CALCULATION AND OPTIMIZATION OF AERODYNAMIC COEFFICIENTS FOR LAUNCHERS AND RE-ENTRY VEHICLES

Alberto Ferrero*

June 17, 2014

Abstract

This work develops a procedure to calculate the aerodynamic coefficients for hypersonic flight condition. The coefficients are obtained analytically, based on the Newton theory of the hypersonic flow, and numerically by the solution of the Navier-Stokes equations. In FreeFem++ [14] several shapes are analyzed and an open source code for the evaluation of the coefficients is developed. The hypersonic flux is even simulated solving numerically the Euler's equations. These equtions are adopted for computational resons: in fact they permit a relatively rapid simulation of the flux, but they are generally not valid for the hypersonic flux. The advantages and the disadvantages of the two methods, analytic and numerical, are analyzed. The analytic expressions of the aerodynamic coefficients allow the implementation of an optimization algorithm, based on the solution of a constrained problem. The analytic solution of this problem, obtained with the software Mathematica, obtains the optimal geometric configurations of the hypersonic shape, in order to reach a minimal value of drag, in case of studying a launcher, or a minimal value of the ballistic coefficient, in case of studying a re-entry vehicle.

1 Research motivation and goals

The primary goal of this work is to develop a procedure to calculate the aerodynamic coefficients, both analytically and numerically for the hypersonic flight regime, eventually demonstrating the advantage of this approach. The comparison between both is made in order to validate the results and to emphasize their advantages and disadvantages.

It is important to underline that the geometry of many common hypersonic vehicles of interest such are basically made by simple shapes like sphere-cones, blunted bi-conics and spheres and it is relatively simple to express these curves analytically [1]. Considering the hypersonic flow simplifications given by the Newton's aerodynamic theory, the corresponding aerodynamic coefficients can also be developed analytically.

Having an analytic approach can be useful because it allows to have exact calculations of the aerodynamic coefficients currently approximated by numerical methods. Additionally, these relations eliminate the large aerodynamic tables enabling the designer to do rapid simulations of hypersonic flight. This is essential above all in a conceptual design and for global optimization, where the phase space is often large [1]. From this point of view, obtaining an analytic expression for the aerodynamic coefficients results essential in order to solve the optimization process without solving by numerical simulation the Navier-Stokes equations. In fact it can be studied as a constrained optimization problem, whose solution can be found or analytically or with simpler numerical algorithm.

In fact, after having calculated the aerodynamic coefficients, it is possible to optimize the shape of the studied vehicle: for launchers the optimization is done minimizing the drag, for the re-entry vehicles maximizing the air resistance, respecting the structural and thermal limits and other constrains.

Contrary to commercial codes, this work will be developed for being free and user-friendly, in order to create a fast and easy instrument for helping the engineer in the preliminary design of hypersonic vehicles. For this reason the work will be developed in the FreeFem++ environment. This is a completely open-source software that uses finite elements to solve integrals. Due to its capacity of solving partial differential equations (PDE), it will be used for obtaining a direct comparison between the numerical and the analytical calculation of the aerodynamic coefficients. The analysis is made simulating the hypersonic flux by the Euler's equations, an approximation of the Navier-Stokes equations.

The result of this work will be a code in which the future users can obtain analytically the aerodynamic

*under the supervision of P. J. S. Gil

coefficients, eventually validate the results with numerical simulations, and know the optimal configuration for the chosen shape.

2 Aerodynamic coefficient calculation

In order to valuate the forces and the moments acting on a hypersonic vehicle, it is necessary to know the six aerodynamic coefficients for the drag C_D, the lateral force C_S, the lift C_L, the roll moment C_l, the pitch moment C_m and the yaw moment C_n. These coefficients are depending on the shape of the vehicle and on the direction of the air flow, that can be defined as:

$$\hat{V}_\infty = \begin{bmatrix} -\cos(\alpha)\cos(\beta) & -\sin(\beta) & -\sin(\alpha)\cos(\beta) \end{bmatrix}^T \quad (1)$$

where α is the attack angle and β is the side-slip angle, referred to a body reference frame, as shown in Figure 1.

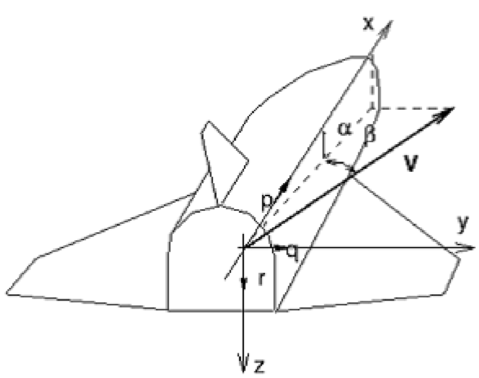

Figure 1: Body-axes reference frame [7]

In order to develop the analytic calculation of the aerodynamic coefficients, the Newton theory of the hypersonic flight regime is analyzed. The results can be compared with the ones given from an usual computational fluid mechanics (CFD) approach, that studies the evolution of the flux around the body solving numerically the Navier-Stokes equations.

2.1 Newton method

At the base of the Newton theory for the hypersonic flux, the motion of the fluid is described as a system of particles traveling in rectilinear motion that in case of striking a rigid surface, they lose all the momentum normal to the surface conserving only the momentum tangential to the surface as it is shown in Figure 2.

