Research Report

Supersonic Speed Rockets Research Report

19.08.2021

Translation Date: 20.08.2021

Halit Yusuf Genç

İ.T.Ü. PARS Rocket Group, Aerodynamics Department

Terminology

1) Introduction

In this research report, we will talk about the methods followed in the aerodynamic design processes of rockets that exhibit supersonic flight and the appropriate designs obtained according to these researches.

The detailed source of this research, which concerns the aerodynamics section, has been added to the references, and this bibliography contains information and analyzes that will be very useful for both the mechanical and electrical-electronic departments. I highly recommend anyone interested in these topics to at least take a look.

The report we examined as a reference is the research report of a team that will participate in the IREC competition from the "Two-Stage Supersonic Rocket" branch by The University of Akron.

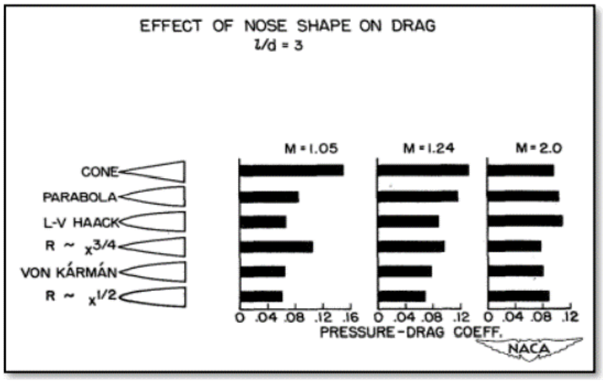
Link:

https://ideaexchange.uakron.edu/cgi/viewcontent.cgi?article=2100&context=honors_research_projects

According to the Report, the General Aerodynamic Conditions Required for the Healthy Flight of a Two-stage Supersonic Rocket are as follows:

- An equilibrium ballast system should be used in the design of the launch vehicle.
- The wing flutter factor of safety must be at least 1,3 prior to manufacture.
- The fin attachment system will allow fin designs to be changed at will.
- The margin of stability of the launch vehicle and each stage shall be no more than 3,50.
- The margin of stability of the launch vehicle and each stage shall be at least 1,75.
- Upper stage vanes will not physically interfere with the separation system.

2) Nose Cone



- The X^1/2 shape is a potentially lower drag option at supersonic speeds, according to this chart published by The National Advisory Committee for Aeronautics (NACA). However, our reference team did not prefer this nose cone due to the difficulty of the production stage and the fact that it is not possible to purchase this nose cone ready-made within the current possibilities.

- The thinness ratio, commonly referred to as the nose cone's aspect ratio, is important to reduce the "wave drag coefficient" value, which is the drag experienced when the rocket reaches the critical Mach number. Tests by NACA have indicated that the ratio of fineness, which is the ratio of the length of the nose cone to its diameter, is ideal for supersonic speeds at a ratio of about 5:1. In addition, since the 5:1 ratio nose cone is lighter than the 5.5:1 ratio nose cone, it has been determined as the most suitable option.

- In addition to these, while the NACA studies indicate that Haack Series profiles are the most efficient for speeds at 1.2 Mach, it should be underlined that the entire flight process is not at this speed. Therefore, teams need to make calculations using applications such as OpenRocket and mathematical formulas. According to the research results of our reference team, a Von Karman type nose cone; It will be the most suitable nose cone choice in terms of drift, altitude, cost and ease of production.

- Fiberglass or carbon fiber can be preferred as production material. Although carbon fiber is lighter as a material, it also brings with it the problem of preventing the communication of the devices inside the rocket. Carbon fiber is a good choice if you don't have that kind of mission statement.

