ISSN (Print) : 0974-6846
Indian Journal of Science and Technology, Vol 9(42), DOI: 10.17485/ijst/2016/v9i42/95791, November 2016 ISSN (Online) : 0974-5645

Drag Reduction by Rotation of Frontal Shell in Case of Axisymmetric Slender Body

Sanjay Darvekar[1*] and Subrahmanyam Kanney[2]

[1]TSSM's Bhivarabai Sawant College of Engineering, Pune-411041, India; sanjay.kdarvekar@gmail.com
[2]Gayatri Vidya Parishad College of Engineering, Visakhapatnam - 530048, Andhra Pradesh, India; acme18@gmail.com

Abstract

Objectives: The reduction of drag is one of the primary considerations for the engineers since it facilitates the increase in speed with limited engine power. Drag reduction also provides energy saving which makes it an everlasting engineering study. This work is focused on utilizing the inevitable disadvantageous forces like Coriolis force and Munk-moments to favor the performance of the axisymmetric slender body with some modifications. **Methods/Analysis:** This conceptual study deals with the analysis of the drag force on a cylinder with rotating head, underwater at different conditions by utilizing the Computational Fluid Dynamics (CFD) simulation software ANSYS FLUENT. Certain fluid is made to flow over the axisymmetric body at various speeds, while the cylinder head is rotating at specific angular velocity. **Findings:** A common conclusion is drawn from the observations of contours of pressure drag, shear stress, turbulent kinetic energy and wake for spinning and non-spinning heads of the cylinder. The significant reduction in drag was observed with the combination of specific angular velocity of the rotating head and the axial velocity of the axisymmetric body. **Novelty/Improvement:** A novel concept of drag reduction by externally powered rotating frontal shell, with specific speed, is presented.

Keywords: Axisymmtric Head, Axial Velovity, Angular Velocity, CFD FLUENT, Spinning Drag Force

1. Introduction

The movement of an object in space is not usually easy since the object is exposed to external forces resisting to its motion called drag. The drag force depends on the medium, shape, velocity and orientation of the object[1,2]. It is composed of two components, the wall shear stresses due to viscous effects and normal stresses due to pressure[3]. In order to carry out these integrations the shape of the body and the distributions of pressure and shear stresses must be known. This is usually achieved with the aid of CFD tools or experiments[4].

Axisymmetric bodies have a wide range of applications, both in aero and hydrodynamics and are undergoing researches to improve its performance. Previous research works on axisymmetric bodies was focused on predictions of the individual dynamic coefficients, and all the drag reduction techniques were limited only to the fields of body shaping, polymer induced drag reduction, drag reduction with micro bubbles, riblets etc[5]. As an example, aerodynamic drag reduction of truck was carried out by[6]. However, this work is an attempt to explore in the direction of the multi-body dynamics.

Even if we consider rockets which are statically stable, the dynamic stability is not promised, and the coning motion appears due to certain forces which destabilize the path of the projectile motion[7]. Those forces are Coriolis force and Munk-moments etc., which gives rise to coning motion. This coning was eliminated, to stabilize the path, in all most all the cases by appending fins, which adds drag indeed[8]. The reason for eliminating this coning motion is, they elevate the gyroscopic moments and are incremental and non-linear.

Only the frontal area section and its profile play the major role in determining the drag on an axisymmetric slender body[9] because that is where the fluid makes the initial contact covering a reasonable area which decides the boundary layer. The resultants are stagnation, line

*Author for correspondence

formation (at an angle), shock wave, boundary layer thickness, shear stress (skin friction) etc. This paper presents a new concept of utilizing the coning motion to reduce the drag on axisymmetric slender bodies. CFD analysis on axisymmetric slender body was carried out in FLUENT to study the effect of rotating frontal shell upon drag reduction.

2. Proposed Model

A concept of an axisymmetric body with rotating frontal shell or head is proposed by the authors in this paper. An attempt is made to utilize the undesirable Coriolis forces and Munk-moments acting upon the underwater body while moving axially, to reduce the drag. Rather than restraining the coning motion caused by these forces with the help of fins, additional spinning can be imparted to the frontal shell of the body[10]. As spinning of the entire body is not desirable, only the head or a front conical portion can be provided with a shell that can rotate freely over it as shown in Figure 1[2]. This model assumes a provision to rotate the frontal shell with various angular speeds. The externally powered spinning shell could help to eliminate line formation, reduce the stagnation area and reduces the skin friction, thereby significantly reducing the drag on the underwater axisymmetric slender body. This conceptual model could also help in the stabilization of the path.