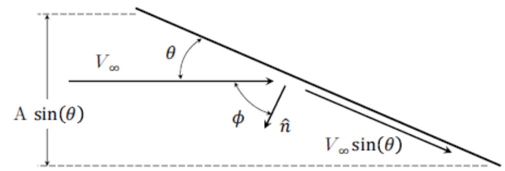

Figure 2: Momentum of a gas particle in Newton assumption [2]

This behavior of the hypersonic flows leads to one of the key hypotheses of the algorithm [2]. The flows tends to change almost instantaneously its direction from the free-stream orientation to a direction tangential to the surface and with this consideration it's possible to simplify the study of the phenomena. In fact it is possible to write an approximate definition of the velocity vector over the body surface, which is found by considering the velocity on the body only with the tangential component of the free-stream velocity [2]:

$$\mathbf{V}_{body-local} = \mathbf{V}_\parallel = \mathbf{V}_\infty - \mathbf{V}_\perp = \mathbf{V}_\infty + (\hat{\mathbf{n}} \cdot \mathbf{V}_\infty) \cdot \hat{\mathbf{n}} \quad (2)$$

where $\mathbf{V}_{body-local} = \mathbf{V}_\parallel$ means that the velocity vector considered on the body has only the tangential direction and it is possible to consider the normal component of the velocity as $\mathbf{V}_\perp = -(\hat{\mathbf{n}} \cdot \mathbf{V}_\infty) \cdot \hat{\mathbf{n}}$, for $\hat{\mathbf{n}}$ the outward normal vector to the surface of the body.

With this hypothesis it is possible to make two assumptions:

- it is necessary to know the velocity field on the vehicle's surface for characterize the flow, so the forces that act on the body [2]
- it is possible to obtain the inviscid pressure on the vehicle's surface, simply by considering the loss of normal momentum in the almost instantaneous change of flow direction from normal to tangential. This is the fundamental hypothesis of Newton method [2].

Starting from the second assumption, it is possible to define the pressure coefficient for the Newton flow model, only dependent on the relative inclination that the surface has with the free-stream [1]:

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2} = 2\sin^2(\theta) \quad (3)$$

Using conventional aircraft body axes show in Figure 1 and the corresponding free-stream velocity vector $\mathbf{V}_\infty$, defined in Eq.(1) as function of angle of attack α and side-slip β, the aerodynamic force coefficients along the body axes are [1]:

$$\begin{bmatrix} C_D \\ C_S \\ C_L \end{bmatrix} = \frac{1}{A_{ref}} \iint_S \mathbf{df} = \frac{1}{A_{ref}} \iint_S C_p \begin{bmatrix} \hat{\mathbf{n}}_x \\ \hat{\mathbf{n}}_y \\ \hat{\mathbf{n}}_z \end{bmatrix} dA \tag{4}$$

and for the moments [1]:

$$\begin{aligned} \begin{bmatrix} C_l \\ C_m \\ C_n \end{bmatrix} &= \frac{1}{A_{ref} l_{ref}} \iint_S \mathbf{r} \times \mathbf{df} = \\ &= \frac{1}{A_{ref} l_{ref}} \iint_S C_p \begin{bmatrix} (\mathbf{r}\times\hat{\mathbf{n}})_x \\ (\mathbf{r}\times\hat{\mathbf{n}})_y \\ (\mathbf{r}\times\hat{\mathbf{n}})_z \end{bmatrix} dA \end{aligned} \tag{5}$$

The position vector $\mathbf{r}$ defines all the points of the surface A by a parametrization it two variables (u, v) [1].

$$\mathbf{r} = \begin{bmatrix} f(u,v) & g(u,v) & h(u,v) \end{bmatrix}^T \tag{6}$$

where $f(u, v)$, $g(u, v)$, and $h(u, v)$ describe the x, y, and z location of a point on the surface of the vehicle as a function of the surface parametrization (u, v).

One of the most important choice of the designer is on the type of parametrization used. In fact the selection of (u, v) parameters dramatically affects the possibility of having a closed-form solution for the integrations [1].

Additionally, due to the convention adopted on the surface outward-normal $\hat{\mathbf{n}}$, the choice of u and v also influences the expression for the differential area dA of the integrations. So the outward-normal vector to the surface can be defined as [1]:

$$\hat{\mathbf{n}} = \frac{\mathbf{r}_u \times \mathbf{r}_v}{\|\mathbf{r}_u \times \mathbf{r}_v\|} \tag{7}$$

where $\mathbf{r}_u = \frac{\partial \mathbf{r}}{\partial u}$ and $\mathbf{r}_v = \frac{\partial \mathbf{r}}{\partial v}$ are the partial derivative and it is possible to define the differential area as $dA = \|\hat{\mathbf{n}}\| = \|\mathbf{r}_u \times \mathbf{r}_v\|$.

Defining $\sin(\theta) = \mathbf{V}_\infty \cdot \hat{\mathbf{n}}$, so as a relation between the normal vector and the velocity of the free-stream, it is possible to obtain a new definition of the pressure coefficient:

$$C_p = 2\sin^2(\theta) = 2\left(\hat{\mathbf{V}}_\infty \cdot \hat{\mathbf{n}}\right)^2 = 2\left(\hat{\mathbf{V}}_\infty \cdot \frac{\mathbf{r}_u \times \mathbf{r}_v}{\|\mathbf{r}_u \times \mathbf{r}_v\|}\right)^2 \tag{8}$$

As the particles are supposed to move straight and to change their direction only when impacting a surface, on the back side of a plate there would be no impact at all. So for the region of the body which does not impact directly the hypersonic flow $C_p = 0$, that means no modification on free-stream pressure. This is called shadow region. In order to evaluate the shadowed region in which $C_p = 0$, it is necessary a study of the wetted surface, depending on the angle of attack and side-slip of the velocity. So it is required a preliminary study for fixing the limits of the integration, $[u_{min}, u_{MAX}]$ and $[v_{min}, v_{MAX}]$. The solution of this problem is not trivial, especially if the surface is parametrized with trigonometric functions. For this region only convex shapes are supported.

The reference area and reference length for each shape are computed based on the parametrization used. The reference area is computed as the projected area of the shape on the $y - z$ plane, considering the entire surface unshadowed $\alpha = \beta = 0$. For composted shapes, A_{ref} is always the projected area of the biggest section. The reference length is computed as the maximum span of the vehicle in the x direction [1].

