



Research Report

Ventilation Holes for Supersonic Rockets Research Report

09.10.2021

(Translated at 16.10.2021)

Halit Yusuf Genç

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

Terminology:

I. Introduction.....	3
II. Ventilation Holes.....	4
III. Sample Analysis.....	9
IV. References.....	27

1) Introduction

In this research report, we shared with you a detailed and exemplary research process that examines why rockets that exhibit supersonic flight need air vents, what these holes are for, where and how they should open.

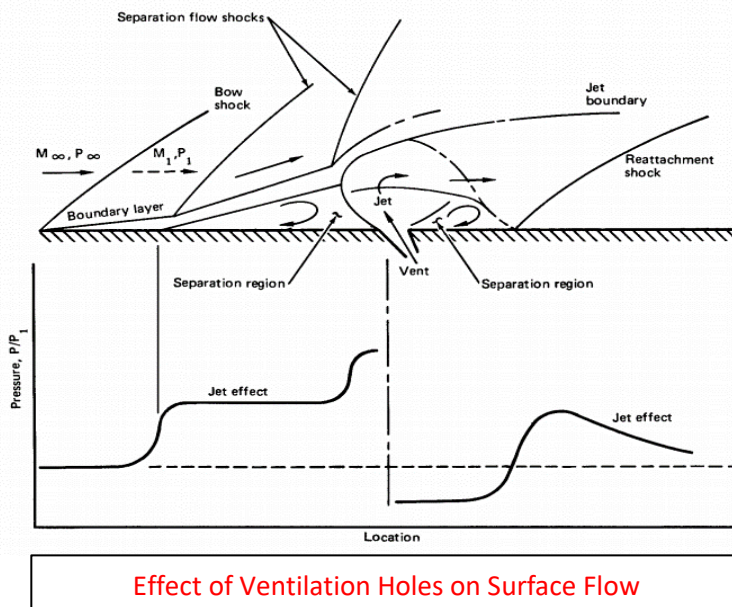
During a long research process of about 2 months, we wanted to present the report as clear and clear as possible by dealing with stages such as reading detailed articles, ANSYS analysis, Openrocket and SOLIDWORKS designs.

2) Ventilation Holes

It should be noted that when a rocket flies at supersonic speeds, the rocket body will be subjected to several pressures. However, as a result of the decrease in pressure at high altitudes and the acceleration of the air flowing around the accelerating rocket, the external pressure on the lateral surfaces of the rocket will decrease over time. 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), a situation called flow choke 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.

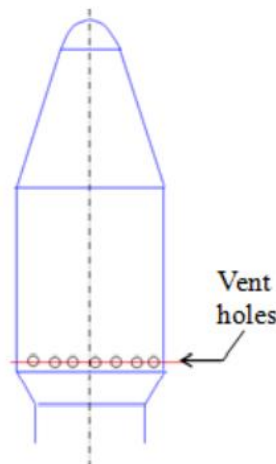
Another important consideration is the location of the holes. This mentioned location is in a different place in each rocket design. However, the common point in all of them is that the pressure on the surface where this hole will be opened should be minimum. If the flow in the body of your rocket is 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, they 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.



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, they also 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 scale, 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 incorrect 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 (C_d), another important value for air vents, is the ratio of actual discharge to theoretical discharge in 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.

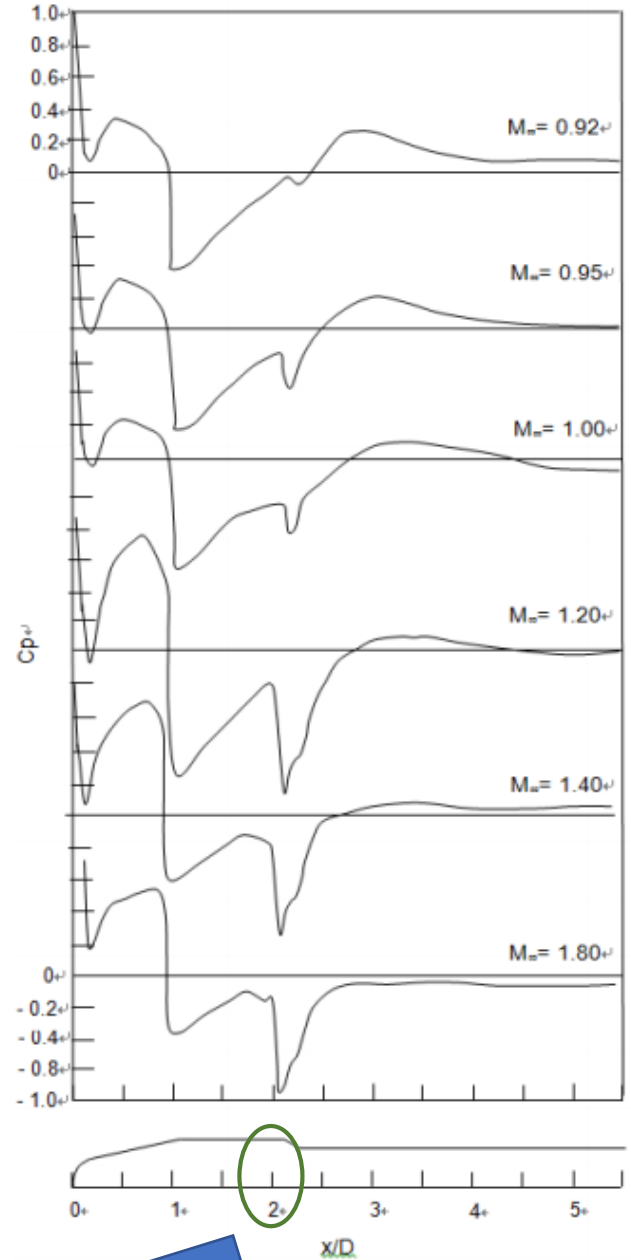
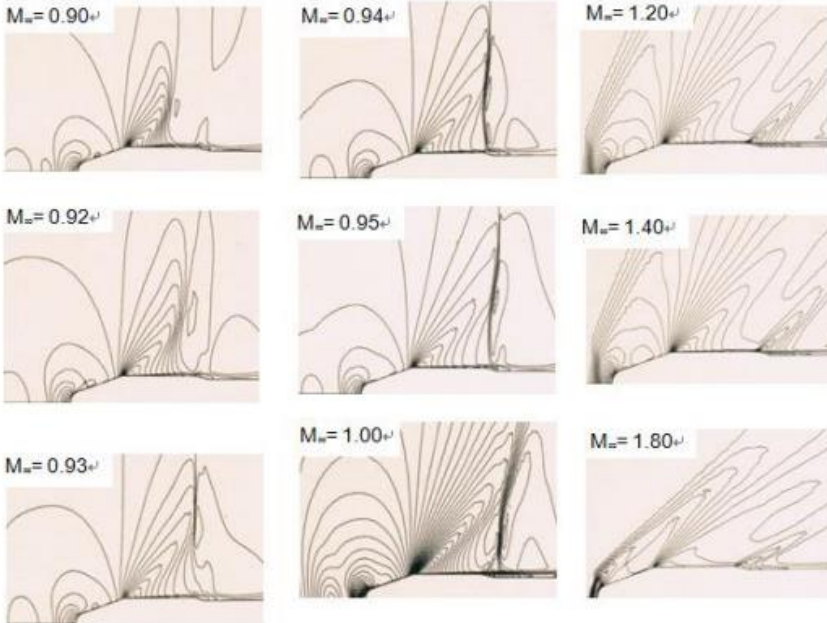
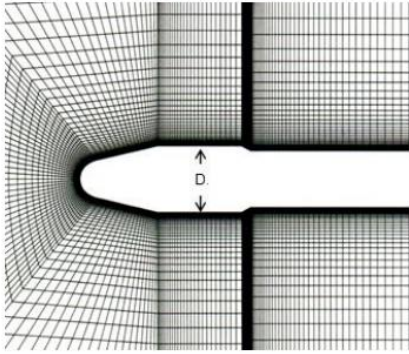
Knowing the cross-sectional area and nozzle area of the upstream along with the pressure difference, it is possible to calculate a theoretical flow rate through a pipe using conservation of mass and momentum. However, these theoretical flow rates do not take into account the momentum losses in the pipe, which are mostly located in the nozzle region, due to factors such as friction and turbulence. This is exactly where the value we call C_d 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 C_d 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 by 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 vent holes 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, it is possible to determine the position of the holes with 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 - C_p) value is minimum are ideal regions for us. We have marked these areas on the rocket in the graph on the right for easier understanding:



Ideal Location for Vent Holes

Important Footnotes for Air Vents:

In high altitude rocketry; as we mentioned before, since the main purpose of these holes is to protect the nose cone, the optimum position where the holes will be drilled will necessarily be near the nose cone. However, these holes to be drilled must not affect the separation system. If the hole is opened to the locations where the separation system is present, our rocket will not be able to build up enough pressure inside the hull and our rescue system will not be able to get out, even though it receives the command to leave when the target reaches altitude. In this respect, opening the holes under the payload, in a position between the parachutes, seems to be a reasonable choice.

