ESCUELA TÉCNICA SUPERIOR DE INGENIEROS INDUSTRIALES Y DE TELECOMUNICACIÓN

Titulación :

INGENIERO INDUSTRIAL

Título del proyecto:

OPTIMIZING CONVENTIONAL AND UNCONVENTIONAL NOZZLES FOR INCREASING THE THRUST

Diego Moreu Otal

Pablo Sanchís Gurpide

Pamplona, 22 de Julio del 2010

Author’s Declaration

Statement 1
This work has not previously been presented in any form to Glyndŵr University or at any other institutional body whether for assessment or for any other purposes. Save for any express acknowledgements, references, and/or bibliographies cited in the work, I confirm that the intellectual content of the work is the result of my own efforts and no other person.

Statement 2
It is acknowledged that the author of this work shall own the copyright. However, by submitting this copyright work for assessment, the author grants to the University a perpetual royalty-free license to do all or any of those things referred to in section 16(I) of the copyright, designs, and patents act 1988 (viz: to copy work; to issue copies to the public; to perform or show or play the work in public, to broadcast the work or make an adaptation of the work).

Signed:____.

Date:____.

ABSTRACT

This project arises from the fact nowadays the engineering process is becoming more and more experimental, so the knowledge on design software is very important in order to save money and time.

Under these circumstances, new mathematical methods are being applied to design softwares to develop new tools for solving a huge number of design analyses.

In this particular case, the study will be done with Fluent, a software based on Computational Fluid Dynamics, CFD, which fundamental basis are the Navier-Stokes equations, that define any single-phase fluid flow. This software will perform a 3 dimensional analysis of conventional and unconventional nozzles which will be designed in Gambit, software that allows building the geometry and mesh of the models that later will be analyzed in Fluent.

The final purpose of this project is to optimize the thrust produced by several designs of conventional and unconventional nozzles, compare its performance characteristics and understand and be aware of the effects the heat transfer has over the nozzles; and the needed of cooling techniques to minimize it.

ACKNOWLEDGEMENTS

The acknowledgement of this project must be dedicated to all the people who have helped and support me.

First of all, to my supervisor Mr Md. Salim Miah for his dedication in this project and all his knowledge, that has been very helpful to me.

I want also to thank to Mr Shafiul Monir for his help in the issues related to the software work and his understanding.

TABLE OF CONTENTS

CONTENTS OF FIGURES:

1. INTRODUCTION

The nozzle of a rocket engine is a carefully shaped aft portion of the thrust chamber that controls the expansion of the exhaust gas so that the thermal energy of combustion is effectively converted into kinetic energy in order to propel the rocket. The nozzle is the biggest component of the rocket engine and has a very important influence on the overall engine performance and represents a large fraction of the engine structure.

The computation of rocket nozzle performance is of critical importance to the rocket engine industry. Sometimes, the rocket engine manufacturer needs to compare various engine operating systems before deciding on a specific configuration. There are two aspects of critical importance, the nozzle performance and the heat transfer characteristics. These two parameters are interrelated, mainly in those cases involving cooling techniques. Under this circumstances, the actively cooled boundary layers serves to thermally protect the nozzle wall from the exhaust gases of the rocket engine core flow field. So the coolant permits the rocket engine to operate at conditions otherwise not available, such as high chamber pressures.

The rocket engine manufacturer has to compare as well between several configurations of conventional and unconventional nozzles depending on the rocket performance desired. He also needs to reach a compromise between the thrust produced by each different design and the heat transfer that is possible evacuate from it, because unfortunately, the most thrust required the higher heat transfer will occur through the wall.

Computational Fluid Dynamics can be used in order to obtain this compromise between thrust and heat transfer. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze different kind of problems that involve fluid flows. This software gives you the power to simulate flows, heat and mass transfer, moving bodies, etc, through computer modeling. CFD const of Fluent, which is the software of simulation of computational fluid dynamics most used in the world, and Gambit, which is Fluent's software for geometry and mesh generation, although the geometry is possible to import it from other softwares like CAD, CATIA, Solid works, etc.

The purpose of the analysis is to explore the abilities of CFD and design the best nozzles as possible to get maximum parameters of thrust from conventional and unconventional nozzles, studying and understanding the effect heat transfer has in their performance.

Finally, the results of the analyses will be used to compare advantages and disadvantages in performance between conventional and unconventional nozzles.

2. AIMS OF PROJECT

The main objective of this project is to develop a complete analysis in performance with Fluent of several 3 dimensional conventional and unconventional nozzles which will be previously designed in Gambit. Those configurations will be conical and bell-shaped as conventional nozzles and linear aerospike and expansion-deflection nozzle as unconventional nozzles. In the picture below two of these designs are shown.