As demontrated in this section, one fondamental result of the Newton flow theory is that every aerodynamic coefficient is derived from the surface integral of the pressure coefficient. So for more complex noses, the global coefficients can be calculated superpositioning the effects of each basic shapes in which is possible to divide the nose.

As already introduced at the beginning of this chapter, common hypersonic vehicles can be determined through superposition of basic shapes. For example, sphere-cones can be constructed using a spherical segment and a single conical frustum, and bi-conics can be constructed using a spherical segment and two conical frustums ans so on.

Each basic shape used will likely have different reference areas and lengths. Therefore, the superpositioning of basic shapes cannot be performed by simply adding the aerodynamic coefficients from each shape. Rather, the aerodynamic coefficients of each basic shape must be scaled to a common reference area and length.

Analyzing the simple sphere-cone nose shown in Figure 3, the drag coefficient $C_{D_{SC}}$, for example, is calculated using the superposition of the effects, where the reference area is the base area of the sphere-cone and the reference length is the length of the entire nose [1].

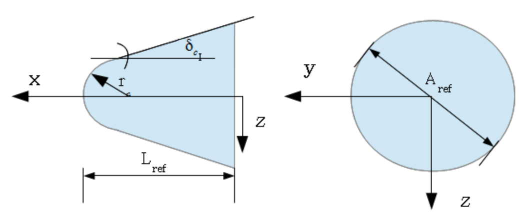

Figure 3: Side and front view of sphere-cone

$$C_{D_{SC}} = C_{D_C} \frac{A_{ref_C}}{A_{SC}} + C_{D_{SP}} \frac{A_{ref_{SP}}}{A_{SC}} \tag{9}$$

Eq.(9) is only valid only if the flux is studied with the Newton model. In this case only shape that makes the nose add its contribution to the total value of the aerodynamic coefficient. This approach permits to create several hypersonic noses just composing a little number of basic shapes.

2.2 Navier-Stokes method

The Navier-Stokes equations describe the motion of fluid substances. These equations arise from applying Newton's second law to fluid motion, together with the assumption that the stress in the fluid is the sum of a diffusing viscous term (proportional to the gradient of velocity) and a pressure term.

They describe the conservation of mass, momentum and energy of all the particles in a limited system. They can be expressed in the general form as [3]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (10)$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial(\rho u_j u_i)}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial p}{\partial x_i} = 0 \quad (11)$$

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho u_j E)}{\partial x_j} - \frac{\partial}{\partial x_j}\left(k\frac{\partial T}{\partial x_j}\right) - \frac{\partial}{\partial x_j}(\tau_{ij}u_i) = 0 \quad (12)$$

in which the Eq.(10) defines the conservation of mass, Eq.(11) the conservation of momentum, equation Eq.(12) the conservation of energy.

In the study of fluid mechanics, numerical methods such as the finite element and finite difference methods are often used to approximate the fluid flow problems. Considering an incompressible flux, $\rho = const$, and an inviscid flux, $\mu = 0$, it is possible to reduce the complexity of the Navier-Stokes equations. The resulting formulation is called Euler's equations and can be written in vector form as [13]:

$$\nabla \cdot \mathbf{u} = 0 \quad (13)$$

$$\rho\frac{\nabla \mathbf{u}}{Dt} = \rho \mathbf{g} - \nabla p \quad (14)$$

A solution of the equations is obtained with FreeFem++, a free numerical integrator of PDE, using finite elements.

Let's consider an approximate model of the Euler's equations, namely the pseudo-compressible approximation, or pseudo-compressible Stokes equations, where a pressure term is added to the continuity equation with a coefficient ε and introducing the kinematic viscosity $\nu = \mu/\rho$ can be written in a dimensionless form as [12]:

$$\nabla \cdot \mathbf{u} + \varepsilon p = 0 \quad (15)$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}\cdot\nabla\mathbf{u} - \frac{1}{Re}\triangle\mathbf{u} + \frac{1}{\rho}\nabla p = \mathbf{g} \quad (16)$$

in which the Reynolds number is $Re = \frac{V_\infty l_{ref}}{\nu}$.

In the FreeFem++ environment Eqs.(15-16) are solved numerically with finite element in order to simulate the hypersonic flux around a 2D shape and finding the field of velocity and pressure, it is easy to evaluate the pressure coefficient as $C_p = \frac{2}{\gamma M_\infty^2}\left(\frac{p}{p_\infty} - 1\right)$ for comparing with the one given by the Newton theory. So the 2D drag coefficient can be calculated, having from the simulation a value of C_p and due to the symmetry of the problem, the 3D drag coefficient is obtained:

$$C_{D_{3D}} = 2\pi C_{D_{2D}} = \frac{2\pi}{l_{ref}} C_p \int_{\partial\Sigma^+} \hat{\mathrm{n}}_x du \quad (17)$$

where $\partial\Sigma^+$ is the upper boundary of the 2D shape.

3 Shape optimization

One of the goals of this work is to obtain analytical expressions for the aerodynamic coefficients. Having these expressions, the optimization process can be done analytically.

In mathematical optimization, constrained optimization is the process of optimizing an objective function with respect to some variables in the presence of constraints on those variables.

In this work the objective function is the drag coefficient C_D or the ballistic coefficient $B^* = B^*(C_D) = \frac{m}{C_D A_{ref}}$ which are to be minimized.

Considering a problem of constrained optimization for an objective function $f(x)$, generally it is expressed as [11]:

$$\begin{cases} \text{find min } f(x), & x \in X \subset R^n \\ h_j(x) = 0, & j = 1, ..., l \\ g_i(x) \gtreqless 0, & i = 1, ..., m \end{cases} \quad (18)$$

in which $x \in X \subset R^n$ are the unknowns of the problem that are limited to the space $X \subset R^n$ by the constraints, $h(x)$ and $g(x)$ are respectively the equality and inequality constraint functions. So the space of the feasible solutions is defined as $X = \{x \in \mathbb{R} : g_i(x) \leq 0, i = i = 1, ..., m, h_j(x) = 0, j = 1, ..., l\}$.