Nose Cone Design	Altitude (ft)	Max Drag (lbf)	Max Velocity (ft/s)
Conical	26,829	192.1	1,233
Ogive	28,251	180.3	1,270
Ellipsoid	27,856	174	1,263
Power Series	28,855	169.1	1,288
Parabolic Series	28,516	174.2	1,279
Haack Series (Von Karman)	28,869	169.3	1,289

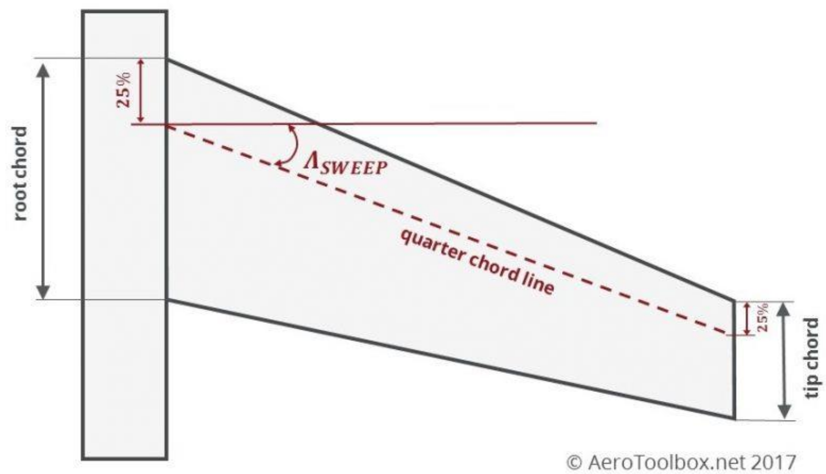
Nose Cone Tip:

- In supersonic flights, teams assume an isentropic flow at the tip of the rocket. It has been deemed appropriate that the tip of the nose cone is made of a more heat-resistant material in order to avoid a situation such as burning the nose cone with the effect of high speed. This material is also determined as aluminum. In the tests, it has been observed that our material will not pose a problem even at 1.2 Mach speed.

3) Fins:

- More fins means more weight and more drag. For these reasons, our reference team has decided to use three fins for each blade set. The team also designed both sets of fins to be kept radially aligned to reduce drag and promote smoother airflow over the body.

- As the "sweep angle" increases in the wing design, the resistance of the vehicle decreases, however, our stability value also decreases. Therefore, a suitable sweep angle must be found to ensure that an adequate margin of stability is maintained while reducing friction as much as possible.



- In researches for this scan angle, OpenRocket and RockSim simulation softwares showed a decrease in the stability margin of the overall rocket design at sweep angles above 70 degrees. Therefore, the team planned to use a leading sweep angle as perpendicular as possible, but below 70 degrees for both stages. Using this, the team decided that they needed to increase the sweep angle as much as possible while staying within the stability requirements set for competition and flight.

- The fact that this scanning angle is kept as large as possible confirms this research in a way when planes at high speeds are examined.

