



**Students'
Space Association**
WARSAW UNIVERSITY OF TECHNOLOGY



Coordinator: Paulina Zurawka
Date: 20/04/2022
Number of pages: 15

Author(s): Jakub Jura

CFD analysis of a rocket across various flight regimes

Tangent-ogive nose cone shape



Contents

1	Introduction	1
2	Theoretical Background	2
2.1	The governing equations of Fluid Mechanics	2
2.1.1	The Green-Gauss-Ostrogradski Theorem	2
2.1.2	Conservation Laws	2
2.1.3	Conservation of mass	3
2.1.4	Conservation of momentum	4
2.1.5	Conservation of Energy	5
2.1.6	The Navier-Stokes Equations	5
2.2	Viscosity	5
2.3	Drag Coefficient	6
3	CFD	7
3.1	Geometry	7
3.2	Meshing	9
3.3	Settings	10
4	Results	12
5	Conclusions	14



1 Introduction

As the fundamental part of any object moving in fluid medium, the geometry of it's frontal part greatly affects it's performance. This is no different in case of model rockets and thus the shape of it's nose cone is a crucial part of aerodynamic optimization. This report takes a closer look at the tangent-ogive nose cone shape on a small scale sounding rocket across wide variety of flight regimes ranging from mach 0.1 up to mach 3.

The analysis is going to be performed using CFD (computational fluid dynamics) which is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows.

The desired property which is going to be obtained as a result of the simulation is drag coefficient which is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, such as air or water. It is used in the drag equation in which a lower drag coefficient indicates the object will have less aerodynamic or hydrodynamic drag.

Many resources were found helpful during the research process, including books[1], reports[2], youtube tutorials as well as numerous lectures and conversations with Paulina Żurawka and Nezar Sahbon whose great leadership skills and vast experience with CFD were irreplaceable and proved crucial in conducting this research.

The results will offer a great insight into the performance of set geometry at different Mach numbers, values of C_d are going to be presented on $C_d(M)$ plot.



2 Theoretical Background

Fluid mechanics is the science of how fluids behave and interact, it's understanding and application allows for prediction of the movement and thus resultant effect on any given objects present within the fluid.

2.1 The governing equations of Fluid Mechanics

Navier-Stokes equations describe motion of viscous fluids allowing for simulation of air-flow around the GROT rocket, this is done sequentially by calculating pressure, temperature, and density in each cell based on information from previous cells of the mesh.

These are the base equations and theorems used to compute behaviour of fluids:

2.1.1 The Green-Gauss-Ostrogradski Theorem

Green-Gauss Theorem relates the volume surface integrals. Given that \mathbf{F} is a vector field and $\delta\Omega$ a smooth surface, the equation is as follows:

$$\int_{\delta\Omega} \mathbf{F} \cdot d\mathbf{S} = \int_{\Omega} \nabla \cdot \mathbf{F} dV \quad (1)$$

2.1.2 Conservation Laws

Fluid mechanics are based on following conservation laws:

- conservation of mass
- conservation of momentum
- conservation of energy

The quantity H is required for their computation, along it's mass specific density h .

$$H(t) = \int_{\Omega} \rho h dV \quad (2)$$



And it's derivative.

$$\frac{d}{dt}H(t) = \frac{d}{dt} \int_{\Omega} \rho h dV = \frac{dH_p}{dt} + \frac{dH_f}{dt} \quad (3)$$

Where $\frac{dH_p}{dt}$ is the rate of change of this quantity in the fluid, and $\frac{dH_f}{dt}$ is it's flux going through the edge $\delta\Omega$ of control volume Ω .

While $\frac{dH_f}{dt}$ can be written as:

$$\frac{dH_f}{dt} = - \int_{\delta\Omega} \rho h \mathbf{v} \mathbf{n} dS \quad (4)$$

2.1.3 Conservation of mass

With prior knowledge, conservation of mass may be derived in following way:

Given that H represents mass:

$$H(t) = M(t)$$

The mass-specific quantity h equals:

$$h = 1$$

Thus it can be written as:

$$M(t) = \int_{\Omega} \rho dV \quad (5)$$

With the derivative of mass:

$$\frac{d}{dt}M(t) = \frac{dM_p}{dt} + \frac{dM_f}{dt} \quad (6)$$

The term $\frac{dM_p}{dt}$ is equal to **0** as the amount of mass cannot change in the control volume. Thus it's written as:

$$\begin{aligned} \frac{d}{dt}M(t) &= \frac{dM_f}{dt} \\ \frac{d}{dt} \int_{\Omega} \rho dV &= - \int_{\delta\Omega} \rho \mathbf{v} \mathbf{n} dS \end{aligned} \quad (7)$$



As the result we obtained two integrals, thus with Green-Gauss-Ostrogradski Theorem, the surface integral may be substituted into a volume integral:

$$\int_{\delta\Omega} \rho \mathbf{v} \mathbf{n} dS = \int_{\Omega} \nabla(\rho \mathbf{v}) dV \quad (8)$$

After substitution into equation (7) and inclusion of $\frac{d}{dt}$ into the integral on the left side of the equation, the following equation remains:

$$\int_{\Omega} \left(\frac{\delta}{\delta t} \rho + \nabla(\rho \mathbf{v}) \right) dV = 0 \quad (9)$$

The volume Ω is chosen, for equation (9) to equal 0 at every part of the domain, thus the inside if this integral must be equal to 0:

$$\frac{\delta}{\delta t} \rho + \nabla(\rho \mathbf{v}) = 0 \quad (10)$$

In the case of constant flow no variable is dependant on time, thus (10) reduces to:

$$\nabla(\rho \mathbf{v}) = 0 \quad (11)$$

Which becomes:

$$\rho \nabla \mathbf{v} + \mathbf{v} \nabla \rho = 0 \quad (12)$$

2.1.4 Conservation of momentum

The conservation of momentum is derived in similar way, and it yields this equation

$$\frac{\delta}{\delta t} \rho \mathbf{v} + \nabla((\rho \mathbf{v} \otimes \mathbf{v}) - \Xi) = \rho \mathbf{f} \quad (13)$$

And for inviscous fluids it can be written in Euler's version:

$$\rho \left[\frac{\delta}{\delta t} \rho \mathbf{v} + (\mathbf{v} \cdot \nabla) \mathbf{v} \right] = -\nabla p + \rho \mathbf{f} \quad (14)$$

On the other hand the Euler's ignores the presence of sheer stresses which do occur in viscous fluids.



2.1.5 Conservation of Energy

The law of conservation of energy states that the total energy of an isolated system remains constant, it is one of fundamental laws of thermodynamics. It is represented with following equation:

$$\rho \frac{D}{Dt} e = \text{Div}(\Xi) \cdot \mathbf{v} + \Xi : \mathbf{D} + \rho \mathbf{f} \cdot \mathbf{v} + \rho \gamma_h + \nabla \cdot (\lambda \nabla T) \quad (15)$$

Given that $\Xi : \mathbf{D}$ is the tensor scalar product, and the term $\frac{D}{Dt}$ is the substantial derivative of the mass-specific energy e .

2.1.6 The Navier-Stokes Equations

With those fundamentals it's possible to arrive at the semi-empirical Navier-Stokes equations, represented with following formula:

$$\rho \left[\frac{\delta \mathbf{v}}{\delta t} + (\mathbf{v} \cdot \nabla) \mathbf{v} \right] = -\nabla p + \nabla \tau + \rho \mathbf{f} \quad (16)$$

In order to solve NSE, it is necessary to add boundary conditions.

2.2 Viscosity

Another crucial part of fluid mechanics is the Reynolds number, it's value describes whether the flow is dominantly turbulent or laminar, and equals:

$$Re = \frac{vL}{\nu} \quad (17)$$

Given that ν is the kinetic viscosity, v is magnitude of velocity and L is characteristic length.

Viscosity is the amount of internal resistances in the fluid, the dynamic viscosity is described by following equation:

$$\mu = \nu \rho \quad (18)$$

And thus the Reynolds number is:

$$Re = \frac{\rho v L}{\mu} \quad (19)$$



Reynolds number describes the ratio of inertial to viscous forces within the fluid, for low Reynolds numbers the flow is laminar, whereas at high Reynolds numbers it becomes turbulent.

$$Re = \frac{\text{inertial forces}}{\text{viscous forces}} \quad (20)$$

Another important part of fluid mechanics is the deformation tensor described as follows:

$$\int_{\delta\Omega} \mathbf{D} \mathbf{n} dS \quad (21)$$

Given that \mathbf{D} is the tensor of deformation in the normal direction to the surface of a fluid.

