Effect of base cavities on the stability of the wake behind slender blunt-based axisymmetric bodies

E. Sanmiguel-Rojas,1 J. I. Jiménez-González,1 P. Bohorquez,1 G. Pawlak,2 and C. Martínez-Bazán1
1Área de Mecánica de Fluidos, Universidad de Jaén, Campus de las Lagunillas, 23071 Jaén, Spain
2Department of Ocean and Resources Engineering, University of Hawai’i at Manoa, Honolulu, 96822
Hawaii, USA

(Received 25 January 2011; accepted 10 October 2011; published online 18 November 2011)

We extend our previous research on the instability properties of the laminar incompressible flow of density ρ and viscosity μ, which develops behind a cylindrical body with a rounded nose and length-to-diameter ratio L/D = 2, aligned with a free-stream of velocity w∞ [E. Sanmiguel-Rojas et al., Phys. Fluids 21, 114102 (2009); P. Bohorquez et al., J. Fluid Mech. 676, 110 (2011)]. In particular, we analyze the effects of a cylindrical base cavity of length h and diameter Dc on both critical Reynolds number, Re_c = ρw∞Dc/μ, and drag coefficient, C_d, combining experiments, three-dimensional direct numerical simulations, and global linear stability analyses. The direct numerical simulations and the global stability results predict with precision the stabilizing effect of the cavity on the stationary, three-dimensional bifurcation in the wake as h/D increases. In fact, it is shown that, for a given value of Dc/D, the critical Reynolds number for the steady bifurcation, Re_c, increases monotonically as h/D increases, reaching an asymptotic value which depends on Dc/D, at h/D ≈ 0.7. Likewise, for a fixed value of h/D, we have studied the effect of the cavity diameter Dc/D on the critical Reynolds number. No effect on Re_c is observed over the range 0 ≤ Dc/D ≤ 0.6, but Re_c shows a monotonic growth for 0.6 ≤ Dc/D < 1. On the other hand, for steady flows, the drag coefficient decreases with the length of the cavity reaching an asymptotic minimum for h/D ≥ 0.5 and Dc/D → 1. Similar behavior with the cavity length has been observed experimentally and numerically for the second, oscillatory bifurcation, and its associated critical Reynolds number, Re_or.


I. INTRODUCTION

This study is motivated by the desire to explore mechanisms to stabilize the flow behind slender bodies with a blunt trailing edge or, similarly, bullet-like bodies. In particular, we are interested in characterizing the influence of a base cavity on the threshold for the loss of axisymmetry and the onset of vortex shedding in the laminar incompressible flow behind a slender blunt-based axisymmetric body.

It is well established that, at moderate Reynolds numbers below a critical value, Re < Re_c, the flow around axisymmetric bodies is axisymmetric.1 However, as the Reynolds number increases beyond a first bifurcation threshold associated with a steady state mode, a pair of steady streamwise vortices develop in the wake, breaking the axial symmetry.2,3 Thus, in a range of Reynolds numbers above the first bifurcation, the flow is no longer axisymmetric although it still exhibits a plane of symmetry. For instance, Figure 1 illustrates the instantaneous topology of the steady streamwise vortices developing for a blunt-based cylindrical body with a rounded nose and length-to-diameter ratio L/D = 2. The flow is steady for this case and, although it is symmetric in the plane shown in Fig. 1(a), it is not axisymmetric (note that the flow is not symmetric in the plane shown in Fig. 1(b)). Thus, considering the trajectory of a projectile with this sort of wake structure, we can expect that it will deviate from that resulting from an axisymmetric wake structure. Furthermore, as the Reynolds number increases beyond a critical value, Re > Re_or, a second transition to an oscillatory regime occurs, which retains planar (or reflectional) symmetry.2,3 Consequently, stabilization of the second mode is also of practical importance in order to avoid the potential generation of fluctuating dynamic loads.

There are several mechanisms traditionally used to improve the aerodynamics of afterbodies, i.e., geometrical modifications of the base of the body, base bleed, base suction, among others.4 Because of the practical importance of the flow around slender blunt-based axisymmetric bodies, which is different from that of two-dimensional bodies5 mainly due to the presence of a first steady state mode, most previous work has been devoted towards analysis of the effects of base bleed or of the shape of the base body on the total drag and base pressure in the transonic/supersonic regime at Reynolds numbers with a lower limit of Re ∼ O(10^4), e.g., Morel.6 However, very little is known about the sequence of transitions leading to turbulence in wakes behind bluff bodies at Re ∼ O(10^3) – O(10^4), although this knowledge is essential in order to predict the dependence of fluid dynamics variables, such as the base pressure and the total drag, as a function of Reynolds number and the geometry of the body. Subsequently, Sanmiguel-Rojas et al.2 and Bohorquez et al.3 detailed the physics of the instabilities developing in laminar flows with base bleed for Re < 2000. However, the influence of the shape of...
FIG. 1. Contours of constant streamwise vorticity, $\omega_s = \pm 0.05$, for a solid base body at $Re = 340$ (after Bohorquez et al.\textsuperscript{3}). (a) Plan view and (b) side view. The figure shows that the flow is steady and exhibits planar symmetry.

the body base, in particular, the effect of a cavity at the base of the body on wake stability has not previously been studied.

Following Viswanath,\textsuperscript{4} detailed flow field measurements are necessary to shed light on the mechanisms associated with vortex shedding around a base cavity. Therefore, this paper is aimed at analyzing the physics of the wakes generated downstream of bullet-like bodies with cylindrical base cavities with varying diameters and lengths at moderate Reynolds numbers, combining experiments, direct numerical simulations, and global linear stability analyses. We start with the description of the numerical methods and experimental techniques in Sec. II. Next, the cavity effects on the first-stationary and second-oscillatory bifurcations are presented in Secs. III A and III B, respectively. Finally, we present conclusions in Sec. IV.