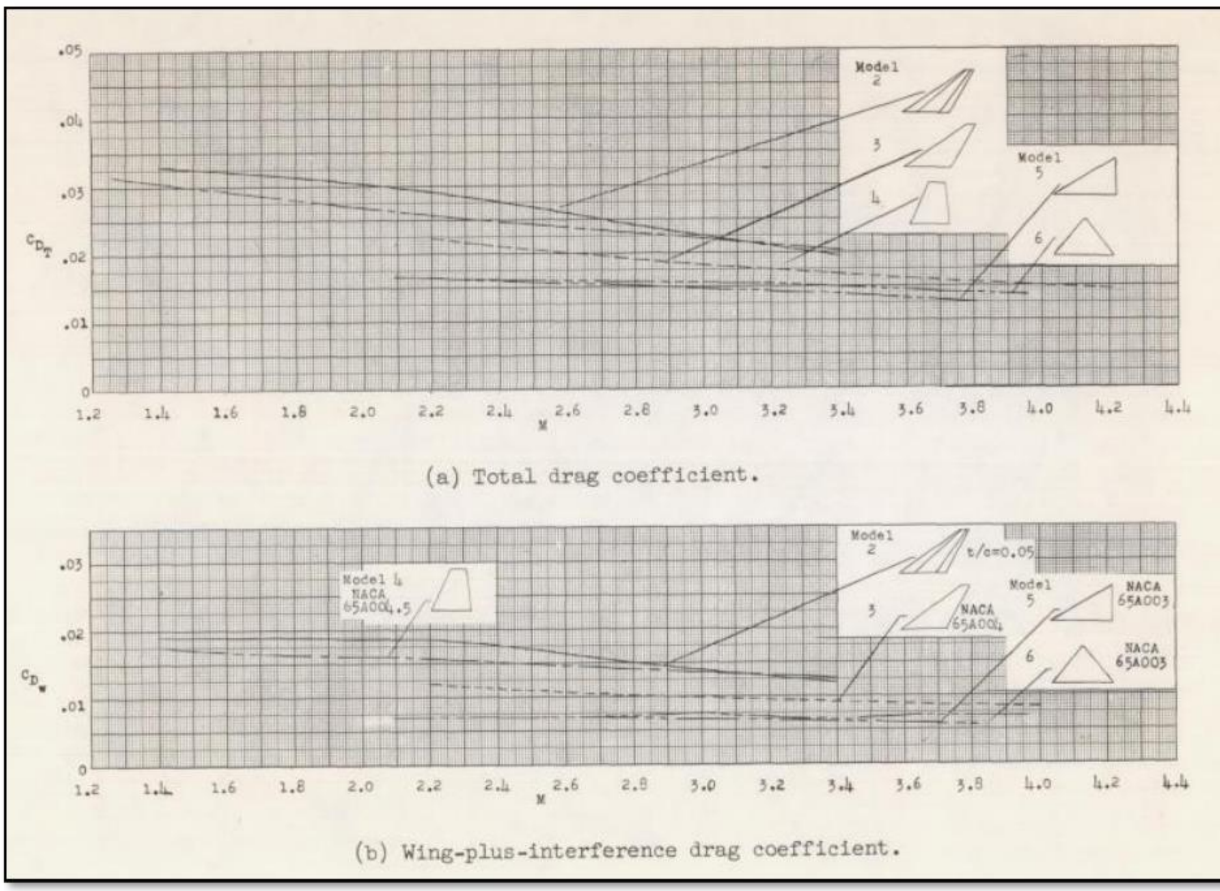


Figure 12 - NACA fin testing results for supersonic speeds

- Based on research from our reference team and flight simulations in OpenRocket, trailing edge scanning was predicted to have no significant effect on altitude or drag, but would affect stability. The team, which is between the following two shapes as suitable fins, turned to the clipped delta wing design for the design to be used in the second stage for ease of production and stability. In addition, the clipped delta design is a very safe choice in case the fin crashes to the ground and breaks or warps during the recovery process of the rocket. Although this design will reduce drag considerably if the cross section of the fin is airfoil, its production is close to impossible.

