MARMARA UNIVERSITY
FACULTY OF ENGINEERING

ANALYZING OF JET FLOW WITH OPEN SOURCE SOFTWARE

Mustafa USCA

GRADUATION PROJECT REPORT

Department of Mechanical Engineering

Supervisor

Prof. Dr. Emre ALPMAN

ISTANBUL, 2021

MARMARA UNIVERSITY
FACULTY OF ENGINEERING

Analyzing of Jet Flow with Open Source Software

by

Mustafa USCA

June 24,2021,Istanbul

SUBMITTED TO THE DEPARTMENT OF MECHANICAL ENGINEERING IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE

OF

BACHELOR OF SCIENCE

AT

MARMARA UNIVERSITY

Signature of Author(s) ....M. USCA..
Department of Mechanical Engineering

Certified By ....
Project Supervisor, Department of Mechanical Engineering

Accepted By...
Head of the Department of Mechanical Engineering

ACKNOWLEDGEMENT

I would like to thank my dear supervisor Prof. Dr. Emre ALPMAN for the all his support in preparing this thesis. In this process, we talked every week regularly and he made our study more efficient by helping me to solve the problems I encountered and to see the mistakes that I could not see. In addition, by sharing his valuable information with me and guiding me, he has made me love the CFD field, which includes the thesis topic. With this project, I started to be more interested in CFD field and decided to pursue graduate education after my undergraduate education. I would like to study and learn more about CFD and write my master thesis in this field. It is great to continue working with him and to write my master's thesis under his supervision.

June, 2021 Mustafa USCA

CONTENTS

ABSTRACT

Analyzing of Jet Flow with Open Source Software

Turbojet engines, which were invented with the studies carried out in the 1930s, enabled the rapid development of the aviation industry. With the discovery of jet engines, aircraft reached very high speeds and the speed of sound was exceeded for the first time in 1947. With the studies carried out in the field of jet engines and the rapid development of technology, the concept of jet has gained much more place in our lives. Today, jet engines are used in many advanced technologies such as warplanes, missiles and rockets that allow us to go into space.

In this study, the analysis of jet flow was taken into account by using computational fluid dynamics (CFD). The SU2 program, an open source software developed by Stanford University, was used to examine the jet flow with CFD. Two different geometries were used to examine the state of the flow both inside the nozzle and in the atmosphere. A test case study was conducted to observe the results at different Mach numbers and altitudes, and to observe how different atmospheric pressure and temperature affect the jet flow according to the nozzle outlet pressure and temperature values. Here, a total of eighteen test cases were studied at three different altitudes and three different Mach numbers.

In line with the studies, it has been observed that as the atmospheric pressure and atmospheric temperature increase, the jet flow accelerates, much stronger shock waves are formed and the vortex structures change. In addition, a study was carried out to see the difference between the far field and the wall while defining the markers. It has been observed that the strong shock waves formed when the wall is defined reflect from the wall and directly affect the flow, changing the angles and numerical values. When the far field was defined, it was observed that there is no reflection. Thanks to the large number of test cases, many analysis results were compared with each other and it was determined that the results were consistent.

Keywords: Computational fluid dynamics (CFD), jet flow, test cases

SYMBOLS

$\mathbf{u}$	**:** Horizontal velocity component
$\mathbf{v}$	**:** Vertical velocity component
Re	**:** Reynold's number
$\mathbf{h}$	**:** Step height
$\boldsymbol{\mu}$	**:** Dynamic viscosity
$\boldsymbol{\rho}$	**:** Density
$\mathbf{y}$	**:** Width of the jet
$\mathbf{U_0}$	**:** Centerline velocity
$\mathbf{x}$	**:** Axial distance
$\boldsymbol{\gamma}$	**:** Specific heat ratio
$\mathbf{M}$	**:** Mach number
$\mathbf{M_e}$	**:** Exit Mach number
$\mathbf{P}$	**:** Pressure
$\mathbf{P^*}$	**:** Sonic pressure
$\mathbf{P_0}$	**:** Total pressure
$\mathbf{P_e}$	**:** Exit pressure
$\mathbf{T}$	**:** Temperature
$\mathbf{T^*}$	**:** Sonic temperature
$\mathbf{T_0}$	**:** Total temperature
$\mathbf{T_e}$	**:** Exit temperature

ABBREVIATIONS

CFD **:** Computational Fluid Dynamics

FVM **:** Finite Volume Method

LIST OF FIGURES

PAGE

LIST OF TABLES

PAGE

1. INTRODUCTION

A jet is formed by flow issuing from a nozzle into ambient fluid, which is at a different velocity. If the ambient fluid is at rest the jet is referred to as a free jet; if the surrounding fluid is moving, the jet is called a co-flowing jet. A jet is one of the basic flow configurations which have many practical applications such as in jet engines, combustors, chemical lasers, ink-jet printer heads, among others. The velocity at the exit of the nozzle of a typical laboratory jet has a smooth profile and a low turbulence level, about 0.1% - 0.5% of the mean velocity. Due to the velocity difference between the jet and the ambient fluid, a thin shear layer is created. This shear layer is highly unstable and is subjected to flow instabilities that eventually lead to the formation of large-scale vortical structures (see Figure 1.). The interaction of these structures produces strong flow fluctuations, entrains ambient fluid into the jet flow and enhances the mixing. The shear layer and consequently, the jet spread along the direction perpendicular to the main jet flow.

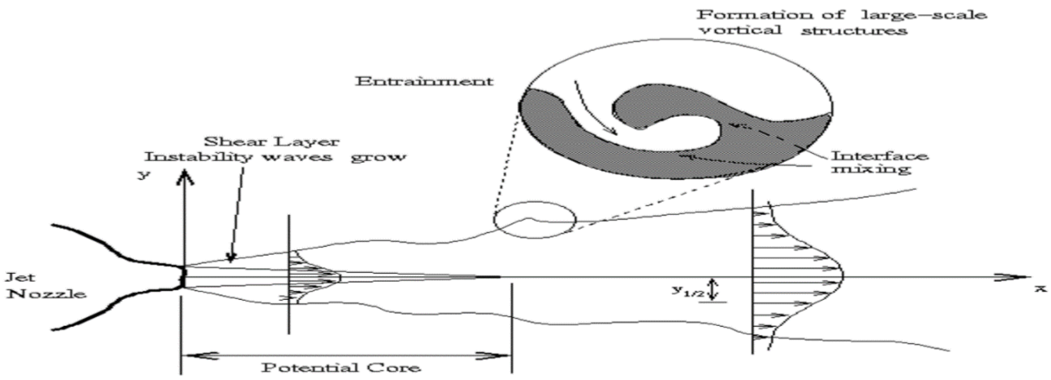


Figure 1.1. Jet spread along the direction perpendicular to the main jet flow

The central portion of the jet, a region with almost uniform mean velocity, is called the potential core. Because of the spreading of the shear layer, the potential core eventually disappears at a distance of about four to six diameters downstream from the nozzle. The entrainment process continues further beyond the end of the potential core region such that the velocity distribution of the jet eventually relaxes to an asymptotic bell-shaped velocity profile as illustrated in Figure 1.2. Also shown in Figure 1.2. is the half-width of the jet, $y_{1/2}$, defined as the distance between the axis and the location where the local velocity equals half of the local maximum or centerline velocity,U_0. The increase in the jet half-width with downstream distance provides a measure of the spreading rate of the jet. Due to the spreading, the jet centerline velocity decreases downstream beyond the potential core region.

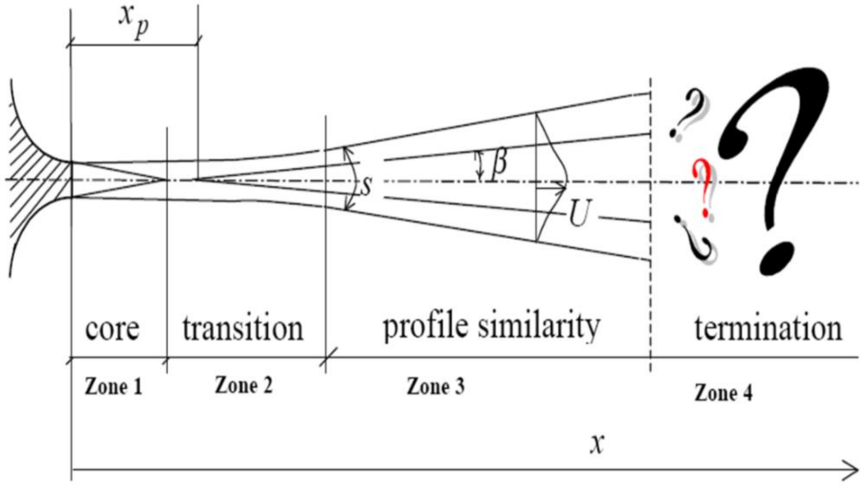


Figure 1.2. Jet expansion zone

From the Figure 1.2. the development of jet is divided in to four zones, related to center line velocity decay.

1) Zone 1, a conical zone where the center line velocity is equal to outlet velocity.
2) Zone 2, a transition zone where the velocity start to decreases often to approximated as proportional to $x^{-0.5}$, where x is the axial distance.
3) Zone 3, where the transverse velocity profile is similar at different value of x and velocity decay is assumed proportional to x^{-1}
4) Zone 4, the jet terminal zone where center line velocity rapidly decreases.

1.1. Literature Research

A jet is a stream of fluid that is projected into a surrounding medium, usually from some kind of a nozzle, aperture or orifice. Jets can travel long distances without dissipating. Jet fluid has higher momentum compared to the surrounding fluid medium. In the case that the surrounding medium is assumed to be made up of the same fluid as the jet, and this fluid has a viscosity, the surrounding fluid is carried along with the jet in a process called entrainment.

Rocket engines and jet engines are driven by jet propulsion. The jet engine, also known as the reactive engine, compresses the air it takes from the atmosphere and heats it by burning it with fuel. The gases that emerge as a result of this heating are rapidly ejected, creating a thrust in the opposite direction. With this power, the vehicle to which the engine is connected is provided to move. These motors were developed based on Newton's laws of motion. According to this law; Every action produces an equal and opposite reaction.

From 1903 to 1935, alternative piston-action internal combustion engine and propeller assemblies were used in aircraft propulsion systems. In 1935, the first jet powered airplane (jet airplane) was built by Hans von Ohain. Jet-powered airplanes can go much faster and much higher than other airplanes.

Today, many types of jet engines have been developed. Some of those; turbojet, turbofan, turboprop, turboshaft and ramjet.

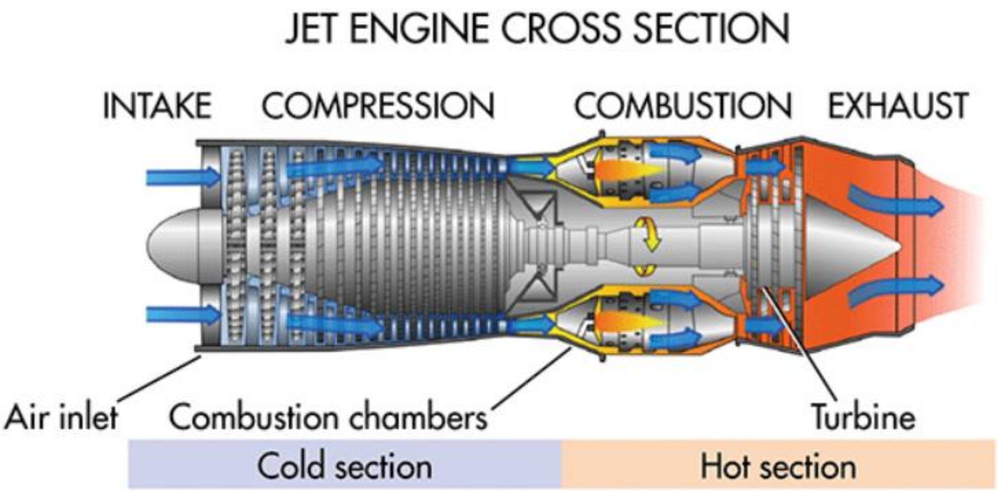


Figure 1.3. Jet engine cross section

There are some structures that are frequently observed in the analysis of a jet stream and in this study. These are shock waves (normal and oblique shocks), shock diamonds, eddies.

1.1.1. Shock Waves

In physics, a shock wave (also spelled shockwave), or shock, is a type of propagating disturbance that moves faster than the local speed of sound in the medium. Like an ordinary wave, a shock wave carries energy and can propagate through a medium but is characterized by an abrupt, nearly discontinuous, change in pressure, temperature, and density of the medium.

For the purpose of comparison, in supersonic flows, additional increased expansion may be achieved through an expansion fan, also known as a Prandtl–Meyer expansion fan. The accompanying expansion wave may approach and eventually collide and recombine with the shock wave, creating a process of destructive interference. The sonic boom associated with the passage of a supersonic aircraft is a type of sound wave produced by constructive interference.