II. NUMERICAL METHODS AND EXPERIMENTAL TECHNIQUES

This section describes the numerical and experimental techniques used in the study of the flow generated downstream of a cylindrical slender body of diameter $D$ and total length $L = 2D$, with an elliptical rounded nose with a 2:1 major-to-minor axis ratio and a blunt trailing edge, aligned with a free-stream of velocity $w_\infty$. The trailing edge was modified by the addition of a cavity inside the body of length $h$ and inner diameter $D_c$, as shown in the flow configuration displayed in Fig. 2. An estimation of the discretization errors is also included as verification of the numerical results.

A. Numerical methods

The flow of a viscous fluid of kinematic viscosity $\nu$ and density $\rho$ around the body is governed by the homogeneous incompressible Navier-Stokes equations

$$\nabla \cdot \mathbf{v} = 0, \quad \frac{\partial \mathbf{v}}{\partial t} + \nabla \cdot (\mathbf{v} \mathbf{v}) = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{v}, \quad (1)$$

where $\mathbf{v} = (u,v,w)^T$ is the velocity field in cylindrical-polar coordinates $(r, \theta, z)$. The equations were made dimensionless, using $D$, $w_\infty$, $D/w_\infty$, and $\rho w_\infty^2$ as length, velocity, time, and pressure scales, respectively, so that the Reynolds number is $Re = w_\infty D/\nu$.

The computational domain consists of a body of length $L/D = 2$, with cavities of diameter $0 \leq D_c/D < 1$ and length $0 \leq h/D \leq 1$, within a coaxial external cylindrical surface of diameter $20 \, D$ and length $60 \, D$. The origin of the coordinate system is located at the base of the body (at the entrance of the cavity), and the outflow on the external cylindrical surface is located $50 \, D$ downstream from the base of the body. Sanmiguel-Rojas et al.\textsuperscript{2} and Bohorquez et al.\textsuperscript{3} have shown that this downstream computational length is sufficient to capture the physics of the problem and to provide an accurate description of the bifurcation process.

We imposed a uniform fluid stream of velocity $w_\infty$ at the inlet of the external cylindrical surface ($z = -10$), no-slip boundary conditions $\mathbf{v} = 0$ on the solid walls of the body, slip boundary conditions $\mathbf{n} \cdot \mathbf{v} = 0$ at the far field external cylindrical surface ($r = 10$), where $\mathbf{n}$ is the outward unit normal vector, and the outflow boundary conditions $\mathbf{n} \mathbf{v} - \nabla \cdot \mathbf{v} \cdot \mathbf{n}/Re = 0$ at the outlet of the computational domain ($z = 50$).

The axisymmetric and the three-dimensional numerical simulations reported in the present work were performed using openfoam\textsuperscript{®}, an open source Computational Fluid Dynamics software package produced by OpenCFD Ltd. (http://www.openfoam.com), based on the finite volume method. The spatial discretization was performed using a van Leer limiter total variation diminishing scheme, TVD, and the time was advanced with a Crank-Nicholson scheme, both of second order of accuracy (see Bohorquez et al.\textsuperscript{3} for further numerical details).

The simplest way to estimate the discretization errors is that based on the Richardson\textsuperscript{7} extrapolation method which assumes that calculations can be done on a grid sufficiently fine to achieve a monotonic convergence. In fact, in a grid refinement study, the value of a given variable estimated from the Richardson extrapolation is the value that would be obtained if the cell grid size tended to zero. Roache\textsuperscript{6} generalized the Richardson extrapolation by introducing $n$th-order methods to provide an estimation of the grid-independent solution as

$$f_{\text{exact}} \approx f_j + \frac{f_{j+1} - f_j}{\ell^n - 1}, \quad (2)$$

where $f$ is any field value or integral quantity discretized on fine ($j$) and coarse ($j+1$) meshes, $\ell$ is the fine-to-coarse grid size refinement ratio in a coordinate direction, and $n$ is the order or convergence rate of the method. The order may be computed from three successive grids using the following expression (see Ferziger and Peric\textsuperscript{8}):

$$n = \frac{\log \left( \frac{f_{j+2} - f_{j+1}}{f_{j+1} - f_j} \right) \log(\ell)}{\log(\ell)}. \quad (3)$$

To ensure that monotonic convergence is reached, the convergence ratio, $R = (f_j+1 - f_j)/(f_j+2 - f_{j+1})$, must fall within the interval $0 < R < 1$ before applying the Richardson
extrapolation. Thus, the grid convergence index can be defined from Eq. (2) as

$$GCI_{j+1,j}(\%) = 3 \left[ \frac{f_{j+1} - f_j}{f_1} \right] \times 100,$$

as a measure of how well the numerical solution approaches the grid-independent, asymptotic value. In addition, the grid convergence index provides a conservative convergence measure for grid refinement studies, based on estimated fractional error derived from the generalization of Richardson extrapolation (see Roache).

Table I summarizes the results of a convergence study performed using three different grids with a refinement ratio of $\ell = 2^{1/3}$: fine (1), medium (2), and coarse (3). In this case, the study has been based on the drag coefficient (see Sec. III A), $C_D$, obtained for a body with a cavity length $h/D = 0.7$ and diameter $D_c/D = 29/30$ at $Re = 400$. Notice that the convergence condition for $C_D$ is monotonic since $R = 0.67 < 1$. The estimated order of the method, $n \approx 1.73$, is slightly lower than the theoretical truncation error (second order in this case) due to the fact that we have used limiter in our method, see Ferziger and Peric. Table I shows that there is a decrease in the grid convergence index as the grid is refined, $GCI_{2,1} < GCI_{3,2}$. However, the grid convergence index obtained with the finest grid, $GCI_{2,1}$, is only slightly lower than that obtained with the medium one, $GCI_{3,2}$, indicating that both meshes are close to the grid-independence solution, and that a further refinement of the grid will not provide substantial improvement in the numerical results. Thus, since the computational time increases by a factor of four in the time-dependent simulations, the medium mesh, consisting of $106 \times 60 \times 460$ nodes, was judged to have the best precision to computational cost ratio and was selected to perform all the calculations included in this work. Parallel computations were carried out in 8 mesh blocks. Fig. 3 shows a detail of the mesh used on the surface of the body.