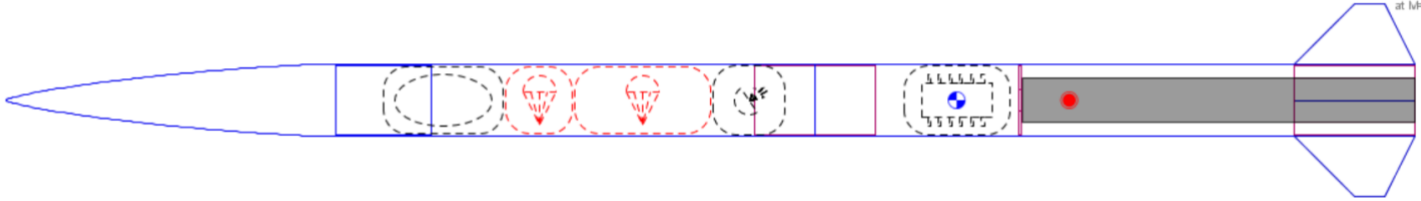
As for the hole size, although rocket tools generally prefer ½ and ¼ inch holes, the most reliable data can be obtained with CFD analysis.

We made an exemplary analysis to show the team members who will come after us better understand the subject and what methods should be followed, and we have shared it with you below.

Sample Analysis

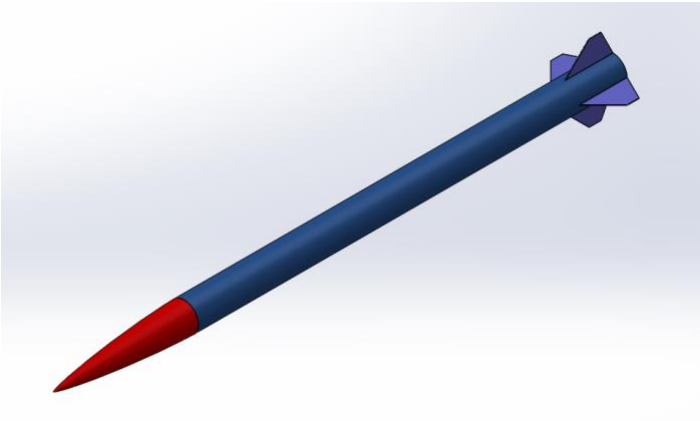
Rocket
 Length 235 cm, max. diameter 12.1 cm
 Mass with motors 13107 g

Stability: 1.56 cal
 CG: 159 cm
 CP: 177 cm
 at M=0.30



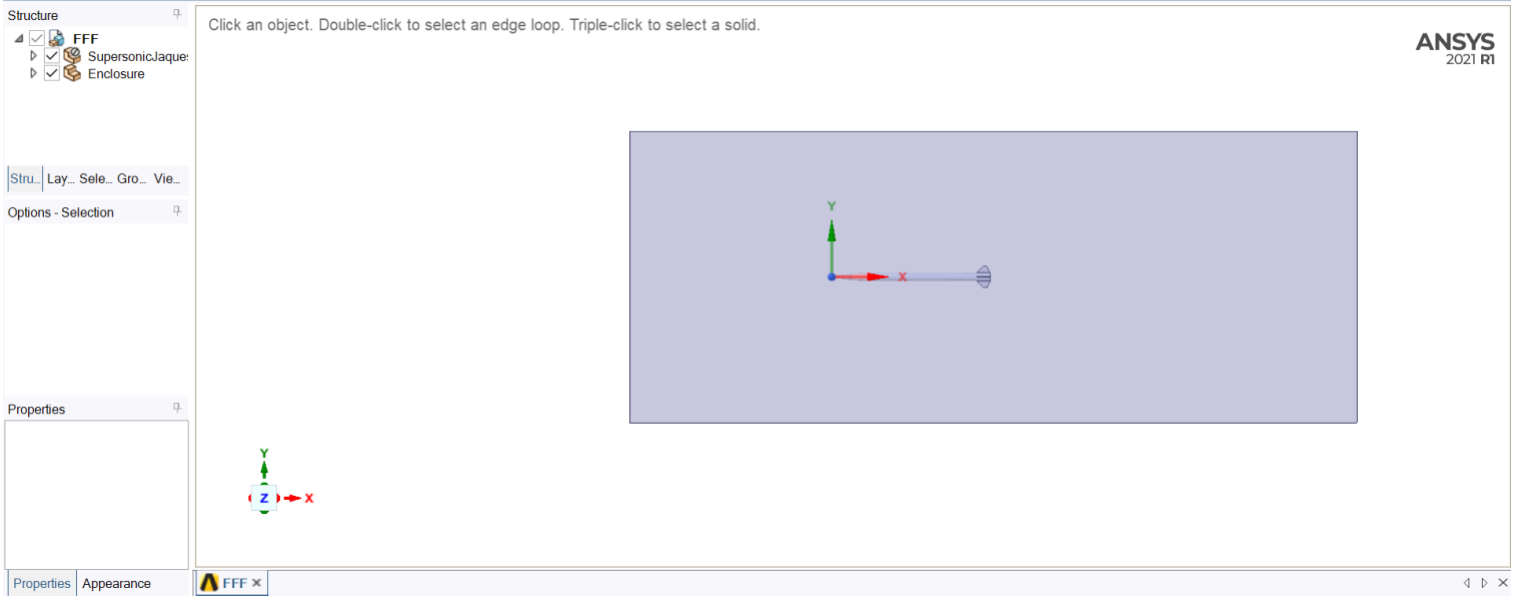
Apogee: 3258 m
 Max. velocity: 378 m/s (Mach 1.12)
 Max. acceleration: 201 m/s²

First, we designed a supersonic rocket using Openrocket. This rocket was designed to have the same body and interior design of our Bürküt rocket. Only some modifications were made on the engine and fins. Then, as in our previous external flow analysis, we drew our rocket on SOLIDWORKS and placed it in the ANSYS analysis.