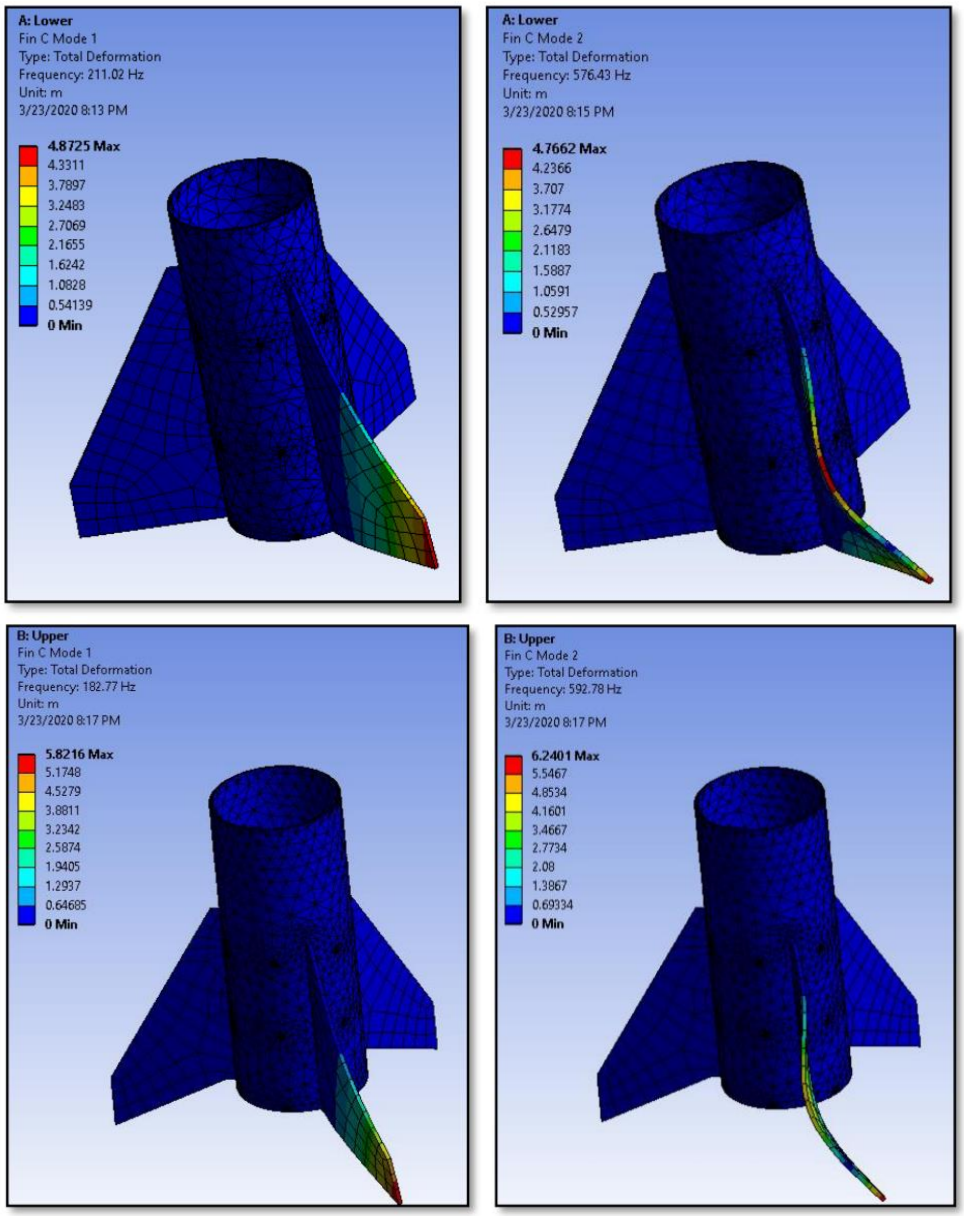
- The local speed of the rockets, when it reaches the speed of sound, will cause a significant amount of drag. This is due to the formation of a near-normal shock wave. Thus, the higher the value of the critical Mach number for the rocket, or the faster the vehicle can travel before its local flow reaches Mach, the less friction the rocket will face.

- To determine this, the team built two flight profiles in ANSYS Workbench and Fluent and found the point where the local stream reaches the speed of sound. The two fin types above were also subjected to this analysis.

- However, because the visualization on ANSYS CFD-Fluent did not give an obvious choice as to which design was more optimal, the team had to compare the rocket's surface velocities with the entry velocities and investigate the incoming numerical results. And as a result of these researches, it was seen that the critical Mach numbers of both designs were quite close to each other. Therefore, which fin was preferred was not very important in terms of drag. This made it clear that the cropped delta selection is the best.

Wing Flutter:

- Wing flutter is the aerodynamic instability of wings due to geometry, material and fluid properties. It can be incorporated into the rocket by designing smaller fins and stronger materials for less rocket stability and additional weight. These fins needs to be subjected to natural frequency testing via ANSYS.



- Based on the results made by our reference team, the resulting natural frequencies are well above and more than twice the maximum range of a sonic boom imperative frequency. Therefore, sonic boom does not cause these structures to resonate due to frequency. This further increased the team's confidence in their wing designs.