For the case of having $f(x) = C_D(x)$, the objective function is often non-linear. A possible solution can be defining the optimization problem with the Karush-Kuhn-Tucker (KKT) conditions, solved with the Newton-like interior point method.

As a design choice, three inequality constrains are selected, in order to solve the problem with the internal point method. So for the considered case $l = 0$ and $m = 3$.

It is important to notice that in case of minimizing the ballistic coefficient B^*, it is necessary to define the mass of the entire re-entry vehicle. It can be divided in a constant part, M_{body}, and a part depending on the shape of the nose, $m(x) = S_{lat}(x)\rho t$, where S_{lat} is the lateral surface of the nose, ρ is a mean density of the nose, t is a mean thickness. So the ballistic coefficient can be expressed as:

$$B^* = \frac{M_{body} + m(x)}{C_D(x)A_{ref}} = \frac{M_{body} + S_{lat}(x)\rho t}{C_D(x)A_{ref}} \tag{19}$$

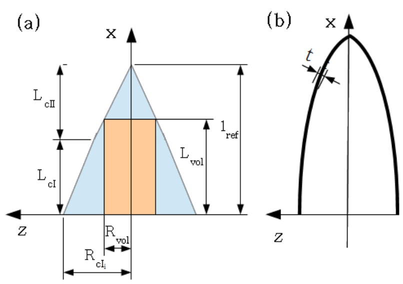

Figure 4: Constrains on the cargo volume (a) and on the nose mass (b)

CARGO VOLUME It can be useful to define a internal volume that has to be maintained during the optimization process. In fact the noses of the launchers are often used for containing the payload, so it is necessary to maintain a certain volume for the cargo. It is considered a cylindrical volume with height L_{vol} and radius R_{vol}, so $Vol = \pi L_{vol} R_{vol}^2$, as shown in Figure 4(a). As design parameter is chosen the height of the volume, that corresponds at a radius, using the equation of the nose shape $f(x)$:

$$R_{vol} = f(x = L_{vol}) \tag{20}$$

where $L_{vol} \leq l_{ref}$ and with simple geometrical consideration is possible to define the volume high as a function of the longitudinal dimension of the shape as $L_{vol} = L_{vol}(L_{fig})$.

So fixed the cargo volume Vol^*, the optimization process has to change the shape of the nose maintaining or improving this internal volume.

$$g_1 = \pi L_{vol} R_{vol}^2 \geq Vol^* \tag{21}$$

and substituting in Eq.(21) the Eq.(20) it is possible to obtain the dependence of the inequality constrain g_1 from the geometrical variables L_{fig},

$$g_1 = Vol^* - \pi L_{vol}(L_{fig}) \left(f\left(L_{vol}(L_{fig})\right)\right)^2 \leq 0 \tag{22}$$

MAXIMUM HEAT In case of having a blunted configuration, it is necessary to fix the maximum heat rate $\dot{q}_{MAX}$ that the material can afford. By definition, the heating rate exchanged with the aircraft and the Earth atmosphere $\dot{q}$ in $[W/m^2]$ that can be expressed with the empirical formulation [9]

$$\dot{q} = 1.83 \cdot 10^{-4} V_\infty^3 \sqrt{\frac{\rho_\infty}{r_n}} \tag{23}$$

in which r_n the vehicle's nose radius. This approximation is only valid in hypersonic flight regime and it is generally used for the first estimations of the heat flux. So the second inequality constraint, that imposes that the heat flux have to be lower than the maximum is

$$g_2 = 1.83 \cdot 10^{-4} V_\infty^3 \sqrt{\frac{\rho_\infty}{r_n}} - \dot{q}_{MAX} \leq 0 \tag{24}$$

A value of $\dot{q}_{MAX} = 120\,\frac{kW}{m^2}$ is set as default. This is the limit value for a non-ablative shield of a re-entry vehicle [10].

Of course in the unblunted configuration this condition is not taken into account.

TOTAL MASS The optimization process has to maintain or even reduce the mass of the nose m^*. As shown in Figure 4(b), the mass of the nose can be simply defined as

$$m^* = S_{lat}\rho t \tag{25}$$

The lateral surface is a function of the geometrical features of the nose so it can be expressed as

$$S_{lat} = 2\pi \int\limits_{L_{fig}} f(x)\sqrt{1 + \left(\frac{df(x)}{dx}\right)^2}\,dx \tag{26}$$

so the third inequality of the optimization problem can be written as

$$g_3 = S_{lat}(L_{fig}, R_{fig})\rho t - m^* \leq 0 \tag{27}$$

Having defined the three inequality constraints of the optimization problem, it is important to notice that all the constraints and the objective function too are nonlinear.

4 Bi-conic blunted nose

The calculation of the aerodynamic coefficients and the shape optimization process are made for several type of hypersonic shapes: conic, bi-conic, ogive, parabolic and Bézier curves noses in blunted or in unblunted configuration. In this section the results obtained for a bi-conic blunted configuration are presented.

The geometrical design parameters of the cone are defined by the length along the axis of revolution L_{cI_i} and L_{cI_f}, by the beginning radius R_{cI_i} and the ending radius R_{cI_f}, in which the notation $_{cI}$ is referred at the cone I. So the cone half-angle is easily defined as $\delta_{cI} = \arctan\left(\frac{R_{cI_i} - R_{cI_f}}{L_{cI_f} - L_{cI_f}}\right)$. The same relations are valid for the second cone too, following the Figure 5.