Unlike solitons (another kind of nonlinear wave), the energy and speed of a shock wave alone dissipates relatively quickly with distance. When a shock wave passes through matter, energy is preserved but entropy increases. This change in the matter's properties manifests itself as a decrease in the energy which can be extracted as work, and as a drag force on supersonic objects; shock waves are strongly irreversible processes.

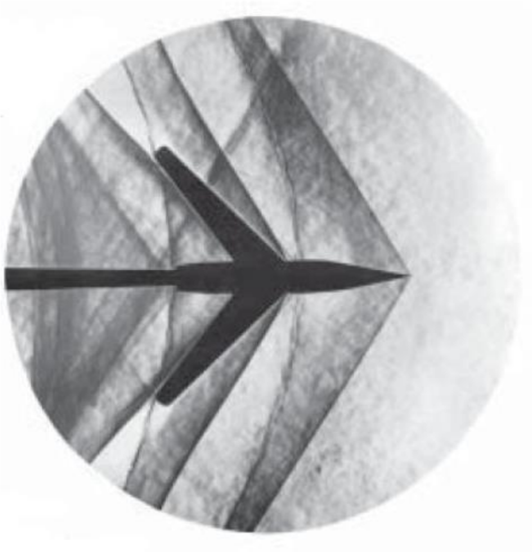


Figure 1.4. Schlieren photograph of an attached shock on a sharp-nosed supersonic body

Figure 1.5. Circular marks where the expanding spherical atmospheric shock waves from the gun firing meet the water surface

1.1.1.1. Normal Shock Waves

If the shock wave is perpendicular to the flow direction it is called a normal shock. A normal shock occurs in front of a supersonic object if the flow is turned by a large amount and the shock cannot remain attached to the body. The detached shock occurs for both wedges and cones. A normal shock is also present in most supersonic inlets. Across the normal shock the flow changes from supersonic to subsonic conditions.

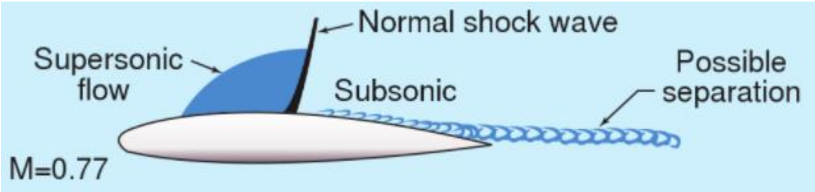


Figure 1.6. Normal shock wave on a transonic flow airfoil

1.1.1.2. Oblique Shock Waves

An oblique shock wave is a shock wave that, unlike a normal shock, is inclined with respect to the incident upstream flow direction. It will occur when a supersonic flow encounters a corner that effectively turns the flow into itself and compresses. The upstream streamlines are uniformly deflected after the shock wave. The most common way to produce an oblique shock wave is to place a wedge into supersonic, compressible flow. Similar to a normal shock wave, the oblique shock wave consists of a very thin region across which nearly discontinuous changes in the thermodynamic properties of a gas occur. While the upstream and downstream flow directions are unchanged across a normal shock, they are different for flow across an oblique shock wave.

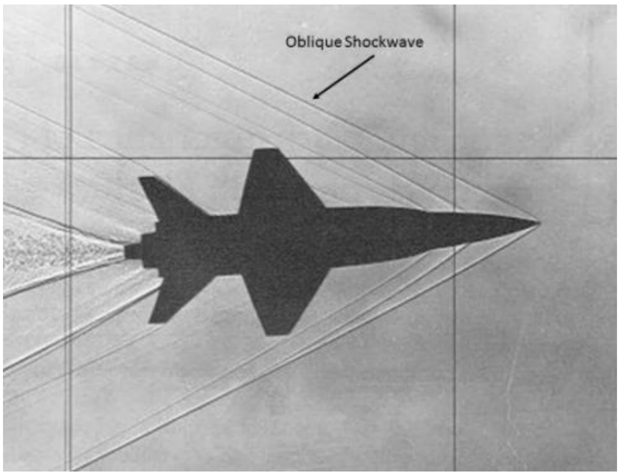


Figure 1.7. Oblique shock waves on X-15 flight vehicle

1.1.2. Shock Diamonds

Shock diamonds (alternatively known as “Mach disks”) occur when gas is exiting a nozzle at supersonic speeds, at a different pressure than the outside atmosphere. At sea level, the exhaust pressure might be lower than the thick atmosphere. And then at very high altitudes, the exhaust pressure might be higher than the thin atmosphere. So, these shock diamonds can appear just as a rocket is taking off, or at high altitude when it shifts into supersonic speed. A classic example is the space shuttle blasting off, but another famous example is when Chuck Yeager’s X-1 rocket plane reached Mach 1.

Figure 1.8. Shock diamonds in Chuck Yeager’s X-1

1.1.3. Vortexes

In fluid dynamics, a vortex (plural vortices/vortexes) is a region in a fluid in which the flow revolves around an axis line, which may be straight or curved. Vortices form in stirred fluids, and may be observed in smoke rings, whirlpools in the wake of a boat, and the winds surrounding a tropical cyclone, tornado or dust devil.

Vortices are a major component of turbulent flow. The distribution of velocity, vorticity (the curl of the flow velocity), as well as the concept of circulation are used to characterise vortices. In most vortices, the fluid flow velocity is greatest next to its axis and decreases in inverse proportion to the distance from the axis.

In the absence of external forces, viscous friction within the fluid tends to organise the flow into a collection of irrotational vortices, possibly superimposed to larger-scale flows, including larger-scale vortices. Once formed, vortices can move, stretch, twist, and interact in complex ways. A moving vortex carries some angular and linear momentum, energy, and mass, with it.

Figure 1.9. Formed vortices behind a plane

1.2. Importance, Aim and Method of the Thesis

Jet flow is a very important research subject in both fundamental fluid dynamics and engineering applications. Jet flow has the essences of fluid dynamics, such as free and wall-bounded shear flows, turbulent flow, eddy and large vortical structures and their stability and control, and so forth. Jets are a common configuration used in many mixings and thrusts producing devices, and the enhancement of jet flow mixing is frequently desirable in a broad range of engineering applications from aerodynamics to hydraulics. In this study, the structure of jet flows, which are of great importance, and the effects of different ambient conditions on the jet flow are investigated using computational fluid dynamics techniques.

For this purpose, SU2 program, which is an open source software, was used for simulation study. SU2 is a computational fluid dynamics program used to solve problems related to heat transfer and fluid mechanics using the finite volume method (FVM). For a comprehensive and detailed analysis, the test case study was conducted to observe the results at different Mach numbers and altitudes, and to observe how different atmospheric pressure and temperature affect the jet flow according to the nozzle outlet pressure and temperature values. In total, analyzes were made on 18 different test cases.

In the first part of the thesis, which consists of four chapters, the need for the project work is presented with its reasons, the definition of the problem is made and the purpose of the study is stated. In the second section, the geometries prepared for simulation and the mesh structures created in these geometries are shown. Boundary conditions and test cases are explained as well as all other necessary information is given. In the third chapter, the results of the test case studies are shown and these results are compared with each other and the situations of the jet flow under different conditions are discussed. In the fourth and last chapter, the findings are summarized and some suggestions are made related to the subject.

2. Material and Method

In this study, SU2, which is an open source software, was used as mentioned before. But SU2 alone is not enough. Different programs are needed to create the geometry, perform the mesh operation, and display the results. The programs used are as follows:

- Gmsh – Creating geometry and meshing
- SU2 – Performing numerical operations
- ParaView – Viewing the outputs

2.1. Theoretical Backgrounds

The simulations are done by solving 2D Navier-Stoke equation for unstable density flow. The continuity and momentum equation for a steady state 2D unstable density flow are given as follows:

Continuity equation,

$$\frac{d}{dx}(\rho u) + \frac{d}{dy}(\rho v) = 0 \tag{1.1}$$

For energy equation,

$$\frac{\partial}{\partial t}\left[\rho\left(e + \frac{V^2}{2}\right)\right] + \nabla \cdot \left[\rho\left(e + \frac{V^2}{2}\vec{V}\right)\right] = \rho\dot{q} + \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) - \frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial x} + \frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{xy})}{\partial x} + \frac{\partial(u\tau_{yy})}{\partial y} + \rho\vec{f} \cdot \vec{V} \tag{1.2}$$

Momentum equation,

X-Momentum:

$$u\frac{du}{dx} + v\frac{du}{dy} = -\frac{1}{\rho}\frac{dP}{dx} + \mu\left[\frac{d^2u}{dx^2} + \frac{d^2u}{dy^2}\right] \tag{1.3}$$

Y-Momentum:

$$u\frac{dv}{dx} + v\frac{dv}{dy} = -\frac{1}{\rho}\frac{dP}{dy} + \mu\left[\frac{d^2v}{dx^2} + \frac{d^2v}{dy^2}\right] \tag{1.4}$$

Other equations for the boundary conditions,

$$T = T_0\left(1 + \frac{\gamma-1}{2}M^2\right)^{-1} \tag{1.5}$$

$$P = P_0\left(1 + \frac{\gamma-1}{2}M^2\right)^{-\frac{\gamma}{\gamma-1}} \tag{1.6}$$

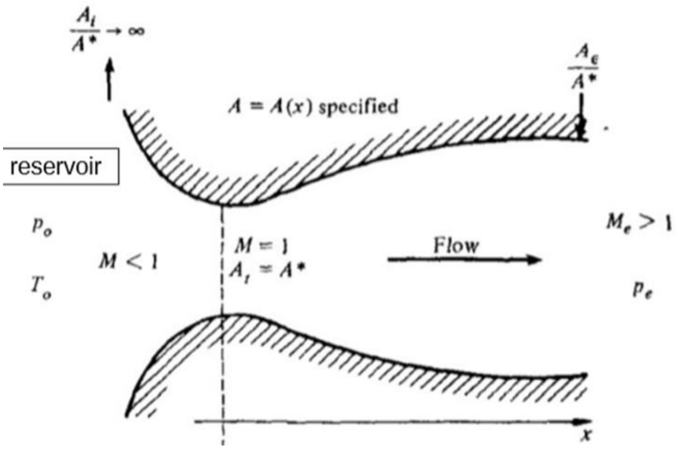


Figure 2.1. Geometry for reservoir and nozzle

2.2. Geometry

Two different geometries were needed in this study. The first is for the analysis of the flow inside the nozzle, and the second is for the analysis of the flow outside the nozzle. Geometries have a simple 2D structure. No need for complex or 3D dimensional geometries.

The geometry in Figure 2.2. is designed for the analysis of the flow through the nozzle. The nozzle has a symmetrical structure. Therefore, only half of the nozzle was drawn in order to halve the analysis time. Flow comes from the left side, that is, from the combustion chamber, and exits from the right side. The lower side is defined as the axis of symmetry. This design is inspired by rocket nozzles. That's why the outlet is designed larger than the inlet. The reason why it is designed large is to increase the output Mach number.

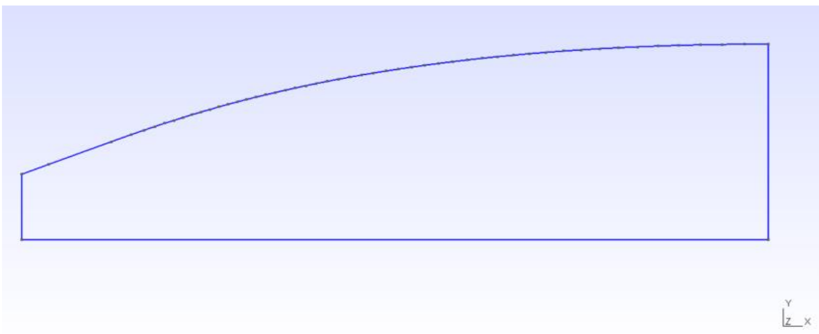


Figure 2.2. Geometry of nozzle for the internal flow

The geometry in Figure 2.3. is designed for the analysis of the flow out of the nozzle. There are two different inlets in this geometry. One is defined for nozzle outlet. The other is defined to indicate atmospheric conditions. In test case studies, the upper part is defined as Wall. This situation causes reflection of strong shock waves in some cases.

Apart from the test case studies, an extra analysis was made to observe how this situation affected the results when far field was defined instead of the part defined as wall. This topic has been discussed in the results and discussions section.
In order to better evaluate the results, this geometry is designed in much larger dimensions compared to the nozzle. Here, the length of the nozzle is represented as h, while the short side of the geometry is represented as 5h and the long side as 30h. Again, in this geometry, the symmetry feature was used in order to reduce the analysis time in half.