In addition to performing three-dimensional numerical simulations, we also carried out a global, linear stability analysis. For the stability analysis, the flow was decomposed into an axisymmetric mean base field, $(V, P) = [U, 0, W, P] (r, z)$, and a three-dimensional unsteady disturbance flow, $q = [q, p'](r, 0, z, t)$. Since this base flow is axisymmetric, the three-dimensional perturbation may be decomposed into both time and azimuthal exponential dependencies, $q = [q(r, z), p(r, z)]^T \exp(\alpha t + im\beta)$, where $\alpha$ and $\beta$ are the growth rate and the angular frequency of the global mode, respectively. Solving the eigenvalue problem, $\mathbf{A} \mathbf{q} = \sigma \mathbf{q}$ (see Sammiguel-Rojas et al.,5 for more details on the linear matrices $A$ and $B$). We used homogeneous boundary conditions for $\mathbf{v}$ at the inlet, solid walls and far field external cylindrical surface, the outflow boundary conditions $\mathbf{p} n - \nabla \mathbf{v} \cdot \mathbf{n} / Re = 0$ at the outlet and, finally, the axisymmetric boundary conditions at the axis $(r = 0)$. The linear operators of matrices $A$ and $B$ were discretized using compact finite difference schemes with truncation error of sixth order for non-uniform meshes.11,12 Compared to the traditional finite difference expressions, the compact finite difference schemes provide a better representation of the shortest length scales. As such, they approach spectral methods while allowing improved freedom in the choice of mesh geometry. Figure 4 shows a detail, inside the cavity, of a typical axisymmetric mesh used to perform global linear stability analyses.

As in the three-dimensional case, Table II shows the convergence study performed, based on the growth rate associated to the leading eigenvalue, $\sigma$, for a body with a cavity of length $h/D = 0.7$ and diameter $D_c/D = 29/30$ at $Re = 400$ and $|m| = 1$. In this case, a single refinement rate of $\ell = 2^{1/2}$ was applied for all the spatial coordinates and, as in the three-dimensional study, three meshes were used: fine (1), medium (2), and coarse (3). The convergence ratio was $R = 0.29 < 1$ and the estimated global order of the method was $n \approx 3.54$, which resulted in a much lower value than the theoretical truncation error (order six). The difference between the theoretical and estimated orders of the
numerical method is a consequence of the use of fourth order stencils for the compact finite differences in the boundary conditions to reduce the band wide of the matrix $A$, which considerably decreased the computational cost of the calculations. Therefore, the global order of our method was clearly dominated by the truncation error of the boundary conditions. On the other hand, the local refinement of the grid close to the corners was limited by the time step for the three-dimensional unsteady simulations, and by the iteration rate for the linear stability study, so that corner singularities can affect convergence rates for both methods. Table II shows that the grid convergence index is reduced by a factor of three for the successive grid refinements, $GCI_{3,1} < GCI_{3,2}$. However, considering that $GCI_{3,2}$ overestimates the relative error between the solution obtained with the medium mesh and the grid-independent solution (the actual relative error for the medium mesh solution is $< 5\%$), and the fact that the computational time increases by nearly a factor of four between the medium (2) and fine (1) meshes, we performed the global stability analyses using the medium mesh (composed by 218 $\times$ 690 nodes).

With approximately $1.2 \times 10^5$ grids points, the eigenvalue problem was too large to be solved with a standard QZ-algorithm. Therefore, the shift-invert-Arnoldi method based on the Krylov subspace, using both the Jacobian-free and the generalized minimum residual methods, was applied to reduce and invert matrix $A$ in order to find the eigenvalues in the complex plane.

**B. Experimental set-up**

The experiments were carried out in a vertical wind tunnel of $20 \times 20$ cm$^2$ cross section. Different bodies made of aluminium, with diameter $D = 1$ cm and length $L = 2$ cm and cavity diameter $D_c/D = 0.86$ and lengths $h/D = 0$, 0.2, 0.4, 0.7, and 1.0 were used to determine the effect of the cavity length on the stability properties of the flow. The bodies were carefully aligned with the free-stream and held with a 1 mm diameter supporting rod (see Fig. 5). A hot-wire anemometer was used to measure the fluctuations of the stream-wise component of the velocity, $w'(t)$, at a given position downstream from the body base. The temperature was monitored to precisely calculate the density and viscosity of the air stream. Details on the experimental apparatus can be found in Bohorquez et al.

### TABLE II. Growth rate for the leading eigenvalue and grid convergence index obtained for a body with a cavity of length $b/D = 0.7$ and $D_c/D = 0.8$.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>$n_r$</th>
<th>$n_z$</th>
<th>Mesh nodes</th>
<th>$\sigma_r$</th>
<th>$GCI_{r+1,j}$ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>308</td>
<td>975</td>
<td>$\sim 2.4 \times 10^5$</td>
<td>-0.02858</td>
<td>1.61462</td>
</tr>
<tr>
<td>2</td>
<td>218</td>
<td>690</td>
<td>$\sim 1.2 \times 10^5$</td>
<td>-0.02895</td>
<td>5.42818</td>
</tr>
<tr>
<td>3</td>
<td>154</td>
<td>488</td>
<td>$\sim 0.6 \times 10^5$</td>
<td>-0.03021</td>
<td>—</td>
</tr>
</tbody>
</table>

**III. RESULTS AND DISCUSSION**

In this section, we examine the effect of a base cavity on the stability properties of the flow past a slender blunt-based axisymmetric body. We have limited the study to a body of length-to-diameter ratio, $L/D = 2$, Global linear stability analyses, unsteady three-dimensional numerical simulations, and wind tunnel experiments were performed to determine the critical Reynolds numbers for the first-steady and the second-oscillatory bifurcations and their dependence with the cavity dimensions. A validation study, comparing the numerical solutions with reference experimental data, is also included to estimate the modeling errors.

**A. Cavity effects on the first-stationary bifurcation**

Following the global linear stability analysis performed with compact finite differences, as described above, for a solid base body of length $L/D = 2$ and Reynolds numbers higher than $Re_{cr} \simeq 326.2$, the flow experiences a first transition to a steady, non-axisymmetric regime, although it exhibits planar symmetry. This first bifurcation corresponds to a

![FIG. 5. Scheme of the experimental facility.](image-url)
three-dimensional \((m = 1)\) and steady \((\sigma = 0)\) perturbation. This result is in excellent agreement with that obtained for the same body using spectral methods by Sanmiguel-Rojas et al.,\(^3\) \(Re_{c,s} \approx 327\), with a relative error in the prediction of the critical Reynolds number smaller than 0.25%, or the value found by three-dimensional numerical simulations by Bohorquez et al.,\(^3\) \(Re_{c,s} \approx 319\).

