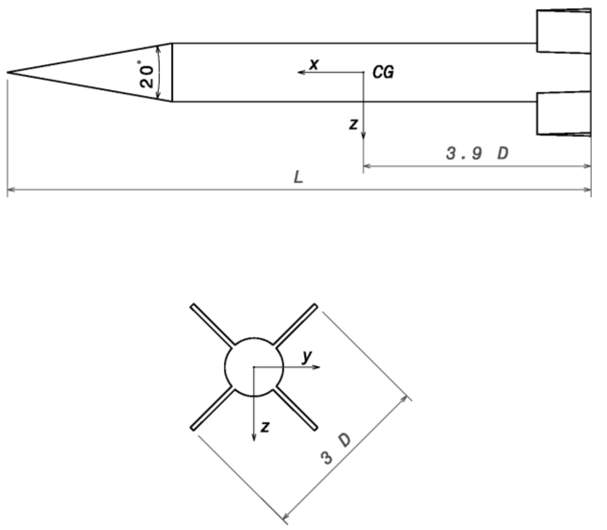

Figure 1. Investigated model and used coordinate system

3. FLOW SIMULATION

The following subsections describe all the steps of the performed flow computations, starting from the geometric model, formed computational meshes, to setting up the problems in ANSYS FLUENT.

3.1. Geometry

The finner geometry is modeled completely in accordance with the previously mentioned geometric features. The surrounding computational domain is cylindrical, extending 10 L in axial direction, starting 1.5 m fore and ending 3.5 m aft of the coordinate beginning (where CG is located 195 mm from the base), and with the diameter of 8 L.

3.2. Computational mesh

Since the starting geometry is rather simple, so are the generated unstructured, hybrid meshes. To ensure that grids do not significantly affect the results, two different meshes are generated (containing approximately 1.4 and 2.4 million control volumes, respectively) and analyzed (by comparing the coefficients of x-components of the computed aerodynamic force). In both cases, global sizing is prescribed (with limited maximal edge and face sizes, and with very fine minimal, proximity and defeature sizes) to adequately represent the model surfaces. Also, several layers of thin, prismatic cells encompass the walls to better capture considerable changes of flow quantities within the thin boundary layer. The first cell height is $y_1 = 0.01$ mm resulting in the satisfactory value of dimensionless wall distance ($y^+ < 5$).

Details of the generated finer computational mesh are given in Fig. 2.

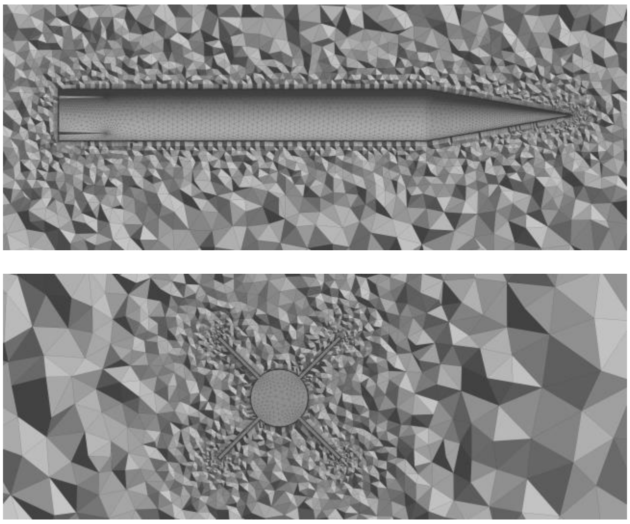

Figure 2. Details of the generated computational grid

3.3. Initial, steady flow case

In both cases (for both roll and pitch damping), a converged steady case and solution are required. ANSYS FLUENT is employed for resolving the flow fields, i.e. solving the governing flow equations by finite volume method. In this initial case, the flow is considered steady, compressible (density-based solver is used), turbulent (k-ω SST turbulence model is employed for closing the flow equations), fluid (air) is assumed as an ideal gas, where the viscosity changes with temperature are modeled according to the Sutherland law.