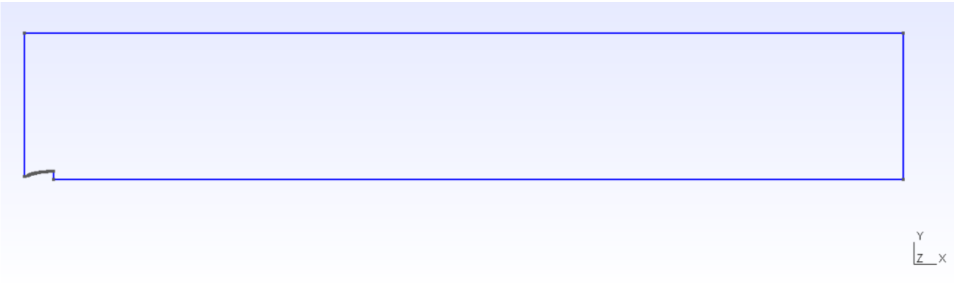


Figure 2.3. Geometry for the external flow

2.3. Mesh Generation

The flow in the nozzle is not affected by atmospheric conditions and has only one inlet. Therefore, the analysis here is simpler compared to the flow outside the nozzle. The mesh process was also made with this situation in mind. Mesh elements consist of triangles and element size is defined as 0.2.

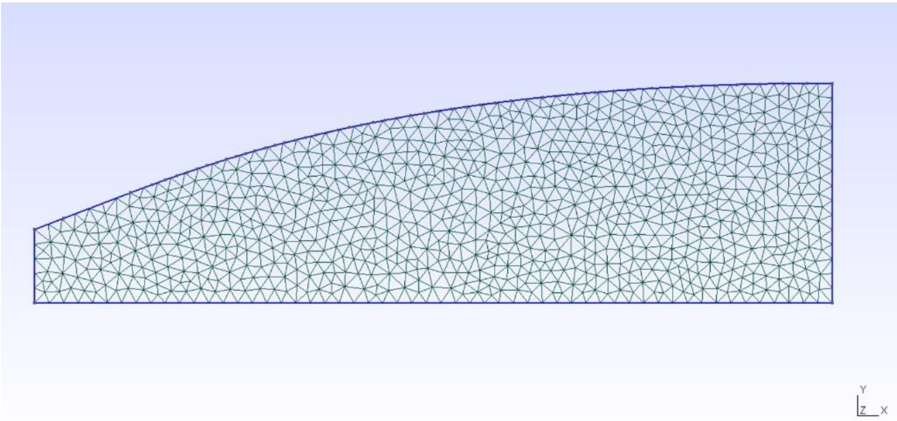


Figure 2.4. Meshed nozzle geometry

The flow outside the nozzle has a much more complex structure. When the vehicle is in motion, the air outside also has a velocity. In addition, atmospheric conditions such as pressure and temperature directly affect the flow. Due to this situation, the mesh structure created has a much finer structure. Mesh elements consist of triangles and element size is defined as 0.5. Here, the jet enters the geometry from the nozzle exit. Therefore, the important places are the exit of the nozzle. As it moves away from the nozzle outlet, the jet loses its force. Therefore, while there are very small elements around the axis of symmetry, relatively larger elements are located as they approach the wall. Here the growth rate is defined as 5.0.

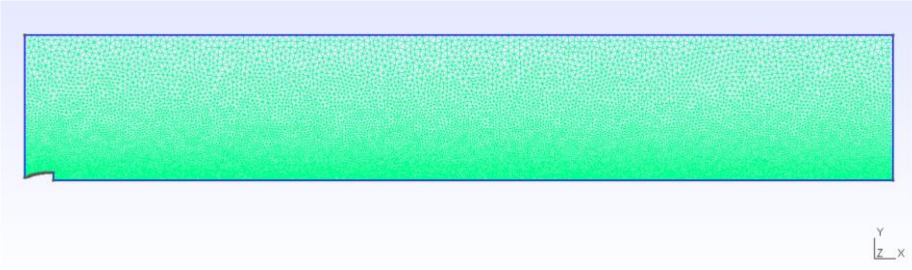


Figure 2.5. Meshed external flow geometry

2.4. Boundary Conditions and Test Cases

It was stated that the flow in the nozzle was not affected by atmospheric conditions. Therefore, the boundary conditions are directly dependent on the conditions in the combustion chamber. The nozzle and the combustion chamber are connected to each other by a structure called throat. The values in the combustion chamber and the throat are different from each other. Therefore, the flow entering the nozzle is not directly dependent on the conditions in the combustion chamber, but on its continuation, the throat. When Figure 2.1. is examined, parameters such as Po, P* and Pe are seen. Here, Po represents the pressure in the combustion chamber, while P* represents the pressure value in the throat, the flow entering the nozzle. In addition, Pe represents the nozzle outlet pressure. Nozzle output values are important and another boundary conditions parameters for this study. Because, in the second stage of the study, the analysis of the flow outside the nozzle was made. Factors such as pressure, temperature, Mach number coming out of the nozzle constitute the input conditions in this analysis.

Table 2.1. Boundary conditions

Parameter	Value	Unit
M_e	2.637	
P_0	7000000.000	Pa
P^*	3697972.514	Pa
P_e	331090.842	Pa
T_0	3600.000	K
T^*	3000.000	K
T_e	1505.525	K

It was stated that the flow outside the nozzle was dependent on atmospheric conditions. Therefore, three different altitudes were determined for the test case study. Temperature and pressure values at these altitudes are defined as free stream temperature and pressure values (see Table 2.2.).

Table 2.2. Pressure and temperature values for different heights

Height (meter)	Pressure (hPa)	Temperature (ºC)
0	1013.25	15
3012	700	-4.5
7185	400	-31.6

The number of test cases is kept high to show how much the jet flow depends on factors such as altitude, speed, pressure and temperature. First of all, three different altitudes were determined and Mach number was increased each time as the altitude increased. Then, in order to examine the factors such as pressure and temperature in more detail, the nozzle outlet pressure is assumed to be half, equal and twice the atmospheric pressure, respectively. The nozzle outlet temperature is assumed to be equal to and twice the atmospheric temperature. The most important factor affecting jet flow is pressure. Therefore, pressure values are kept more diverse compared to temperature values. In total, 18 test cases were studied and the results were examined and compared with each other. In addition, an extra study has been done to see the difference between Wall and far field. While strong shock waves reflect from the wall, it does not reflect when the headlight field is defined. This situation directly affects the jet flow. In addition, viscosity effects are neglected in this study.

Table 2.3. Test cases

Test Case	Height (meter)	Mach Number	Pressure	Temperature
1	0	0	atm x 0.5	atm x 1
2	0	0	atm x 1	atm x 1
3	0	0	atm x 2	atm x 1
4	0	0	atm x 0.5	atm x 2
5	0	0	atm x 1	atm x 2
6	0	0	atm x 2	atm x 2
7	3012	0.5	atm x 0.5	atm x 1
8	3012	0.5	atm x 1	atm x 1
9	3012	0.5	atm x 2	atm x 1
10	3012	0.5	atm x 0.5	atm x 2
11	3012	0.5	atm x 1	atm x 2
12	3012	0.5	atm x 2	atm x 2
13	7185	1.5	atm x 0.5	atm x 1
14	7185	1.5	atm x 1	atm x 1
15	7185	1.5	atm x 2	atm x 1

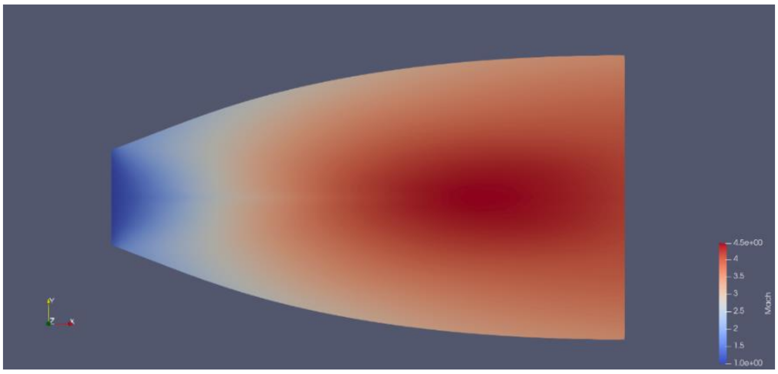
16	7185	1.5	atm x 0.5	atm x 2
17	7185	1.5	atm x 1	atm x 2
18	7185	1.5	atm x 2	atm x 2

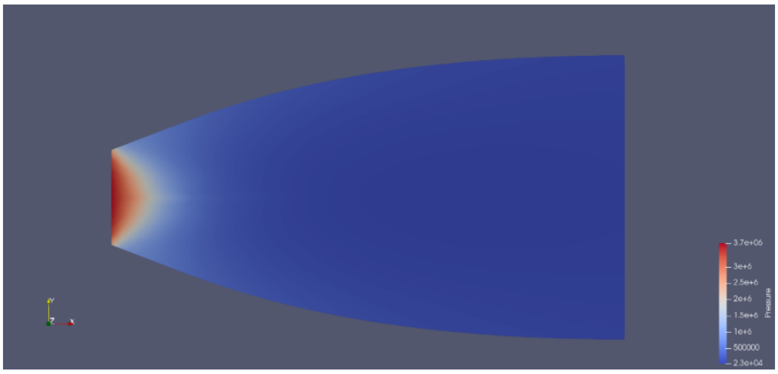
3. RESULT AND DISCUSSION

As mentioned before, jet flow was investigated for three different altitudes in this study. The first altitude was determined as sea level, that is 0 meters, and the speed was 0 Mach. The goal here is to simulate ground tests for a vehicle. The second altitude determined is 3012 meters. The speed of the vehicle at this altitude is determined as 0.5 Mach. In this way, the observation of the jet flow in subsonic flying vehicles was made. The third altitude was determined as 7185 meters. The speed of the vehicle was increased to 1.5 Mach. Thus, the observation of the jet flow in supersonic flight was made. Mach number, pressure and temperature of the jet stream were investigated in the studies. Stream lines were examined in order to better interpret the results and to better observe the vortex-like structures.

3.1. Analysis of Jet Flow Inside the Nozzle

The jet inside the nozzle is not affected by atmospheric conditions. Therefore, there is no need for any test case study for the nozzle. The reason that affects the flow here is the combustion chamber. The jet reacts depending on the temperature and pressure in the combustion chamber.





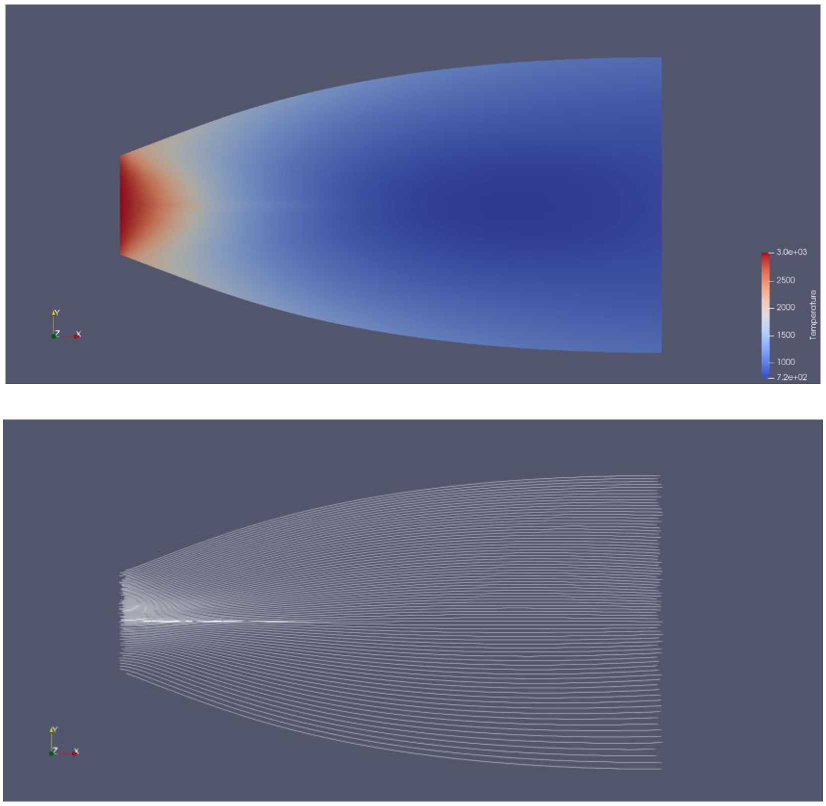


Figure 3.1. Analysis results of the jet flow in the nozzle

The jet has a mach number of 1 as it passes through the throat. The nozzle has an expanding design. Because of this design, the jet accelerates through the nozzle. There is no swirl structure or fluctuation in the flow inside the nozzle. Such structures are seen in atmospheric conditions.

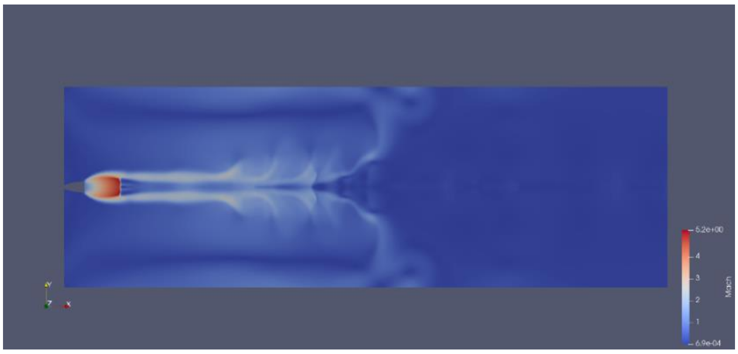
3.2. Analysis of the Jet Flow Outside the Nozzle

The jet outside the nozzle is affected by atmospheric conditions. In order to better examine the effect of atmospheric conditions on the jet, test case study was conducted in this section. This study was carried out at three different altitudes and three different speeds.

