



25<sup>th</sup> ABCM International Congress of Mechanical Engineering  
October 20-25, 2019, Uberlândia, MG, Brazil

**COB-2019-1697**

## **COMPUTATIONAL AND EXPERIMENTAL ANALYSES OF THE AIRFLOW OVER A ROCKET MODEL**

**Vitor Gabriel Ferraz Leite Rebouças de Farias**

**Victor Carpigiani da Silva Bezas**

Universidade Federal de Uberlândia - Av. João Naves de Ávila, 2121 - Santa Mônica, Uberlândia - MG, Brazil. CEP: 38400-902  
vitor\_gflrf@hotmail.com , bezasyt@gmail.com

**Felipe José Oliveira Ribeiro**

**Odenir de Almeida**

Universidade Federal de Uberlândia - Av. João Naves de Ávila, 2121 - Santa Mônica, Uberlândia - MG, Brazil. CEP: 38400-902  
feliperibeiro.ufu@gmail.com, odenir.almeida@ufu.br

**Abstract.** *This work has the goal to present the results of computational and material experiments of the flow over a stationary rocket, analyzing the drag coefficient and how it behaves according to the Reynolds number. The initial hypothesis is that the fluid velocity is between the interval in which the flow is considered incompressible, that is, from air flows in velocities that ranged from 12 m/s to 24 m/s. The computational fluidodynamic simulations (CFD) were made by Ansys Fluent software and for its solver the Shear-Stress Transport (SST)  $k-\omega$  model was used which accounts for the transport of the turbulence shear stress in the definition of the turbulent viscosity. The material experimentation was carried out in the Center of Research in Experimental Aerodynamics' (CPAERO's) wind tunnel, in order to validate the computational results. In the CFD simulation, an average drag coefficient of 0.63 was obtained and with the wind tunnel experiment, a value of 0.30, representing a relative error of 109 % for the Reynolds range from  $4.10^5$  to  $1.10^6$ . Future research will be developed to better understand the reasons that led to the discrepancy in the results.*

**Keywords:** *Rocket, Wind Tunnel, CFD, Aerodynamics, Fluid Mechanics*

### **1. INTRODUCTION**

Rocket manufacturing is one of the most challenging and complex tasks, since it brings to the limit almost all fields of the engineering involved. Such fact is due to the high velocities, forces and altitudes normally experimented by such vehicles (Newman, 2002). The aerodynamic forces are crucial for the study of the rocket's dynamics, as, for low altitudes flights, the forces from the interaction between rocket and atmosphere is present through all the flight trajectory.

In the industry there are two big fields of aerodynamic analysis approaches: the computational and experimental methodologies. The experimental study is the one with higher reliability (Mani *et al.*, 2004), while the computational implementations are capable of resulting in a more detailed observation, but requires validation.

Wind tunnel testing is indispensable for aerodynamic researches and rocketry development (Zhu, 2019). With the use of additive manufacturing to test experimentally, the prototypes fabrication can be faster and cheaper meaning a way to create a bridge between the Computational Fluid Dynamic (CFD) and experimental ground tests (Tyler and Braisted, 2005).

The use of wind tunnel aerodynamic balance is important during the tests and allows to obtain the forces and moments acting on the body. There are two basic types of aerodynamic balance operation: the internal or external approaches. The first type is designed to fit inside the tunnel and its main advantage is that the rod which connects the body has little influence on the wetted area of the prototype. The external balances are located outside the tunnel and because it has a complicated mechanism of force transmission, it is difficult to determine the influence of the rod on measurements. There are experiments in the aerospace industry that conduct tests with wind tunnels (Gebink *et al.*, 2018) that have an internal balance (DeMar, 1995), but as the used tunnel has greater applicability for automotive testing which has an external balance, this form of instrumentation will not be presented.