The following boundary conditions are defined along all the outer boundaries: Mach number is M = 2.49, angle-of-attack (AoA) is $\alpha = 0°$, static pressure is $p = 2664$ Pa, and static temperature is $T = 139$ K. These static values are defined so that they match the total values in the wind tunnel during the experiments. The reference values used for the computation of aerodynamic coefficients are $A_{\mathrm{ref}} = 0.002$ m^2 and $l_{\mathrm{ref}} = 0.05$ m.

Implicit numerical formulation is employed, with Roe-FDS flux type, and all numerical schemes of 2nd order, where CFL = 2. Aerodynamic coefficients of forces (C_X and C_Z) and moments (C_l and C_m) were monitored, and computations were performed until reaching their convergence (usually 1000 iterations). The expressions of interest are:

$$C_X = \frac{X}{\frac{1}{2}\rho V^2 A_{ref}} = \frac{X}{\frac{1}{2}\kappa p \mathrm{M}^2 A_{ref}}, \quad (1)$$

$$C_Z = \frac{Z}{\frac{1}{2}\rho V^2 A_{ref}} = \frac{Z}{\frac{1}{2}\kappa p \mathrm{M}^2 A_{ref}}, \quad (2)$$

$$C_l = \frac{L_x}{\frac{1}{2}\rho V^2 A_{ref} l_{ref}} = \frac{L_x}{\frac{1}{2}\kappa p \mathrm{M}^2 A_{ref} l_{ref}}, \quad (3)$$

$$C_m = \frac{M_y}{\frac{1}{2}\rho V^2 A_{ref} l_{ref}} = \frac{M_y}{\frac{1}{2}\kappa p \mathrm{M}^2 A_{ref} l_{ref}}, \tag{4}$$

where X and Z are components of aerodynamic force along x- and z-axis respectively, L_x and M_y are roll and pitch moments respectively, ρ is fluid density and $\kappa = 1.4$ is ratio of specific heat.

3.4. Forced continuous roll

To determine the damping in roll, it is possible to perform quasi-steady simulations assuming rotation of the complete computational domain. Here, frame motion is assumed, and additional inertial terms are simply added to the flow equations. This approach is computationally much less expensive than unsteady simulations with actual rotation of the portion of the computational domain. Otherwise, the numerical setting is quite similar to the previously described.

The flow is steady, compressible and turbulent, and air is considered as ideal gas. Frame motion around the axial, x-axis is defined with p = 353 rad/s. Computations are performed until achieving the convergence of the roll coefficient C_l. The obtained results are then post-processed to determine the roll damping (roll derivative with respect to angle rate p) according to the equations:

$$\frac{\partial C_l}{\partial p} \approx \frac{\Delta C_l}{\Delta p}, C_{l_p} = \frac{\frac{\partial C_l}{\partial p}}{\frac{l_{ref}}{2V}} \approx \frac{2V}{l_{ref}} \frac{\Delta C_l}{\Delta p}. \tag{5}$$

3.5. Forced harmonic pitch

To determine the damping in pitch, it is necessary to perform unsteady flow simulations. However, instead of rotating the model and portion of the domain closely surrounding it (which is computationally rather expensive), it is possible to assign variable conditions at the boundaries, in particular the harmonic (sine shaped) change of the angle-of-attack:

$$\alpha(t) = \alpha_0 + A\sin(\omega t), q = \dot{\alpha} = A\omega\cos(\omega t), \tag{6}$$

where $\alpha_0 = 0°$, $A = 0.5°$ and $\omega = 3764$ rad/s.

By writing and including user-defined functions (UDFs) governing the change of velocity direction, in every timestep velocity components corresponding to the harmonic (sine) change of AoA are recomputed and assigned anew to the outer boundaries.

Other than this difference in BCs, the flow is compressible and turbulent, and the remainder of the numerical set-up is the same as before.