The surface of the cone is parametrized by:

$$\begin{cases} u = z(x) = r(x), & r \in [R_{cI_i}, R_{cI_f}] \\ v = \omega, & \omega \in [0, 2\pi] \end{cases} \tag{28}$$

in which ω is the revolution angle as shown in Figure 5. The equation of the semi-cone in the $x - z$ plane is

$$z(x) = -x \tan\delta_c + R_{cI_i} \tag{29}$$

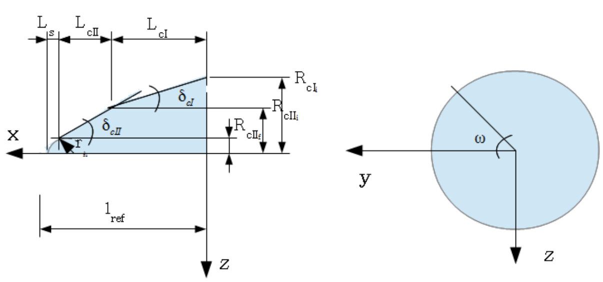

Figure 5: Side and front view of bi-conic cone parametrization

So the position vector is defined as

$$\mathbf{r} = \begin{bmatrix} -\frac{u}{\tan\delta_c} + \frac{R_{cI_i}}{\tan\delta_c} & u\cos(v) & -u\sin(v) \end{bmatrix}^T \tag{30}$$

Eq.(30) is used in Eq.(7) for evaluate the pressure coefficient and then integrated along the surface obtaining the following aerodynamic coefficients:

$$\begin{aligned}
C_{D_c} &= -\frac{1}{A_{ref}} \frac{2\pi \tan\delta_c (R_{cI_f} - R_{cI_i})\left(\cos^2\beta\left(2\cos^2\alpha\tan^2\delta_c + \sin^2\alpha\right) + \sin^2\beta\right)}{\left(\tan^2\delta_c + 1\right)^{3/2}} \\
C_{S_c} &= -\frac{1}{A_{ref}} \frac{4\pi\cos\alpha\cos\beta\left(R_{cI_f} - R_{cI_i}\right)\sin\beta\tan\delta_c}{\left(1 + \tan^2\delta_c\right)^{3/2}} \\
C_{L_c} &= -\frac{1}{A_{ref}} \frac{4\pi\cos\alpha\cos^2\beta\left(R_{cI_f} - R_{cI_i}\right)\sin\alpha\tan\delta_c}{\left(1 + \tan^2\delta_c\right)^{3/2}} \\
C_{l_c} &= 0 \\
C_{m_c} &= -\frac{1}{A_{ref} l_{ref}} \frac{2\pi\cos\alpha\cos^2\beta\left(R_{cI_f} - R_{cI_i}\right)\sin\alpha\left(R_{cI_f} - R_{cI_i} + \left(R_{cI_f} + R_{cI_i}\right)\tan^2(\delta_c)\right)}{\left(1 + \tan^2(\delta_c)\right)^{3/2}} \\
C_{n_c} &= \frac{1}{A_{ref} l_{ref}} \frac{2\pi\cos\alpha\cos\beta\left(R_{cI_f} - R_{cI_i}\right)\sin\beta\left(R_{cI_f} - R_{cI_i} + \left(R_{cI_f} + R_{cI_i}\right)\tan^2(\delta_c)\right)}{\left(1 + \tan^2(\delta_c)\right)^{2}}
\end{aligned}$$

The integrations are performed with the help of Mathematica.

Following the same process, the coefficients for the spherical termination are obtained, for a parametrization made by:

$$\begin{cases} u = \delta, & \delta \in [0, \frac{\pi}{2} - \delta_{tan}] \\ v = \omega, & \omega \in [0, 2\pi] \end{cases} \tag{31}$$

in which ω is the revolution angle and δ is the nose angle. With the parametrization of the curve:

$$\mathbf{r} = \begin{bmatrix} x_c + r_n\cos(u) \\ r_n\cos(v)\sin(u) \\ r_n\sin(v)\sin(u) \end{bmatrix} \tag{32}$$

the aerodynamic coefficients are obtained.

$$\begin{aligned}
C_{D_S} &= \frac{1}{A_{ref}}\frac{1}{3}\pi\sin(\delta_{sp})(\cos(2\delta_{sp})\left(\cos^2\beta\left(2\cos^2\alpha - \sin^2\alpha\right) - \sin^2\beta\right) + \\
&\quad + \cos^2\beta\left(10\cos^2\alpha + \sin^2\alpha\right) + \sin^2\beta) \\
C_{S_S} &= \frac{1}{A_{ref}}\frac{4}{3}\pi\cos\alpha\cos\beta\sin\beta\sin^3(\delta_{sp}) \\
C_{L_S} &= \frac{1}{A_{ref}}\frac{4}{3}\pi\cos\alpha\cos^2\beta\sin\alpha\sin^3(\delta_{sp}) \\
C_{l_S} &= 0 \\
C_{m_S} &= -\frac{x_c}{A_{ref} l_{ref}}\frac{4}{3}\pi\cos\alpha\cos^2\beta\sin\alpha\sin^3(\delta_{sp}) \\
C_{n_S} &= \frac{x_c}{A_{ref} l_{ref}}\frac{4}{3}\pi\cos\alpha\cos\beta\sin\beta\sin^3(\delta_{sp})
\end{aligned}$$

where $\delta_{sp} = \frac{\pi}{2} - \delta_{tan}$ and as aspected due to the symmetry of the shape, the roll moment coefficient is always equal to zero.

An important issue of the Newton theory for the hypersonic flight is the definition of the shadowed area. In that area the flux doesn't interact with the shape, so consequently the pressure coefficient is equal to zero, $C_p = 0$.