On the present paper, the drag coefficient over an rocket outer geometry (The Marvin 1, developed by Equipe de Propulsão e Tecnologia Aeroespacial (EPTA) of the Federal University of Uberlândia (UFU)) is evaluated through computational simulations. Experimental tests were also carried out for validation purposes, using a wind tunnel with an external aerodynamic balance.

## 2. METHODOLOGY

This work aims to present an experimental aerodynamic study on a stationary rocket through a low speed wind tunnel, a classification used for tunnels that reach up to about Mach 0.4 (Barlow *et al.*, 2015). The results from such studies then were compared in terms of drag coefficient with results found in previous work (Alves *et al.*, 2018).

Computational fluid dynamics (CFD) simulation was performed using Ansys Inc. software for the same range of speeds, temperature and pressure as carried in the experimental procedure. In these simulations it was possible to obtain the drag force and drag coefficient for the rocket under such a flow condition.

### 2.1 Physical Model

The geometry of the rocket model was provided by Equipe de Propulsão e Tecnologia Aeroespacial (EPTA) and its dimensions were optimized for a 500 m apogee launch. The rocket has a length of 603 mm and a diameter of 60,3 mm, as well as an elliptical nose of length 120,6 mm and trapezoidal fins with dimensions given in Fig. 1.

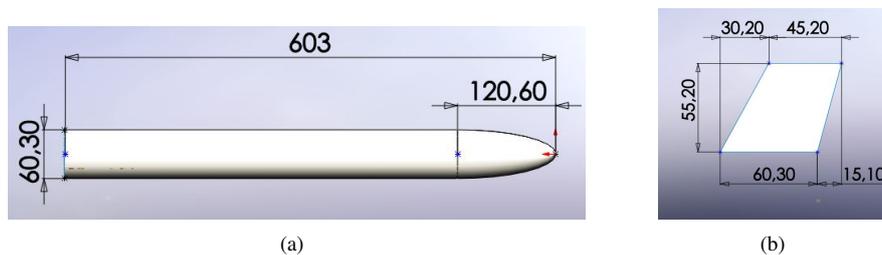


Figure 1. Rocket model: a) entire fuselage and b) rocket's fins.

### 2.2 Computational Methodology

The computer simulation was performed using the Ansys Fluent software, which includes the definition of the domain, the mesh, the boundary conditions to the numerical solution to obtain the results. In this context, a steady and incompressible flow was defined for the Solver, being solved with Shear-Stress Transport (SST)  $k-\omega$  model which accounts for the transport of the turbulence shear stress in the definition of the turbulent viscosity and show better agreement with experimental data for adverse pressure gradient boundary-layer flows (Menter, 1994).

The Simple algorithm was used as a pressure-velocity coupling scheme that through the relation between velocity and pressure corrections allows to reinforce mass conservation and obtain the pressure field. In addition, a second-order spatial discretization of the convection terms was chosen in the solution equations, which allows more precise results to be obtained for triangular and tetrahedral meshes. Finally, with a convergence criterion for the residues of order  $10^{-6}$ , 5000 iterations were made for each flow velocity.

The computational domain was defined for this analysis as a three dimensional fluid region where the transport equations for each mesh element will be solved, presenting a geometric shape that is similar to a box as shown in Fig. 2.

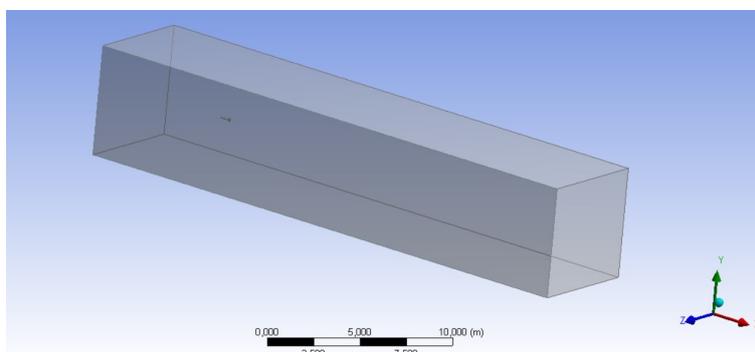


Figure 2. Computational fluid domain

This domain must be large enough to contain the region of influence promoted by the interaction between the rocket and the airflow, and it should also be estimated to what extent the mat effects occur to define the limits of the box. However, as the computational resources are limited, the mesh was optimized and scaled according to Tab. 1.