Again, during the computation, the periodic aerodynamic coefficients are monitored and registered, and afterwards, their derivatives are determined from the saved files/data:

$$\begin{aligned} C_m(t) &= C_{m_o} + Ak\left(C_{m_{\dot{\alpha}}} + C_{m_q}\right)\cos(\omega t) \\ &= C_{m_o} + Ak\bar{C}_{m_q}\cos(\omega t), k = \omega D/2V, \end{aligned} \tag{7}$$

where D is the model diameter. We can assess the derivative of interest:

$$\bar{C}_{m_q} = \frac{2V}{n\pi AD}\int_{t_a}^{t_b} C_m(t)\cos(\omega t)\,\mathrm{d}t, \tag{8}$$

by numerically computing this integral. The difference $(t_b - t_a) = nT_t = 2\pi n/\omega$ is the integer (n) number of time periods T_t. Usually, the computations were performed for at least 5-10 periods T_t, and the effect of time step dt was also investigated (d$t = T_t/180$ and d$t = T_t/360$).

4. RESULTS AND DISCUSSION

Illustrations of the computed steady, nearly axisymmetric flow at zero AoA by Mach number contours and vortical structures are given in Figs. 3 and 4. The initial oblique shock is apparent, as well as the smaller ones forming at the wedges, and the considerable wake induced by the model base.

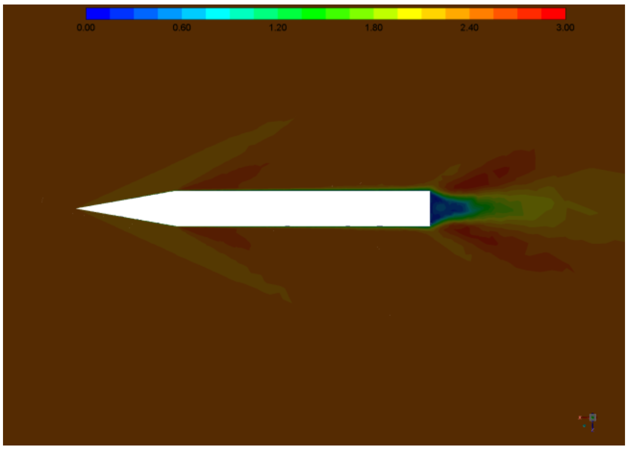

Figure 3. Computed velocity field in the symmetry xz-plane

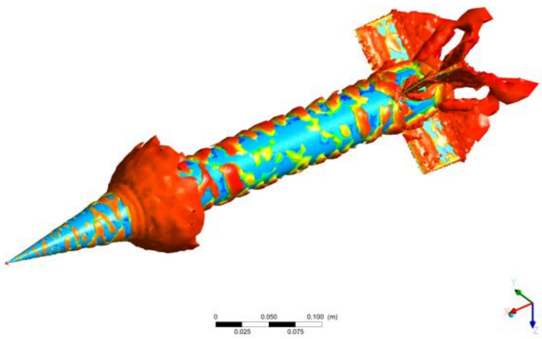

Figure 4. Computed vorticity structures originating from the model surfaces

The computed damping in roll and pitch are provided in Tables 1 and 2. Given the sensitivity of these derivatives and simplified computational set-ups, it can be concluded that rather satisfactory correspondence between the two sets of data is achieved.

Table 1. Computed and measured damping in roll

Mode	C_{l_p}
Numerical (mesh 1)	−16.07
Numerical (mesh 2)	−16.05
Experimental	−16.8 ± 0.8

Table 2. Computed and measured damping in pitch

Mode	$\bar{C}_{m_q}$
Numerical (dt = T_t/180)	−135
Numerical (dt = T_t/360)	−139
Experimental	−180 ± 45

The registered (resulting periodic) change/output in aerodynamic pitch coefficients with respect to time and AoA is illustrated in Fig. 5.