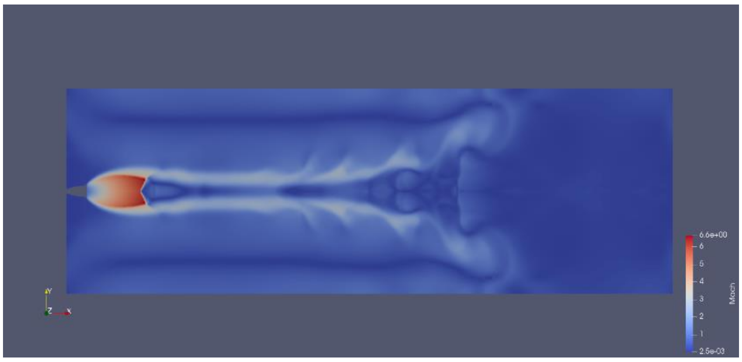
3.2.1. Analysis of 0 Meter and 0 Mach Jet Flow

In the first stage, test cases 1-6 were examined. In test cases 1-3, the nozzle outlet temperature was kept equal to the atmospheric temperature and the nozzle outlet pressures were accepted as half, equal and twice the atmospheric pressure, respectively. In test cases 4-6, the nozzle outlet temperature was accepted as twice the atmospheric temperature and the nozzle outlet pressures were accepted as half, equal and double the atmospheric pressure, respectively. The effects of pressure and temperature on the jet were investigated by varying the pressure and temperature values.

Pressure half and temperature equal

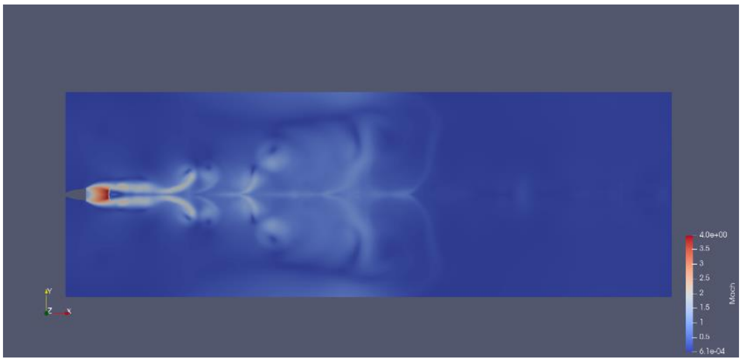


Pressure and temperature equal

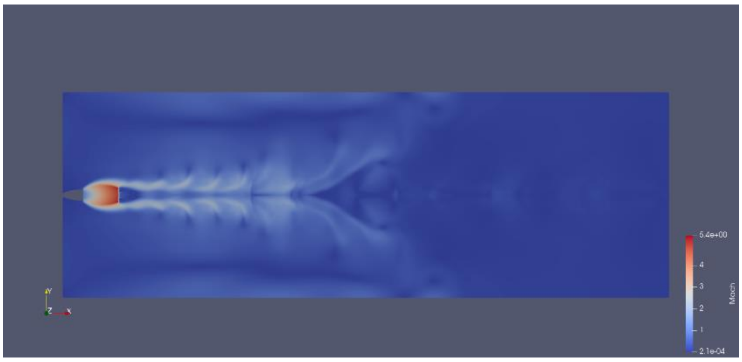


Pressure double and temperature equal

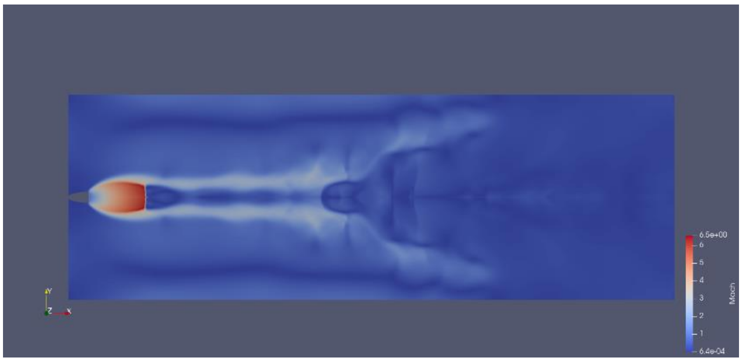
Figure 3.2. Mach number results for test cases 1-3



Pressure half and temperature double



Pressure equal and temperature double



Pressure and temperature double

Figure 3.3. Mach number results for test cases 4-6

It is observed that as the pressure increases, the Mach number increases and stronger normal shock waves are formed. With the increase in the Mach number, the jet reached longer distances and affected its surroundings more. When the results are examined, it is seen that the vortex structures formed with the increase in pressure become stronger. When the nozzle outlet pressure is twice the atmospheric pressure, the Mach number increased to 6.6 and a hypersonic flow was formed (see Figure 3.2.). While normal shock wave occurs in other cases, it is seen that an oblique shock wave occurs as a result of excessive acceleration of the jet in this case.

When the effect of temperature is examined, it cannot be said that it has a net effect like pressure. When the temperature increases, Mach number increases in some cases and decreases in some cases.

Pressure half and temperature equal

Pressure and temperature equal

Pressure double and temperature equal

Figure 3.4. Pressure results for test cases 1-3

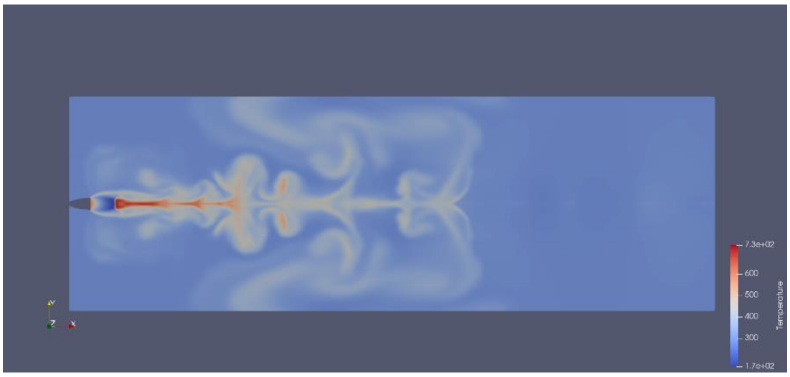
Pressure half and temperature double

Pressure equal and temperature double

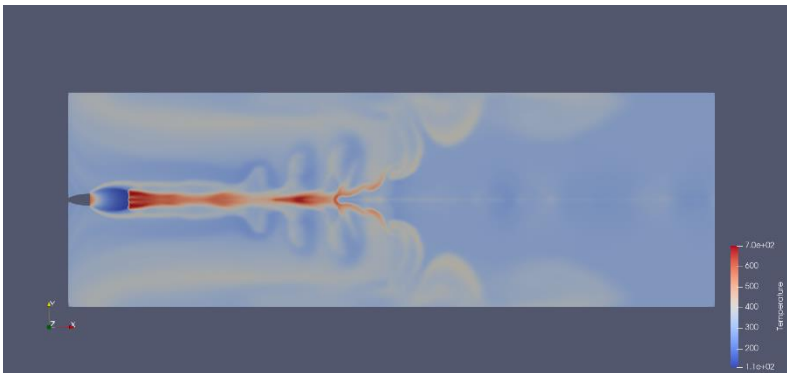
Pressure and temperature double

Figure 3.5. Pressure results for test cases 4-6

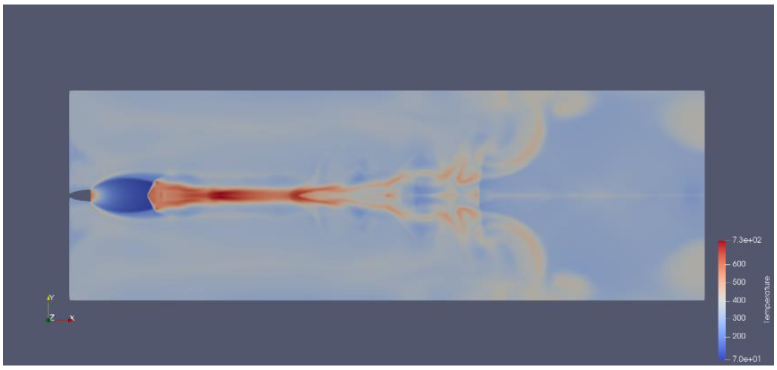
When the nozzle outlet pressure was increased, naturally the pressure in and around the flow also increased. Similarly, stronger shock waves occur. The jet coming out of the nozzle has high pressure, but after exiting the nozzle, the pressure drops quickly and the lowest pressure is observed. This pressure, which is at a very low level, rises abruptly with the shock wave effect. When the nozzle outlet temperature is increased, it does not cause any increase in the pressure contour, unlike the Mach number. While the maximum pressure does not change in some cases, it decreases in some cases.



Pressure half and temperature equal

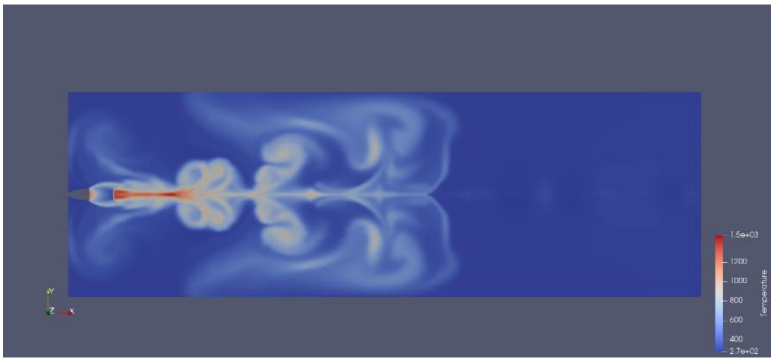


Pressure and temperature equal

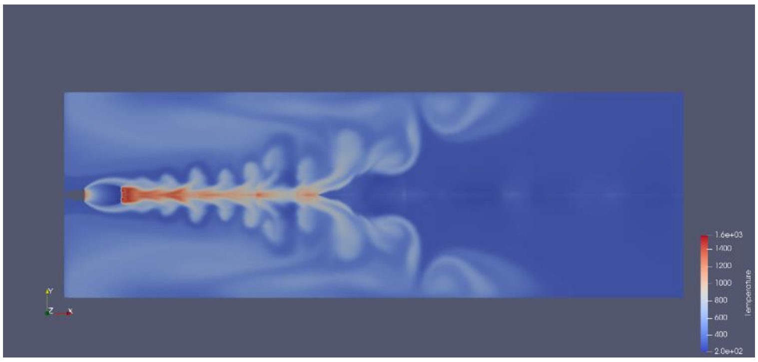


Pressure double and temperature equal

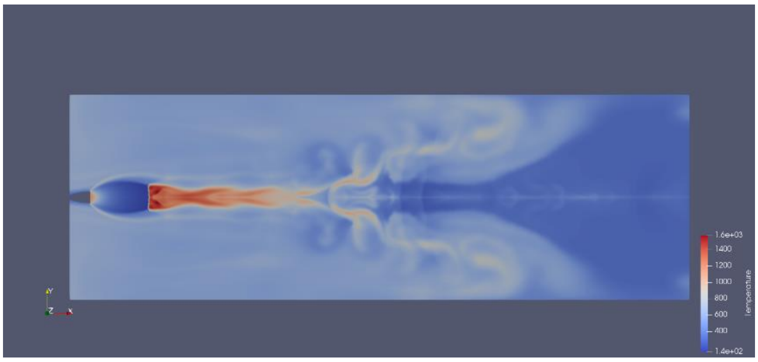
Figure 3.6. Temperature results for test cases 1-3



Pressure half and temperature double



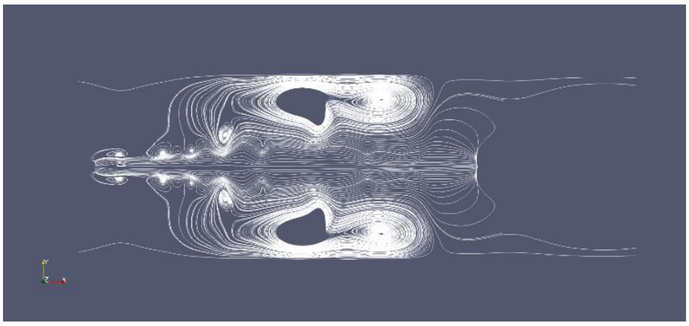
Pressure equal and temperature double



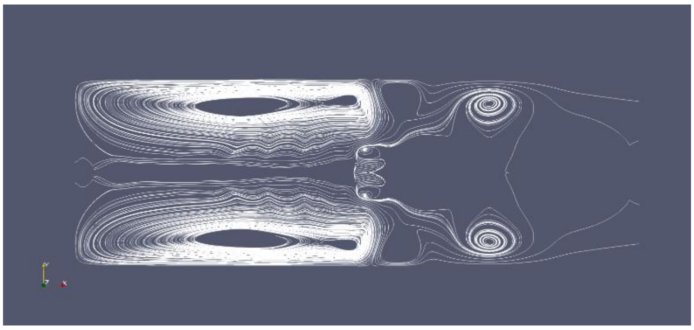
Pressure and temperature double

Figure 3.7. Temperature results for test cases 4-6

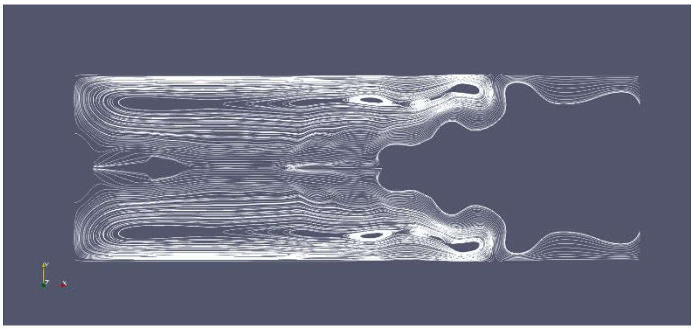
It is observed that when the nozzle outlet pressure is increased, much stronger shock waves are formed. In cases where the pressure is doubled, oblique shock waves may occur instead of the normal shock wave. In addition, when the results are examined, it is seen that the jet stream first thickens, then thins and then thickens again. This indicates the presence of shock diamonds. Increasing the pressure does not cause the maximum temperature to increase, but takes the jet to greater distances.