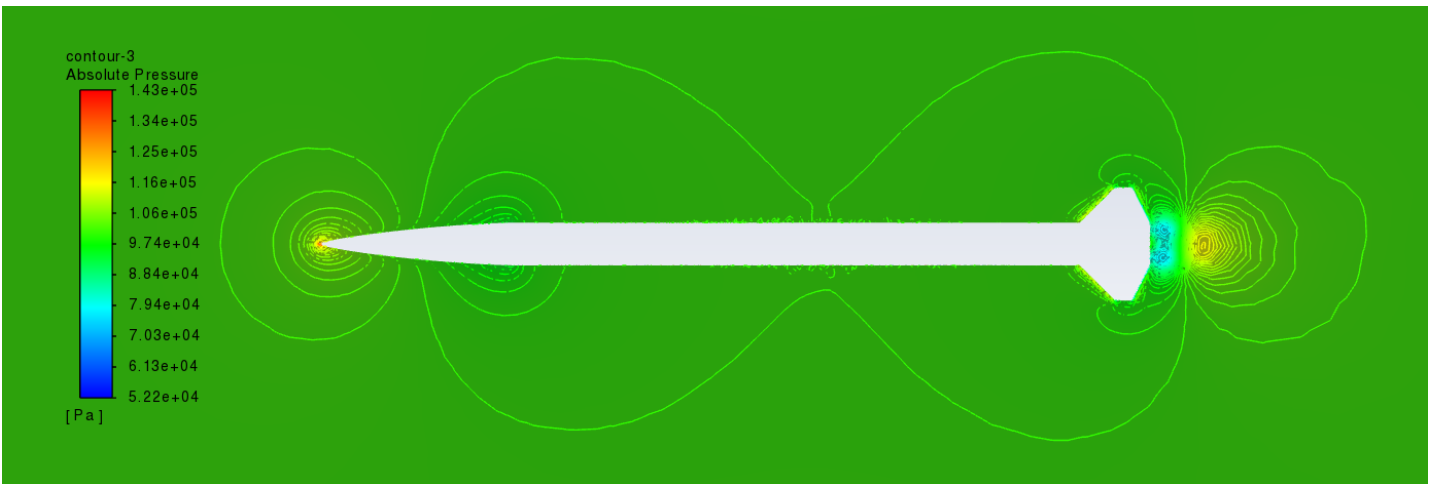
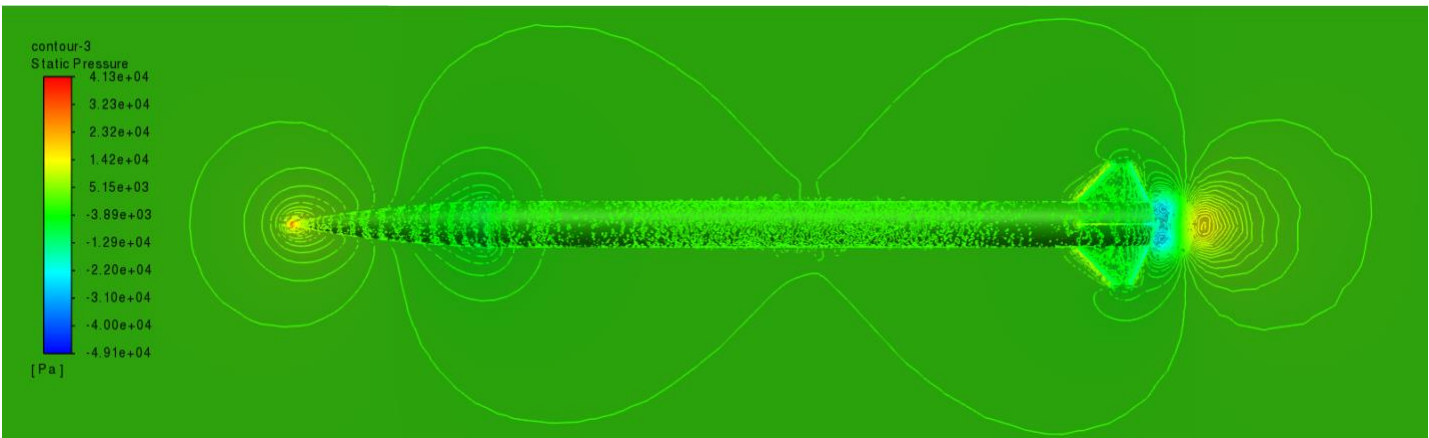
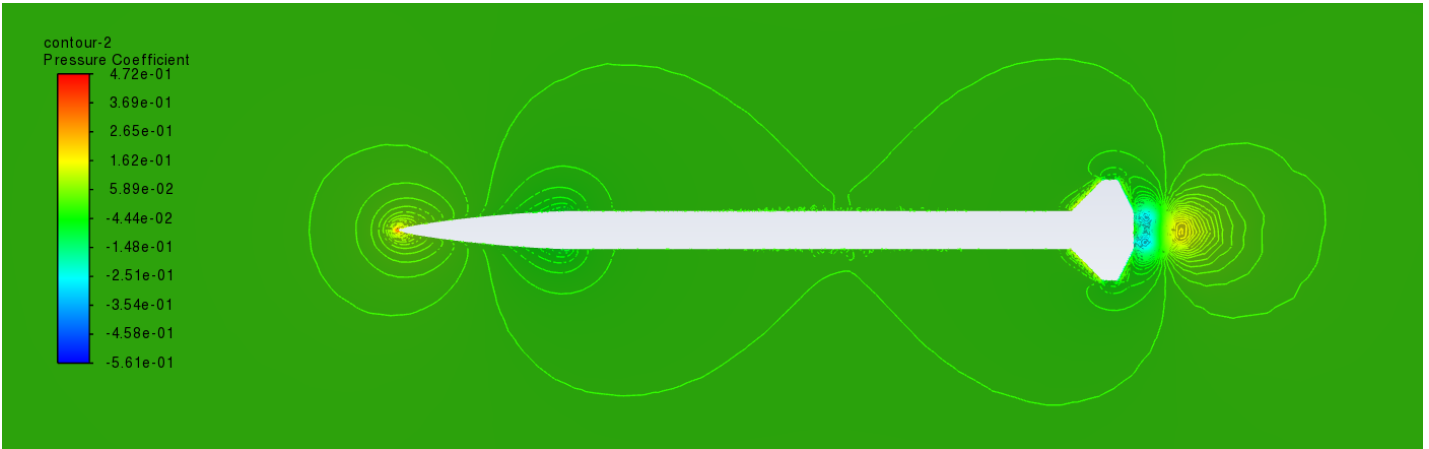
Our initial goal in our ANSYS analysis was to identify suitable locations for holes. In this way, we would have been able to show you more clearly what we talked about at length above.

We used the SpaceClaim application to have a practical geometry selection. With the Enclosure command, we have the control volume you want very quickly. Then all we had to do was disable the rocket itself by saying Suppress For Physics. And in less than a minute, we comfortably designed our control volume.

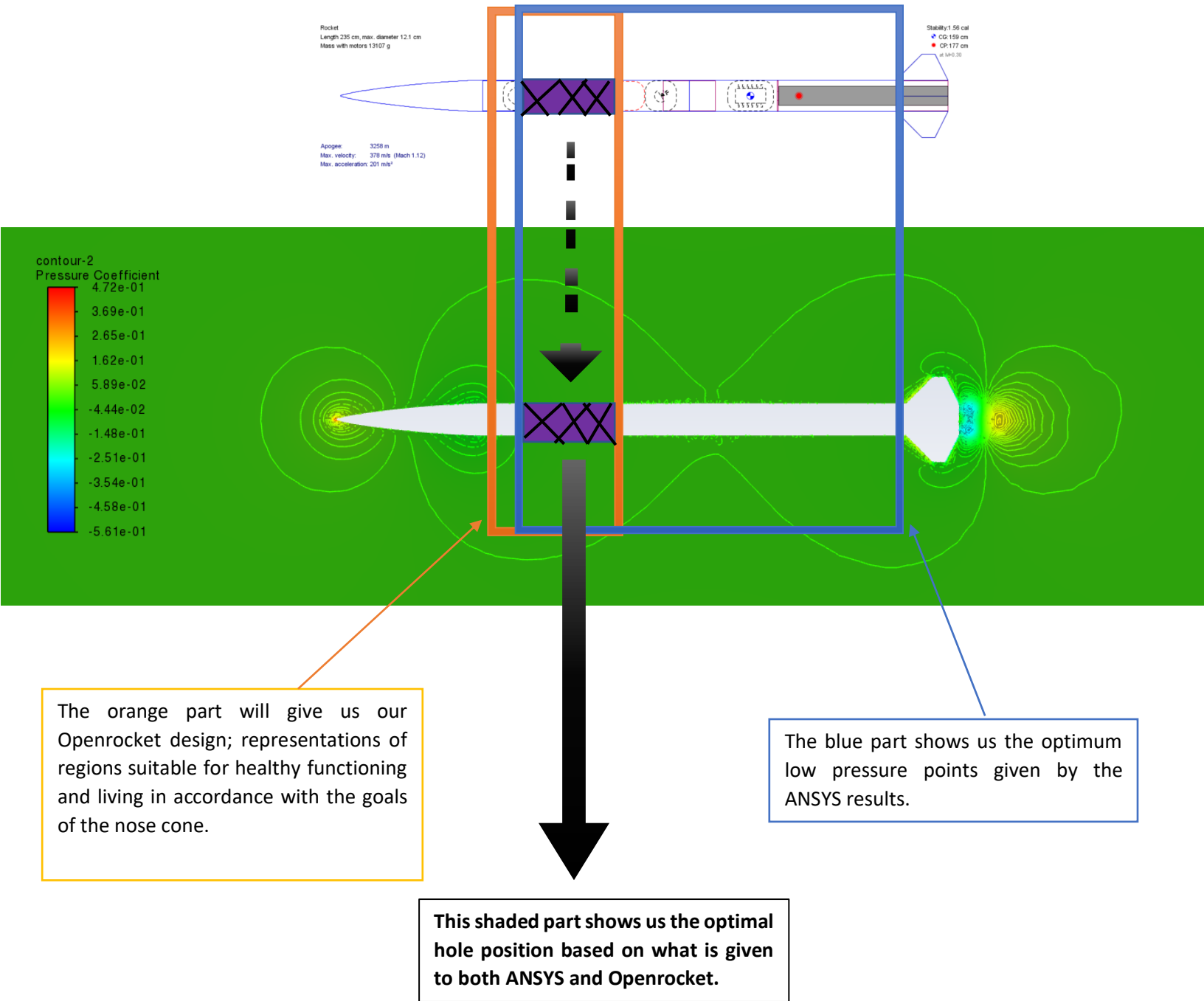


After the geometry was completed, we moved on to the Meshing part. There was no need to pay too much attention to the fins in this part. The quality of the mesh on the nose cone and body would be more than enough for us. While it would not be possible for us to increase the Orthogonal Quality above 0.1, it was possible for us to improve the Skewness and Aspect Ratio values. The 4mm face sizing we gave to the nose cone, body, back of the fuselage and fins has done our job more than enough. When we checked the value above 1000 in Aspect Ratio, our body gave a very good result. Since there is no point in increasing our number of personnel at this point, we have directly analyzed our rocket in this quality. Since this initial analysis of the hole location was not as detailed and important as we would deal with in hole size, it wouldn't be a problem if we did a rough analysis like this.