Wing Flutter Formula:

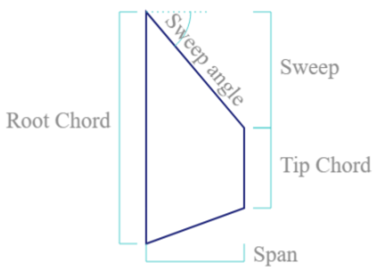
$$V_f = \alpha \sqrt{\frac{G}{\frac{1.337(AR)^3(P)(\lambda + 1)}{2(AR + 2)\left(\frac{t}{C_r}\right)^3}}} \quad ft/s$$

Individual calculations of values are available below. First, the calculations should be made and then these values should be put into the above formula. **Units in the formula ARE ACCORDING TO AMERICAN STANDARDS! (inch, feet, lb, psi, Fahrenheit)**

Cr ($Root\ Chord$fin) : Length of the edge of the fin that contacts the fuselage (in.)

Ct ($Tip\ Chord$fin) : Length of outermost edge of(in.)

A single fin:



t ($Thickness$) : Fin thickness (in.)

b ($Semi - Span$) : A fin span (in.)

G ($Shear\ Modulus$ of Aluminum) : Aluminum shear modulus (psi)

h ($Height\ of\ Maximum\ Velocity$) : Height at where rocket reached to the maximum speed (ft.)

$$S = \frac{(C_r + C_t)}{2} b \quad in^2 \qquad AR = \frac{b^2}{S} \qquad \lambda = \frac{C_t}{C_r}$$

$$T = 59 - 0.00356(h) \quad °F \qquad \alpha = \sqrt{1.4(1{,}716.59)(T + 459.7)} \quad ft/s$$

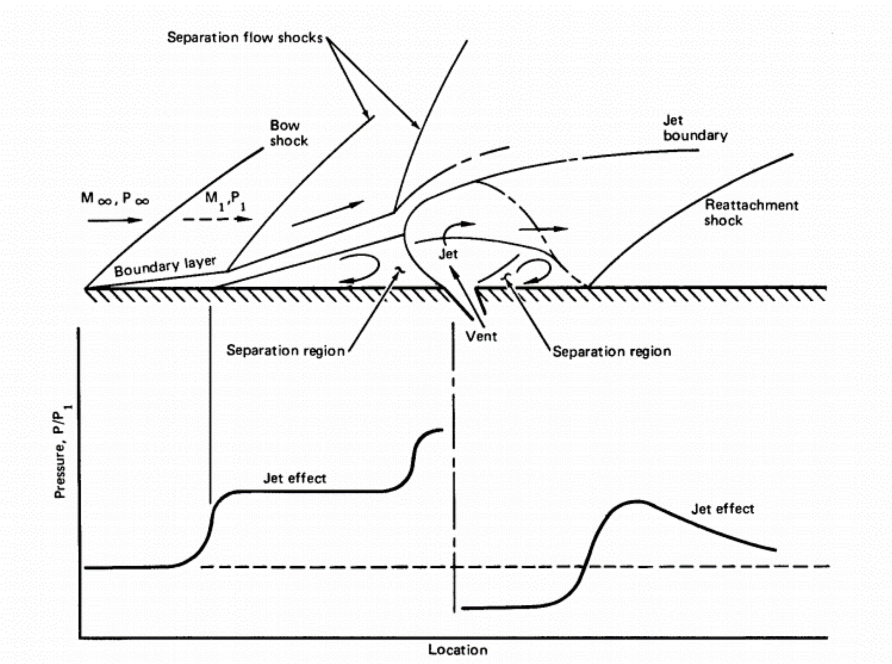
$$P = \frac{2116}{144}\left(\frac{T + 459.7}{518.6}\right)^{5.256} \quad \frac{lb}{in^2}$$

4) Vent Holes

It should be noted that when a rocket performs flights at supersonic speeds, the rocket body will be subjected to serious pressure. In addition, there is an extra load to be placed on the rocket due to the decrease in pressure at high altitudes. These pressures must be equalized in the interior and exterior of the rocket so that the rocket is not damaged by the internal and external pressure differences, and if you are using a rocket carrying a payload, the nose cone of the rocket does not open prematurely. This can be accomplished by drilling air vents into the designs of the rockets.