The conventional drag analysis was carried out in ANSYS FLUENT while certain fluid is made to axially flow over the axisymmetric body at a specific speed and the frontal shell of the body is made to rotate with certain angular velocity. The Drag force F_D is taken into account at every stage of permutation of fluid velocity and Angular velocity.

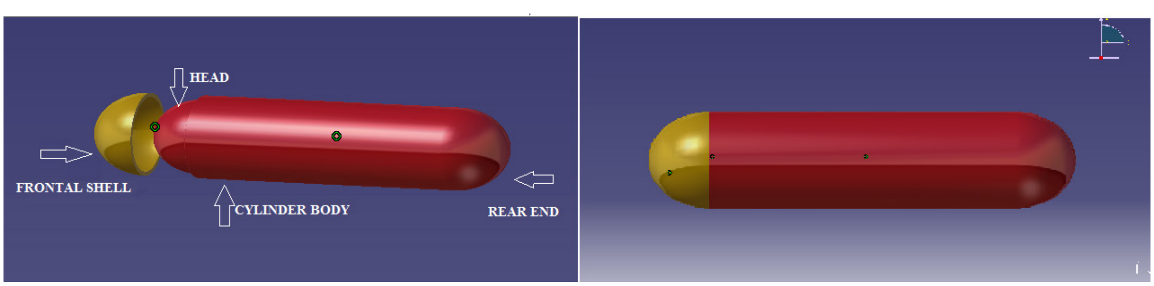
3. Methodology

3.1 Meshing

As this is a conceptual model to study the behavior of the spinning axisymmetric body in a fluid, the dimensions are explicit assumptions by taking the L/D ratio of 3.5.The model is created in ICEM (mesh program) by using the basic entities such as lines and circles and prepared for grid generation. The model includes cone1, cone 2, and cylinder. The conceptual model is subjected to two different motions as axial or linear motion of the entire body and the angular motion of the frontal shell and the domain sizes are 7D and 4D respectively. To model these motions two domains were created, domain-one for axial motion, and domain two for spinning motion of the frontal shell, as shown in Figure 2. The complex part is meshing a model with two different domains (flow in linear and angular directions) so in order to accomplish this, the two domains are meshed separately and merged in the final stage before exporting the model into FLUENT solver. In the FLUENT solver the domains are merged using "interface" options. Thus the solver allows two domains to interact and makes the mutation of boundary conditions feasible. Total 652289 elements were generated with good mesh quality as shown in Figure 3.

3.2 Solver Settings for Cylinder

The solver adopted in CFD simulations is the Ansys FLUENT 6.2.16 code using the k-e turbulence model to solve the three dimensional viscous incompressible flow. Wall boundary conditions are applied along the boundaries of cylinder, cone-2 (Figure2). Velocity-inlet is defined for boundary far-field and boundary inlet. Interface condition is given to boundary interface-1



(a) (b)

Figure 1. Axisymmetric body with frontal shell (a) Exploded view (b) Assembled view.