So by definition, it's necessary to define the tangency condition of the flux, for the given $\mathbf{V}_\infty$. First it is defined the slightest inclination of the shape, by the angle $\delta_{\min}$. Then there is a portion of the shape shadowed, only if it is satisfied this condition:

$$\max\{\alpha, \beta\} > \delta_{\min} \tag{33}$$

If the condition of shadowed area, Eq.(33), is satisfied, the flux interacts only with an half of the shape. Cutting the shape with a plane $y-z$, it's obtained a circle defined by the angle ω. Projecting the components of the air speed, $V_y = -\sin\beta$ and $V_z = -\cos(\beta)\sin(\alpha)$ on the $y - z$ plane, it is possible to identify the direction of the velocity in this plane. The velocity vector in the $y - z$ plane can be defined with as:

$$\omega_i = \arctan\left(\frac{\cos(\beta)\sin(\alpha)}{\sin(\beta)}\right) \tag{34}$$

So finally the half of the shape wetted by the flux, so unshadowed for an arc of π, is given by $v \in [\omega_i - \frac{\pi}{2}\,;\,\omega_i + \frac{\pi}{2}]$.

In case of having a blunted nose, so if the nose has a spherical segment at the end, an other assumption is necessary. Due to the parametrization chosen for this surface, the parameter $u = \delta$ defines the half-angle of the spherical segment. For this segment the $\delta_{\min}$ is given by the tangency of the spherical segment with the backward part of the nose. So in case of that the shadow condition is satisfied, Eq.(33), the unshadowed surface of the spherical segment on the $x-y$ or $x-z$ plane, is given by $u \in [0\,;\,\max\{\alpha, \beta\}]$.

In case of having a back part of the body with a different inclination, can happen that even if one part of the nose has shadowed areas, not all the nose has. Figure 6 shows the possibility to have the second part of the nose unshadowed, due to its higher inclination.

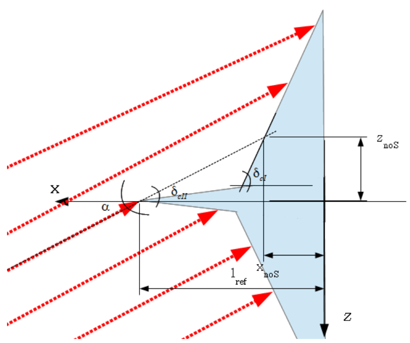

Figure 6: Partially shadowed condition for a bi-conic nose

The unshadowed area can be defined geometrically by the interaction of the velocity vector and the back part of the nose as:

$$\begin{cases} x_{noS} = \frac{l_{ref}\tan(\max\{\alpha,\beta\}) - R_{cI_i}}{(\tan(\max\{\alpha,\beta\}) - \tan\delta_{cI})} \\ z_{noS} = R_{cI_i} - x_{noS}\tan\delta_{cI} \end{cases} \tag{35}$$

Defined the possible shadowed regions of the nose, the coefficients are calculated following the superposition of the effects as:

$$\begin{array}{rcl} C_D &=& C_{D_{cI}} + C_{D_{cII}} + C_{D_S} \\ C_S &=& C_{S_{cI}} + C_{S_{cII}} + C_{S_S} \\ C_L &=& C_{L_{cI}} + C_{L_{cII}} + C_{L_S} \\ C_l &=& C_{l_{cI}} + C_{l_{cII}} + C_{l_S} \\ C_m &=& C_{m_{cI}} + C_{m_{cII}} + C_{m_S} \\ C_n &=& C_{n_{cI}} + C_{n_{cII}} + C_{n_S} \end{array} \tag{36}$$

The results obtained analytically are first confirmed calculating the coefficients numerically with the panel method. This method penalizes the surfaces in laminar plates and computes numerically the coefficients. Then a comparison with the simulation of the Stokes equations is made.

For the Navier-Stokes problem, an hypersonic flux at Mach=6 have been analyzed, in a Stratosphere conditions of pressure, density and temperature, according to the ISA standards as presented.

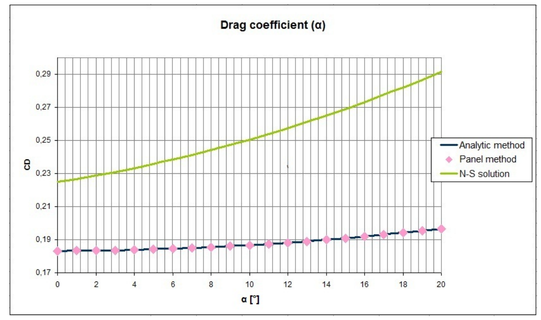

Figure 7: Diagram of $C_D \propto \alpha$ for the three different methods, conic nose

The standard deviation of each coefficient has been calculated for testing the validity of the analytical approach. The results obtained with the analytical approach are perfectly confirmed by the panel method, with a difference less than $1.15{\cdot}10^{-3}\%$. This is because both are derived from the Newton theory for the hypersonic flux. A bigger local deviation of the dates has been registered comparing the analytical approach with the solution given by the Navier-Stokes method. As aspected, with all the approximations done for computing the Navier-Stokes equations, the results aren't extremely precises, so even the fitting with the analytic dates can't be perfect. Evaluating the drag coefficient for a quite wide range of angle of attack, the standard deviation between the two approaches results globally less than 26%.

This result can be due to the fact that the conical nose can be described in a 2D geometry as two plates. For this geometrical reason the Navier-Stokes equations give a similar result to the panel method because in both of the approaches the shape analyzed can be reduced to a sum of plates. However, since that the Newton method doesn't take into account turbulence or eventual presence of flux separation, surely present with higher angle of attack, improving the free-stream inclination, the two methods give more different results.

Compared Methods	%
Analytical/Panels	$1.15{\cdot}10^{-3}$
Analytical/Navier-Stokes	25.5

Table 1: Standard deviation of the analytic dates, compared with the numerical results for the conic nose

With more complicated shapes as the ogive, parabolic or Bézier noses the gap between the values of the analytic approach and the Navier-Stokes simulation is bigger.