Pressure half and temperature equal

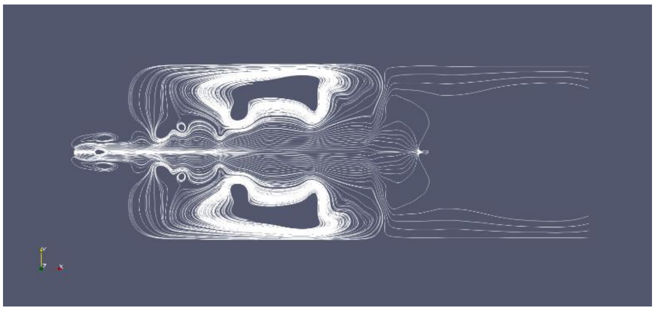


Pressure and temperature equal

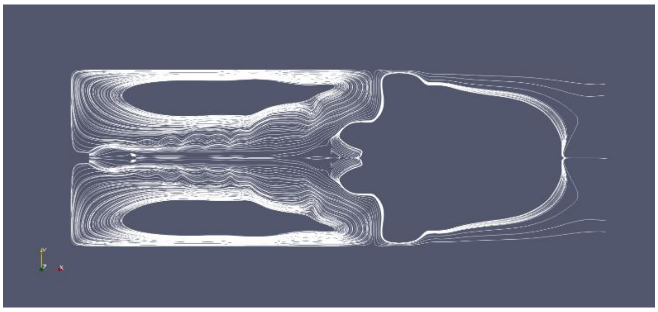


Pressure double and temperature equal

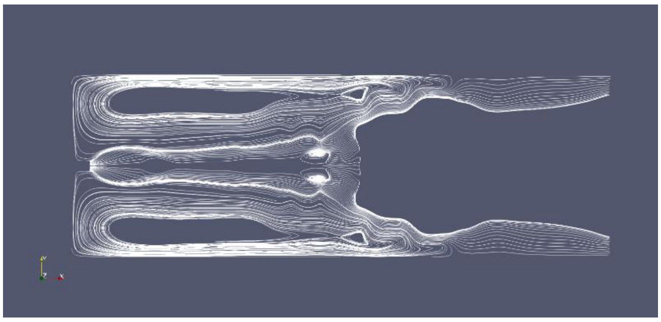
Figure 3.8. Streamlines for test cases 1-3



Pressure half and temperature double



Pressure equal and temperature double



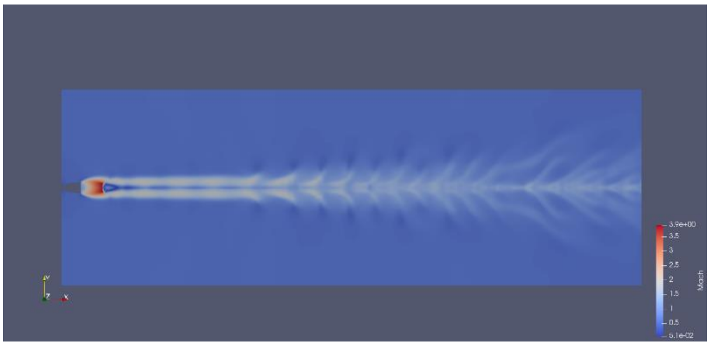
Pressure and temperature double

Figure 3.9. Streamlines for test cases 4-6

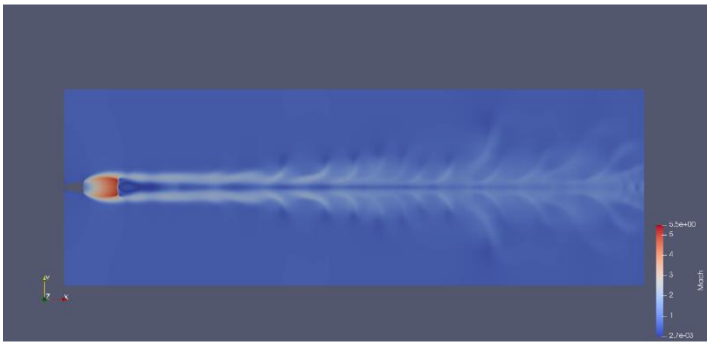
If the pressure is increased, larger vortices occur. If the temperature is increased, the vortex structures change. In addition, the fluctuations in the jet are clearly visible in the contours. As a result of the jet being affected by atmospheric conditions, the jet frequently expands and becomes compressed. These fluctuations occur as a result of the succession of these events.

3.2.2. Analysis of 3012 Meter and 0.5 Mach Jet Flow

In the second stage, test cases numbered 7-12 were examined. In test cases 7-9, the nozzle outlet temperature was kept equal to the atmospheric temperature and the nozzle outlet pressures were accepted as half, equal and twice the atmospheric pressure, respectively. In test cases numbered 10-12, the nozzle outlet temperature was accepted as twice the atmospheric temperature and the nozzle outlet pressures were accepted as half, equal and double the atmospheric pressure, respectively. The effects of pressure and temperature on the jet were investigated by varying the pressure and temperature values.



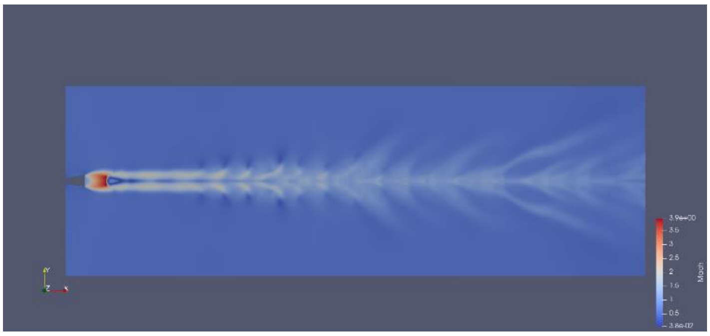
Pressure half and temperature equal



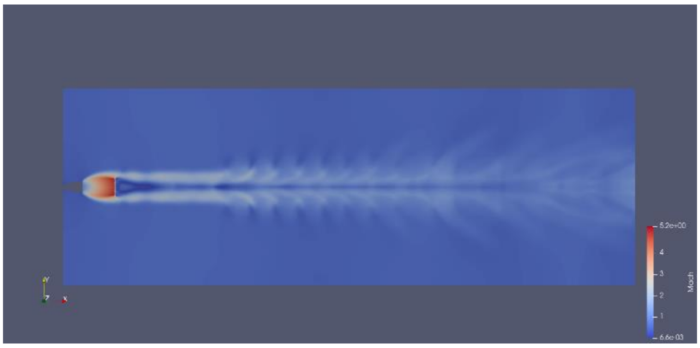
Pressure and temperature equal

Pressure double and temperature equal

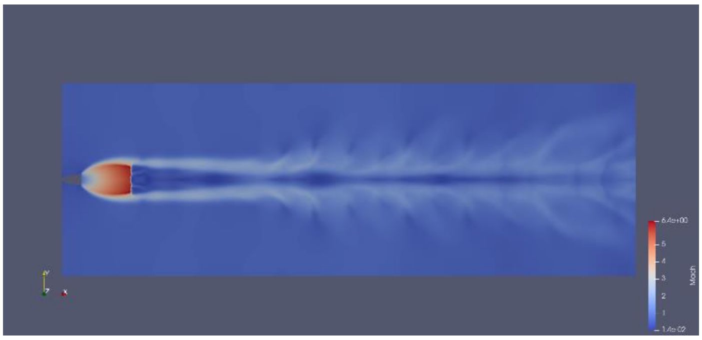
Figure 3.10. Mach number results for test cases 7-9



Pressure half and temperature double



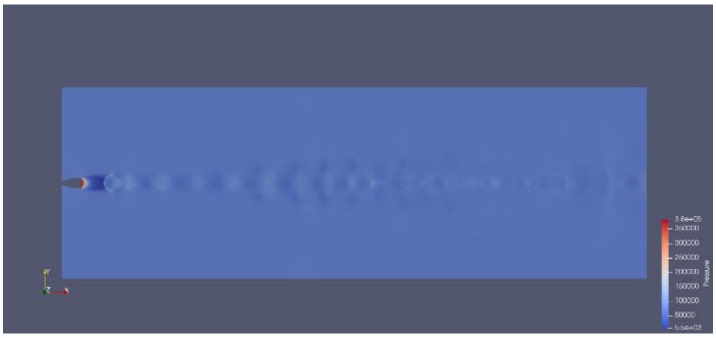
Pressure equal and temperature double



Pressure and temperature double

Figure 3.11. Mach number results for test cases 10-12

When the jet flow of the vehicle standing on the ground is compared with the jet flow of the vehicle moving at a speed of 0.5 Mach at 3012 meters, it is seen that the maximum Mach number decreases slightly as the altitude increases, although the vehicle also has speed. This is related to the decrease in atmospheric pressure as you go up. As the pressure decreases, the flow velocity of the jet also decreases. As in the first stage, the temperature does not have a clear effect on the flow, such as increasing or decreasing. In some cases, increasing the temperature increases the Mach number, while in some cases it decreases the Mach number.

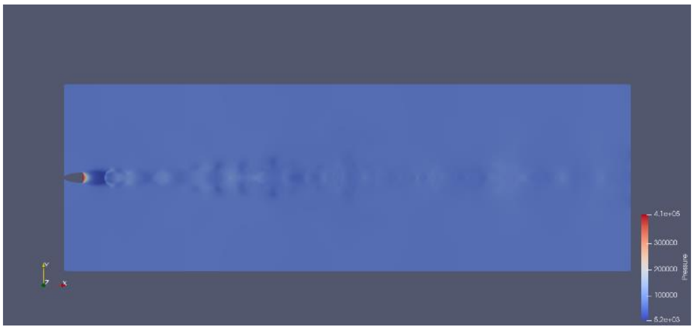


Pressure half and temperature equal

Pressure and temperature equal

Pressure double and temperature equal

Figure 3.12. Pressure results for test cases 7-9



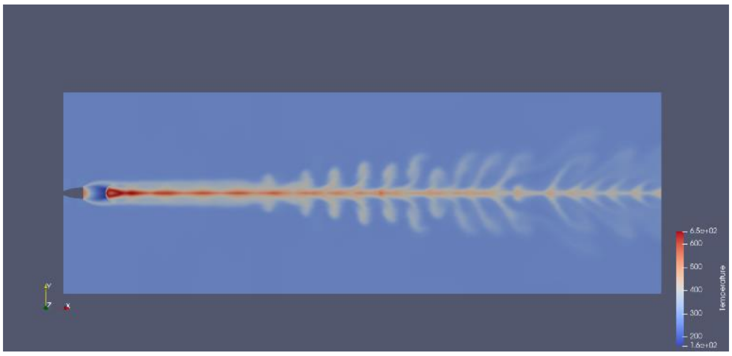
Pressure half and temperature double

Pressure equal and temperature double

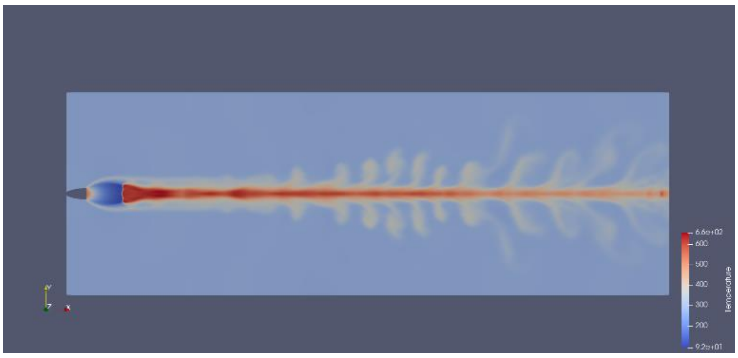
Pressure and temperature double

Figure 3.13. Pressure results for test cases 10-12

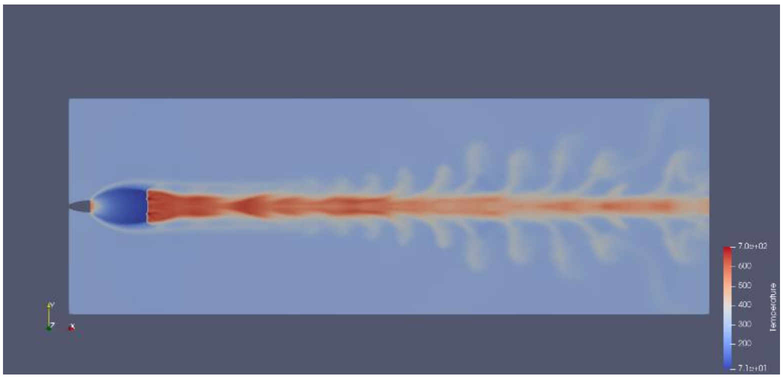
In a stationary environment, the jet scatters around. However, while the vehicle is in motion, the air around it also has a velocity. In this case, the jet cannot disperse as much as in a stationary environment. It moves in the direction of the exit. In test cases 1-6, the jet changes the conditions around it, while in cases 7-12 it changes the conditions more in the direction of the outlet. When Figure 3.12 is examined, shock diamonds can be seen clearly. Especially when the nozzle outlet pressure is lower than the atmospheric pressure, more shock diamond structures are formed. It was also stated that as the altitude increased, the atmospheric pressure decreased. This situation appears numerically in pressure charts (see Table 2.2.).