The mesh was created in Fluent Meshing, containing approximately three million triangular and tetrahedral elements with refinement near the rocket and in its mat. A layer of prisms elements is used in the contour of the rocket in order to correctly capture the boundary layer as shown in Fig. 3.

Table 1. Domain dimensions

Axis	Inferior limit [m]	Superior limit [m]
x	-6,03	25,341
y	-3,105	3,105
z	-3,105	3,105

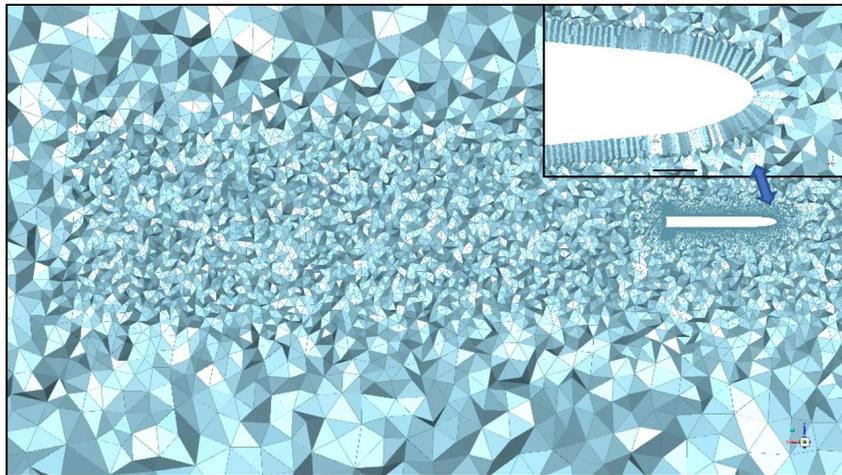


Figure 3. Mesh detail near the rocket

Thus, it was possible to define the boundary conditions for the six faces of the domain (box), as referenced in Fig. 4, and on the outer surface of the rocket. For inlet, outlet and symmetry (four) faces, velocity-inlet, pressure-outlet and symmetry were chosen, respectively. Moreover, it is valid to say that in inlet and outlet conditions turbulent intensity of 1% and turbulent viscosity ratio of 10 were defined. On the rocket, the wall option was used with the non-slipping condition and the reference values for the simulations such as frontal reference area, pressure, temperature, airflow velocity and specific air mass were the same as the experimental procedure.

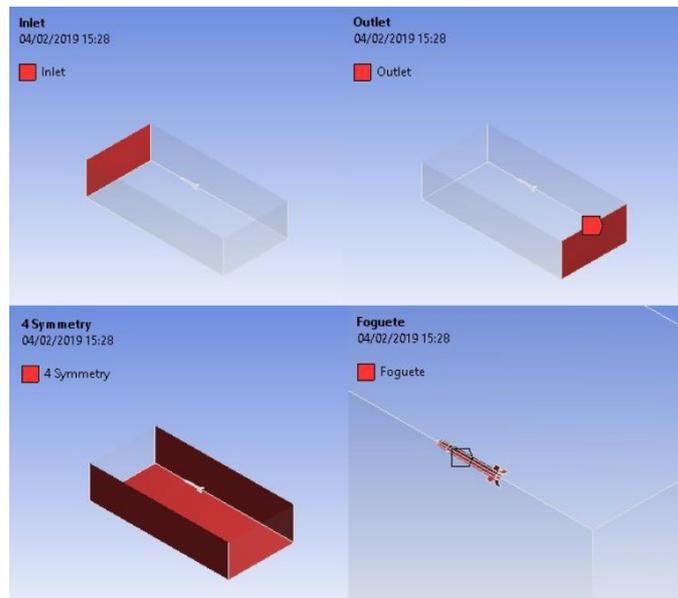


Figure 4. Named selection for domain faces: inlet, outlet, 4 Symmetry and foguete (rocket).