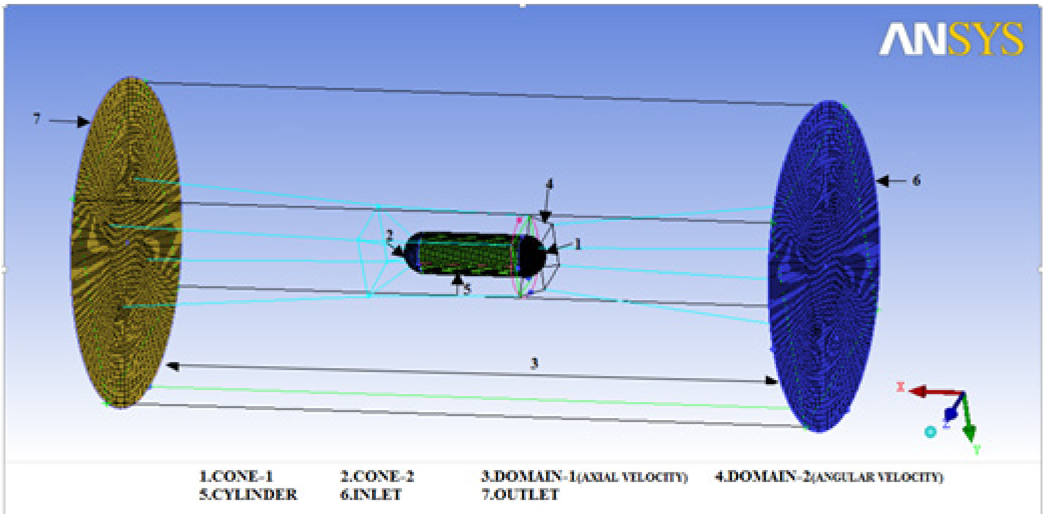


Figure 2. Illustration of meshed parts.

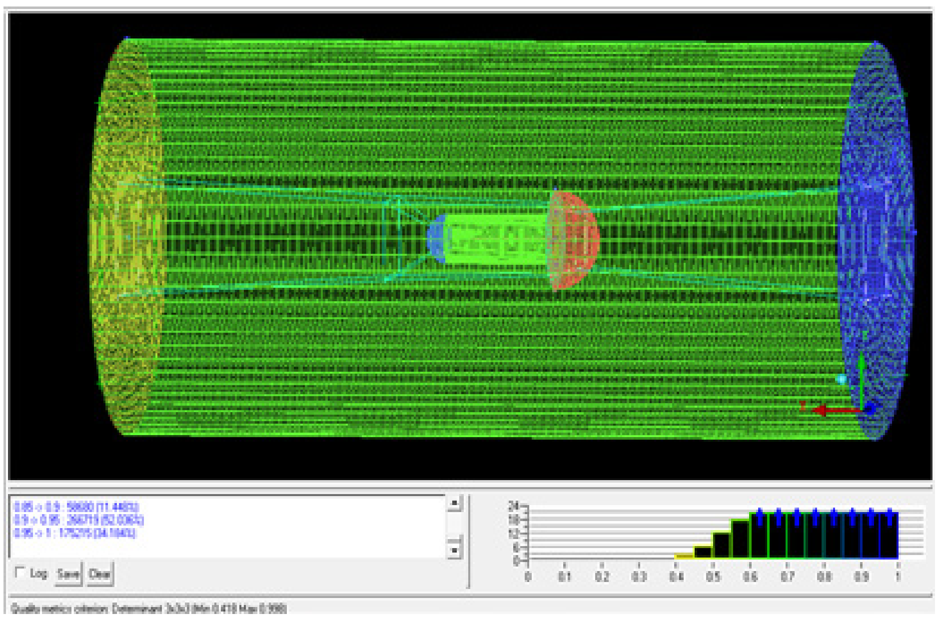


Figure 3. Grid over the entire domain.

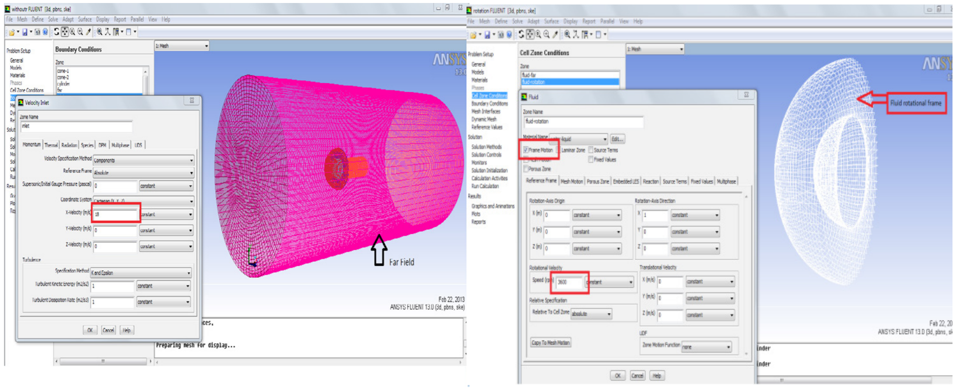
and boundary interface-2. The pressure-outlet is given to the boundary outlet. Cylinder details and cylinder flow characteristics are presented in Table 1 & Table 2 respectively.

Cell zone conditions: The axial and angular velocity parameters are assigned in this cell zone window. In order to set the fluid in the domain-2 in rotation, frame motion in the cell zone menu "Fluid rotation" is enabled and the corresponding angular velocity is assigned as shown in the Figure 4.