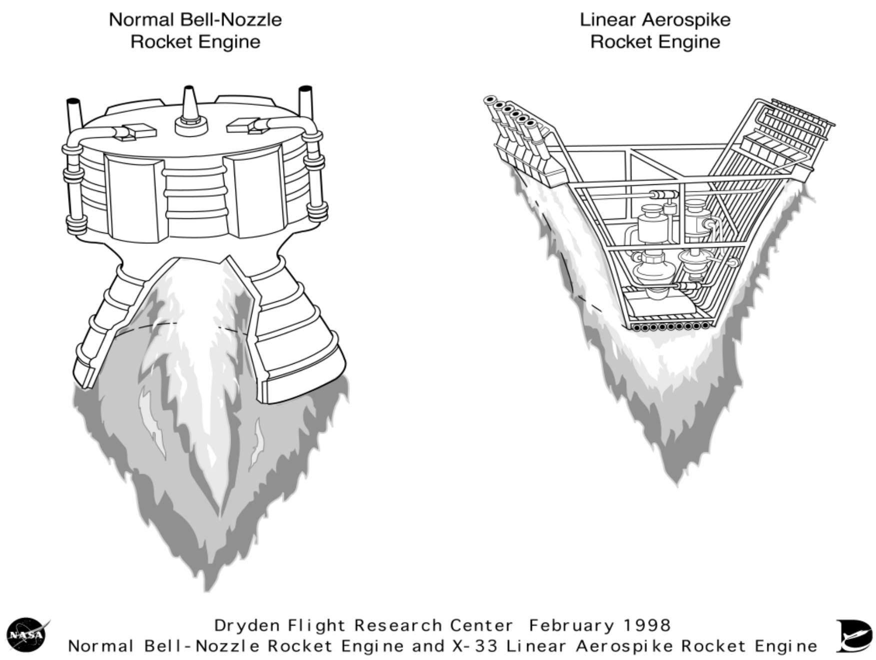


Fig 2.1: Bell-shaped and linear aerospike nozzles. [1]

Another important target will be study, be aware and understand the problems and effects the heat transmission has in the performance of rocket engines, mainly in their chamber and nozzle throat. And find out the different cooling techniques available nowadays to minimize these effects.

In order to analyze different nozzles, gaining knowledge in CFD and familiarizing with the software will be a very important goal as well. I will have to learn designing and meshing techniques with Gambit and how to analyze and get the desired results of these designs with Fluent.

With the help of these softwares, each one of all the configurations will be redesigned by changing its initial contour and parameters in order to maximize their thrust.

Finally, with all the results obtained from each design a comparison in performance between the designs will be done, and the advantages and disadvantages between conventional and unconventional nozzles will be shown.

As personal experience, this project has helped me to go deeper into the design of elements from a different angle to the common. It means, not only the design of a concrete nozzle, it has been the design with a view to its use and functionality. So I have to develop skills to analyze and design the nozzles from the point of view of knowing that this element should fit in a complete structure, bearing in mind all kinds of restrictions.

3. REVIEW OF LITERATURE

3.1. INTRODUCTION

During the human history, men have been figuring out many methods, some quite surprising, to travel from the Earth to other planets. Sometimes these methods were based on physical principles he thought he knew well but which in practice could never have been successful and other were simply ingenious products of his imagination that did not respond to the laws of nature.

Of all the ways of transport envisioned, the only one which would be viable for space flight would be the one propel by rockets, machine that was already known from a very long time, but had not deserved enough attention and was relegated to some aspects very minor in its practical applications. One of the advantageous characteristics of the rocket is that its speed can be adjusted so that the warming caused by air resistance while passing through the lower layers of the atmosphere is tolerable. At the time of the take off the speed of the rocket is lower, but it is increasing gradually as it reaches the upper layers, where the resistance is minimal, and finally reaches its maximum speed in the vacuum where the air resistance is zero.

To understand better the immense possibilities of the rockets in the field of Astronautics, let´s review its basic principles. First of all we must start with the idea that a rocket is a flying device that follows the laws expounded by Isaac Newton in his famous third law of motion: "Every action has a reaction equal in magnitude and opposite in direction". (Sir Isaac Newton 1686).

This reaction force can be verified experimentally by observing the recoil of a gun or other weapon, when it fires its missiles. This is what occurs when the rocket expels the gases produced during combustion obtained by the chemical reaction of its two propellants: the fuel and the oxidizer.

Despite of they use the same law of reaction to produce movement, there are some notable differences between rocket engines and jet engines. In the jet engines, the oxygen needed for combustion is obtained from the air drawn from outside, so it can work only in the atmospheric layers dense enough to be able to provide this gas in the quantities required, which automatically invalidates this type of engines for high altitudes and especially for flights into space. So rocket engines have the advantage of containing in its interior all oxygen needed, the oxidizer, either mixed with fuel, or in independent tanks for liquid fuel rockets, makes of it a truly autonomous totally independent of external environment and therefore able to operate in vacuum.

This kind of engine, called anaerobes, get their best performance in those areas where the lack of atmospheric air minimize any resistance to their displacement, which makes them ideal drivers of space vehicles.

3.2 PROPULSION

To travel through the space, a rocket needs a propulsion system capable of providing acceleration to them. Like I mentioned before, due to the vacuum, any acceleration should be based on Newton's third law or law of action and reaction.

A spacecraft changes its velocity by its propulsion system. Due to inertia, the more mass the ship has, the more difficult will be accelerating it. This is why we often speak of the momentum of a rocket, and to quantify the change of momentum we define the impulse. So, the aim of the propulsion is create impulse. When the spacecraft take off from the Earth, the propulsion method used must overcome the gravitational force. In order to put the rocket into orbit we need a tangential speed that generates a centripetal force enough to compensate the effect of the gravity.

In order to get the impulse there are two needs:

- Mass of reaction (propellant)
- Energy

The impulse obtained from a particle which has been ejected, if its mass is m and its velocity is equal to $m * v$. But this particle is ejected with a kinetic energy equal to $m * v^2/2$. In a solid fuel rocket, liquid, or hybrid, the propellant must be burnt, providing energy and the products obtained from the reaction are driven to the outside of the rocket through the nozzle, providing reaction mass.

The engine most commonly used for spacecraft propulsion is the rocket engine, because it is able to generate an enormous power and, unlike other types of engines, it does not need air. Despite the great power of rocket engines, the huge distances across the universe demands engines with a higher specific impulse, able to get more speed with the same weight of propellant. To achieve this purpose the ion thruster is being developed, which may be ten times more efficient because of its higher output. Although nowadays, there is not any motor available for interstellar travel. However, there are several alternatives to jet engines like the solar sails which can get thrust from solar radiation, solar wind, or even laser or microwave rays sent from the Earth. In the future, other methods of propulsion like the "curvature engines” or Warp drives can be developed.