After solving 1000 iterations with the SST-KW solution, which took about 9 hours, we got the results we wanted. After the solution, we will look at the Pressure Coefficient and Absolute Pressure contours. The results are listed below:

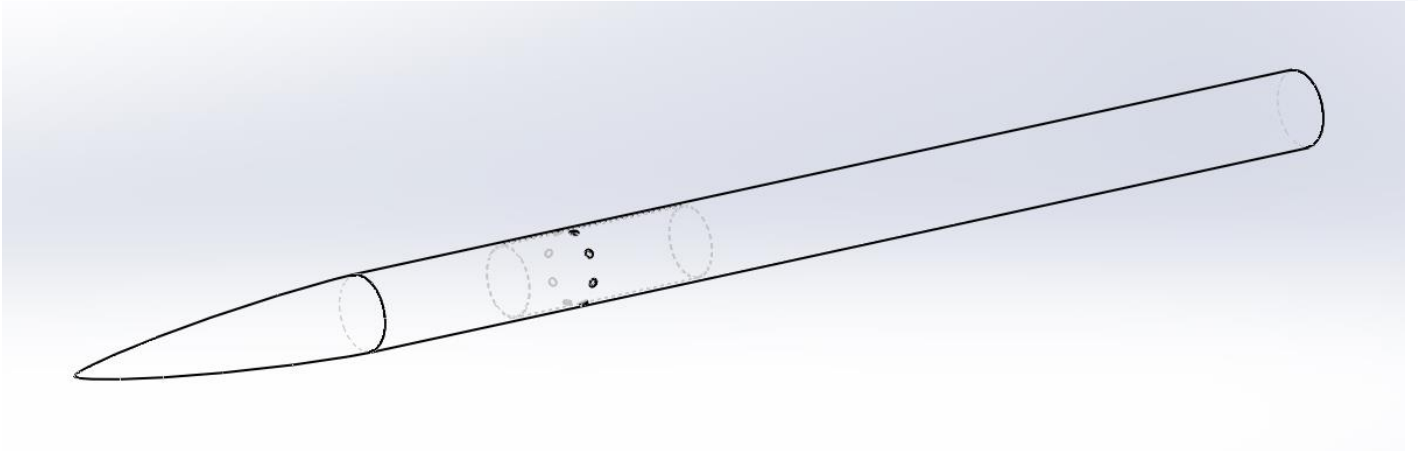
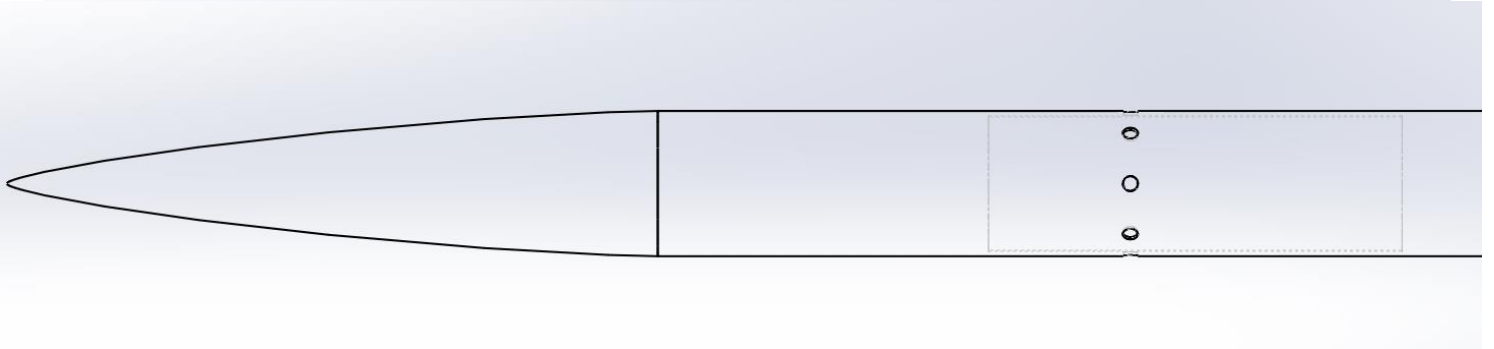


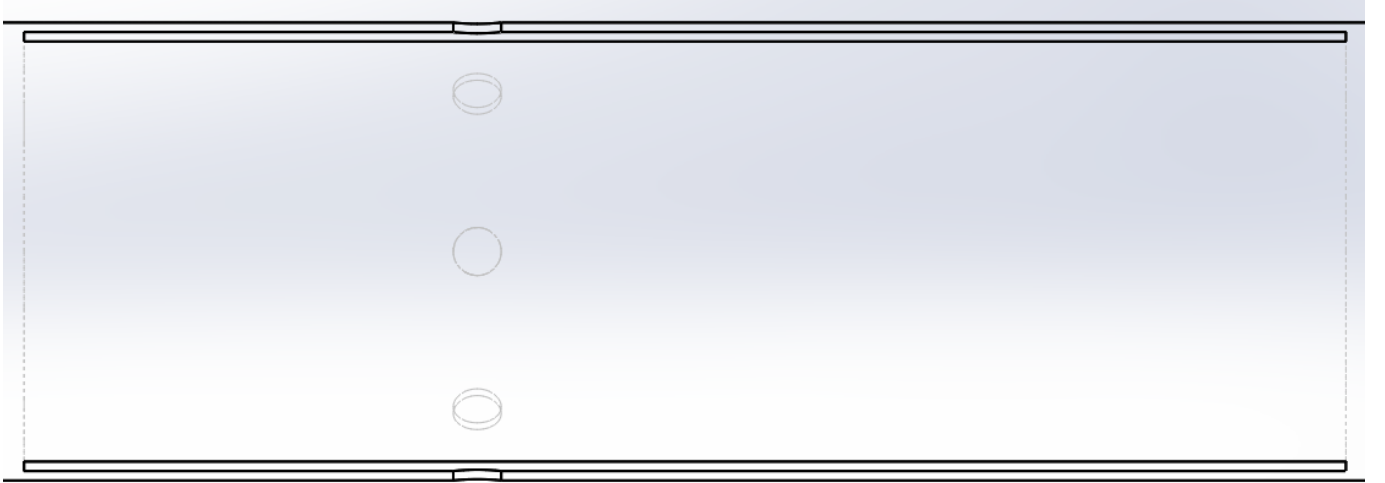
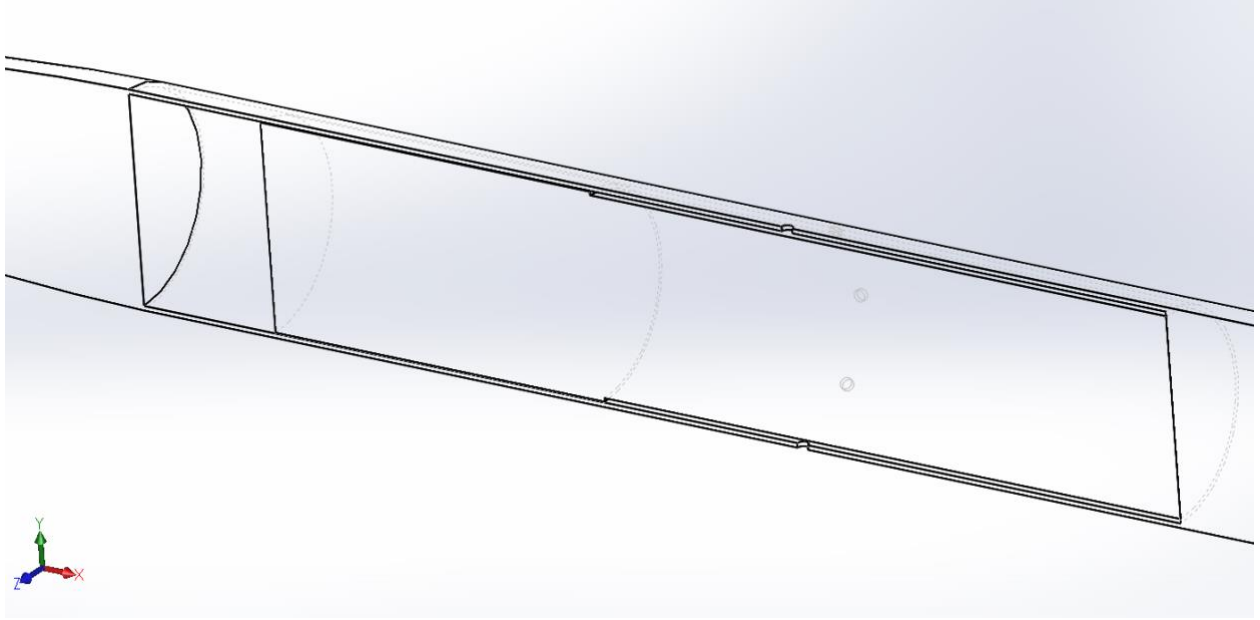
As can be seen, our absolute pressure and C_p values gave the same results visually. As we explained at the beginning of the topic, the air evacuation holes should be opened to positions where the C_p value is small. Although the lowest C_p value on the body is the rear body, a hole to be drilled here will not mean anything. In order to reduce the pressure on the nose cone, the hole must be in the front body and in a place that will not affect the separation system. With the collaboration of Openrocket and ANSYS, we can show this more clearly the ideal location for our holes.