Then the solution control parameters were adjusted and the flow field was initialized by clicking on "solution is initialized". Iteration process was then initialized by clicking an option - "run calculation". When the solution converges, it automatically saves and stops.

Table 1. Cylinder Details

Cylinder	Controllable pitch cylinder
Principal Dimensions	Diameter, D
Domain size	Cylindrical domain of length 7D, Dia 4D.
Mesh count	652289 Hexahedral cells.



(a) (b)

Figure 4. Illustrations of assigning velocities (a) assigning axial velocity and (b) assigning angular velocity.

Table 2. Cylinder flow characteristics

Pressure Link	SIMPLE
Pressure	Standard
Discretization scheme for convective fluxes and turbulence parameters	Quadratic Upwind (QUICK)
Turbulence model	K-e
Solver	Steady, Moving Reference Frame

4. Results and Discussion

The drag force analysis was carried out for three different linear velocities of an axisymmetric body, such as 9, 18, 36 m/s. For each linear velocity, five different angular velocity conditions on frontal shell were considered to study the drag phenomena, such as no rotation, unconstrained free rotation, angular velocity (*N*) 900, 1800, 3600 RPMs respectively. From the Figure 5 and Figure 6, for the same axial velocity of 18m/s, it is observed that the total pressure on the cylinder without rotation is high (240.3 KPa) and in case of a cylinder with rotating front it is much less (111.0 KPa).

Similarly, from Figure 7 and Figure 8 it is observed that the wall shear stress on the cylinder with axial velocity 18 m/s, without rotation model is very high (2.421 KPa) in comparison with cylinder with rotational model (1.703 KPa). The downstream pressure (the central region of the figure), as seen in Figure 9 and Figure 10, is also less in case of a cylinder with rotational model compared to the cylinder without rotation for the same axial velocity.

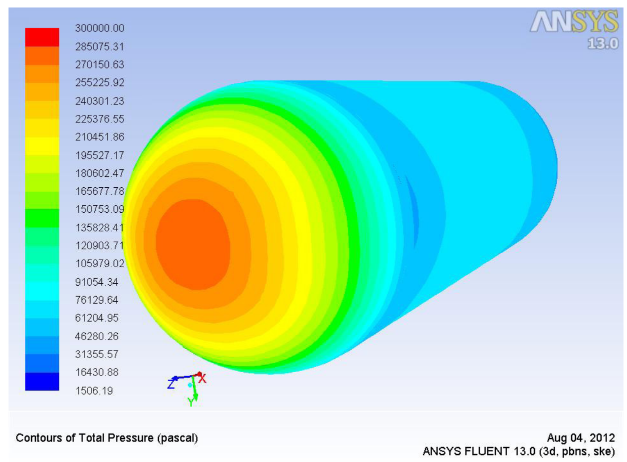


Figure 5. Total pressure on cylinder for V = 18m/s & without rotation.

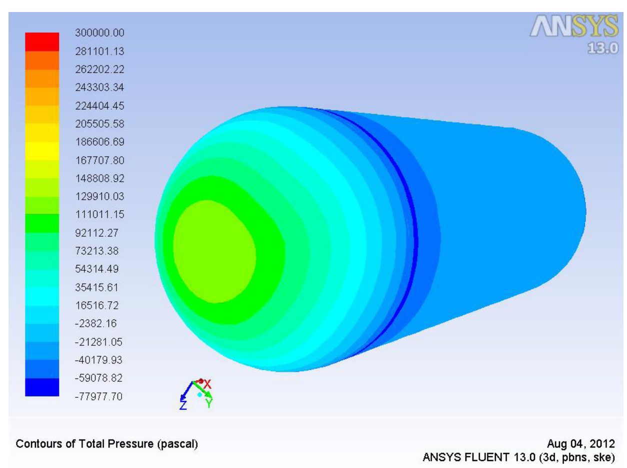


Figure 6. Total pressure on cylinder for V = 18 m/s & N = 900 RPM.