3.2.1 Thrust

A rocket engine, whether it is part of a massive engine or a low-powered device used for adjusting the stability of the satellite's orbit, operate the same way, the mass is accelerated and expelled, creating a reaction force according to Newton's third law. The rocket is therefore a container of mass and energy. The mass, at the beginning located in the container, becomes a gas releasing kinetic energy. This gas escapes through the nozzle at high velocity while there is a change of momentum in the rest of the rocket which produce a reaction force.

An equation for the rocket propulsion can be obtained from a system of two elements: a rocket of mass m and a gas of mass dm which leaves the rocket through the nozzle. The momentum of these two elements combined can be expressed by the following equation:

(EQ 1) $$N = mv + \delta m(v - v_e)$$

Where N is the momentum, v the velocity of the rocket and v_e is the relative velocity of the gas leaving the nozzle.

The dynamic reduction in the mass of the rocket due to the increase of gas can be expressed as:

(EQ 2) $$\frac{d(\delta m)}{dt} = -\frac{dm}{dt}$$

Assuming that the escape velocity is constant, the acceleration of the gases driven through the nozzle is zero, so:

(EQ 3) $$\frac{dv_e}{dt} = 0$$

The momentum can be derived respect to time and the result equated to zero. And like dm is very small and in the limit tends to zero, and using the above two equations, we get:

(EQ 4) $$m\frac{dv}{dt} = -\frac{dm}{dt}v_e$$

The first part of the equation above can be seen as a reaction force F acting on the rocket, so the equation can be written as:

(EQ 5) $$F = -\frac{dm}{dt}v_e$$

The minus points out the fact that the gas flow is interpreted as a positive number, and the exit velocity and the reaction force acts in opposite directions.

The pressure acting in the inner and outer walls of the rocket engine, releasing kinetic energy through the transformation of mass in exhaust gases, are shown in the figure below:

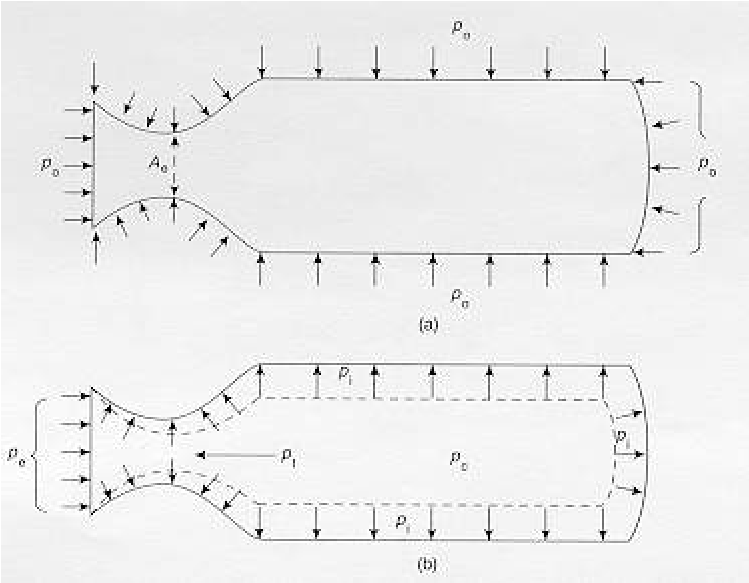


Figure 3.1: Pressure in the interior of a rocket engine. [2]

The sum of these forces produces the thrust that propels the rocket. This thrust can be expressed by the following equation:

(EQ 6) $$F - \int_s p ds - \int_{s_i} p_i \, ds_i + \int_{s_o} p_o \, ds_o$$

Where p is the pressure acting in the walls of the rocket, ds is a differential surface of the rocket and s is the total surface of the rocket motor, p_i is the pressure in the inner surface of the walls and p_o is the pressure in the outer surface.

Applying the theorem of momentum to the combustion of the gas contained in the inner walls of the rocket, the integral can be evaluated. In particular, identifying the velocity of change in direction with the force acting in the gas content within the walls of the rocket and assuming that pressure forces are symmetric, we get:

(EQ 7) $$\int_{p_i} p_i ds_i = A_e p_e - m v_{e_a}$$

Where A_e is the nozzle exit area, m is the output velocity of exhausted gases; Ve_a is the mean axial component of relative velocity of the rocket exhaust.

$m * Ve_a$ represents the momentum axial flow through the exit plane. The exit velocity usually consists of two orthogonal components, the velocity in the direction of the rocket, Ve_a, and the orthogonal component of the rocket, V_{ep}. In order to simplify, it is assumed that $V_{ep} = 0$, so:

$Ve=Ve_a$. And now the equation will be:

(EQ 8) $$\int_{p_i} p_i \, ds_i = A_e p_e - m v_e$$

The integral of the pressure acting on the outer walls of the rocket can be evaluated considering the resultant force due to atmospheric pressure Po and the fact that the sum of these forces is zero on the outer surface of the rest of the container.

(EQ 9) $$\int_{p_o} p_o \, ds_o = -p_o A_e$$

We assume that the pressure has axial symmetry. Substituting the integrals of internal and external pressures in the expression of the total pressure force, we obtain an expression for calculating the thrust of the rocket:

(EQ 10) $$F = \dot{m} v_e + A_e (p_e - p_o)$$

We have to keep in mind that V_e and P_e are averaged over the surface of the nozzle exit.