To show the effect of the cavity length, \(h/D\), on the critical Reynolds number, we have performed global linear stability analyses varying \(h/D\) from 0 to 1. Figure 6(a) displays the effect of the cavity length, \(h/D\), on the critical Reynolds number of the first bifurcation, \(Re_{c,s}\), for a cavity with diameter \(D_c/D = 29/30\). Note that \(Re_{c,s}\) increases with \(h/D\), showing the stabilizing effect of the cavity, until it reaches an asymptotic value for \(h/D > 0.7\). Thus, the value \(h/D \approx 0.7\) can be considered as a critical cavity length, beyond which no further improvement in \(Re_{c,s}\) is observed. The addition of a cavity of \(h/D > 0.7\) results in an increase in the critical Reynolds number from \(Re_{c,s} = 326.2\) for a body without cavity to \(Re_{c,s} = 512\), representing a considerable stability increase of about 57%. In Fig. 6(a), we have also plotted (squares) critical Reynolds numbers determined from the three-dimensional numerical simulations for \(h/D = 0, 0.2, \) and 0.7, with the aim of comparing the results provided by the global linear stability analyses (the procedure followed to determine \(Re_{c,s}\) from the numerical simulations can be found in Bohorquez et al.\(^3\)). Note that the relative differences between the global stability and the numerical simulations are lower than 2.25% in the worst case, indicating that the numerical meshes were correctly selected in the verification stage, see Sec. II A.

Likewise, using global linear stability analysis, we have studied the effect of the diameter of the cavity \(D_c/D\) on the critical Reynolds number, \(Re_{c,s}\). Figure 6(b) displays the evolution of \(Re_{c,s}\) with \(D_c/D\) for two different cavity lengths, namely \(h/D = 0.2\) and 0.7. It can be observed that, for both cavity lengths, the critical Reynolds number exhibits a similar behavior with varying diameter. In fact, the critical Reynolds number, \(Re_{c,s}\), does not change for cavity diameters \(D_c/D \lesssim 0.6\), independent of the cavity length, however, it increases monotonically for \(0.6 < D_c/D < 1\). Figure 6(b) shows that the maximum critical Reynolds number is achieved for \(D_c/D = 1\), a limit impossible to reach in practical applications since it corresponds to a zero-wall cavity thickness. For the three-dimensional model, mathematical singularities developed at the edge-corner of the wall cavity with a thin thickness, which lead to numerical instabilities. Therefore, the maximum cavity diameter studied numerically in the present work was limited to \(D_c/D = 0.99\).

In addition to examining the effect of the cavity length on the critical Reynolds number of the first, steady bifurcation, \(Re_{c,s}\), we have also performed numerical simulations to evaluate the effect, if any, of the cavity dimensions on the drag coefficient, defined here as,

$$ C_D = \frac{F_D}{\frac{1}{2} \rho u^2 \pi D^2} = \frac{F_D^p + F_D^v}{\frac{1}{2} \rho u^2 \pi D^2} = C_D^p + C_D^v, $$

FIG. 6. (a) Dependence of the critical Reynolds numbers corresponding to the first, stationary bifurcation, \(Re_{c,s}\), with the cavity length, \(h/D\), for a cavity diameter of \(D_c/D = 29/30\). Comparison between the results given by global linear stability analyses (solid line) and the three-dimensional numerical simulations (squares), for the stationary bifurcation. (b) Effect of the cavity diameter, \(D_c/D\), on the critical Reynolds numbers for the stationary bifurcation. Cavity lengths \(h/D = 0.2\) (dashed line) and \(h/D = 0.7\) (solid line).

FIG. 7. Effect of the cavity length on the (a) pressure drag coefficient \(C_D^p\), (b) viscous drag coefficient \(C_D^v\), and (c) drag coefficient \(C_D\), for a body of cavity diameter \(D_c/D = 29/30\) and Reynolds numbers \(Re = 300\) (dashed line) and \(Re = 400\) (solid line). The drag coefficients \(C_D^{300}, C_D^{300}, \) and \(C_D^{400}\) correspond to a body without base cavity, i.e., \(D_c/D = 0\).
Dcobtained from the steady axisymmetric numerical simulations. Thus, we have plotted in Fig. 9 the pressure contours length, it is useful to examine the base flow in more detail. stand the asymptotic behavior observed with the cavity flow pressure over the surface of the body, to better understand the asymptotic behavior observed with the cavity length for a body without a base cavity, a local minimum is located at the center of the base, i.e., near \( r = 0, z = 0 \). However, as the cavity length is increased, this pressure minimum begins to attenuate, consistent with the decrease of the pressure drag coefficient with \( h/D \) reported in Fig. 7(a). The asymptotic behavior of \( C_D^p \) for \( h/D > 0.7 \) described above can be more clearly understood from Figs. 9(c) and 9(d), which show that the pressure distribution at the end of the cavity is nearly the same for a body with a cavity of \( h/D = 0.7 \) as that obtained for a body with a cavity of \( h/D = 1 \). Moreover, the pressure field outside the cavity remains unchanged for \( h/D > 0.7 \). These observations illustrate the asymptotic value achieved by the pressure drag coefficient for \( h/D \geq 0.7 \), as reported in Fig. 7(a).