Pressure half and temperature equal

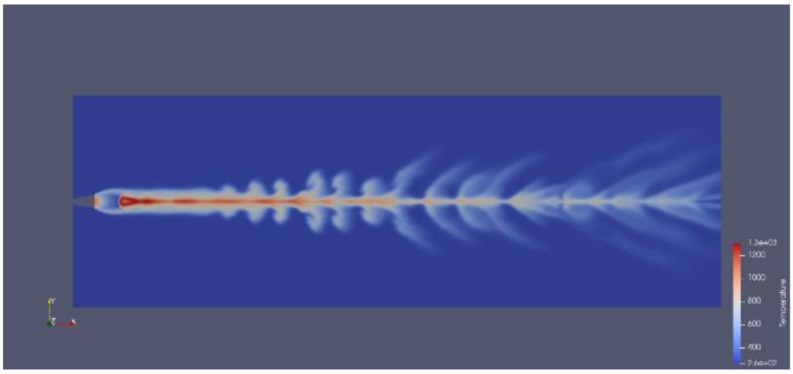


Pressure and temperature equal

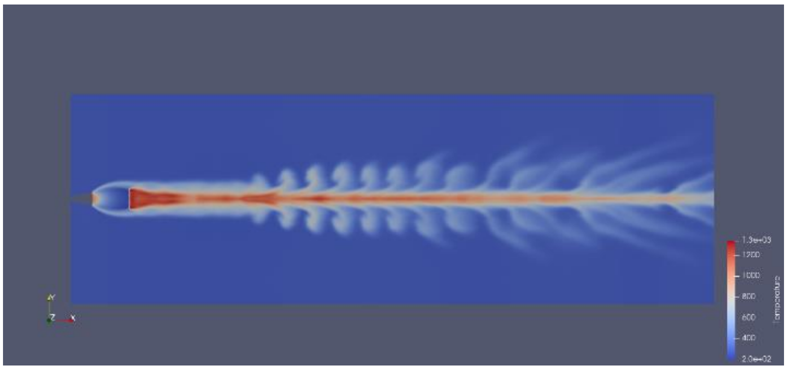


Pressure double and temperature equal

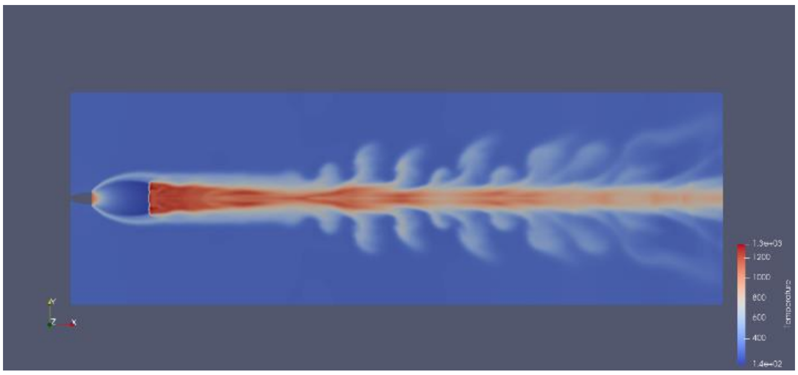
Figure 3.14. Temperature results for test cases 7-9



Pressure half and temperature double



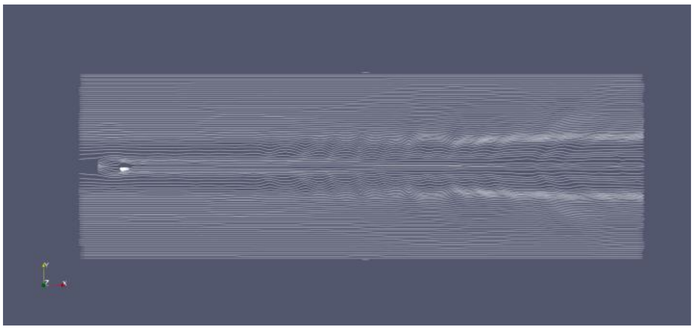
Pressure equal and temperature double



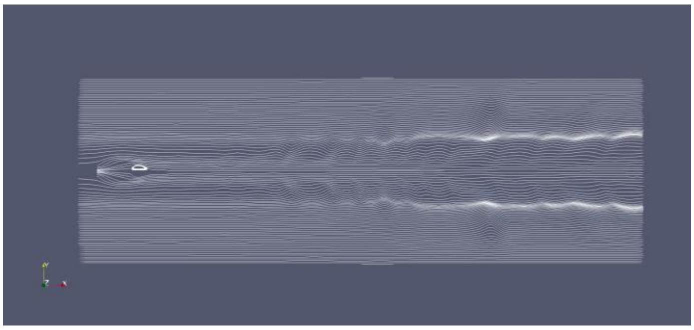
Pressure and temperature double

Figure 3.15. Temperature results for test cases 10-12

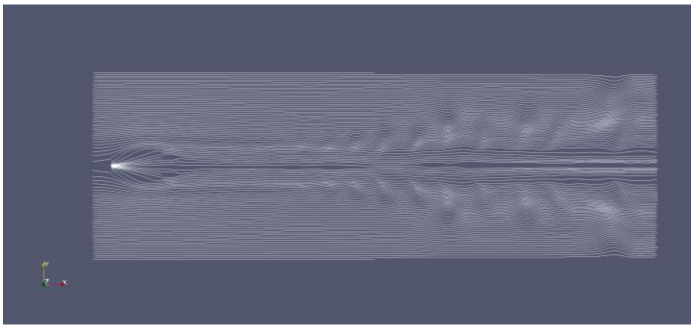
Increasing the pressure causes the environment around the jet to heat up. Looking at the points where the temperature is highest in the results, it is seen that the temperature reaches the highest level where the shock wave occurs. A lower temperature is observed at the nozzle outlet compared to where the shock wave occurs. After the shock wave, the jet loses its heat and continues its movement. At 3012 meters, the temperature is below 0 degrees Celsius. Therefore, a rapid heat transfer takes place between the jet and the atmosphere.



Pressure half and temperature equal

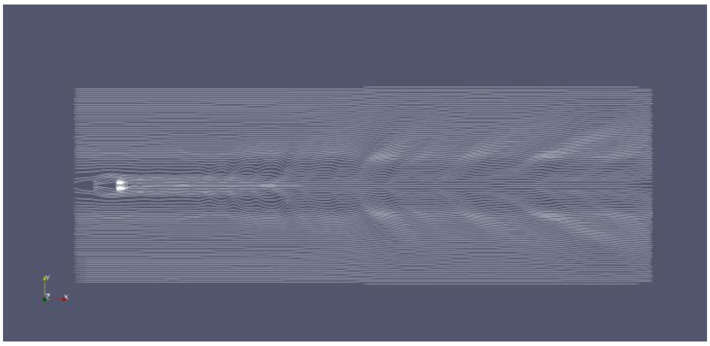


Pressure and temperature equal

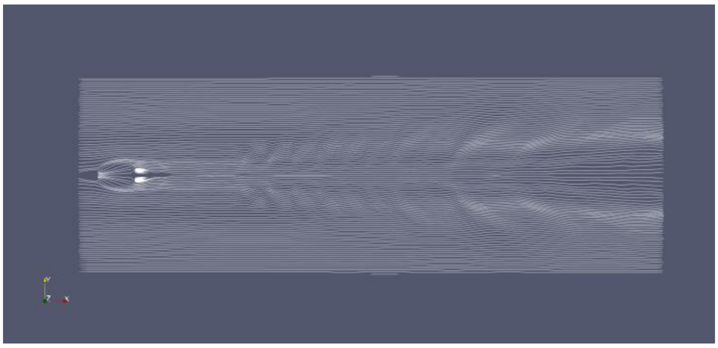


Pressure double and temperature equal

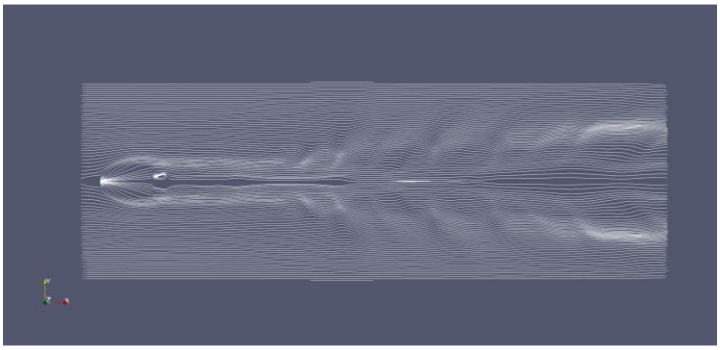
Figure 3.16. Streamlines for test cases 7-9



Pressure half and temperature double



Pressure equal and temperature double



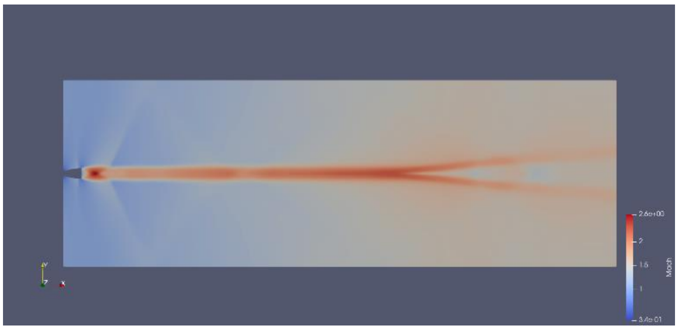
Pressure and temperature double

Figure 3.17. Streamlines for test cases 10-12

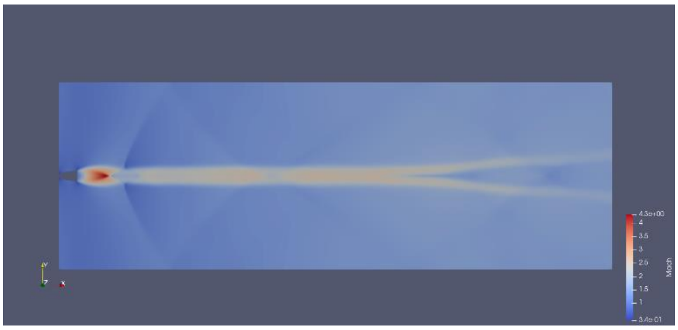
When the stream lines are examined, it is seen that the vortices formed at 0 meters do not occur at a height of 3012 meters. This is because there is also a flow in the air due to the speed of the vehicle. The velocity of the air in these cases is 0.5 Mach. This level of speed does not allow for eddy structures. The vortex structure is seen only in the parts where shock waves occur. In addition, when looking at the stream lines, the fluctuation in the jet caused by compression and expansion is clearly seen.