The dimensions of the holes should be calculated analytically according to the rocket design and flight speed. If our hole is too small for the current conditions, the balancing between the internal and external pressure cannot be achieved and as a result of the pressure discharge reaching critical levels (Approximately 1 Mach), the situation we call flow drowning will occur. While this doesn't seem like a problem from an isentropic point of view, it will actually cause some shock waves. Therefore, drowned flows are undesirable in engineering calculations and such situations should be avoided by making necessary calculations.

Another important consideration is the location of the holes. This mentioned location is in a different place in each rocket design. However, the common point in all of them is that the pressure on the surface where this hole will be opened should be minimum. If the flow in the body of your rocket is not linear, then the rocket should be subjected to air tunnel testing or detailed CFD (Computational Fluid Dynamics) analysis. With this test, positions that do not have high pressure and are free from shock should be determined and the ventilation holes of the rocket should be opened to these positions. In an old study published by NASA in 1970, he suggested that the difference between the static pressure on the pavement surface of the payload at the location where the vent is to be opened and the ambient pressure at the same location should be close to zero.

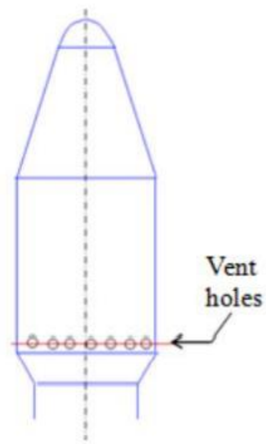


Effect of Ventilation Holes on Surface Flow

We can explain this situation with a superficial example:

• For example, the external pressure in the nose cone of a flying rocket is greater than the internal pressure, and if we drilled air vents on the nose cone, then a one-way entry into the rocket starts and serious disruptions occur in our flow. However, as we said, this is a very superficial example. In the main part of the work, much more different and complex problems will arise. Because it is difficult to predict precisely the behavior of the air vents at the speed at which the rocket will be found. The flow in the ventilation holes is quite complex. Although approximate estimations can be made theoretically on a few assumptions, determining the paths and flow characteristics of the flow without experimental data will cause us to obtain erroneous results. In addition to these, it should be noted that the ambient temperature and the velocity change over time can change the behavior of these holes.

R. C. Mehta



In the same article that NASA published in 1970, he stated that they primarily conducted scaled experiments in air tunnel experiments, but these scaled analyzes did not match those of one-to-one dimensions. Because in this scaling process, there are many factors that can be forgotten to be scaled, and there are some interactions that may occur in real calculations but remain invisible because they are scaled. Just reducing the size of the rocket and changing its speed to the value corresponding to that measurement will give you erroneous results. Due to the fact that the distances between the holes are shorter in small scales, ignoring many situations such as the interaction of the gases coming out of the holes, the change of the boundary layer thickness, the change of pressure ratios, the change of the mass flow rate will cause you to get erroneous results in these scaled analyzes. In addition to the above, trying to scale the rocket will require serious attention and complicated calculations, as the analysis will also depend on time.

The analysis of such vents is performed by solving the first-order nonlinear differential equation using a fourth-order Runge-Kutta method, and the solutions are supported by the experimental results obtained. And in this way, the correct hole design is obtained.

Discharge Coefficient and Hole Dimensions

Coefficient of discharge (Cd), another important value for air vents, is the ratio of actual discharge to theoretical discharge at a nozzle or other constriction. That is, it is the ratio of the mass flow rate at the discharge end of the nozzle to that of an ideal nozzle expanding an identical working fluid.