Finally, let us now briefly describe the effect of the cavity diameter, \( D_c/D \), on the axisymmetric mean base flow, with the aim of understanding the stability behavior shown in Fig. 6(b). To that end, we have performed axisymmetric, steady numerical simulations for different cavities of length \( h/D = 0.7 \) and diameters \( D_c/D \) = 0.6, 0.8, and 0.9 at \( Re = 400 \). Radial velocity profiles starting at the trailing edge, \( U(r = 0.5, z) \), have been plotted in Fig. 10(a). Notice that, as \( D_c/D \) increases, the local maximum of the radial velocity, located approximately at \( z = 0.16 \), decreases while moving slightly closer to the body base. However, Fig. 10(a) also shows that the local maxima corresponding to bodies with cavity diameters of \( D_c/D = 0 \) and \( D_c/D = 0.6 \) have similar amplitudes, which might explain the weaker stabilizing effect of the cavity on the flow for \( D_c/D < 0.6 \). The downstream evolution of the axial mean velocity along the axis, \( W(r = 0, z) \), has been shown in Fig. 10(b). The figure shows that the minimum axial velocity at the axis increases slightly as \( D_c/D \) increases, decreasing the magnitude of the recirculating velocity. Thus, it seems that for smaller cavity diameters, the recirculating velocities are larger, producing larger radial velocities close to the rear corner, which perturb the free stream at \( r > 0.5 \) and destabilize the flow.

Having described the stability effects of adding a cavity of a given length, \( h \), and diameter, \( D_c \), at the base of the body (up to 40% increase of \( Re_{sz} \) and drag reduction of nearly 1%), we would like to analyze the stability characteristics of the flow in greater detail, as given by the global linear stability analysis. The eigenvalue spectra obtained for the azimuthal mode, \( |m| = 1 \), for four bodies of lengths

---

TABLE III. Drag coefficients of a solid base body, \( C_{D0}, C_{Dp0}, \) and \( C_{Dv0} \), at Reynolds numbers \( Re = 300 \) and \( Re = 400 \).

<table>
<thead>
<tr>
<th>Re</th>
<th>( C_{D0} )</th>
<th>( C_{Dp0} )</th>
<th>( C_{Dv0} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>300</td>
<td>0.718</td>
<td>0.208</td>
<td>0.510</td>
</tr>
<tr>
<td>400</td>
<td>0.619</td>
<td>0.192</td>
<td>0.427</td>
</tr>
</tbody>
</table>

---

been able to observe that the effect of the cavity on pressure and viscous components of the drag force is very different. Figure 7 shows the effect of the cavity length on the drag coefficients, \( C_D, C_{Dp}, \) and \( C_{Dv} \), relative to those obtained for the solid base body, \( C_{D0}, C_{Dp0}, C_{Dv0} \), respectively (see Table III). Furthermore, Fig. 7(a) indicates that, in the range of Reynolds numbers reported here, the viscous drag coefficient increases up to 4% due to the effect of the viscous stresses acting on the inner wall of the cavity. Finally, for these Reynolds numbers, the combined pressure and viscous effects result in a slight reduction of the drag coefficient with increasing cavity depth, until no further improvement is observed for \( h/D \geq 0.5 \), as shown in Fig. 7(c). Nevertheless, more substantial decreases in \( C_D \) can be anticipated for higher Reynolds numbers,\(^\text{16}\) for which \( C_{Dp0} < C_{Dv0} \).

A similar numerical study was carried out to determine the effect of the cavity diameter on the drag coefficient (Fig. 8). As in the case of varying cavity length, discussed above, a decrease in the pressure drag coefficient of up to 10% with varying \( D_c/D \) has been observed (Fig. 8(a)), while \( C_{Dv0} \) increases by approximately 4%, as displayed in Fig. 8(b). The behaviors of \( C_{Dp0} \) and \( C_{Dv0} \), with \( D_c/D \) translate into a small decrease of \( C_D \) with \( D_c/D \) for \( D_c/D > 0.5 \), reaching a minimum value of about \( C_D = 0.993 C_{D0} \) (0.7% decrease) in the limit \( D_c/D \rightarrow 1 \) as observed in Fig. 8(c).

Since the drag coefficient is an integral value of the base flow pressure over the surface of the body, to better understand the asymptotic behavior observed with the cavity length, it is useful to examine the base flow in more detail. Thus, we have plotted in Fig. 9 the pressure contours obtained from the steady axisymmetric numerical simulations for four bodies with a cavity of diameter \( D_c/D = 29/30 \) and lengths \( h/D = 0, 0.2, 0.7, \) and 1 at \( Re = 400 \). As shown in Fig. 9(a), in the case of a body without a base cavity, a local minimum is located at the center of the base, i.e., near \( r = 0, z = 0 \). However, as the cavity length is increased, this pressure minimum begins to attenuate, consistent with the decrease of the pressure drag coefficient with \( h/D \) reported in Fig. 7(a). The asymptotic behavior of \( C_D^p \) for \( h/D > 0.7 \) described above can be more clearly understood from Figs. 9(c) and 9(d), which show that the pressure distribution at the end of the cavity is nearly the same for a body with a cavity of \( h/D = 0.7 \) as that obtained for a body with a cavity of \( h/D = 1 \). Moreover, the pressure field outside the cavity remains unchanged for \( h/D > 0.7 \). These observations illustrate the asymptotic value achieved by the pressure drag coefficient for \( h/D \geq 0.7 \), as reported in Fig. 7(a).

Finally, let us now briefly describe the effect of the cavity diameter, \( D_c/D \), on the axisymmetric mean base flow, with the aim of understanding the stability behavior shown in Fig. 6(b). To that end, we have performed axisymmetric, steady numerical simulations for different cavities of length \( h/D = 0.7 \) and diameters \( D_c/D \) = 0.6, 0.8, and 0.9 at \( Re = 400 \). Radial velocity profiles starting at the trailing edge, \( U(r = 0.5, z) \), have been plotted in Fig. 10(a). Notice that, as \( D_c/D \) increases, the local maximum of the radial velocity, located approximately at \( z = 0.16 \), decreases while moving slightly closer to the body base. However, Fig. 10(a) also shows that the local maxima corresponding to bodies with cavity diameters of \( D_c/D = 0 \) and \( D_c/D = 0.6 \) have similar amplitudes, which might explain the weaker stabilizing effect of the cavity on the flow for \( D_c/D < 0.6 \). The downstream evolution of the axial mean velocity along the axis, \( W(r = 0, z) \), has been shown in Fig. 10(b). The figure shows that the minimum axial velocity at the axis increases slightly as \( D_c/D \) increases, decreasing the magnitude of the recirculating velocity. Thus, it seems that for smaller cavity diameters, the recirculating velocities are larger, producing larger radial velocities close to the rear corner, which perturb the free stream at \( r > 0.5 \) and destabilize the flow.