3.2.3. Analysis of 7185 Meter and 1.5 Mach Jet Flow

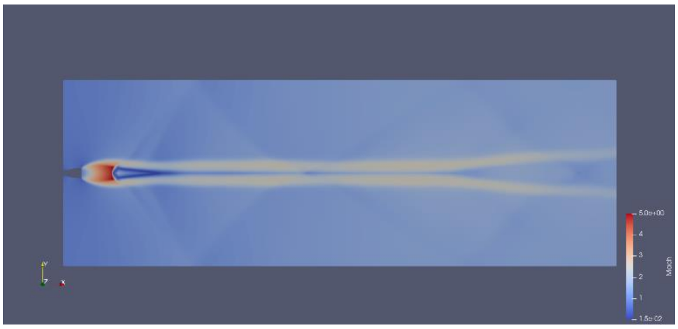
In the third stage, test cases 13-18 were examined. In test cases 13-15, the nozzle outlet temperature was kept equal to the atmospheric temperature and the nozzle outlet pressures were accepted as half, equal and twice the atmospheric pressure, respectively. In the 16-18 test cases, the nozzle outlet temperature was accepted as twice the atmospheric temperature and the nozzle outlet pressures were accepted as half, equal and double the atmospheric pressure, respectively. The effects of pressure and temperature on the jet were investigated by varying the pressure and temperature values.



Pressure half and temperature equal



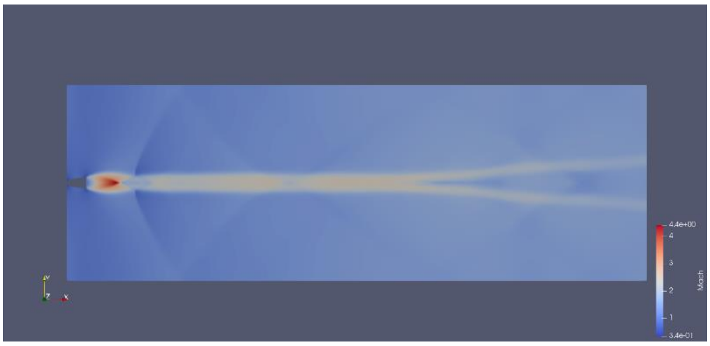
Pressure and temperature equal



Pressure double and temperature equal

Figure 3.18. Mach number results for test cases 13-15

Pressure half and temperature double



Pressure equal and temperature double

Pressure and temperature double

Figure 3.19. Mach number results for test cases 16-18

Normal shock waves at high altitude and high speed are not seen. Instead, there are very strong oblique shock waves. These oblique shock waves reach up to the wall and are reflected from the wall. These waves are so strong that they reflect completely from the wall and create a shock diamond image. Although we increase the speed of the vehicle to 1.5 Mach at this altitude, the maximum speed that the jet can reach is lower than other altitudes due to atmospheric conditions.

Pressure half and temperature equal

Pressure and temperature equal

Pressure double and temperature equal

Figure 3.20. Pressure results for test cases 13-15

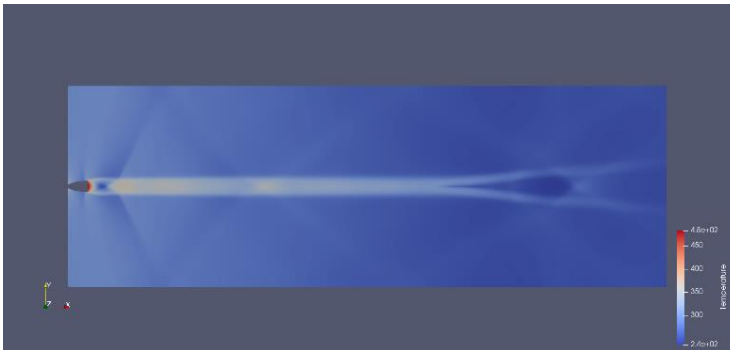
Pressure half and temperature double

Pressure equal and temperature double

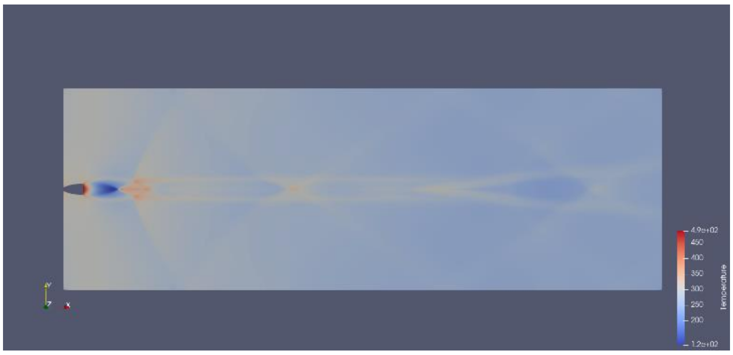
Pressure and temperature double

Figure 3.21. Pressure results for test cases 16-18

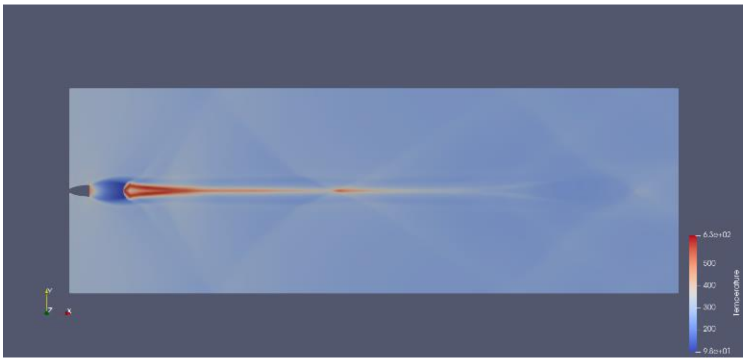
Since the air has a speed of 1.5 Mach and the exit speed of the jet from the nozzle is 2.637 Mach, serious pressure differences are not seen. The jet coming out of the nozzle has high pressure, but when it encounters the atmosphere, it suddenly reaches very low values. Then, this pressure rises again with the effect of shock waves. In the pressure graphs here, pressure changes due to strong oblique shocks and their reflections are observed.



Pressure half and temperature equal

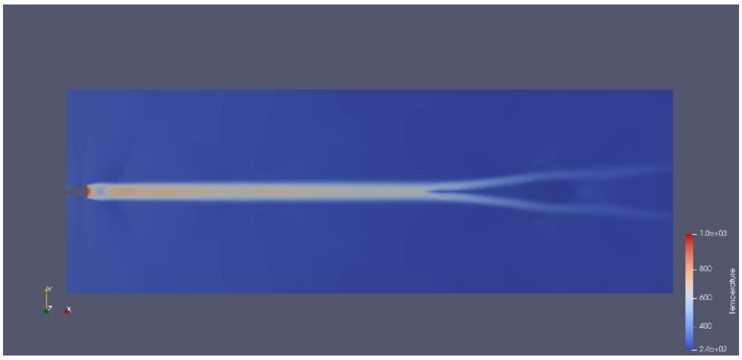


Pressure and temperature equal

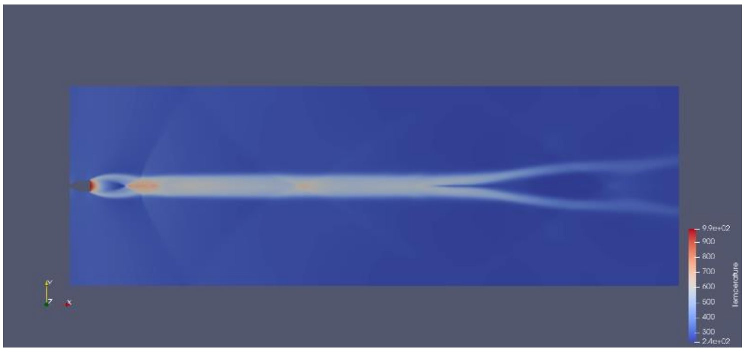


Pressure double and temperature equal

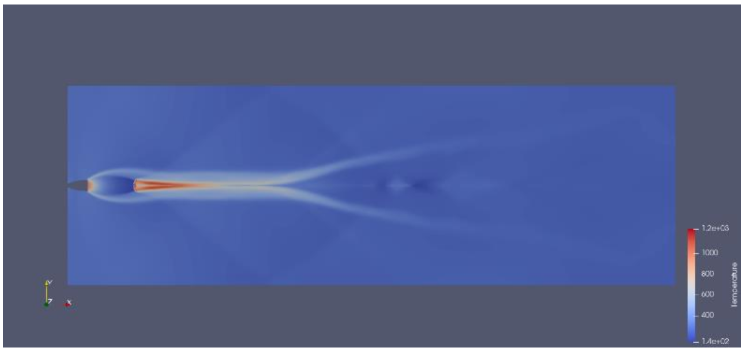
Figure 3.22. Temperature results for test cases 13-15



Pressure half and temperature double



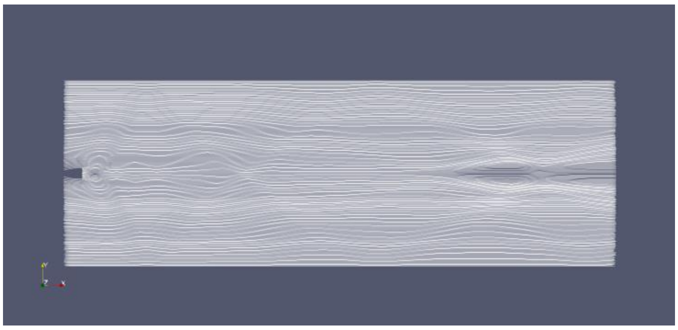
Pressure equal and temperature double



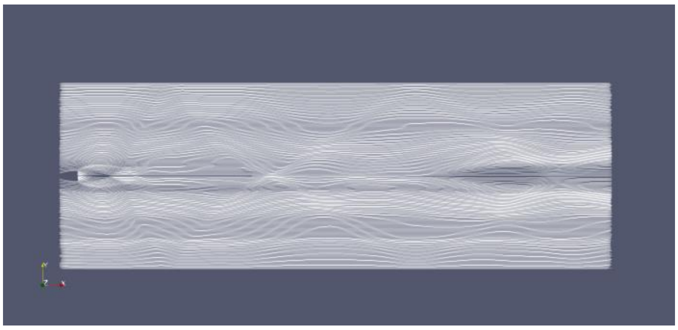
Pressure and temperature double

Figure 3.23. Temperature results for test cases 16-18

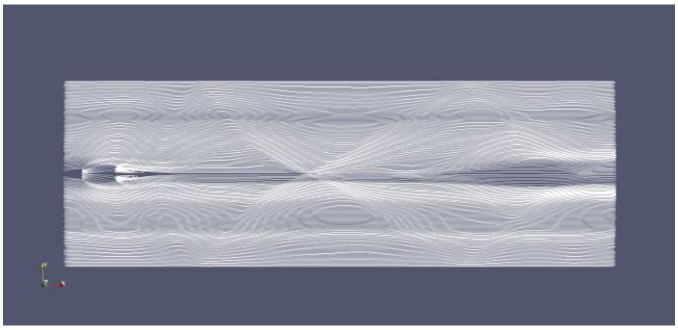
When the results are examined, it is seen that there is a serious increase in temperature such as 200 K only when the nozzle outlet pressure is twice the atmospheric pressure. A slight increase or decrease is observed when the nozzle outlet pressure is equal to or equal to half the atmospheric pressure. Similar effects are seen in pressure and temperature increase as in other cases.



Pressure half and temperature equal

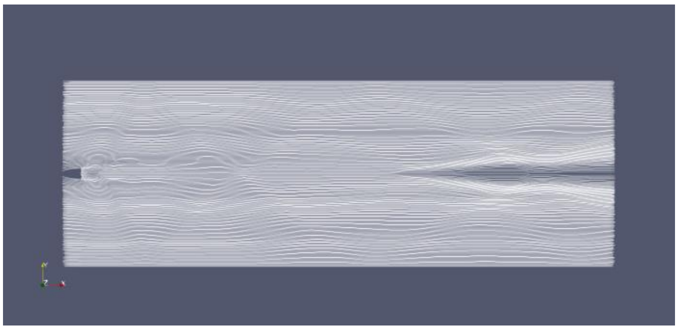


Pressure and temperature equal

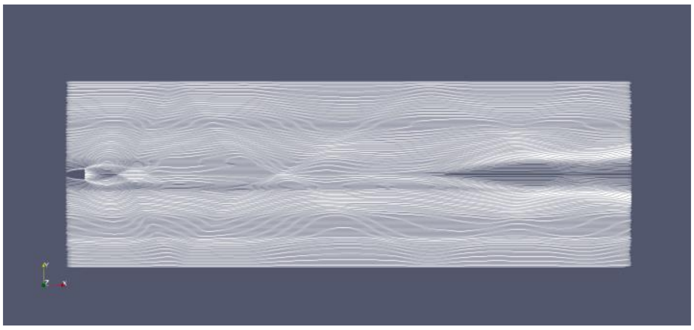


Pressure double and temperature equal

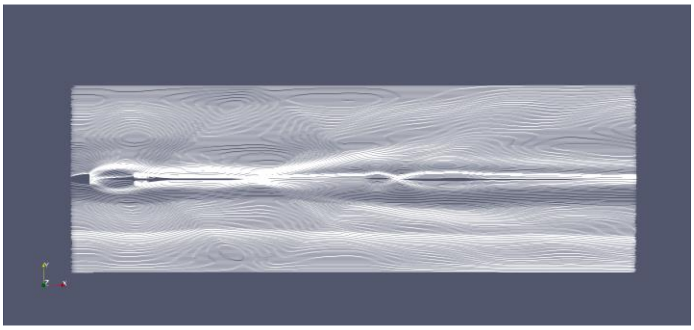
Figure 3.24. Streamlines for test cases 13-15



Pressure half and temperature double



Pressure equal and temperature double



Pressure and temperature double

Figure 3.25. Streamlines for test cases 16-18

The speed of the air has been increased to 1.5 mach. This situation causes no serious interaction between the jet and the air flow as in previous studies. Both types of flow are supersonic flow. It is seen that as the pressure increases, more vortices are formed.