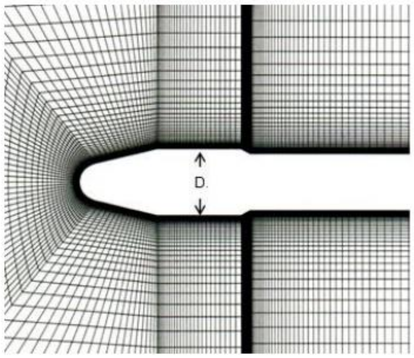
Knowing the cross-sectional area and nozzle area of the upstream along with the pressure difference, it is possible to calculate a theoretical flow rate through a pipe using conservation of mass and momentum. However, these theoretical flow rates do not take into account the momentum losses in the pipe, mostly located in the nozzle region, due to factors such as friction and turbulence. This is exactly where the value we call cd comes into play. The discharge coefficient can be thought of as a correction factor for real flowmeter devices. Correlation equations can also be used to calculate the actual corrected flow rate over a given counter. The Cd value is also a function of the Reynolds number or the ratio of the inlet and outlet cross-sectional areas of the respective flow. It is also possible to calculate the flow rate using these discharge coefficient values together with the nozzle geometry and the measured pressure difference.

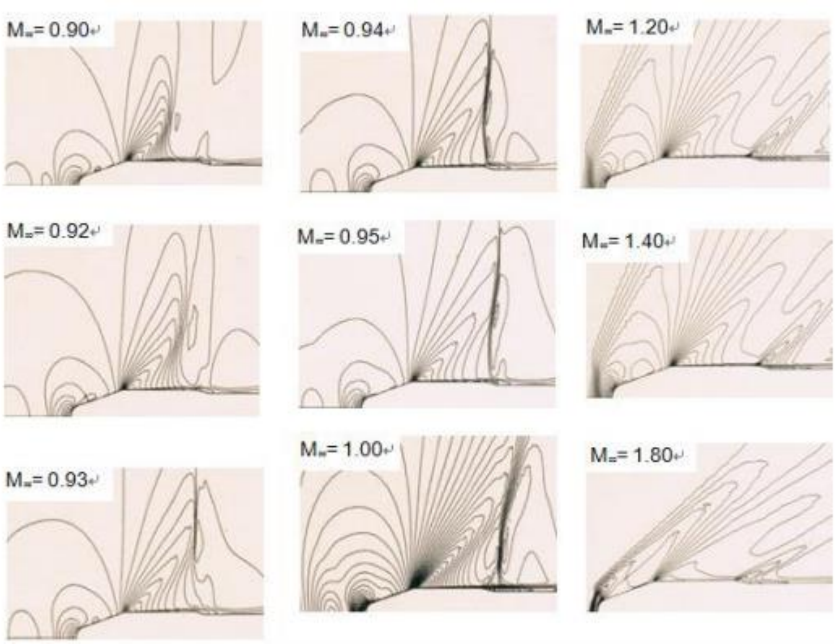
So to summarize briefly; Since there is no flow meter to measure the air entering and leaving the holes in our rocket, we can easily make these calculations using the discharge coefficient. These values will show us how much the flow rates, whether volumetric or mass, are in and out of the hole, according to the Mach velocities passed during the flight time. Or, to make a more interesting use, rockets with different sized vents but of the same design can be simultaneously subjected to CFD analysis and we can also obtain the optimal hole size depending on the discharge coefficient. The greater the discharge coefficient, the less likely the loss due to factors such as friction and turbulence will be. This method is very useful for determining the ideal hole sizes.