Having described the stability effects of adding a cavity of a given length, \( h \), and diameter, \( D_c \), at the base of the body (up to 40% increase of \( Re_{sz} \) and drag reduction of nearly 1%), we would like to analyze the stability characteristics of the flow in greater detail, as given by the global linear stability analysis. The eigenvalue spectra obtained for the azimuthal mode, \( |m| = 1 \), for four bodies of lengths

---

**FIG. 8.** Effect of the cavity diameter on the (a) pressure drag coefficient \( C_D^p \), (b) viscous drag coefficient \( C_D^v \), and (c) drag coefficient \( C_D \), for a body with a cavity length of \( h/D = 0.7 \) at Reynolds numbers \( Re = 300 \) (dashed line) and \( Re = 400 \) (solid line), respectively.
h/D = 0, 0.2, 0.7, and 1, respectively, and a cavity diameter \( D_c/D = 29/30 \) (close to the optimum \( D_c/D \rightarrow 1 \)) at Reynolds number \( Re = 400 \), have been plotted in Fig. 11. It can be observed in Fig. 11(a) that, at this Reynolds number, the flow around a body without base cavity, \( h/D = 0 \), exhibits an unstable (\( \sigma_r > 0 \)) but steady (\( \sigma_r = 0 \)) regime (eigenvalue marked with a square), characterized by the loss of axial symmetry.\(^3\) However, as the cavity length is increased, the growth rate of the unstable mode, \( \sigma_r \), decreases, and the eigenvalue moves towards the stable half-plane defined by \( \sigma_r < 0 \). This trend continues for increasing values of \( h/D \) until \( h/D \approx 0.2 \), for which the unstable value is neutralized, as shown in Fig. 11(b). Increasing the cavity length from \( h/D = 0.2 \) to \( h/D = 0.7 \) produces a decrease of \( \sigma_r \) towards more negative values (Fig. 11(c)) until no further changes are observed in \( \sigma_r \) for bodies of cavity length larger than \( h/D = 0.7 \) as depicted in Fig. 11(d). This result is in agreement with the asymptotic behavior found in both the critical Reynolds number, represented in Fig. 6(a), and the drag coefficient, displayed in Fig. 7(c), for \( h/D > 0.7 \). Furthermore, the spatial structure of the global mode is also significantly affected by the increase of the cavity length, as revealed by an examination of the normalized eigenfunctions shown in Fig. 12. For a solid body to \( \min \quad \frac{\|^u_\infty\|^2}{\|^u_\infty\|^2}; \quad \frac{\|^w_\infty\|^2}{\|^w_\infty\|^2}; \quad \frac{\|^p_\infty\|^2}{\|^p_\infty\|^2} = \Re \{ (\hat{u}, \hat{\hat{v}}, \hat{\hat{w}}, \hat{\hat{p}})/q_\infty \} \), corresponding to the eigenvalues marked with a square in Fig. 11. One of the important effects of the cavity is that the near-field structure is confined within the recess, concurrent with a decrease in eigenfunction amplitudes. In fact, the most intensive part (negative values) of both radial \( \hat{\hat{u}} \) and azimuthal \( \hat{\hat{w}} \) normalized eigenfunctions decrease in magnitude and move inside the cavity as \( h/D \) increases. It is worth mentioning that the eigenfunction with the largest amplitude, for all the cavity lengths, is that corresponding to the axial velocity component \( \hat{\hat{w}} \). Concerning the pressure normalized eigenfunction, Fig. 12(a) shows that, for body without cavity, the minimum occurs at the base of the body, near the axis. However, as the cavity length increases, the minimum in the normalized pressure eigenfunction moves radially and concentrates near the corner of the cavity, see Figs. 12(c) and 12(d). Note that the minimum value of \( \hat{\hat{p}} \) reduces in magnitude from \( \hat{\hat{p}}_{\text{max}} \approx -0.15 \) for a solid body to \( \hat{\hat{p}}_{\text{max}} \approx -0.06 \) for a body with a cavity of length \( h/D > 0.7 \). Finally, it is apparent in Fig. 12 that the shape and the values of all the normalized eigenfunctions are nearly identical for the bodies of length \( h/D = 0.7 \) and \( 1.0 \), which corroborates the asymptotic behavior of the flow stability with varying cavity length, discussed earlier.

B. Cavity effects on the second-oscillatory bifurcation

It is known that the flow past around slender bodies, spheres, or disks experiences a second oscillatory bifurcation

\[ i = (\hat{u}, \hat{\hat{v}}, \hat{\hat{w}}, \hat{\hat{p}})/q_\infty \].

\[ \frac{\|^u_\infty\|^2}{\|^u_\infty\|^2}; \quad \frac{\|^w_\infty\|^2}{\|^w_\infty\|^2}; \quad \frac{\|^p_\infty\|^2}{\|^p_\infty\|^2} = \Re \{ (\hat{u}, \hat{\hat{v}}, \hat{\hat{w}}, \hat{\hat{p}})/q_\infty \} \]

\[ \min \quad \frac{\|^u_\infty\|^2}{\|^u_\infty\|^2}; \quad \frac{\|^w_\infty\|^2}{\|^w_\infty\|^2}; \quad \frac{\|^p_\infty\|^2}{\|^p_\infty\|^2} = \Re \{ (\hat{u}, \hat{\hat{v}}, \hat{\hat{w}}, \hat{\hat{p}})/q_\infty \} \]

\[ \min \quad \frac{\|^u_\infty\|^2}{\|^u_\infty\|^2}; \quad \frac{\|^w_\infty\|^2}{\|^w_\infty\|^2}; \quad \frac{\|^p_\infty\|^2}{\|^p_\infty\|^2} = \Re \{ (\hat{u}, \hat{\hat{v}}, \hat{\hat{w}}, \hat{\hat{p}})/q_\infty \} \]