The part we scanned with purple was determined as the suitable area to open the holes. Our next step will be to drill these holes through SOLIDWORKS and then put our rocket back into ANSYS analysis using the right methods. In this way, we aim to find the optimum hole size by trying different hole sizes on ANSYS by trial and error. The hole size, which is the highest of the Cd or mass flow rate values we will obtain, will be our optimum hole size.

Below is a SOLIDWORKS design as an example to qualify. It is the design of the halls with a thickness of 2.5mm for the places where the air can enter in the form of a hollow cylinder in itself. In this way, a room plan in the rocket where air can enter. Get it to this state, when the air we designed is filled and entered, the air will be more visible. In this design, which is made without blades, it has been excluded from the design that the blades will be affected in our analysis, making it difficult to be inside the structure. Planned designs are ¼ and ½ inch in model, each part size is designed for two different rocket designs.





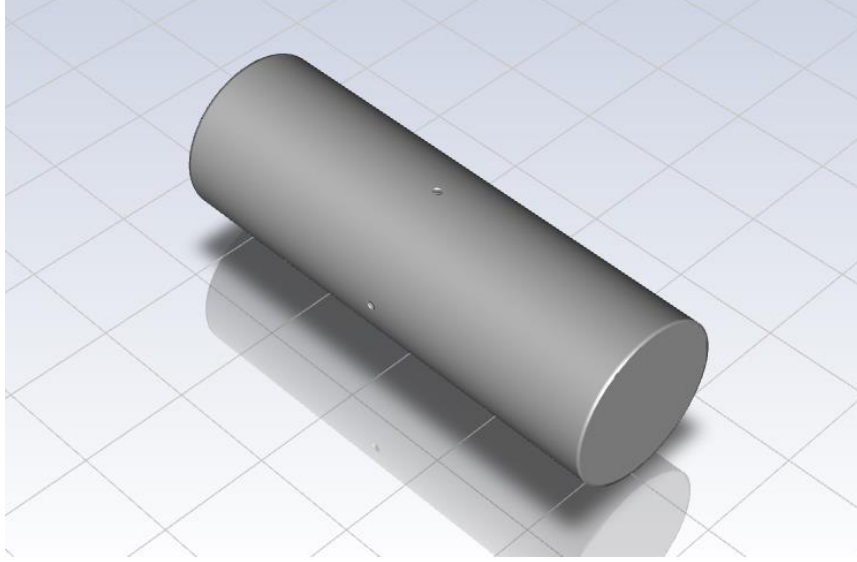
Attempts to Obtain Mass Flow Rate and Volume Flow Rate

Before analyzing the rockets we designed, we made a small experiment on how to obtain Mass Flow Rate and Volume Flow Rate values from perforated geometries in this way. We drilled holes in a partially evacuated cylinder and inserted it into ANSYS analysis. It was chosen in this way because the mesh throwing and unwinding would be easier and shorter than the original rockets. The drawing was made in SOLIDWORKS and transferred to ANSYS as parasolid. The Enclosure command in SpaceClaim was used for geometry. A quality and simple mesh was made and the setup part was started. We wanted to use the speed of our rocket for the setup part, so 378 m/s became our inlet speed. We did the rest in the same way as the rocket outer flow. SST-K Omega and Coupled methods were used. Our main point was to get the Mass Flow Rate and Volume Flow Rate values. For this, we added a graphic from the Report Plot section. We said New Plot, by choosing the New Surface option, we first assigned parallel planes to the ends of the holes. Two planes were enough for this experiment. After assigning a plane to the top and left holes, we chose the Mass Flow Rate and Volume Flow Rate values from the options above to be a separate graph for each plane.

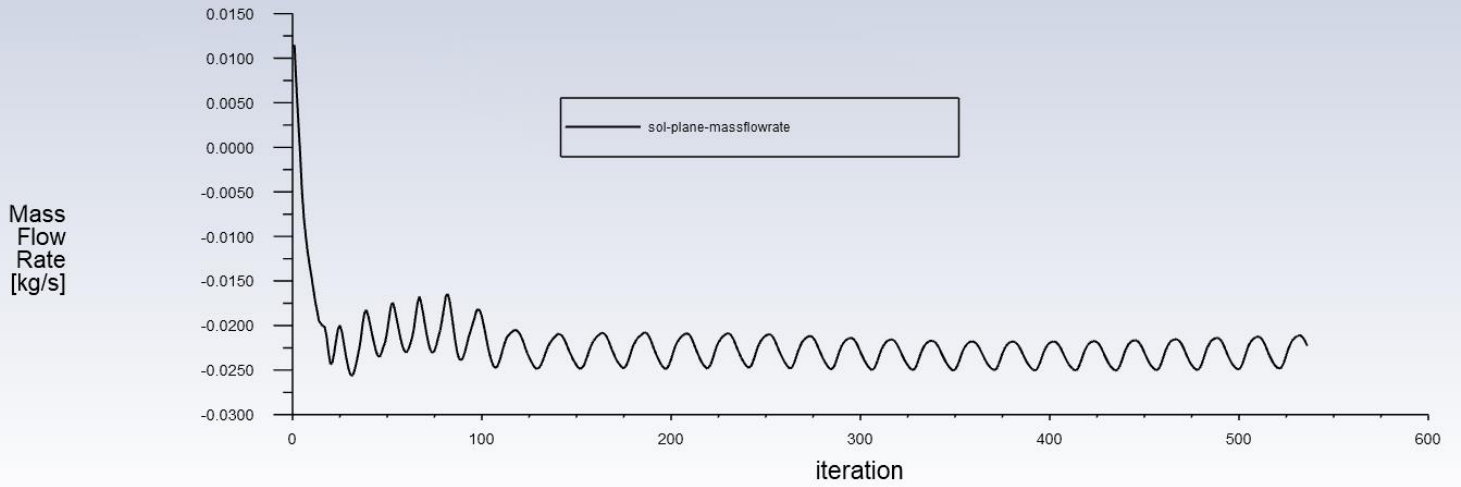
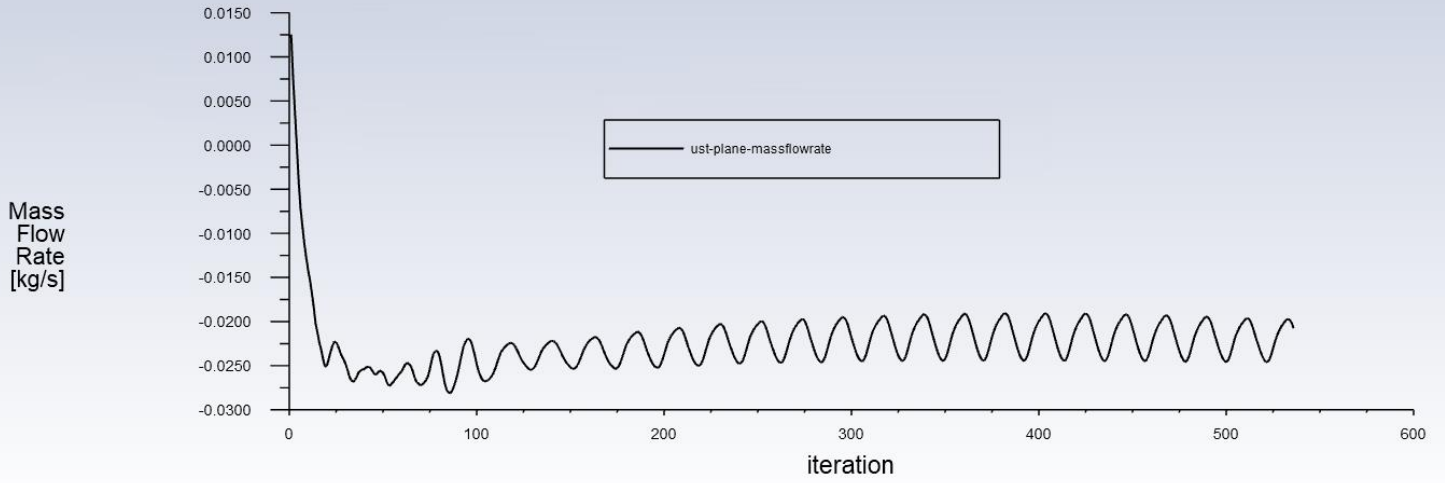
We will illustrate this process again in more detail in the rocket's own analysis. However, to put it briefly, we used the Surface---Mass Flow Rate or Volume---Volume Flow Rate preferences instead of the Force---Drag/Lift preference that we had previously taken with the old method using the interface part we used Drag and Lift.