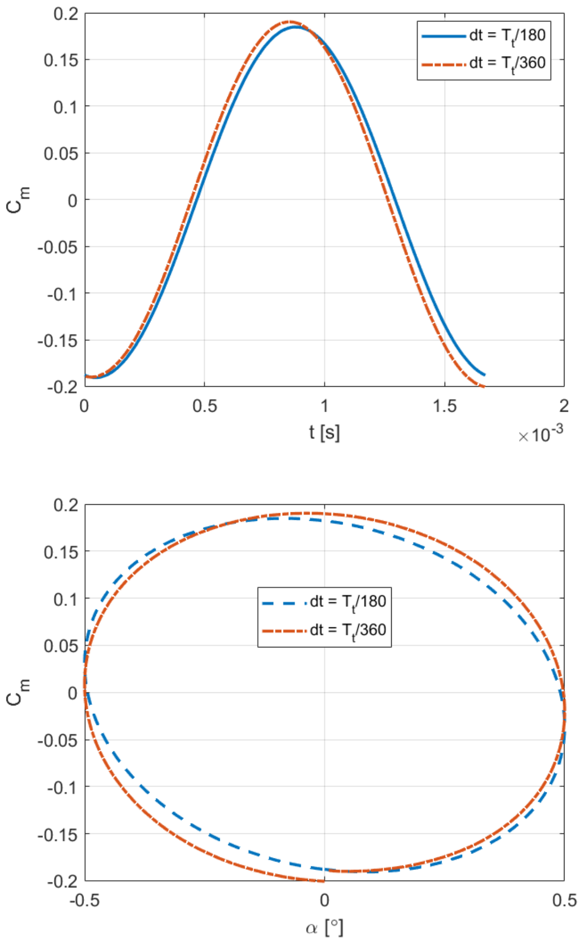

Figure 5. Periodic response to forced harmonic pitch motion

5. CONCLUSION

This study confirms the considerable sensitivity of aerodynamic derivatives in roll and pitch that significantly change with Mach and Reynolds numbers, as well as angles-of-attack. Furthermore, their accurate determination, both experimental and numerical, is extremely difficult and costly (in both time and resources). For the time being, by employing standard numerical models, it is possible to estimate aerodynamic damping with approximate relative error amounting to 20%. In future studies, to achieve higher levels of accuracy, finer meshes and more advanced computational approaches should also be considered and tested.

Acknowledgement

This research work was supported by the Ministry of Science, Technological Development and Innovation of the Republic of Serbia through contract No. 451-03-65/2024-03/200105 from 5 February 2024.

References

[1] BHAGWANDIN,V.A., SAHU,J.: *Numerical prediction of pitch damping stability derivatives for finned projectiles*, ARL-TR-6725, U. S. Army Research Laboratory, Aberdeen Proving Ground, 2013.

[2] BHAGWANDIN,V.A., SAHU,J.: *Numerical prediction of pitch damping stability derivatives for finned projectiles*, Journal of Spacecraft and Rockets, 51(5) (2014) 1603-1618.

[3] SAMARDŽIĆ,M., ISAKOVIĆ,J., MILOŠ,M., ANASTASIJEVIĆ,Z., NAUPARAC,D.B.: *Measurement of the direct damping derivative in roll of the two calibration missile models*, FME Transactions 41(3) (2013) 189-194.

[4] SAMARDŽIĆ,M., ISAKOVIĆ,J., ANASTASIJEVIĆ,Z., MARINKOVSKI,D.: *Apparatus for measurement of pitch and yaw damping derivatives in high Reynolds number blowdown wind tunnel*, Measurement: Journal of the International Measurement Confederation 46(8) (2013) 2457-2466.

[5] SHELTON,A., MARTIN,C., SILVA,W.: *Characterizing aerodynamic damping of a supersonic missile with CFD*, AIAA Aerospace Sciences Meeting, Kissimmee, 2018.

[6] GUO,C., REN,Y.-X.: *The computation of the pitch damping stability derivatives of supersonic blunt cones using unsteady sensitivity equations*, Advances in Aerodynamics 1(1) (2019) 17.

[7] TATAR,M., MASDARI,M.: *Investigation of pitch damping derivatives for the Standard Dynamic Model at high angles of attack using neural network*, Aerospace Science and Technology 92 (2019) 685-695.

[8] YAYLA,K.: *Numerical Prediction of Aerodynamic Stability Derivatives of a Projectile*, M.Sc. - Master of Science, Middle East Technical University, 2021.