\[ \min \quad \frac{\|^u_\infty\|^2}{\|^u_\infty\|^2}; \quad \frac{\|^w_\infty\|^2}{\|^w_\infty\|^2}; \quad \frac{\|^p_\infty\|^2}{\|^p_\infty\|^2} = \Re \{ (\hat{u}, \hat{\hat{v}}, \hat{\hat{w}}, \hat{\hat{p}})/q_\infty \} \]
at a critical Reynolds number, $Re_{cr}$, occurring after a first stationary transition. In the case of spheres or disks, global linear stability theory based on axisymmetric base flows predicts the second, oscillatory bifurcation reasonably well. However, in the case of slender bodies, in particular for the bullet-like body of aspect ratio $L/D = 2$ considered in this work, linear stability theory, with an axisymmetric base flow, fails to predict the second oscillatory bifurcation. Thus, in this section, we have limited our study to three-dimensional numerical simulations and experimental studies, performed to determine the effect of the cavity dimensions on the second oscillatory bifurcation.

Oscillations of the wake downstream of the solid-base body are clearly observed at moderate Reynolds numbers, e.g., $Re = 500$, with three-dimensional unsteady numerical simulations, as displayed in the vorticity contours plotted in Fig. 13(a). However, the introduction of a cavity of length $h/D = 0.7$ and diameter $D_c/D = 0.86$ suppresses the oscillatory mode at this Reynolds number, as shown in Fig. 13(b). In place of the oscillatory mode, we can observe in Fig. 13(b) that the flow has become stationary and, although it is not axisymmetric, exhibits planar symmetry. Notice from Fig. 6(b) that for a Reynolds number $Re = 500$ and a cavity length $h/D = 0.7$, the cavity diameter $D_c/D$ should be larger than 0.96 to completely stabilize the flow and to recover the axisymmetric solution. Fig. 13(c) also includes a detail of the vorticity contours inside the cavity, showing that the vortical flow extends into the cavity. Thus, it can be observed that the recirculating region forming behind the body penetrates inside the cavity modifying the dynamic of the wake and, consequently, changing the nature of the instability from oscillatory to steady at this Reynolds number.

We also experimentally measured the critical Reynolds number for the second, oscillatory bifurcation, $Re_{cr}$, for a bullet-like body of length-to-diameter ratio equal to 2, with cavities of diameter $D_c/D = 0.86$ and lengths varying from $h/D = 0$ to $1$. The fluctuations of the streamwise component of the velocity, $w'(t)$, were measured at $r = 0$ and $z = 3$ with a hot-wire anemometer. Figure 14 shows two examples of the hot-wire measurements of $w'(t)$, along with their corresponding power spectral density, obtained at $r = 0$, $z = 3$ for two bodies with cavity lengths of $h/D = 0$ and 0.2 and $L/D = 2$ with $Re = 465$. It can be clearly observed that the amplitude of the energy spectrum decreases with the addition of a base cavity, illustrating its stabilizing effect. Notice that the scale in the energy spectrum displayed in Fig. 14(b) is ten times smaller than that shown in Fig. 14(a). It is
also notable that the Strouhal number, defined here as 
\[ St = \frac{f c D}{w_1} \],
where \( f_c \) is the most energetic frequency in the power spectral density, does not change with variations in cavity length.

To determine the critical value of the Reynolds number, \( Re_{ct} \), we examined the evolution of the squared amplitude of the streamwise velocity fluctuations as a function of the Reynolds number, \( w_0^2(Re) \). We define \( w_0^2 \) following Sanmiguel-Rojas et al. \(^2\) as
\[
w_0^2 = \int_{f_c - \Delta f_c}^{f_c + \Delta f_c} PSD(f) \, df,
\]
where \( PSD(f) \) is the power spectral density obtained from the velocity measurements, \( f_c \) is the characteristic shedding frequency, and \( \Delta f_c \) and \( \Delta f_e \) correspond to the frequency interval around \( f_c \) at which the power spectral density drops below 5% of the peak value. The linear increase of \( w_0^2 \) with the Reynolds number near the critical value indicates that the transition to the oscillatory regime corresponds to a Hopf bifurcation. \(^18\) The critical value of the Reynolds number was determined from a linear regression of the experimental measurements of \( w_0^2 \) near criticality. Figure 15 displays the dependence of the energy of the velocity fluctuations, \( w_0^2 \), at \( (r = 0, z = 3) \) on the Reynolds number near the critical point, for cavities of different lengths, \( h/D = 0, 0.2, 0.4, 0.7, \) and \( 1.0 \). The figure shows that critical Reynolds number, \( Re_{ct} \), increases with the cavity length, \( h/D \), corroborating the earlier observations on the stabilizing effects of the cavity.

Figure 16 summarizes the effect of the cavity length on the values of the critical Reynolds numbers corresponding to the oscillatory mode obtained experimentally (solid circles) and from three-dimensional simulations (solid triangles). Because of the considerable computational expense, a smaller number of Reynolds number cases was examined at only four cavity lengths \( (h/D = 0, 0.2, 0.7, 1.0) \) for the three-dimensional simulations. As a result, the oscillatory bifurcation was not determined following the same procedure as for the experiments. Instead, we show in Figure 16 an upper limit at which the flow is always unstable (oscillating) and a lower limit where the flow is always stable (steady), obtained from the numerical simulations, so that the critical Reynolds

---

**FIG. 14.** Hot-wire measurements of the fluctuations of the streamwise component of the velocity, \( w'(t) \), and the corresponding power spectral density at \( r = 0, z = 3 \) for \( Re = 465 \) and two cavities of diameter \( D_c/D = 0.86 \) and lengths (a) \( h/D = 0 \) and (b) \( h/D = 0.2 \). Note that the scale in the energy spectrum displayed in (b) is ten times smaller that shown in (a).

**FIG. 15.** Energy of the streamwise velocity fluctuations, \( w_0^2 \), versus the Reynolds number, obtained experimentally, at \( (r = 0, z = 3) \) for cavities of diameter \( D_c/D = 0.86 \) and different lengths, \( h/D = 0, 0.2, 0.4, 0.7, \) and \( 1.0 \).
number corresponding to the oscillatory bifurcation, $Re_{cs}$, is bounded by these limits.