Another very important term to remember is the rigid rotation tensor \mathbf{R} . It defines the vorticity of the fluid, which is also crucial when predicting a fluids behaviour.

When a flow is turbulent, vortices are formed, they are also called **eddy's**.

With these fudamentals we have all necessary basics for the simulation of fluid flow.

2.3 Drag Coefficient

Drag coefficient is composed of a few parts, including:

- **form drag** which is generated by the cross-sectional area of the rocket's geometry.
- **skin friction** which is dependent on the roughness of the rocket's surface
- **wave drag** which appears at transonic velocities and higher, when shock waves are formed on the body of the rocket

The sum of all the parts is C_d which is being measured in this investigation.



3 CFD

Computational Fluid dynamics is based on a few discretization methods, of which Finite Volume Method was used.

The main rules of CFD is to integrate NSE over the entire volume by calculating values in each cell of the mesh based on the information from surrounding cells.

For this purpose Ansys Fluent[®] was used.

For this to be possible it's necessary to perform several steps:

3.1 Geometry

The geometry of the nose cone was described with following equation for tangent-ogive nose cone shape:

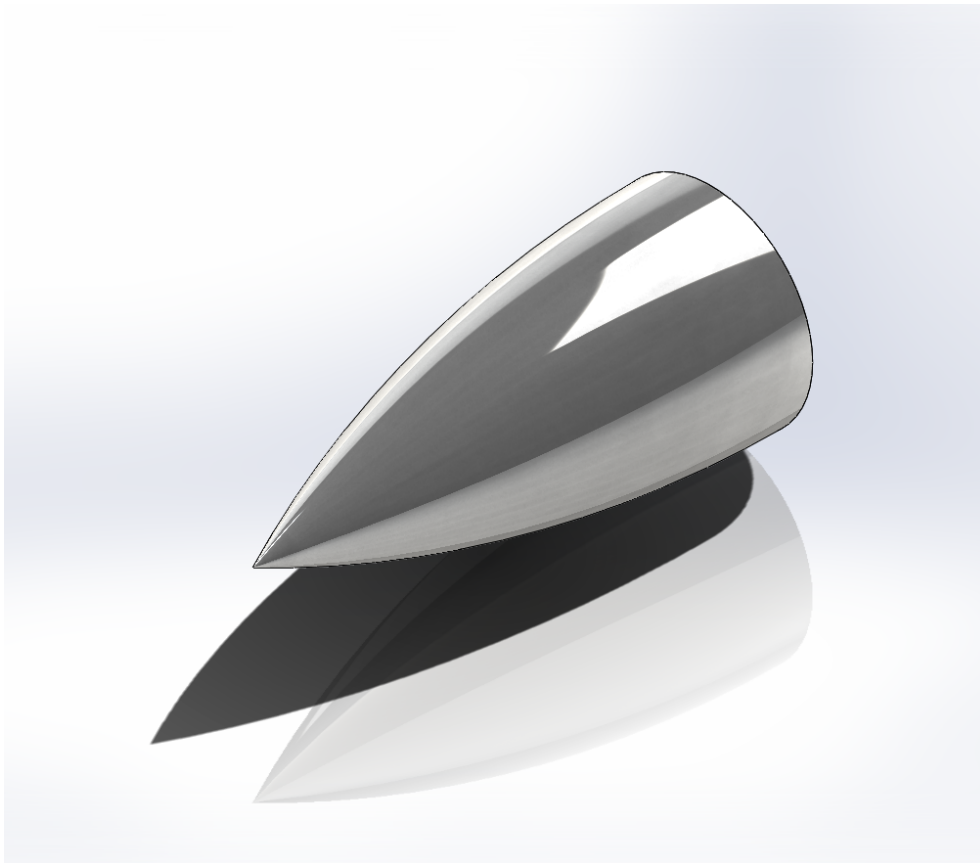


Figure 1: Tangent-ogive nose cone



$$\rho = \frac{R^2 + L^2}{2R}$$

The radius y at any point x , as x varies from 0 to L is:

$$y = \sqrt{\rho^2 - (L - x)^2} + R - \rho$$

Figure 2: Tangent-ogive nose cone equation

Additionally two separate cases were analyzed, one with slanted base of the rocket, and the other with flat base, both had the nose cone created by revolution of parametric line defined by the equation mentioned prior.