The results came out pretty good. Mass Flow Rate and Volume Flow Rate gave close values for both holes. We managed to screenshot these values.

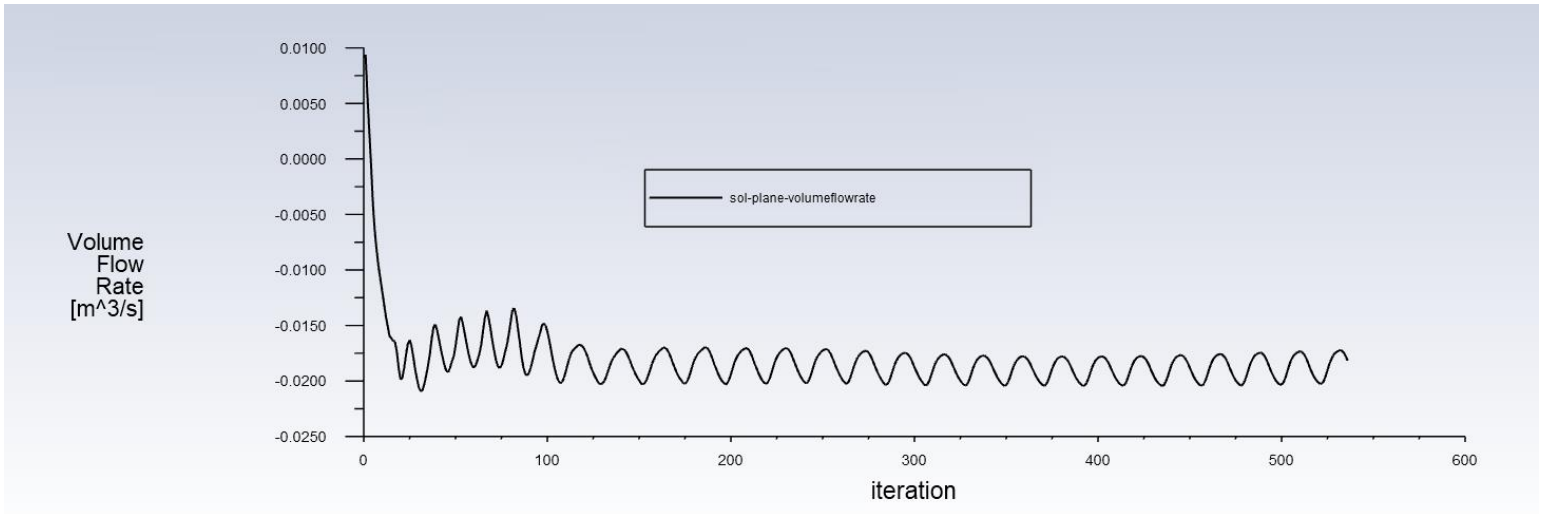
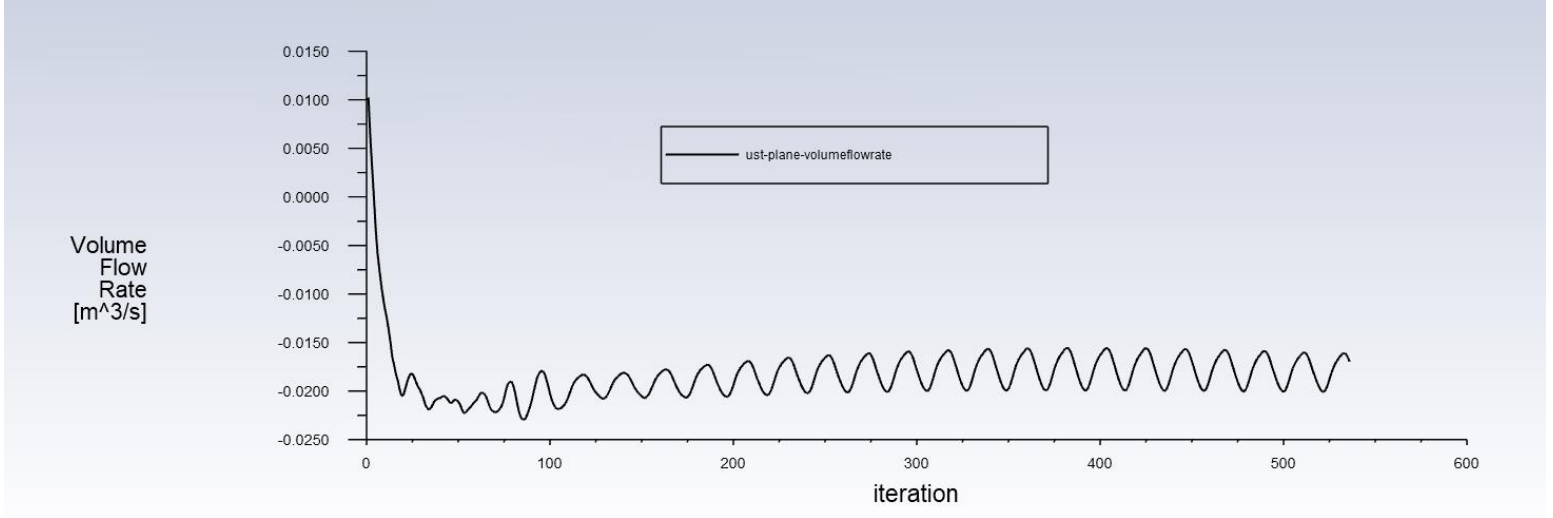
Unfortunately, when we tried to get vectors to observe the flow path, ANSYS crashed and we lost our analysis. However, at least we were able to recover our Mass Flow Rate and Volume Flow Rate values. As a footnote, it is also worth mentioning: Apart from Mass Flow Rate and Volume Flow Rate, you will also see a plain Flow Rate value. The Flow Rate here is actually the Flow Rate of Pressure. And we have nothing to do with this value, which is already zero.