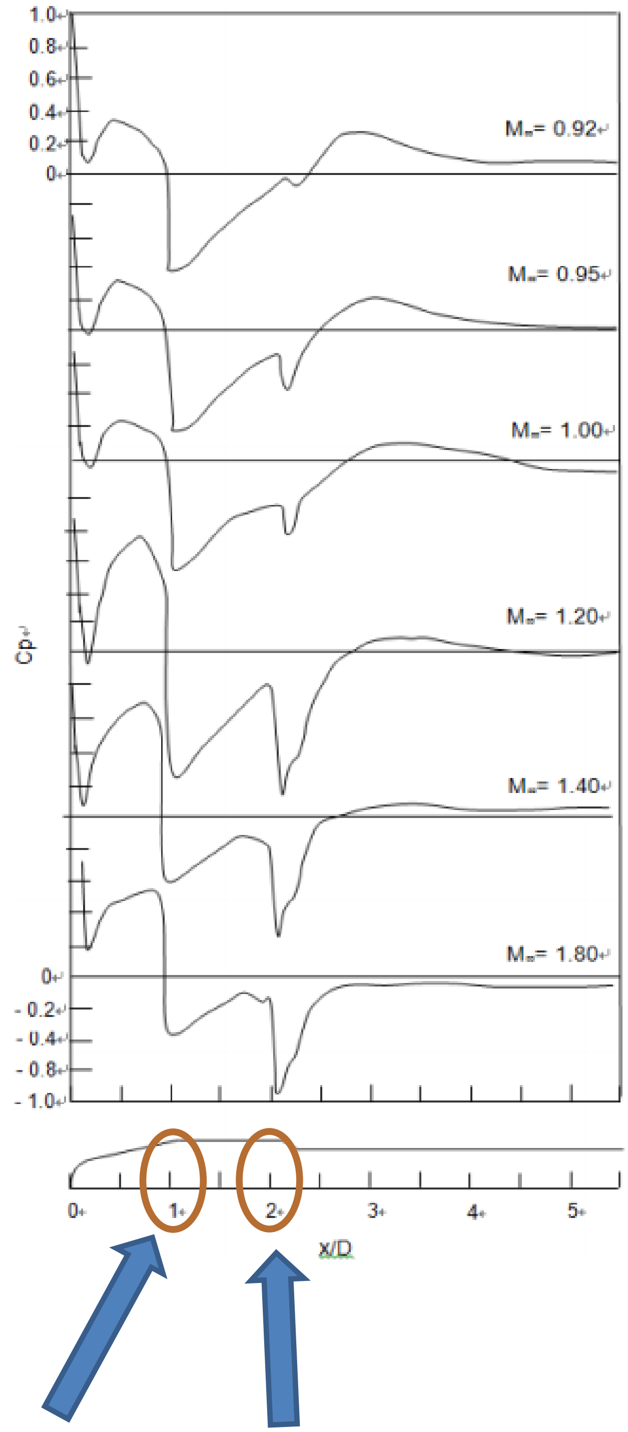
Pressure Coefficient and Hole Positioning

Along with the hole size, the position of the holes can also be determined by CFD analyses. If we include the unperforated version of the current rocket design into the external flow analysis, the regions with the lowest pressure coefficient values on the rocket body are our ideal hole regions, as stated by NASA. The holes should be opened symmetrically with equal angles to the ideal areas to be selected.

Below is an example of analysis made by Noorul Islamic University. As can be seen, the position where the hole will be drilled also partially changes according to the changing Mach numbers. The regions where the pressure coefficient (Coefficient of pressure - Cp) value is minimum are ideal regions for us. We have marked these regions on the rocket in the graph on the right for easier understanding:







Ideal Locations for Vent Holes

5) References

Links:

https://ideaexchange.uakron.edu/cgi/viewcontent.cgi?article=2100&context=honors_research_projects

https://en.wikipedia.org/wiki/Supersonic_aircraft

https://www.apogeerockets.com/education/downloads /Newsletter68.pdf

http://everyspec.com/NASA/NASA-SP-PUBS/download.php?spec=NASA_SP-8060.00000228.pdf

http://www.techno-press.org/fulltext/j_aas/aas4_4/aas0404005.pdf

https://www.google.com/url?sa=t&rct=j&q=&esrc=s&source=web&cd=&ved=2ahUKEwjxh-fQ9rjyAhUbhv0HHWoJC-0QFnoECCIQAQ&url=http%3A%2F%2Fwww.qats.com%2FQpedia-Article%2FQpedia_Feb08_Discharge_coefficient_in_flow_calculation_and_measurement&usg=AOvVaw3ak9-VsxZZQGP2f_t9IBbr

Department: Aerodynamic
Research, Analysis, Report: Halit Yusuf Genç
Direction: Umut Engin, Muhammet İkbal