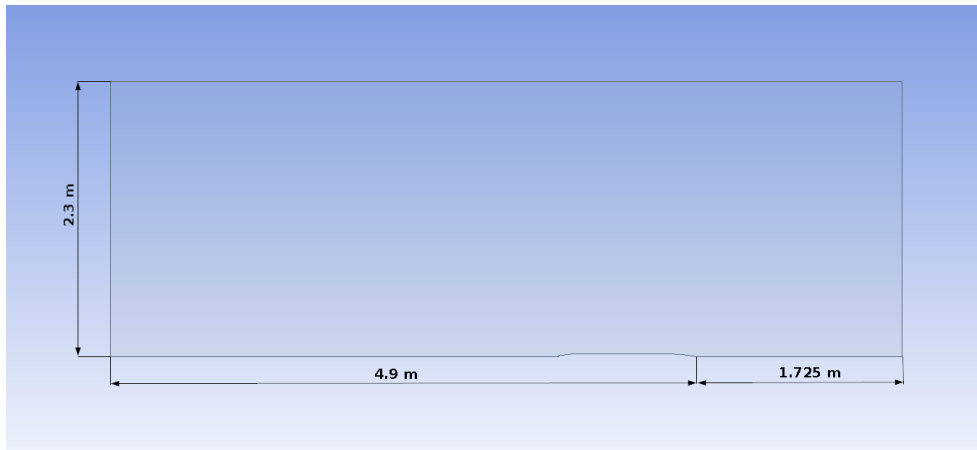


Figure 3: Domain geometry

Also the domain independency study was performed which validated domain size with following geometry.

3.2 Meshing

The mesh is a key factor in CFD calculations. It is a discrete model, which allows the code to perform proper calculations. The mesh, in other words is the result of dividing the control volume into finite elements. It could be explained that data is stored in the center of each cell. Between these points, the CFD code assigns a linear function connecting them. The process is then repeated with every cell in the mesh. It is in this way, that the software generates a result.



The entire fluid domain was divided into a mesh consisting of nearly 50000 elements to achieve desired accuracy.

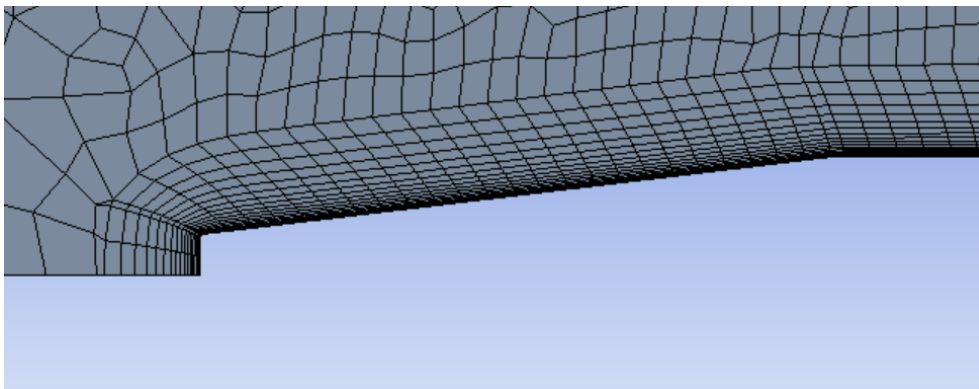


Figure 4: Mesh zoomed in to the back of the rocket

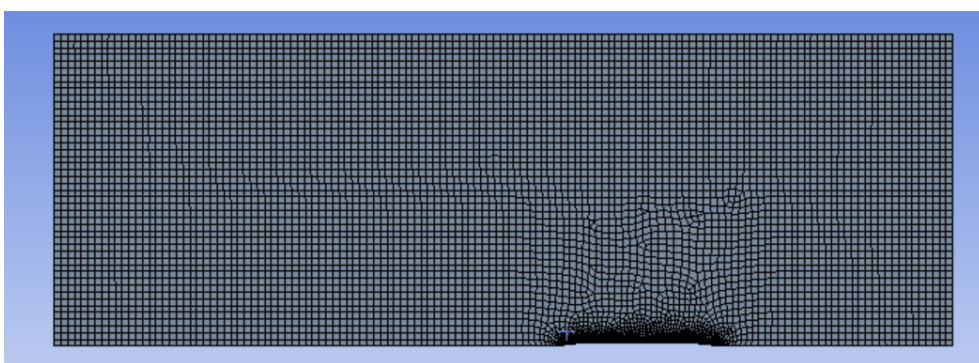


Figure 5: Entire mesh

3.3 Settings

In the Ansys Fluent[®] software following settings were used:

Simulation was set to density-based solver allowing us to find solutions for compressible flow. The turbulence model was set to SST k- ω , which is able to determine stress tensor components necessary for calculating diffusion. Energy equation was enabled as well.



Boundary conditions were set such that operating pressure was set to 0 Pa while gauge pressure to 101325 Pa, and temperature equaled 288 K. For turbulence specification method, intensity and length scale was chosen. Pressure far-field was set at turbulent intensity equals 0.5% and turbulent length scale equals 0.001 m . Respectively for pressure outlet it was set at 5% and 0.5 m. During analysis the Mach number equaled values from 0.1 to 3.

For calculations initially first order upwind scheme was used and then additional calculations were performed with second order upwind scheme in order to achieve better stability and precision. Solution steering was enabled for controlling flow type, which was useful as analysis covers wide range of Mach numbers.

Scaled residuals were also added as a way to keep track of calculation errors.

A residual is the local error of the outcome in each cell of the mesh. It creates a matrix including all the errors created while iterating.

Divergence threshold was set to residuals equalling $1e-5$.



4 Results

Results are presented as the graph of $C_D(M)$ for both slanted and flat bottom cases.

Initially a following table of resultant values of C_d was generated.

Velocity Mach Number	Drag Coefficient Straight base	Drag Coefficient Slanted base
0.1	0,487	0,353
0.3	0,438	0,284
0.6	0,425	0,253
0.8	0,431	0,240
1.0	0,591	0,334
1.2	0,564	0,334
1.6	0,487	0,299
2.0	0,407	0,267
2.5	0,330	0,234
3.0	0,275	0,207

Table 1: Comparison of the Drag Coefficients and the Mach Number

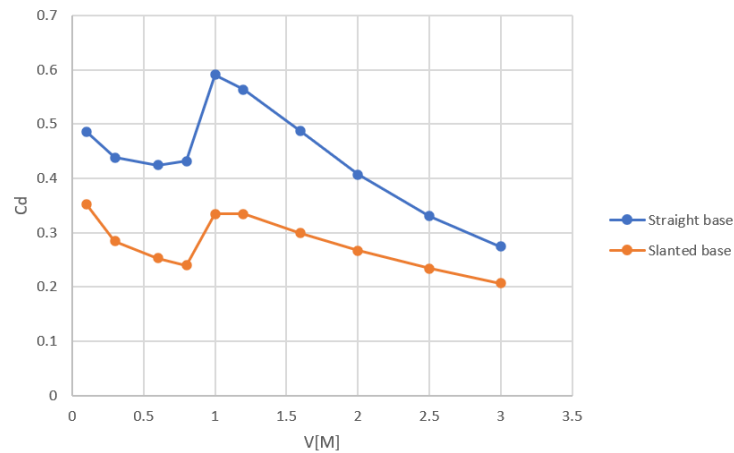


Figure 6: Drag coefficient to Mach Number plot

Additionally shockwaves are visible in the contour plot:



Figure 7: Shockwave on dynamic pressure contour plot



5 Conclusions

As visible on the graphs the coefficient of drag is much higher for geometry with flat base, which is caused by the pocket of low pressure forming behind the rocket and increasing pressure drag, additionally there is significant spike in coefficient of drag at mach 1 due to crossing the sound barrier and formation of shockwaves. Final assessment of the performance of this nose cone shape will be possible with comparison between other geometries from other members of this project.



References

- [1] John David Anderson and J Wendt. *Computational fluid dynamics*, volume 206. Springer, 1995.
- [2] Lucas de Almeida Sabino Carvalho and Geovanio Claudino Cavalcante Filho. Cfd analysis of drag force for different nose cone design. In *Conference proceedings, October*, 2019.