### 2.3 Material Methodology

The model of the rocket was manufactured in a 3D printer Prusa i3 Mk2s with PETG (Polyethylene terephthalate) polymer filament in scale 1:1. The rocket was divided into segments in relation to the 3D printer maximum volume, which is 250 x 210 x 250 mm. The first manufactured part is the base and it has a project hole in the pressure center for

coupling with the wind tunnel aerodynamic rod Fig. 5(c). The fins were printed separately as shown in Fig. 5(a).

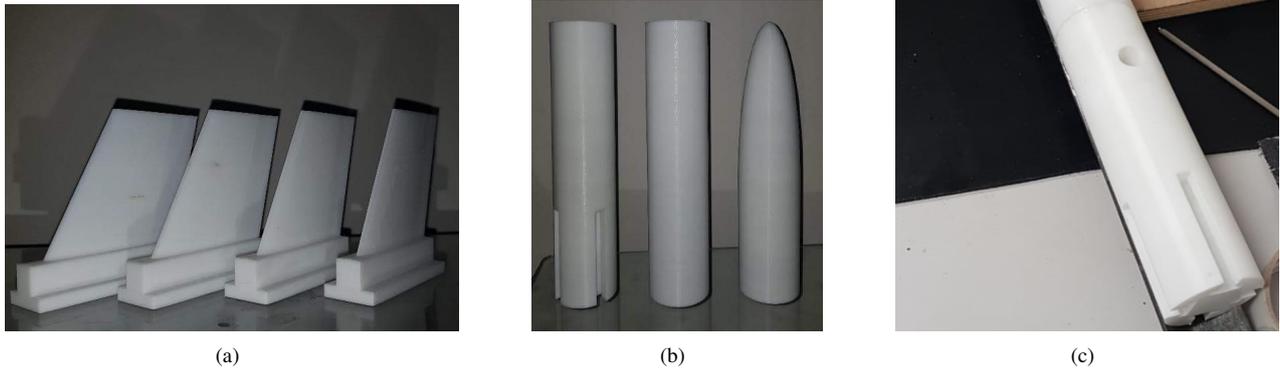


Figure 5. 3D printed parts: a) Fins, b) three parts of the fuselage, and c) detail of the hole for coupling with the wind tunnel rod.

All the parts were assembled, glued, sanded and painted resulting in the final rocket geometry. An aluminum cylinder was machined as a support for the measuring rod of the balance, being inserted in the previously mentioned hole and properly fixed with glue. It's important to emphasize that the aerodynamic balance used in this study requires the fixation by an axle. In this case, the balance is physically placed in the sidewall of the Wind tunnel test-section and it is believed that this setup could influence the final results.

The printing principle of additive manufacturing is layer-by-layer deposition. This fact influences the surface roughness of the model and has a direct impact on the results obtained in wind tunnel tests (Penna *et al.*, 2003). Then, for the surface finish of the rocket was used polyester resin and automotive polyurethane paint. The abrasive finish with sandpaper and subsequent reinforcement with resin resulted in a smooth and adequate surface for the experimental test as shown Fig. 6.