The first term on the right side of the equation represents the increase of momentum in the gases accelerated and expelled through the nozzle and it is called the momentum of thrust. The second term is the resultant of the pressures acting on the surface of the nozzle exit Ae and is called the pressure thrust. Therefore, using these concepts we can express the total thrust as:

Total thrust = Momentum of thrust + Pressure thrust

3.3 ROCKET ENGINES

As it is shown in figure 3.1, a common rocket engine consists on the injector, the combustion chamber and the nozzle.

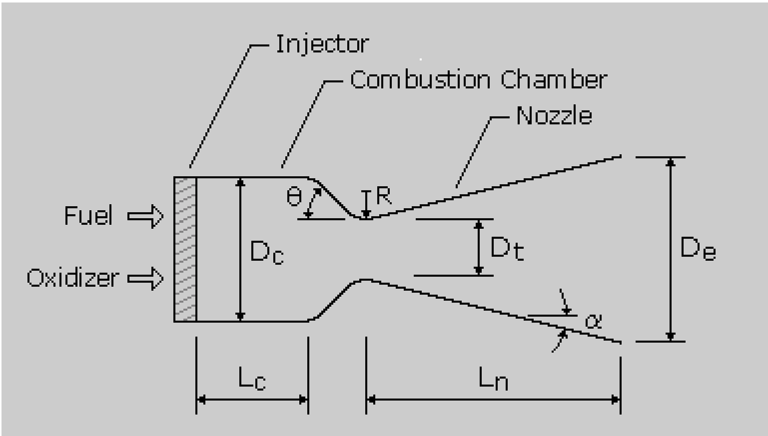


Figure 3.2: Parts of a rocket engine. [3]

The injector is a part of the engine made of small holes which objective is to provide the fuel and oxidizer to the chamber in a point short in distance away from the injector plate, once there the propellants burnt at high pressure. This part of the engine has to be strong enough to handle the high pressure and temperature generated in the combustion process and has length enough to ensure that complete combustion has been produced before the flow enter in the nozzle. Due to the high temperature and heat transfer, the chamber and the nozzle are usually cooled.

3.3.1 Nozzle

Like it has been shown in the figure 3.1, the nozzle is the last part of a rocket engine. Its function is to obtain kinetic energy from the chemical-thermal energy generated during the combustion process in the chamber. So, it changes the low velocity, high pressure and temperature gas flow into a high velocity, low pressure and temperature flow. Like it has been already mentioned, a huge portion of the thrust comes from the product of mass and velocity, so a very high gas speed is desirable.

There are, therefore, three different types of nozzles depending on the mach number of the flow at the exit of the nozzle: subsonic, sonic, and supersonic.

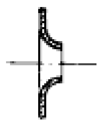
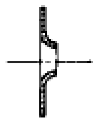
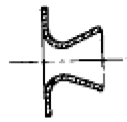
	Subsonic	Sonic	Supersonic
Throat velocity	$v_1 < a_t$	$v_t = a_t$	$v_t = a_t$
Exit velocity	$v_2 < a_2$	$v_2 = v_t$	$v_2 > v_t$
Mach number	$M_2 < 1$	$M_2 = M_t = 1.0$	$M_2 > 1$
Pressure ratio	$\frac{p_1}{p_2} < \left(\frac{k+1}{2}\right)^{k/(k-1)}$	$\frac{p_1}{p_2} = \frac{p_1}{p_t} = \left(\frac{k+1}{2}\right)^{k/(k-1)}$	$\frac{p_1}{p_2} > \left(\frac{k+1}{2}\right)^{k/(k-1)}$
Shape			

Table 3.1: Properties of different types of nozzle. [4]

In a rocket engine, the supersonic nozzle is needed in order to get more thrust because it achieves the highest degree of conversion of enthalpy into kinetic energy. In all the rockets the ratio between inlet and outlet pressures are large enough to induce supersonic flow, but if the absolute chamber pressure drops under 1.78 atm approximately, there will be subsonic flow in the divergent portion of the nozzle during operations at sea levels. This condition occurs only for a short time during the stop and start transients.

Traditional nozzles have different ways of function depending on the altitude that it is operating. In the picture below, it is shown how a bell nozzle works as it operates from sea level to high altitude.

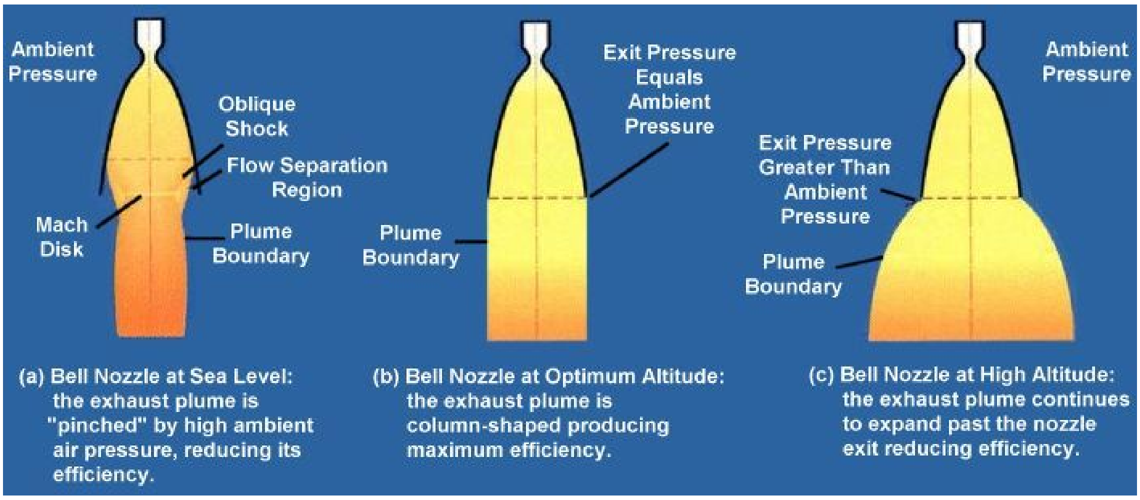


Figure 3.3: Effects on expansion due to ambient pressure. [5]