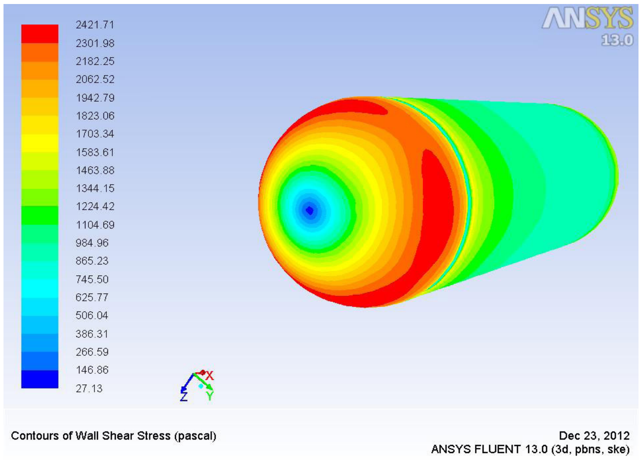


Figure 7. Wall shear stress for V = 18 m/s & without rotation.

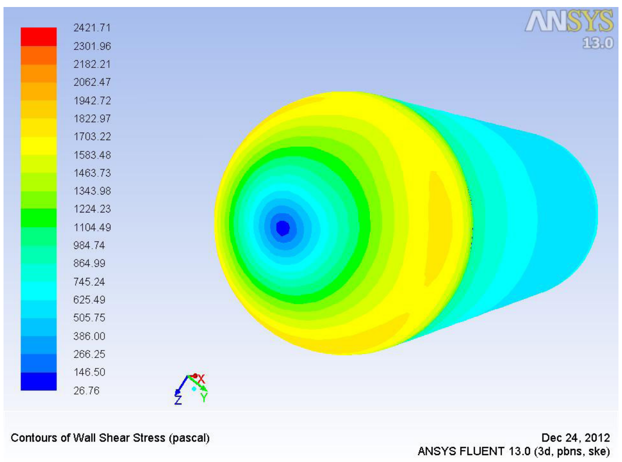


Figure 8. Wall shear stress for V = 18 m/s &N = 900RPM.

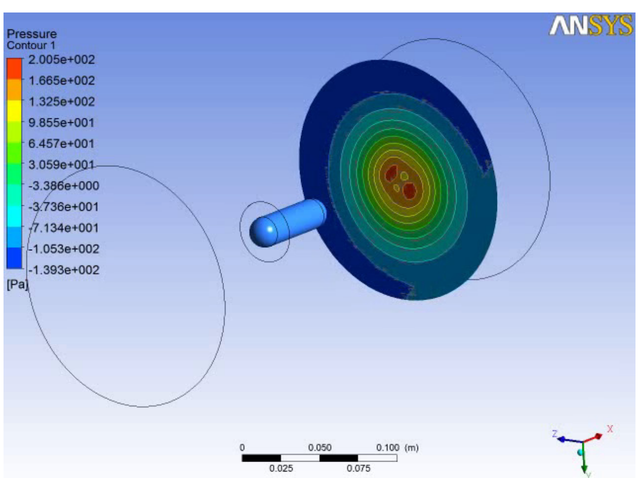


Figure 9. Pressure downstream for V=18m/s & without rotation.

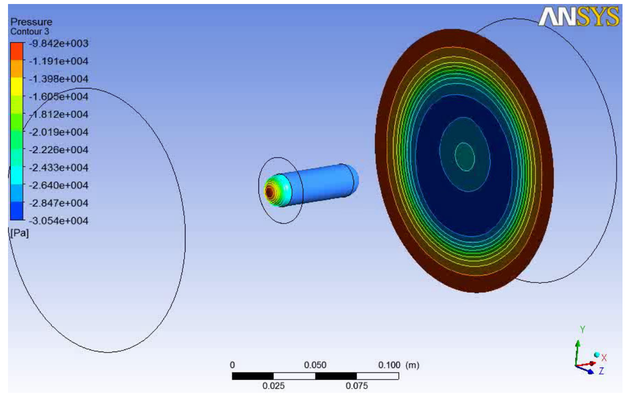


Figure 10. Pressure downstream for V = 18m/s & N = 900 RPM.