However the first big advantage on the analytic approach is demonstrated, the computational time of the numerical method results an order of magnitude bigger than the other. In fact the simulation of the flux can take tens of second, while the calculation of the coefficients via the analytic equation is instantaneous.

The second advantage of having an analytic expression is that is can be used as objective function for a constrained optimization problem. The results of the minimization of the drag coefficient are presented.

Table 2 defines the geometrical features and the values of the drag coefficients after and before the optimization.

Configuration	Bi-conic (blunted)
Fixed parameters $[m]$	$R_{cI_i} = 2.5$
Free parameters $[m]$	$L^*_{cI} = 2 \rightarrow 1.009$ $R^*_{cI_f} = R^*_{cII_i} = 2 \rightarrow 2.340$ $L^*_{cII} = 3 \rightarrow 2$ $R^*_{cII_f} = 1 \rightarrow 1.866$
$C_D \rightarrow C_{Dopt}$	$0.4498 \rightarrow 0.4304$
% reduction	4.3

Table 2: Results of optimized drag coefficient for bi-conic nose

The optimization achieves a reduction of the drag coefficient around the 4.5%. It is obtained reducing the length of the nose in order to obtain a shape more cylindrical, respecting the limitation on the mass and the cargo volume. Reducing the inclination of the cone, even the surface exposed to the normal part of the velocity field is reduced, so globally the drag can be lower.

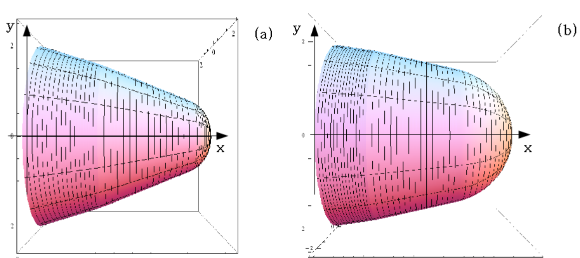

Figure 8: Blunted bi-conic nose with drag coefficient C_D optimized, in design (a) and optimized configuration (b)

Table 3 defines the geometrical features and the values of the ballistic coefficients after and before the optimization.

Configuration	Bi-conic (blunted)
Fixed parameters $[m]$	$R_{cI_i} = 5$
Free parameters $[m]$	$L^*_{cI} = 1 \rightarrow 0.394$ $R^*_{cI_f} = R^*_{cII_i} = 4 \rightarrow 5$ $L^*_{cII} = 3 \rightarrow 2.4$ $R^*_{cII_f} = 2.5 \rightarrow 0.653$
$B^* \rightarrow B^*_{opt}$	$667.3 \rightarrow 250.1$
% reduction	70.8

Table 3: Results of ballistic coefficient optimization for conic nose family

It is important to underline that the optimization process not always converges. In fact it is necessary to introduce geometrical parameters that are already close to the optimal shape in order to have a starting point that allows the convergence. Generally it is enough to check that the radius of the blunted part is satisfying the constraint on the heat flux, so $\dot{q}(r_n) < \dot{q}_{MAX}$. Moreover a limitation on the reduction of not more than 80% on the length of the nose has been imposed.

The results confirm that the spherical termination gives the bigger contribute to the drag, but the conical part is the one that influences more the result. With a study on the sensitivity of the geometrical parameters, it can be easily demonstrated that the cones parameters influence more the final solution. In order to minimize the ballistic coefficient or maximizing the drag, the spherical part is reduced until the heat flux constraint is valid, due to maximizing the conical part.

A physical interpretation of this behavior is given by the Newton model itself, from which the coefficients are calculated. In fact for improving the drag it is necessary a higher pressure coefficient. It is calculated from the loss of normal momentum of the free-steam particles that hit the shape. It is easy to notice that having a bigger conic portion of the nose, with an higher angle δ_c, the normal collision of the particles along the nose results more effective. In fact for the relative angle θ that tends to $\frac{\pi}{2}$, the pressure coefficient increases, $C_p = 2\sin^2\theta$.

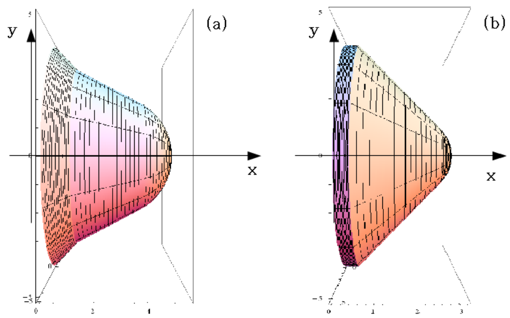

Figure 9: Blunted bi-conic nose with ballistic coefficient B^* optimized, in design (a) and optimized configuration (b)

5 Conclusions

Noticing that generally the noses of the hypersonic vehicles can be often referred to simple shapes of revolution, the aerodynamic coefficients of some basic shapes were calculated, following the Newton theory for the hypersonic regime. Often, the integration of the pressure coefficient along the different surfaces is not a trivial problem. The integrations have been done with the help of the software Mathematica. It is necessary to underline that the selection of the parametrization of the shapes strongly influences the possibility of obtaining an analytic solution. Although Mathematica is a very powerful software, human insight was still critical to obtain the formulation of the aerodynamic coefficients.It was often necessary to add mathematical assumptions to the integrals and model the equations in order to obtain the analytic expressions of the coefficients.

A model of superposition of the effects [1] was adopted in the case of the Newton theory to determinate the coefficient of composed shapes. With this model each shape gives its contribution to the total value of the aerodynamic coefficient so conic, bi-conic, ogive, parabolic and Bézier noses have been analyzed in unblunted or blunted configuration, that is made adding a spherical termination to the nose. In this papaer only the the blunted bi-conic nose is presented. Validated the results from the analytic approach with the panel method, the formulation of the aerodynamic coefficients was also implemented in FreeFem++.