As it is discussed in the picture above, these nozzles have less efficiency at high or low altitude due to the different pressure between the ambient pressure and the exhaust gases. When the rocket is flying at low altitudes, the exit pressure is too low and the atmospheric pressure pushes the flow inward. So, the gases become to separate from the walls of the nozzle reducing the thrust produced. This condition is known as overexpansion. Otherwise, at high altitude, the exit pressure is much higher than the ambient pressure, so the gases keep expanding after the nozzle exit. Since this additional expansion is located outside the nozzle, it does not exert thrust on the nozzle, it is lost. And this is known as underexpansion.

As both overexpansion and underexpansion reduce the efficiency of the nozzle, the optimum design should avoid these conditions. If somehow the exit area at launch could be minimize and increased as the rocket ascends, the nozzle could be optimized for each altitude and the thrust would be maximized. An ideal nozzle would be able to continually adjust its contour, length and area ratio in order to maximize the thrust at each altitude.

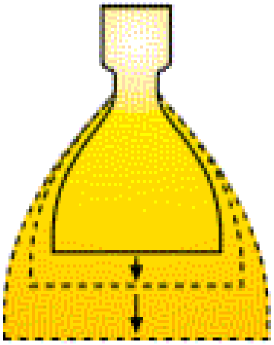


Figure 3.4: Ideal nozzle with altitude compensation. [6]

However, a nozzle that is continually changing its geometry is not practical, and the designer instead has to make a series of tradeoffs in selecting the area ratio of the nozzle.

A wide number of different proven nozzle configurations are available today and depending on their geometry they have been classified in conventional and unconventional nozzles.

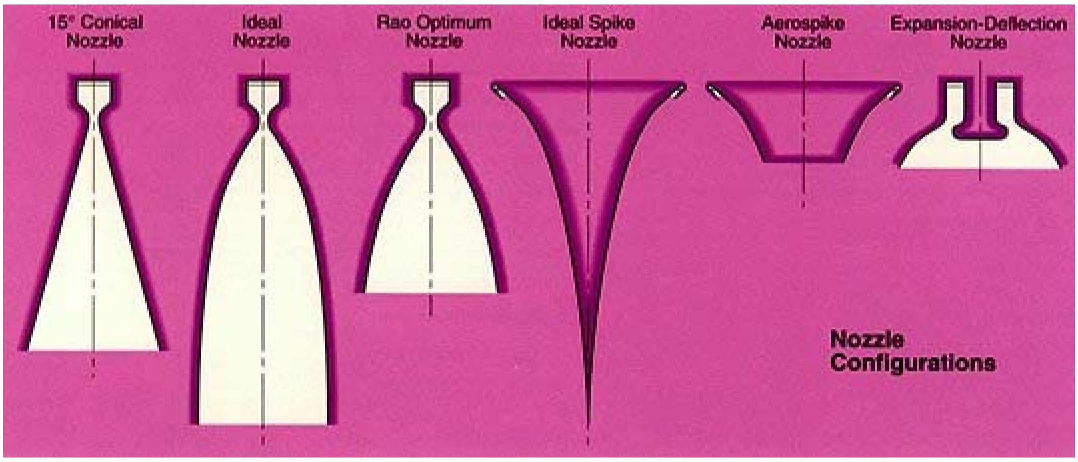


Figure 3.5: Nozzle configurations. [7]

- **Conventional nozzles**

Are known as conventional nozzles all of them that are formed of a convergent section followed of a throat and a divergent section.

The converging nozzle section and the throat have never been very important in order to achieve high performance because the subsonic flow in this section can be quiet easily turned into very low pressure drop with any radius, cone angle or wall contour. The shape of the throat is not very critical to performance as well because any radius or curve is accepted. But the principal difference in performance between different nozzle configurations is the contour of the diverging supersonic-flow section.

The *conical nozzle* is the oldest and maybe the simplest configuration of a nozzle. It is relatively easy to fabricate and nowadays it is still used in many small nozzles but it is rarely used in rocket nozzles. The most usual conical nozzle has a divergence cone angle of 30º and its exit momentum and therefore the exit velocity is 98.3% of the velocity of an ideal nozzle. This correction factor, λ, comes from the equation 11 due to the nonaxial component of the exit gases.

(EQ11) $$\lambda = \frac{1+\cos\alpha}{2}$$

If the divergence angle is small, most of the momentum produced is axial, so this configuration provides a high specific impulse, but long nozzles have a penalty in the rocket propulsion system mass and also in design complexity. On the other hand large divergence angle designs provide short and lightweight nozzles but with lower performance because the high ambient pressure causes overexpansion and flow separation. So a compromise between these parameters is needed, typically between 12º and 18 º half angles, depending on the application and flight path.

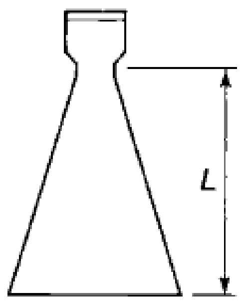


Figure 3.6: Conical nozzle. [8]

The *bell-shaped nozzle* has been the most usual nozzle shape in rocket engines. After the throat, it has a high angle expansion section (20º to 50º) followed by a gradual reversal of nozzle contour slope until at the end of it, the divergence angle is less than a 10º half angle. The most common shape used is the parabola because they are a very good approximation for the bell-shaped contour curve, and the lengths of these nozzles are approximately 20% shorter than a 15º conical nozzle of the same area ratio.