From the tabulation and contour figures it is clear that the drag force on the rotating model has declined when compared with stationary model. The physics behind this is that the Centrifugal force of fluid particles near the wall of cylinder body makes the fluid particles to move towards the axis, which lead to the reduction in Boundary layer thickness[8]. The provision of a thin rotating frontal shell prevents the coning motion to transfer from the frontal head to rest of the body, which prevents the nonlinear gyroscopic moments and avoid destabilization. For a spinning body the stagnation point moves away from the surface of the shell[5] as the motion is axisymmetric thus reducing good amount of drag. Skin friction (shear stress) is also very less on a spinning frontal shell rather than on the stationary slender body as the former accounts to rolling friction. Shell spin rate molds to all the nonlinear forces in case of free rotation. The idea is to create a shell in the front region, which is allowed to freely rotate utilizing the unfavorable forces like Coriolis force and Munk moments. The mass of the shell is far less than the entire body so it does not affect the gyroscopic moments. As only the shell is allowed to rotate it does not allow the rolling moment to translate over entire body, thus it effectively neutralizes the roll moment which builds up in the case of without rotation model. The drag forces which were estimated using CFD solver, FLUENT, on the without rotation model were compared with Numerical values. To numerically determine the drag force Equation (1) was used.

$$F_D = \frac{C_D A \rho V^2}{2} \qquad (1)$$

From the Table 3 it is observed that the error between CFD solver and numerical values is within the acceptable range. Table 4 shows the drag force at all angular and axial velocities.

Table 3. Drag forces Comparisons of Numerical and Simulated results

Velocity 'V' m/s	Drag Force (N)		Error [%]
	Numerical	Simulation	
9	8.76	10.47	19.5
18	35.0	31.27	-10.6
36	140.1	103.23	-26.2

Table 4. Drag force in Newton at all angular and axial velocities

Velocity 'V' m/s	Without rotation	Unconstrained [Free rotation]	N=900 RPM	N=1800 RPM	N=3600 RPM
9	10.47	10.25	10.24	10.233	9.56
18	31.227	18.3	13.08	13.33	13.24
36	103.23	100.34	100.41	99.3	93.07

5. Conclusion

A concept of an axisymmetric body with rotating frontal shell or head is proposed. As part of continuing research on the flow about slender bodies of revolution, a coning motion for hydrodynamics model was tested by simulation. A model of the axisymmetric slender body was created in CFD solver and was simulated to have an axial flow of the fluid and the rotating frontal shell. From the results it can be concluded that by rotation phenomenon, the drag force has decreased with a good margin for the specific combination of V= 18m/s and N=900 RPM. Decrease in pressure gradient in the downstream wake region, of rotating model also proves that the frontal shell has a great influence on boundary conditions and gives favorable results when set in rotational motion. Future scope includes experimental testing of the concept and deriving relation between axial velocity and rate of spinning of frontal shell for the optimal result.

6. References

1. Kumar DS. Fluid Mechanics. 3rd Edition. S.K. Kataria and Sons. India, 1988.
2. Frank MW. Fluid Mechanics. 5th edition. McGraw-Hill. New York, 1980.
3. Kornilov VI. Turbulent Boundary Layer on a Body of Revolution with Periodic Blowing/Suction. Thermo-physics and Aeromechanics. 2006; 13(3):343–57.
4. Suman KNS, Nageswara Rao D, Das HN, Bhanu Kiran G. Hydrodynamic Performance Evaluation of an Ellipsoidal Nose for a High Speed under Water Vehicle. Jordan Journal of Mechanical and Industrial Engineering. 2010; 4:641–52.
5. Davood H, Hongli P, Guoliang Yu. On the estimation of the falling velocity and drag coefficient of torpedo anchor during acceleration. Ocean Engineering. 2011; 42:135–46.
6. Raut G, Chilbule Chaitanya M, Upadhyay A, Kumar MJ. Impact of Profile modifications and rear view mirror on aerodynamics and fuel economy of goods carrier. Indian Journal of Science and Technology. 2015; 8(S2):224–30.
7. Xuerui M, Xing YS, Yong X. Coning motion stability of wrap around fin rockets. Science China Ser E-Tech Sci. 2007; 50(3):343–50.
8. Quek R. Computational Modelling of High Speed Finned Slender Body Aerodynamics. The Singapore engineer. 2008 Jul.
9. Some Aspects of Submarine Design. Defense Science and Technology Organization. DSTO–TR–1920, Australia, 2006.
10. Boatright JA. A Coning Theory of Bullet Motions. Precision Shooting Magazine. 2011; 59(1):34–44.

Nomenclature

V	Axial velocity (m/s)
C_D	drag coefficient
F_D	Drag Force (N)
A	Area (m^2)
N	Angular velocity of Frontal shell (RPM)

Greek symbols

ρ	density (kg/m^3)