The FreeFem++ environment allows having a completely open-source code for the calculation of the aerodynamic coefficients for hypersonic vehicles and allows to simulate the flux along the shapes, using finite elements to solve the Navier-Stokes equations. The solution given by the numerical simulation however presents sometimes a sensible gap with the values given by the analytic calculation. This is due to many factors: first of all, methods as finite differences or finite volumes are usually preferred for solving the Navier-Stokes equations. Moreover the simplifications made on the equations and on the flux model don't allow the results to be very precise. In fact the density of the flow is considered constant (or semi-constant) and the effects of the viscosity μ were ignored. Also the effects of the temperature are not considered in the equations, effects that are even more important when the velocity of the flux tends to hypersonic values.

Having as objective the developing of a procedure for the calculation of the aerodynamic coefficients in a preliminary phase of the design, the approximations given by the Newton model are adopted. The results of the simulation confirm the validity of this model for high hypersonic flight condition. For lower hypersonic Mach number the solution given by the two methods, analytic and numerical, have a difference up to around 25%, depending on the complexity of the shape, the angles of attack or side-slip, the presence of turbulence or flux separation.

However the study of the evolution of the flux by the numerical simulation of the Navier-Stokes equations has proved the sensible advantage of having an analytic model in terms of Cpu time, as expected. In fact the results of the analytic calculation are obtained in fractions of second, while the ones given by the numerical simulation can take tens of seconds, with a lower precision for the considered model of the flux.

Saving computational time is fundamentally important in a preliminary phase of the design of a hypersonic vehicle, in which is usually preferred obtaining faster results and in which is important to understand which are the better design choices, taking into account an higher level of approximation. Moreover, by the implementation of the analytic model in the FreeFem++ environment, the code results completely free, instead of the commercial codes, and there is the open possibility to improve the numerical model in order to obtain an higher level of details in the evolution of the hypersonic flux. In fact, the implementation of the Euler's equations results inappropriate for describing the hypersonic regime and a better model for the evolution of the flux have to be developed for obtain higher fidelity results in the numerical simulation.

Finally, the analytic expressions of the aerodynamic coefficients allow to build a simple but very effective optimization procedure. The analytic formulations permit to solve the problem of minimizing the drag or the ballistic coefficient, solving a constrained optimization problem. Due to the high non-linearity of the equations, the solution was obtained using the Newton-like internal point method. This algorithm was implemented in Mathematica and can be added in the

FreeFem++ code too. The results of the optimization are in some cases very effective, reaching generally up to 70% of reduction in the ballistic coefficient, up to 4% of in drag coefficient, for the analyzed case.

In conclusion, in this work is presented an interesting approach on the hypersonic aerodynamic coefficients, obtained in an analytic way that allows its optimization, depending on the goals. The advantage in terms of computational time is shown in comparison with the numerical simulation. The biggest advantage of having analytic expressions consist in the possibility of implementing directly the coefficients in the study of the optimal condition needed, obtaining good results.

As open future develops, it is possible to implement the existing database of the studied shapes, adding new or creating new combinations. Having already a FreeFem++ code, it could be possible to improve the fluid model governed by the Euler's equations, in order to obtain a numerical model that better describe the evolution of the hypersonic flux. Finally, the Internal Point algorithm can be implemented in the FreeFem++ environment, obtaining a complete opensource software for the calculation of the aerodynamic coefficients and the shape optimization for a wide range of hypersonic vehicles.

References

[1] Michael J Grant. Rapid simultaneous hypersonic aerodynamic and trajectory optimization for conceptual design. Phd thesis, Georgia Institute of Technology. Atlanta, 2012.

[2] Francesco Villa. Algebraic models for aerodynamic coefficients calculation during the atmospheric re-entry. MSc thesis, Politecnico di Milano. Milano, 2011.

[3] Gianluca Iaccarino. Solution methods for the incompressible Navier-Stokes equations. Lecture notes, Standford University. Standford, 2008.

[4] Florian De Vuyst. Numerical modeling of transport problems using FreeFem++ software. Phd thesis, Department of Mathematics-ENS CACHAN. Paris, 2013.

[5] Luca Dedè. Adaptative and reduced basis methods for optimal control problems in environmental applications. Phd thesis, Politecnico di Milano. Milano, 2008.

[6] Jack Moran. An Introduction to Theoretical and computational aerodynamics. Dover publication, New York, 1984.

[7] Antonio Vitale, Federico Corraro, Guido De Matteis and Nicola de Divitiis. Identification from flight data of the aerodynamics of an experimental re-entry vehicle. Dr. Chaoqun Liu (ed.). Arlington, Texas, 2012.

[8] F. S. Milos, Y.-K. Chen, T. H. Squire. Analysis of Galileo Probe heatshield ablation and temperature data. JOURNAL OF SPACECRAFT AND ROCKETS, Volume 36, Number 3. Moffett Field, California, 1999.

[9] Dominic Dirkx, Erwin Mooij. Continuous aerodynamic modeling of entry shapes. AIAA Atmospheric Flight Mechanics Conference, 8-11 August. Portland, Oregon, 2011.

[10] Peter A. Gnoffo. Planetary-entry gas dynamics. Annu. Rev. Fluid Mech, Volume 31, Pages 459-494. Hampton, Virginia, 1999.

[11] Alessandro Agnetis. Introduzione all'ottimizzazione vincolata. Lecture notes, Università di Siena. Siena, 2013.

[12] Florian De Vuyst. Numerical modeling of transport problems using FreeFem++ software. Phd thesis, Department of Mathematics-ENS CACHAN. Paris, 2013.

[13] Leonardo Rubino. EQUAZIONI DI NAVIER-STOKES: la regina della fluidodinamica. Scribd Inc. Torino, 2010.

[14] F. Hecht. New development in freefem++. Journal of Numerical Mathematics, Volume 20, Issue 3-4, Pages 251–266. Paris, 2013.