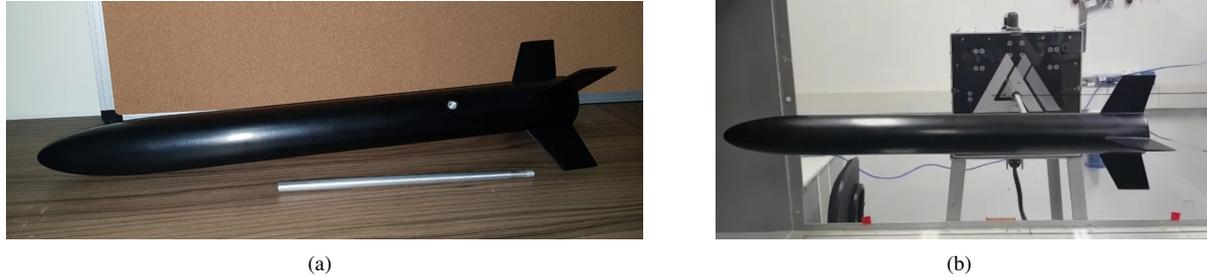


Figure 6. Finished rocket: a) rocket and the rod and b) Rocket-rod coupling on aerodynamic balance.

## 2.4 Experimental Methodology

Before carrying out the tests, it is necessary to calculate the frontal reference area  $S_{ref}$  of the rocket model considering the largest diameter  $D$  through Eq. (1). For a diameter of  $0.0603\text{ m}$ , there is an area of approximately  $0.002856\text{ m}^2$ .

$$S_{ref} = \pi \frac{D^2}{4} \quad (1)$$

In addition, the outside temperature  $T$  of the wind tunnel and ambient pressure  $p$  during the test should also be measured to calculate the specific mass  $\rho$  of the air during the experiment using the ideal gas law as shown in Eq. (2). For a measured temperature of  $28\text{ }^\circ\text{C}$ , pressure of  $1,012.10^5\text{ Pa}$  and the specific constant  $R$  for dry air equals  $287,058\text{ J/kgK}$ , the specific mass of the air results in  $1,1712\text{ kg/m}^3$ .

$$\rho = \frac{p}{R \times T} \quad (2)$$

The Reynolds number was calculated using the length of the model rocket ( $L$ ), the airflow velocity  $v$  and the dynamic viscosity  $\mu$  by the Eq. (3).

$$Re = \frac{\rho v L}{\mu} \quad (3)$$

The viscosity of the gases depends essentially on the temperature (Fox *et al.*, 2011), so it is possible to use the Sutherland equation to obtain the dynamic viscosity value for air by the Eq. (4).

$$\mu = \frac{bT^{3/2}}{S + T} \quad (4)$$

For the air,  $b = 1,458 \cdot 10^{-6} \text{ kg/msK}^{1/2}$  and  $S = 110,4 \text{ K}$ . Then, the temperature being 301 K the dynamic viscosity is  $1,85 \cdot 10^{-5} \text{ Ns/m}^2$ .

The experiment was carried out in the wind tunnel Facility of the Center of Research in Experimental Aerodynamics (CPAERO) of the Federal University of Uberlândia as shown in Fig. 7, which has a test section of  $0,6 \times 0,6 \text{ m}$  and one meter long. The airflow is created by a rotor of 12 blades driven by a 18388 W electrical engine on the upstream of the wind tunnel and the air velocity is driven by an electrical inverter (output from 0 to 60 Hz).