As for the steady mode case discussed in Sec. III A, $Re_{cs}$ exhibits an asymptotic behavior for $h/D \geq 0.7$. The critical Reynolds number, $Re_{cs}$, in the numerical case (solid triangles), is increased from 413 to an asymptotic value inside the range $650 < Re < 680$, which represents an important stability increase of greater than 60%. In the experimental case (solid circles), this asymptotic value is about $Re \approx 580$. The small differences between the experimental and the numerical results are most likely associated with small misalignments between the body and the free-stream in the experimental measurements. To show this, we performed a three-dimensional simulation for a cavity of $h/D = 0.7$ considering a body misaligned $1\degree$ with respect to the free-stream. The results obtained are shown by the open symbols in Fig. 16, again including the upper and lower limits. Notice that the numerical range has changed from $650 < Re < 680$ to $580 < Re < 600$, decreasing the differences between the experimental and the numerical results from about 14% to only 2.5%.

### IV. CONCLUSIONS

In this work, we have shown that the addition of a cavity at the base of an axisymmetric slender body with a blunt trailing edge improves the stability of the wake and decreases the drag coefficient. In particular, we have reported results obtained from linear stability theory, three-dimensional simulations and experiments, for the flow generated behind a cylindrical slender body of diameter $D$ and total length $L = 2D$, with an elliptical rounded nose of 2:1 major-to-minor axis ratio and a blunt trailing edge, aligned with the free stream of velocity $w_\infty$. The body base was modified to form a cavity of length $h$ and inner diameter $D_c$.

In previous work, Sanmiguel-Rojas et al.\textsuperscript{2} and Bohorquez et al.\textsuperscript{3} demonstrated that, as the Reynolds number increases, a bullet-like body of length $L/D = 2$ with a solid base experiences a first, steady bifurcation at a critical Reynolds number in the range $319 < Re_{cs} < 327$ and, a second, oscillatory transition at approximately $412.4 < Re_{cs} < 413$. Prior to the description of the results, we have also performed an error analysis. A verification stage first established the discretization errors for the numerical results. Modeling errors were subsequently determined in a validation stage.

Here, we have first analyzed the influence of the cavity dimensions on the critical Reynolds number, $Re_{cs}$, corresponding to the first, steady bifurcation. We have developed a global linear stability code based on compact finite differences of sixth order, from which we obtain a critical Reynolds number, $Re_{cs} = 326.2$, very close to that reported by Sanmiguel-Rojas et al.,\textsuperscript{3} $Re_{cs} = 327$, for a solid base body. To show the effect of the cavity length $h/D$ on the critical Reynolds number reported above, we have performed global linear stability analyses varying $h/D$ from 0 to 1.0, and have shown that $Re_{cs}$ exhibits an asymptotic behavior for $h/D \geq 0.7$. The cavity length, $h/D \approx 0.7$, can thus be considered as a critical cavity length, beyond which no significant change in $Re_{cs}$ is observed. The critical Reynolds number increases from $Re_{cs} = 326.2$ for a solid body to $Re_{cs} = 512$ for a body with a cavity of length $h/D = 0.7$, which represents an important stability increase of about 57%. In addition, we have also performed three-dimensional numerical simulations with OpenFOAM® with the aim of validating the global linear stability analyses. The relative errors between the global stability and the numerical results are lower than 2.5% in the worst case. Similarly, we have studied the effect of the cavity diameter $D_c/D$ on the critical Reynolds number, $Re_{cs}$. Basically, no effect on $Re_{cs}$ has been observed for the range $0 \leq D_c/D \leq 0.6$, however, a monotonic increase in $Re_{cs}$ has been found in the range $0.6 \leq D_c/D \leq 0.99$. In addition to describing the effect of the cavity length on $Re_{cs}$, we performed steady numerical simulations at $Re = 400$ to determine the effect of the cavity dimensions on the drag coefficient, $C_D$.

Our simulations indicated that, although an improvement of 10%–15% can be achieved in the pressure drag coefficient, $C_{Dp}$, for bodies with a base cavity, the total drag reduction is, at best, approximately 1%. The low value is due to the fact that the improvement in the pressure drag coefficient, $C_{Dp}$, achieved by addition of the cavity, is balanced by an increase in the viscous drag coefficient, $C_{Dv}$, associated with increased
viscous forces acting on the inner wall of the cavity. Nevertheless, the relative importance of the viscous contribution is expected to decrease at higher Reynolds numbers and, thus, substantial drag reductions could be achieved,\textsuperscript{4,6} although a detailed numerical study ought to be done to prevent the onset of cavity resonances, see Weickgenannt and Monkewitz.\textsuperscript{19}

Concerning the second-oscillatory bifurcation, three-dimensional numerical simulations have been performed to determine the effect of the cavity length on the critical oscillatory Reynolds number, \( \text{Re}_{c,\text{os}} \), with comparison to experimental measurements carried out using hot-wire anemometry in a wind tunnel. As in the case of the steady bifurcation, both experimental and numerical results exhibit an asymptotic value for \( h/D \geq 0.7 \).

To conclude, we have demonstrated that a cavity placed at the base of an axisymmetric slender body has beneficial effects on both wake stability and drag reduction, showing an asymptotic behavior at cavity lengths \( h/D > 0.7 \). Our study has been limited to Reynolds numbers below \( \text{Re} = 700 \) due to the high computational cost of the three-dimensional simulations in our facilities. Thus, both a more complete experimental study and three-dimensional direct numerical simulations are needed to examine the behavior of the drag coefficient with varying the cavity dimensions and body lengths and at higher Reynolds numbers.

**ACKNOWLEDGMENTS**

This research has been supported by the Spanish MICINN Project # DPI2008-06624-C03-02, Junta de Andalucía and European Funds under Projects # P07-TEP-02693, P10-TEP-5702, and University of Jaén Project # UJA2010/12/60. G.P. would like to thank the Junta de Andalucía for their support for his research stay at the University of Jaén.


\textsuperscript{9}J. H. Ferziger and M. Perić, Computational Methods for Fluid Dynamics (Springer-Verlag, Berlin, 2002).