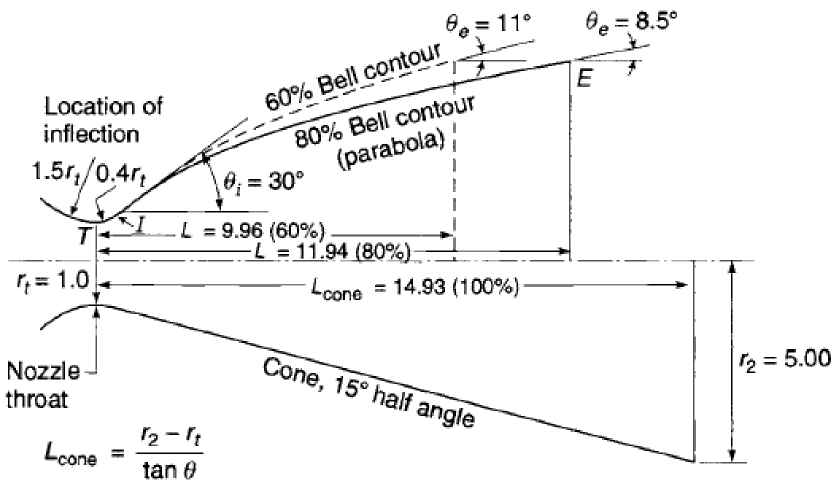


Figure 3.7: Comparison between conical and bell nozzle. [9]

The expansion produced in this nozzle is more efficient than a conical one of similar area ratio and length because the wall contour is designed to minimize losses. Engineers developed this wall contour that changes gradually enough to avoid oblique shocks in order to obtain higher performance and shorter length designs.

For conventional bell nozzles, loss mechanisms may be due to: geometric or divergence loss, viscous drag loss and chemical kinetic loss. The geometric loss occurs because a portion of the nozzle exhaust gases directed away from the nozzle axis results in a radial component of momentum. However, in an ideal nozzle, this flow is parallel to the nozzle axis. The drag force is produced in the walls of the nozzle by the effects of the viscous high-speed flow acting in the opposite direction of the thrust. And the chemical kinetic loss is because due to the rapidly accelerating nozzle flow, the gases do not have enough time to reach full chemical equilibrium. So, the overall nozzle efficiency is given by the combined effects of these losses.

The nozzle designer knows that a long nozzle is perfect to maximize the geometric efficiency, but this produce a higher drag of the nozzle. And in order to reach the chemical equilibrium the radius of curvature of throat region should be higher to produce lower acceleration of the gases at the throat. So, the designer needs a compromise that results in maximum overall nozzle efficiency. Although due to the different pressures in the exit of the nozzle along its flight, conventional nozzles will only be optimum at one particular altitude.

Because there is little to gain in thrust by increasing the nozzle efficiency, most of the work in the development of new configurations has been directed toward obtaining the same efficiency from a shorter nozzle. This new technology is known as unconventional nozzles.

- **Unconventional nozzles**

The need of some form of altitude compensation in the nozzles that were able to operate from sea level to orbit is the reason of the development of unconventional nozzles. Nowadays there are several configurations with the objective of reaching this goal and during this project we are going to focus on the aerospike nozzle and the expansion-deflection nozzle.

In the *annular aerospike nozzle*, gases flow from an annulus at a diameter located some radial distance from the nozzle axis and it is radially inward toward the nozzle axis; which is the opposite concept of a bell nozzle. In an aerospike nozzle, the expansion process is originated on the outer edge of the annulus and, because this point is exposed to the ambient pressure as well, the expansion is limited by the influence of the external environment.