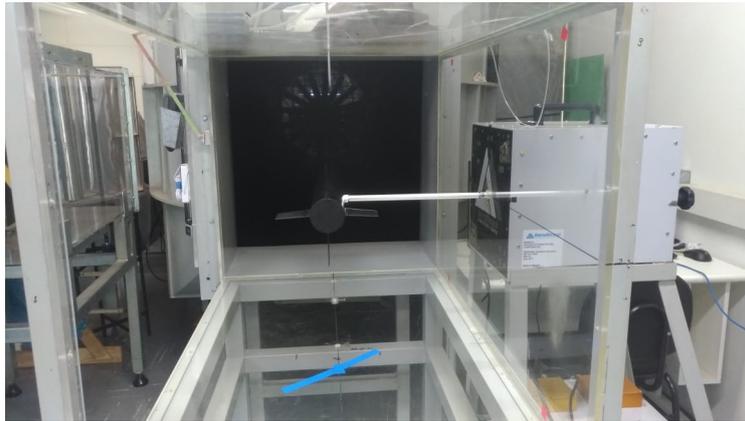


Figure 7. Wind tunnel

The wind tunnel works in conjunction with a three-component external aerodynamic balance that calculates normal force (lift), axial (drag) and pitch moment. The balance model is A-TAB1 capable of measuring drag up to 50 N with accuracy of 0,8 % and a resolution of 0,01 N. In order to compute the drag of the rocket, two configurations were performed. The first one contains both rocket and rod and the second just the rod, represented by the Fig. 8.

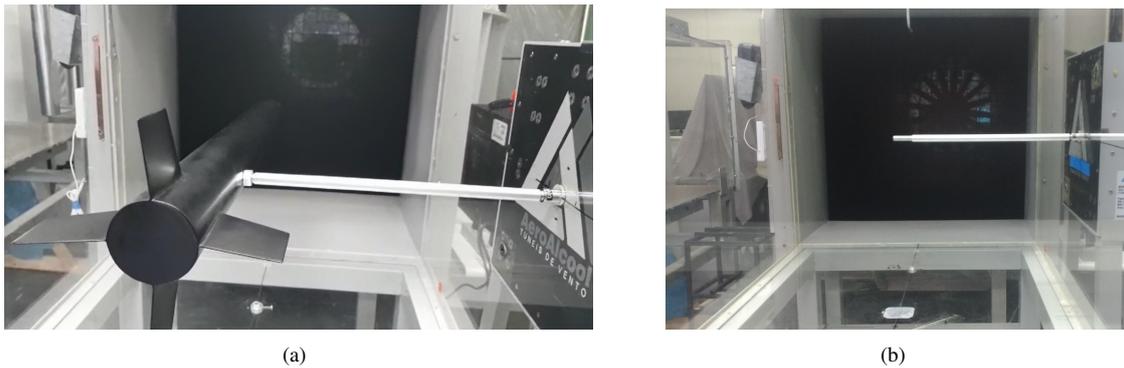


Figure 8. Configurations used on the wind tunnel experiment: a) first configuration and b) second configuration.

As the power provided to rotate the blades was limited, it was chosen for each configuration a flow velocity of 12 m/s up to 24 m/s with intervals of 2 m/s between the tests to obtain the drag force. The tunnel turbulence intensity for this speed range is approximately 1 %. In addition, to ensure that the rocket was parallel to the airflow without angle of attack, an aluminum level was used before the measurements. Finally, by subtracting the calculated drag from the rocket and rod configuration by the drag produced in the test involving only the rod as shown in Fig. 8(b), it was possible to calculate the drag coefficient  $C_d$  of the rocket according to Eq. (5).

$$C_d = \frac{F_d}{\frac{1}{2} \rho v^2 S_{ref}}, \quad (5)$$

where  $F_d$  is the difference between the drag forces obtained by the external balance in newtons.

### 3. RESULTS AND DISCUSSION

During the experimental methodology, it was possible to collect the drag force with the aerodynamic balance and also calculate the drag coefficient and the Reynolds number associated with the airflow velocity which the rocket model was submitted. Experimental results are presented in Tab. 2.

Table 2. Experimental data

Flow velocity [ $m/s$ ]	Drag force [ $N$ ]	Drag coefficient	Reynolds number
12	0,052974	0,22	$4,6 \cdot 10^5$
14	0,085347	0,26	$5,3 \cdot 10^5$
16	0,174618	0,41	$6,1 \cdot 10^5$
18	0,128511	0,24	$6,9 \cdot 10^5$
20	0,155979	0,23	$7,6 \cdot 10^5$
22	0,309996	0,38	$8,4 \cdot 10^5$
24	0,361008	0,37	$9,2 \cdot 10^5$

The computational results obtained by CFD in Ansys can be compared with the experimental results. When comparing these values, a discrepancy is noticed and it is possible to calculate the relative error for the drag coefficients averages in this Reynolds range. The average  $C_d$  for CFD simulations is 0.6313 and the average  $C_d$  for tunnel tests is 0,3014. Then, the relative error is 109 % and does not validate the computational model.