3.2.4. Difference Between Wall and Far Field

Wall and far field definitions in jet flow analysis create serious differences in some cases. For example, very strong oblique shock waves are seen in the jet stream of a vehicle with a speed of 1.5 Mach at an altitude of 7185 meters. When the edges are defined as wall in the analysis geometry, strong oblique shock waves reflect from the wall. This has serious effects on the jet. Strong shock waves do not reflect when edges are defined as far field instead of wall. If the analysis will be done for a jet in the open air, defining the far field, if it will be done in a wind tunnel, defining the Wall will make the result more accurate.

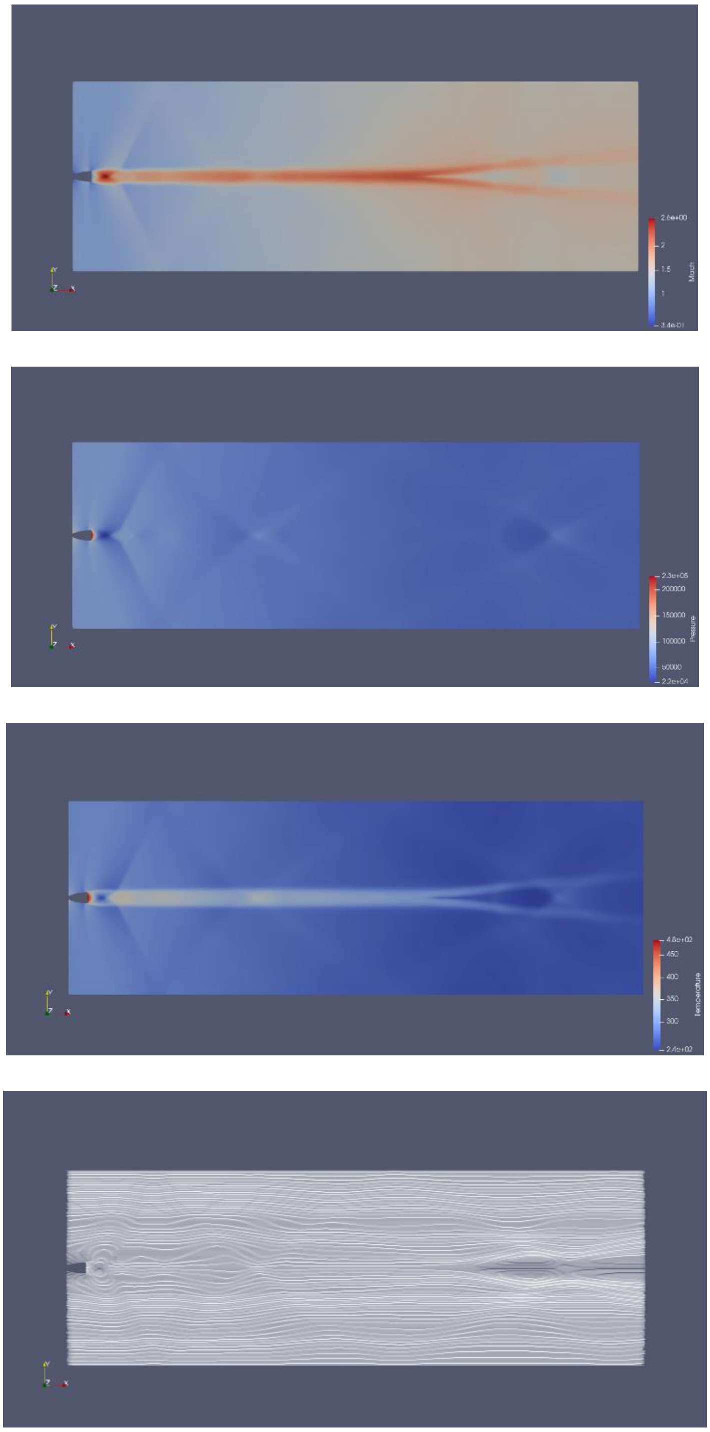


Figure 3.26. Wall defined analysis results

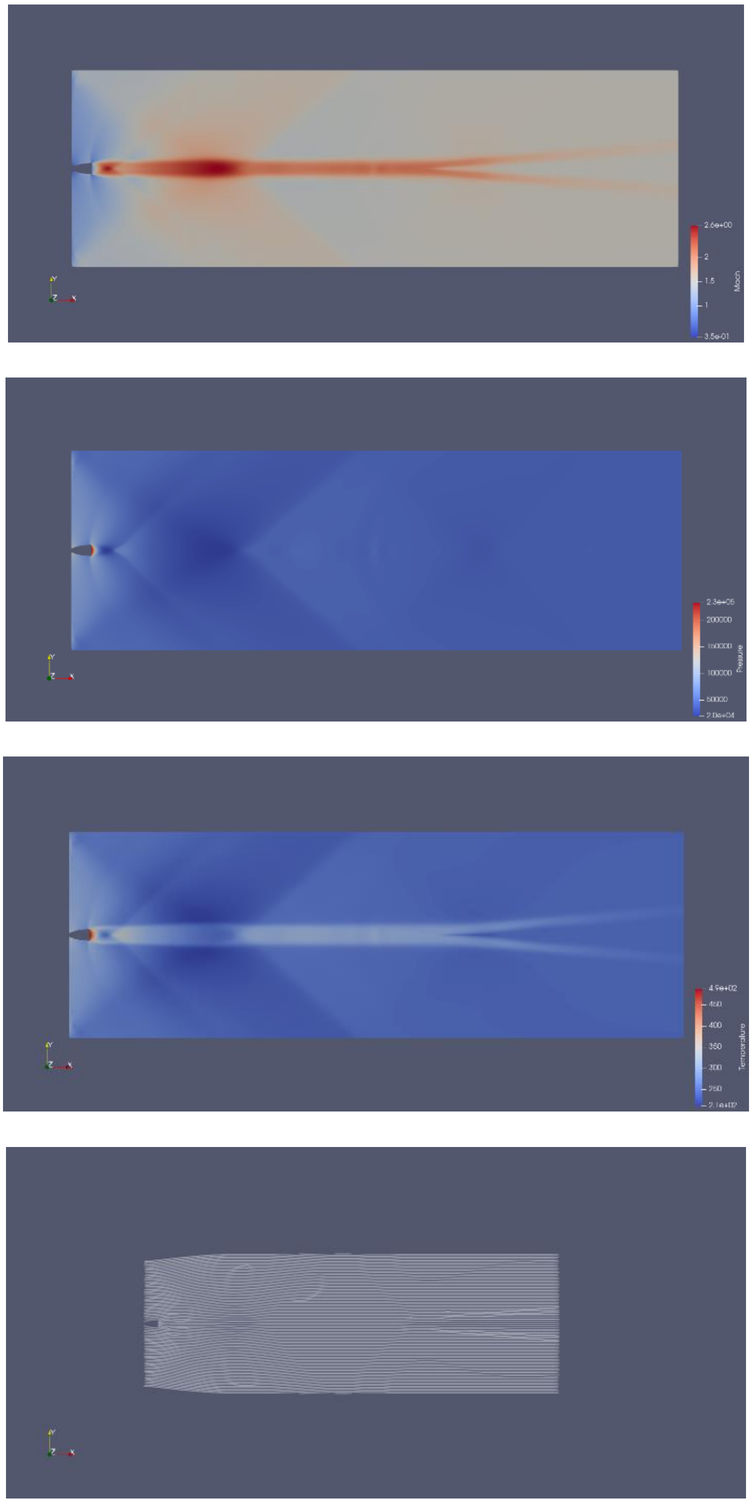


Figure 3.27. Far field defined analysis results

When the figures are examined, it is seen that there are serious differences in the results. When the far field is defined due to reflection, it is seen that the temperature and pressure are lower in some regions. In the Mach number, the situation is the opposite. When the far field is defined, the Mach number increases in some regions. This is due to the Mach number being inversely proportional to pressure and temperature. When shock waves occur, the Mach number drops abruptly, while the temperature and pressure increase. When the stream lines are examined, a more regular flow is seen as a result of the analysis defined by the far field compared to the analysis result defined by the wall.

4. CONCLUSION

In the presented study, the analysis of jet flow was taken into account by using computational fluid dynamics. Jet flow takes place inside and outside the nozzle in two different environments. No structure (shock wave, vortex, etc.) is seen in the flow inside the nozzle. A smooth flow takes place here. In the flow outside the nozzle, these types of structures are formed and varied according to parameters such as height, Mach number, pressure and temperature. Three different heights were used in this study. In addition, the effect of pressure and temperature on the jet was investigated by changing the nozzle outlet pressure and nozzle outlet temperature. As altitude increases, atmospheric pressure and atmospheric temperature decrease. The decrease in pressure causes the Mach number to decrease. Therefore, as the altitude increases, the maximum speed that the jet can reach decreases. The decrease in pressure also causes the ambient temperature to decrease. With the increase in pressure, the jet accelerates and shows its effect to farther points. Meanwhile, the temperature of the environment increases. If we want the jet to accelerate, that is, the acceleration of the vehicle, we need to increase the nozzle outlet pressure. For this, changes should be made in the nozzle design. In addition, the increase in pressure changes the structure of the shock waves. As the pressure increases, stronger normal shock waves are formed. If the pressure is increased too much, these normal shock waves can turn into oblique shock waves.

In this study, the flow is assumed to be inviscid. In this assumption, the Navier-Stokes equation is used in flow analysis. When the equations are examined, it is seen that the temperature is not directly included, but there is density in the equation. Increasing the temperature causes the volume to increase and the density to change. Temperature affects indirectly, but the Mach number is kept constant in the equations used for boundary conditions. This eliminates the effect of temperature. If the Mach number were not constant, the temperature change would have an effect on the flow. In test case studies, it has been observed that temperature changes have little or no effect on the jet. It is quite possible that there are reasons such as meshing operation and rounding error behind this little effect. As a result, it was concluded that there is no effect of temperature on jet flow.

It is seen that a jet in a still environment creates too many vortices. In addition, low pressure increases the number of eddies. This is explained by the pressure-velocity relationship. An increase in pressure causes an increase in the Mach number. The increased velocity prevents the flow from forming vortices. In the test case study, the height and the speed of the vehicle were increased gradually. The speed of the vehicle moves the air in the atmosphere. For example, if the vehicle has a speed of 1 Mach, the air in the atmosphere also has a speed of 1 Mach. Studies show that the number of eddies created by a moving vehicle is quite low. The eddies formed are seen where strong shock waves occur. If a very weak shock wave occurs, then no eddies are formed.

The higher the number of eddies, the higher the noise. To reduce the noise, it is necessary to reduce the number of vortices. For this, the nozzle outlet pressure can be reduced.

REFERENCES

Pandey, K.M., Kumar, V. (2010) CFD Analysis of Four Jet Flow at Mach 1.74 with Fluent Software. International Journal of Chemical Engineering and Applications, 1(1), 302-308.

Hossain, M.A., (2013) Numerical Investigation of Fluid Flow Through A 2D Backward Facing Step Channel. International Journal of Engineering Research & Technology, 2(10), 3700-3708ç

Anderson, J.D., (2002). Modern Compressible Flow: With Historical Perspective, 3rd Edition., McGraw-Hill Education, New York, USA.

Dhamankar, N., Blaisdell, G.A., Lyrintzis, A., (2016). Analysis of Turbulent Jet Flow and Associated Noise with Round and Chevron Nozzles using Large Eddy Simulation. Aeroacoustics Conferences, 30 May – 1 June, 2016, Lyon, France.

Bayraktar, S. (2008) Çapraz Akıştaki Türbülanslı Jet Akışlarının Deneysel ve Hesaplamalı Akışkanlar Dinamiği Analizi. Doktora Tezi, Yıldız Teknik Üniversitesi Fen Bilimleri Enstitüsü, İstanbul, Türkiye.

Compressible Aerodynamics Index, Normal Shock Wave Equations, https://www.grc.nasa.gov/.

Oblique Shockwaves, https://curiosityfluids.com/, (2016.04.08).

Shock Diamonds, https://www.universetoday.com/.

Jet (Fluid), https://en.wikipedia.org/.

Jet Engine, https://en.wikipedia.org/.

Shock Wave, https://en.wikipedia.org/.

Oblique Shock, https://en.wikipedia.org/.

Vortex, https://en.wikipedia.org/.