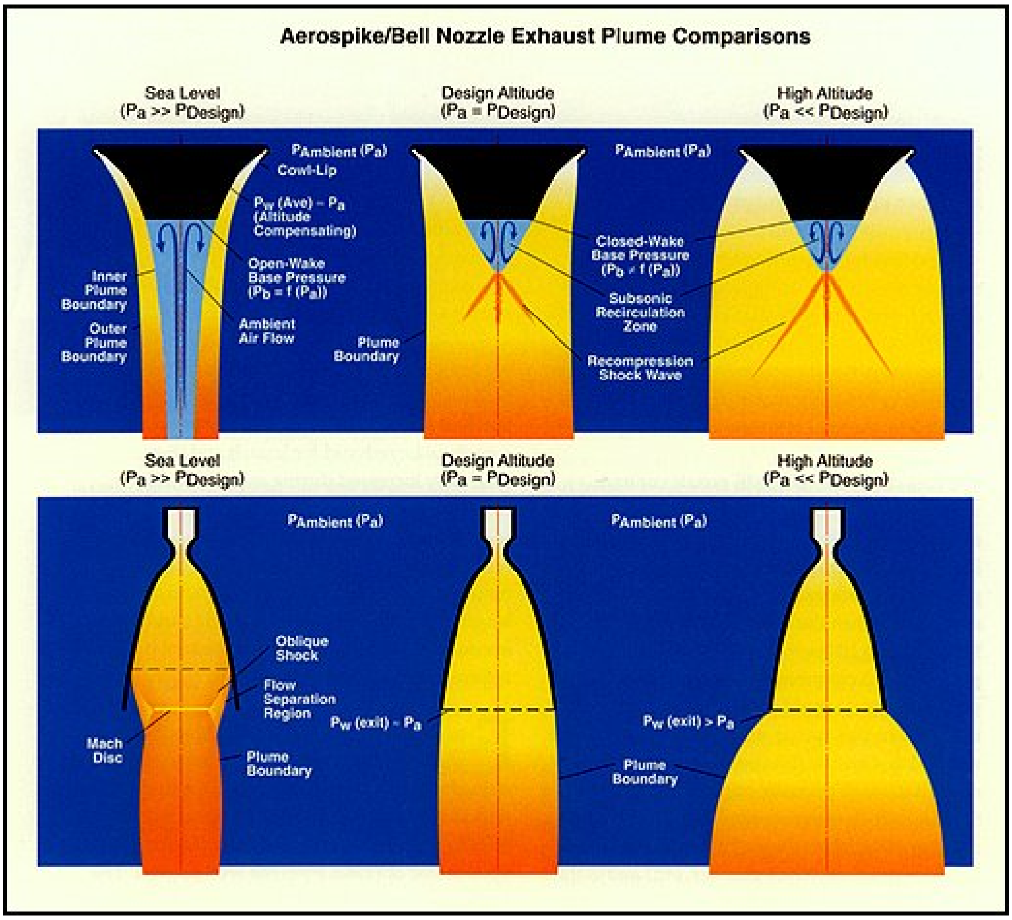


Figure 3.8: Aerospike/Bell nozzle exhaust plume comparisons. [10]

The ambient pressure places a limit on the expansion process and the loss in the thrust produced by the overexpansion does not occur. Also, the benefits of a variable area ratio nozzle are obtained without the need of variable nozzles geometry. Since the atmospheric pressure controls the expansion, the flow at the end of the nozzle changes with altitude. An important advantage is that a high area ratio nozzle, which has high vacuum efficiency, performance perfectly at sea level as well. And similar values of thrust respect a conventional nozzle can be achieved at high altitudes with only one quarter of their length, and better performance in low altitudes.

The *linear aerospike nozzle* is a variation of the annular one, where the combustion chamber is made up of a series of modular chamber segments, and the gas generator engine cycle is used in place of the combustion tap-off cycle. This configuration offers the same performance advantages while offers some unique configurationally advantages due to its linear shape.

The ideal aerospike shape turns out to be very long, quite similar in length to an optimum bell nozzle. Here, the mass flow per unit exit area is approximately uniform over the cross section and the divergence losses are minimal. But this nozzle would be too weight and its length is truncated to the 20% range because the losses in thrust are small in comparison with the save in weight. And even the losses caused by the cut-off spike can be offset by injecting a small amount of gas flow through the base into the recirculating region.

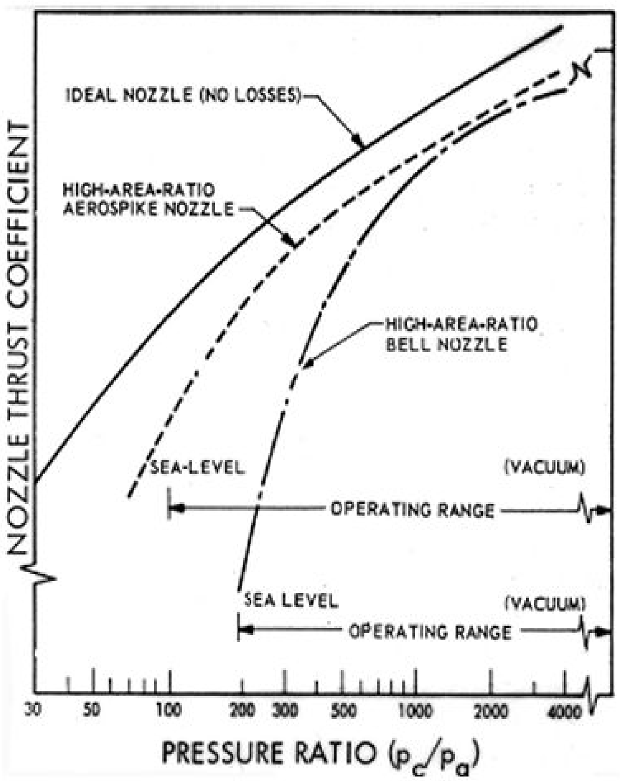


Figure 3.9: Thrust comparison between aerospike and bell nozzles. [11]

The advantages of this nozzle respect a conventional one are:

- Smaller nozzle: Truncates spike can be far smaller than a conventional one for the same efficiency and even can give greater performance for the same length.
- Superior performance: Due to the altitude compensation
- Less risk of failure: The aerospike engine has a simple gas generator cycle with a lower chamber pressure reducing the risk of an explosion.
- Lower vehicle drag: Because it fills the base portion of the vehicle thereby reducing the base drags.

- Modular combustion chambers: The linear aerospike is made up of these thrusters that gives the engine greater versatility and are small, easier to develop and less expensive.
- Thrust vectoring: Due to each combustion chamber can be controlled individually, the rocket can use differential thrust vectoring in order to control the direction. This avoids the need for the heavy gimbals and actuators used to vary the direction of conventional nozzles.