However, the computational results obtained by numerical simulation can be compared with the values for the same geometry using the MFsim code and OpenRocket software (Alves *et al.*, 2018) in Fig. 9. The OpenRocket program allows the development of a rocket using basic data such as its geometry and material and the physical model was recreated in the software where the simulations of the launching of the rocket were executed, thus obtaining data such as speed, acceleration, altitude and relations of several parameters, among them  $C_d \times Re$ . The MFsim code solved the Navier-Stokes equations in its tridimensional transient form, using a block-structured mesh with local adaptability.

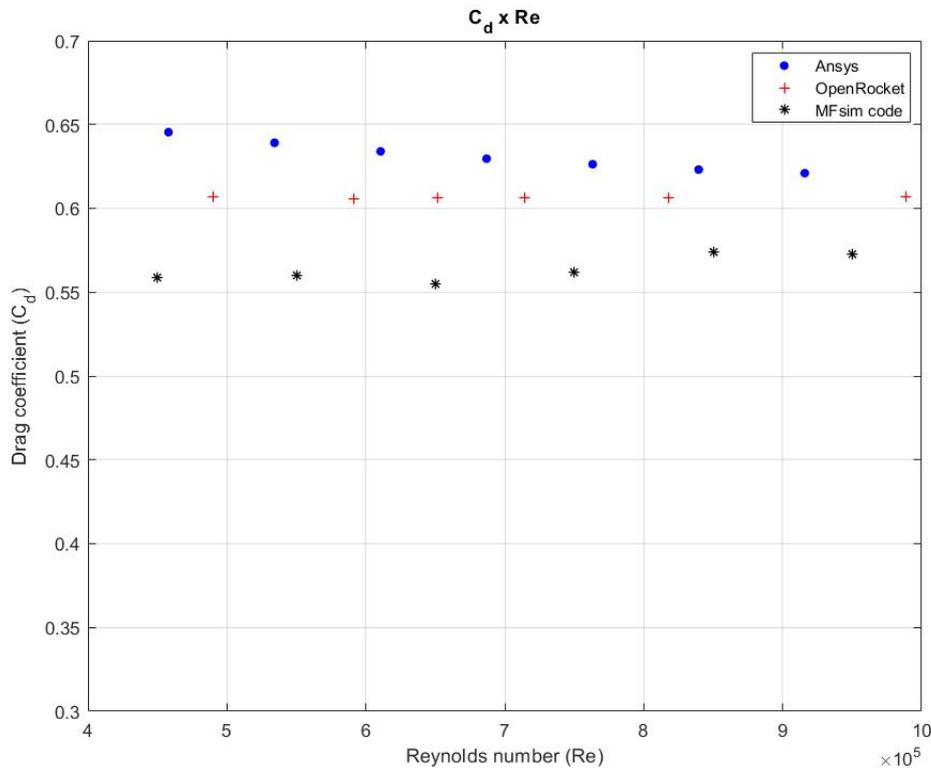


Figure 9.  $C_d \times Re$  graph

It is possible to realize that these values are close and should not be far from 0,6, considering that different computational methods were used that approximate the value of the drag coefficient. However, computer simulation does not exclude the need to compare with experimental values, requiring validation.

Therefore, it can be assumed that there is interference from the fixation rod of the rocket model in the body flow under study, presenting itself as an experimental problem so that it is possible to validate computational results that represent the simulation of a free flight without the presence of the rod. In addition, it is possible to perform a CFD simulation considering both the rocket and the rod for comparison with the experimental results.