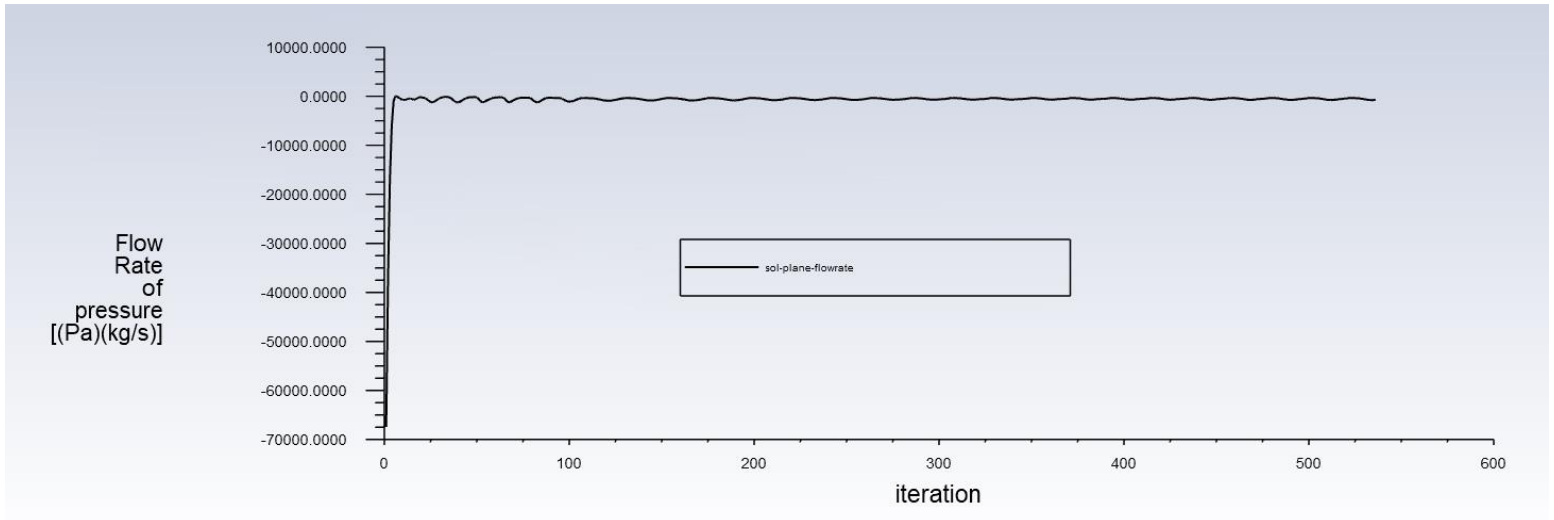
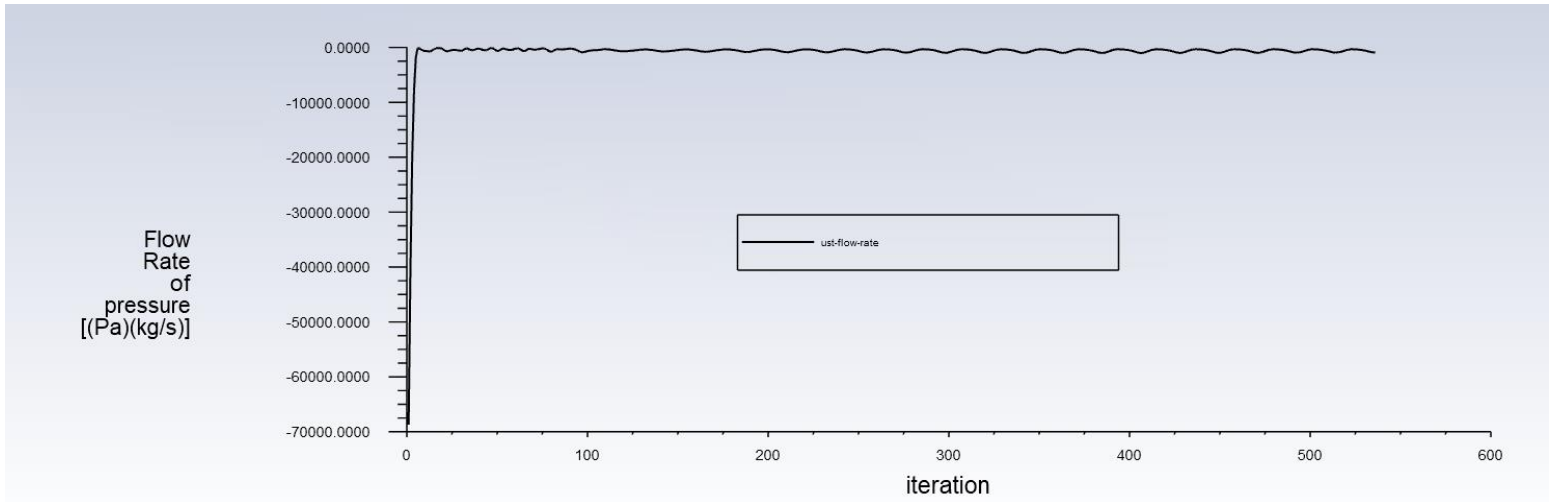
Mass Flow Rate Values:



Volume Flow Rate Values:



Flow Rate of Pressure (Unnecessary for Analysis):

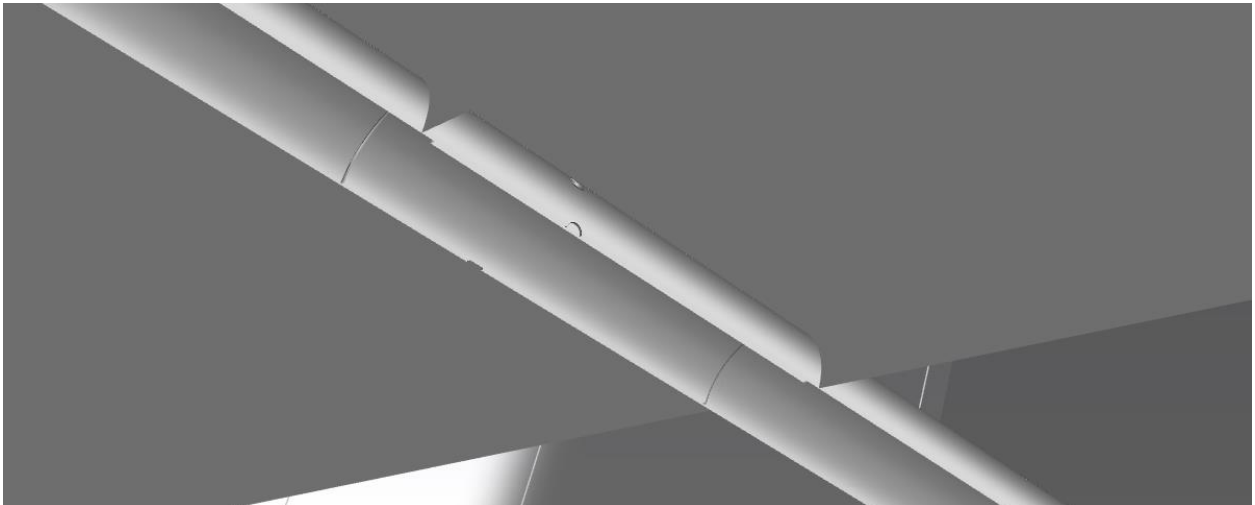
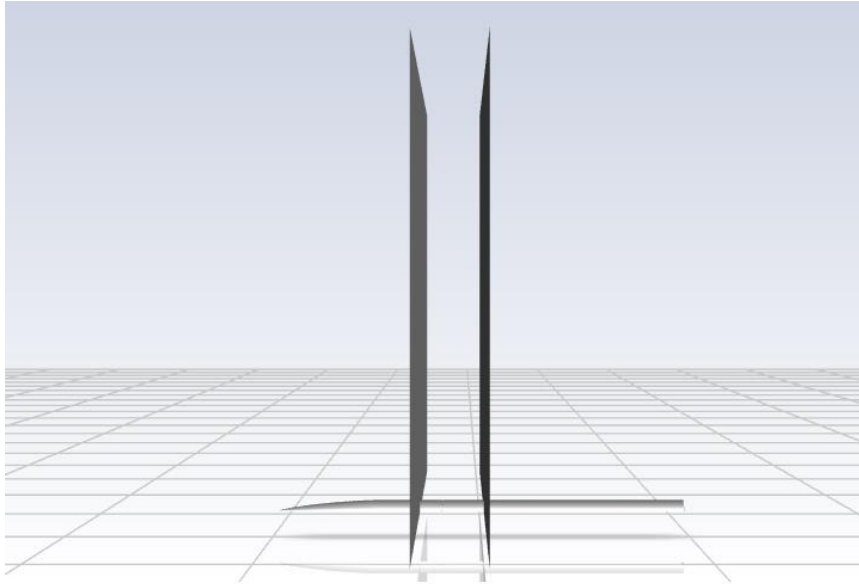


An important point that caught our attention in all these analyzes and realized later on changed the course of the whole project. As it was noticed, all of our graph values were always fluctuating, so there was no complete convergence. The reason was the plane we assigned. If we try to assign a plane in this way on our real rocket, vertical air fluctuations at the back and front of the rocket will affect our flow rate values, as in this analysis, and maybe even not converge. In other words, the values we obtained from all these graphs were **not correct** and healthy values. This revealed a big problem:

“How would we get the Mass flow rate and Volume flow rate values?”

All research was blocked at this point and a solution had to be found. As a result, it was not possible for us to understand whether there was air coming out of the hole or not, unless we had numerical data. Getting pressure and velocity contour didn't give us the images we wanted. Similarly, when we tried to import vector and streamline, either the analysis crashed or the image we saw didn't mean anything to us. We had insufficient evidence to say there is an air outlet, and due to this inadequacy, we had no idea about the hole dimensions.