For this reason, to visualize these effects that promoted the difference obtained in the computational-experimental comparison, it is practical to change the way the rocket is fixed on the balance. Two possible ways would be to use a thinner rod on the side or a new rod fixing behind the rocket to minimize the influence of that part on the drag that will be measured. These results are preliminary and for a next study, new possibilities will be evaluated and the results compared with the computer simulations.

#### 4. CONCLUSION

The initial objective of the project is to validate the theoretical aerodynamic model by the comparison of numerical and empirical methods. The methods used were computational simulation by Ansys and wind tunnel test.

From the computer simulation, it was possible to obtain the approximate values for the drag coefficient of the rocket model according to a Reynolds number associated with each airflow velocity. Thus, the results were compared with other simulations using MFSim code and OpenRocket software. These values were close and the average  $C_d$  obtained with CFD was 0,63.

Finally, through the presented experimental methodology, the values of  $C_d$  for velocity  $v$  of 12 up to 24 meters per second were calculated, obtaining an average drag coefficient of 0,30 for this Reynolds range. There was a notable discrepancy of 109 % between the experimental and computational results, associated with the influence of the rocket fixing rod on flow over the body in study.

These results are preliminary, so further studies will be conducted experimentally to evaluate the correct influence of the rod and the new conclusions will be published in future papers. Thus, it is suggested to change the form of attachment, reducing the influence of the rod on the total drag generated by the system and also perform computational simulation with the presence of the rod fixed on the sidewall of the Wind Tunnel. This research opens the door for the creation of a new methodology for Equipe de Propulsão e Tecnologia Aeroespacial (EPTA) in the development of new rockets, combining the numerical and experimental implementation.

#### 5. REFERENCES

- Alves, I.C., Valim, G.B., Silva, A.R. and Neto, A.S., 2018. "Computational simulation of the flow over a rocket model". In *11th Spring School on Transition and Turbulence - EPTT 2018*. Uberlândia, Brazil.
- Barlow, J.B., Jr, W.H.R. and Pope, A., 2015. *Low speed wind tunnel testing*. INCAS Bulletin, New York, USA, 3rd edition.
- DeMar, J.S., 1995. "Model rocket drag analysis using a computerized wind tunnel". *National Association of Rocketry Research & Development Report*.
- Fox, R.W., Pritchard, P.J. and McDonald, A.T., 2011. *Introduction to Fluid Mechanics*. John Wiley & Sons, Inc., New Jersey, 8th edition.
- Gebbink, R., Wang, G. and Zhong, M., 2018. "High-speed wind tunnel test of the cae aerodynamic validation model". *Chinese Journal of Aeronautics*.
- Mani, M., Naghib Lahouti, A. and Nazarinia, M., 2004. "Experimental and numerical aerodynamic analysis of a satellite launch vehicle with strapon boosters". *The aeronautical journal*.
- Menter, F.R., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications". In *24th Fluid Dynamics Conference*. Orlando, USA.
- Newman, D.J., 2002. *Interactive Aerospaceengineering and design*. Mcgraw Hill, New York, USA, 1st edition.
- Penna, P.J., Su, J.C. and Syms, G.F., 2003. "Experimental and numerical studies of the effects of upper surface roughness on aileron performance". *SAE Technical Paper Series*.
- Pinto, W.J.G.S., 2016. "Numerical and experimental analysis of the flow over a commercial vehicle – pickup".
- Tyler, C. and Braisted, W., 2005. "Evaluation of rapid prototyping technologies for use in wind tunnel model fabrication". In *43rd AIAA Aerospace Sciences Meeting and Exhibit*. Reno, USA.
- Zhu, W., 2019. "Models for wind tunnel tests based on additive manufacturing technology". *Progress in Aerospace Sciences*.

#### 6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.