After long deliberations, a solution was finally found. According to this solution idea, two planes could be assigned so that the flow volume of the holes would be in the middle of the planes, and then the mass flow rate difference between these planes could be determined. In order to observe whether the idea gave healthy results, a design was immediately made and left to the solution. The result was successful as you can see below.



In the Solution section, we can still select Surface---Mass Flow Rate and obtain a graph, but we cannot see the fractional values from the graph because the values are too large. For this, after the solution was finished, we selected the front and rear surfaces that we created from the "Surface Integrals" section in the Solution section and printed the mass flow rate value to the console with the Compute command. Indeed, as we expected, air escaped from the hole and the mass flow rate on the surface at the back was larger than the front. With the same technique, we were able to detect the volume flow rate.

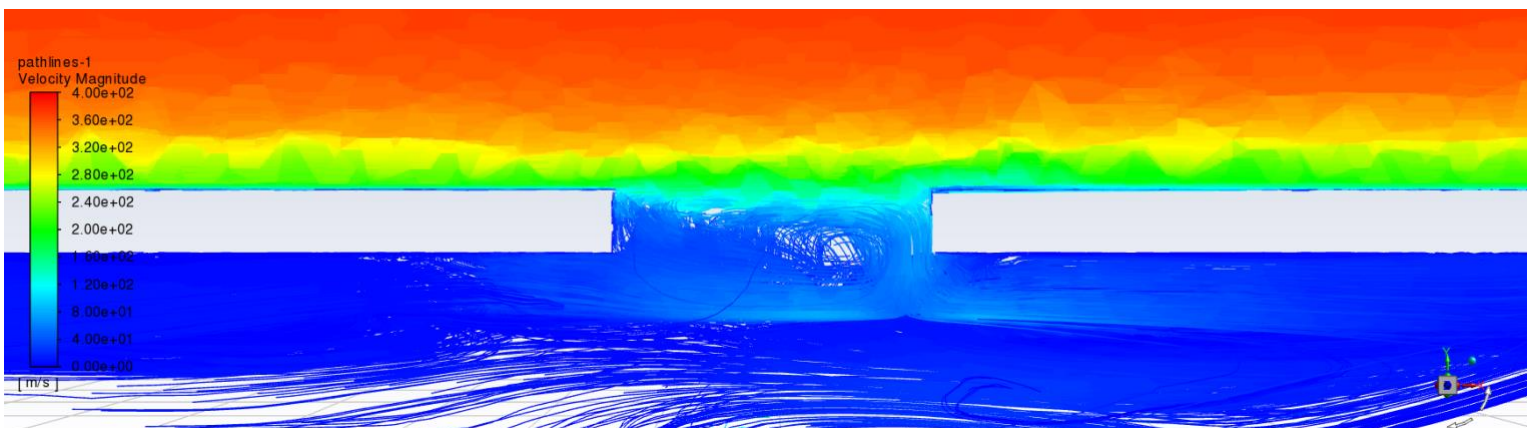
Mass Flow Rate	[kg/s]
arka-taraf	-5788.1168
on-taraf	-5788.1164
Net	-11576.233

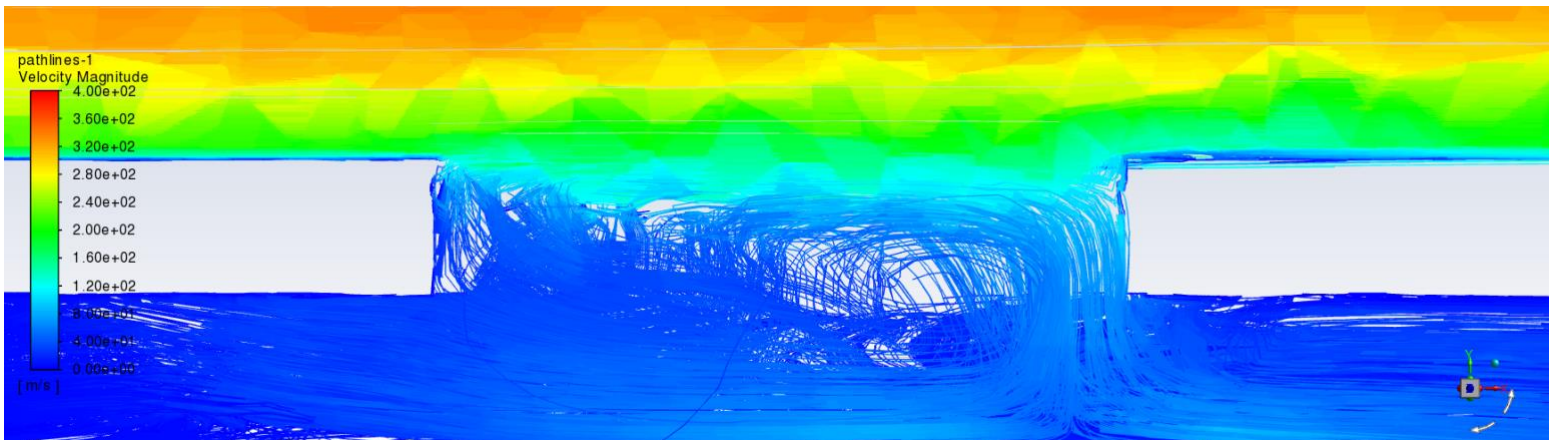
All that remained was to visually observe a flow. At this point, we were able to take a streamline from the solution section over a contour passing through the middle of the rocket on the xy and convert our stream into a video with the Pulse command, and we examined the stream in more detail. The air coming out of the hole was clearly visible. While some of the air was coming out from the end of the hole, most of the remaining air in the hole was contacting the outside high-velocity air and was accelerating back into the hole. After these successful solutions, we were ready to put our real rockets into analysis. We designed and analyzed one 8 hole ½ inch and one 8 hole ¼ inch rocket.

1/2 inch Hole Rocket Results:

Mass Flow Rate	[kg/s]
-----	-----
arka-plane	-17348.911
on-plane	-17348.908
-----	-----
Net	-34697.819

Volumetric Flow Rate	[m ³ /s]
-----	-----
arka-plane	-14162.376
on-plane	-14162.374
-----	-----
Net	-28324.75





According to these results, no problems are observed in 0.5 inch holes. It is suitable for design.

We obtained a total of 0.003 kg/s mass flow rate and 0.002 m³/s volume flow rate from the holes of our 1/2 inch 8-hole rocket.

$$C_d = \frac{\dot{m}}{\rho \dot{V}} = \frac{\dot{m}}{\rho A u} = \frac{\dot{m}}{\rho A \sqrt{\frac{2\Delta P}{\rho}}} = \frac{\dot{m}}{A \sqrt{2\rho\Delta P}}$$

Our discharge coefficient value is our main goal, and as can be seen from the formula above, we can compare the ratio of our mass flow rate and volume flow rate values with our rockets with $\frac{1}{2}$ to $\frac{1}{4}$ inch holes. The rocket with a large Cd value will give our appropriate hole size.

Cd value of our $\frac{1}{2}$ inch 8-hole rocket: $1.5/\text{density of air}$

1/4 inch Hole Rocket Results:

Mass Flow Rate	[kg/s]
-----	-----
arka-plane	-17348.915
on-plane	-17348.901
-----	-----
Net	-34697.816
Volumetric Flow Rate	[m ³ /s]
-----	-----
arka-plane	-14162.38
on-plane	-14162.368
-----	-----
Net	-28324.748

We obtained a total of 0.014 kg/s mass flow rate and 0.012 m³/s volume flow rate from the holes of our 1/4 inch 8-hole rocket.

Cd value of our 1/4 inch 8-hole rocket: 1.166667/density of air

Sample Analysis Result:

$Cd(\max) = 1.5 / \text{density of air}$

As seen in our sample analysis, our 1/2 inch 8-hole rocket represents our ideal hole type, but it also now determines the hole position. We have clearly determined. As long as the methods in this sample analysis are followed, a new supersonic rocket to be designed will no longer be a problem for us. In case of development of ANSYS and technology, this report should be updated and additions should be made.

3) 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=2ahUKEwixh-fQ9rjvAhUbhv0HHWoJC-00FnoECCIOAQ&url=http%3A%2F%2FQpedia%2F%2FQpedia-www.2FQpedia_Feb08_Discharge_coefficient_in_flow_calculation_and_measurement&usg=AOvVaw3ak9-VsxZZQGP2f_t9IBbr

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

Department: Aerodynamics

Research, Analysis, Report: Halit Yusuf Genç

Direction: Umut Engin, Muhammet Ikbal Akyuz