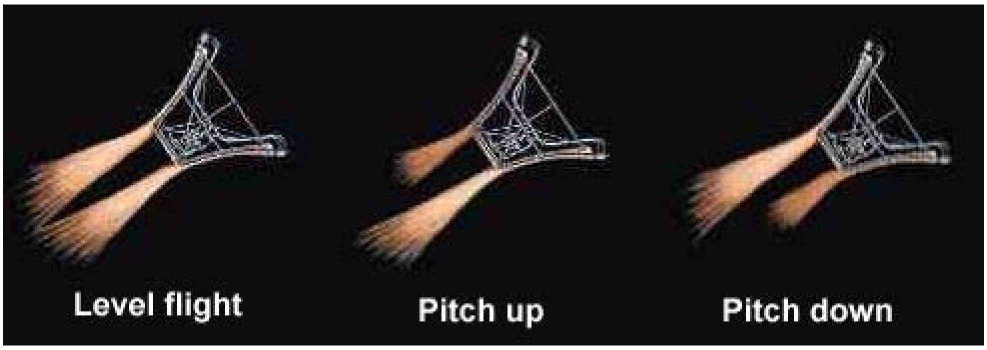


Figure 3.10: Aerospike thrust vectoring. [12]

- Lower vehicle weight: Although the aerospike tends to be heavier than a bell nozzle, it shares many structural elements with the rocket reducing overall weight.

On the other hand, the disadvantages of this configuration are:

- Cooling: The central spike has far greater heat fluxes than a bell nozzle. Although this problem can be solved by truncating the spike to reduce the exposed area and by refrigerating the spike with cold cryogenically-cooled fuel. The secondary flow helps to cool the centre body as well.
- Manufacturing: Because these nozzles are more complex and difficult to build than a bell nozzle. So, it is more expensive.
- Flight experience: Due to its recently discovered, the engineers have less experience about them.

The *expansion-deflection nozzle* is another unconventional nozzle design with the objective of achieves the altitude compensation through interaction of the exit gases with the ambient.

It looks like a standard bell nozzle, but inside there is centre body which deflects the flow towards the walls and the gas flow expands around it. The nozzle is shorter than a conventional one and has some internal oblique shock wave losses. The aerodynamic interface between the gas flow and the ambient air produces an inner boundary of the gas flow in the diverging section, where the gas flow fills the section as the ambient pressure is reduced.

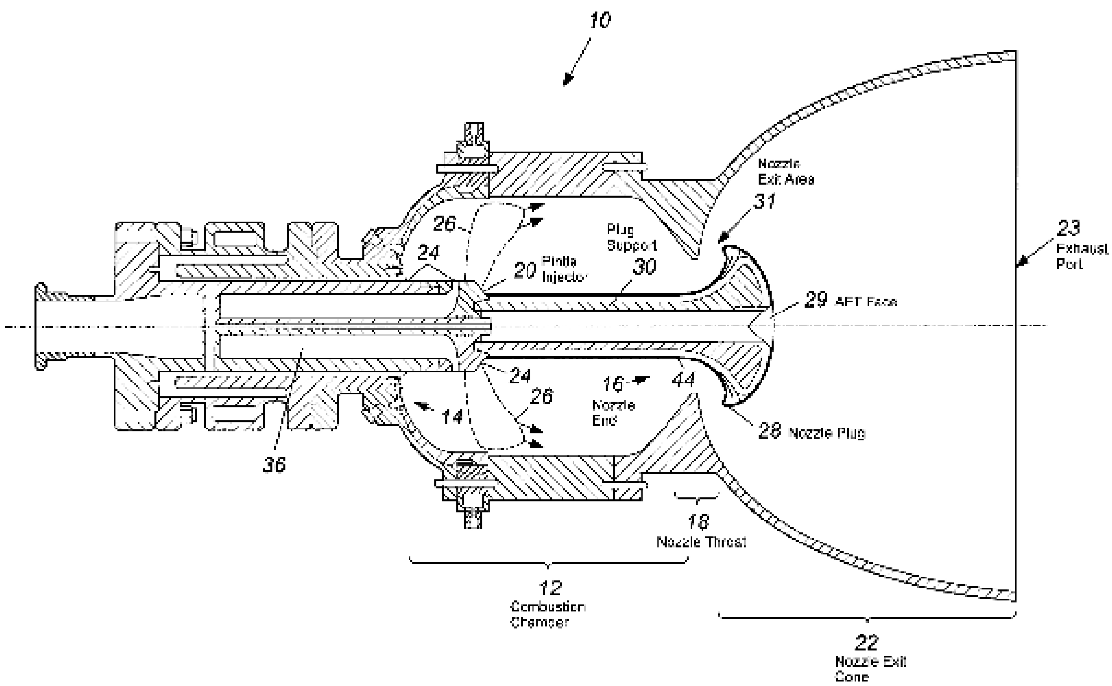


Figure 3.12: Expansion-Deflection nozzle. [13]

This kind of nozzle has long been considered for rockets due to its potential increments in performance offered through altitude compensation. But there are several difficulties with it because a more complex combustion chamber and throat design is needed, and the non-isentropic flow phenomena such as viscosity and shock waves on flow field composition.

3.4 HEAT TRANSFER ANALYSIS

The heat created during the combustion process in a rocket is contained within the exit gases. Most of this heat is expelled along with the gas flow that contains it, although a significant portion of this heat is transferred to the walls of the engine.

In actual rocket development not only is the heat transfer analyzed but the rocket units are tested as well to make sure that heat is transferred satisfactorily under all kinds of operating and emergency conditions. Heat transfer calculations are very useful in order to guide the design, testing and failure investigations. For those rockets combustion devices that are regeneratively cooled or radiation cooled can reach thermal equilibrium so the steady-state heat transfer relationships will be applied. On the other hand, transient heat transfer conditions will be applied during thrust build-up and shutdown of all the rockets and in those where their cooling techniques never reach the equilibrium.

The heat flux is transmitted to all the internal surfaces exposed to the gas flow, the injector, the chamber and the walls of the nozzle. The heat transfer rate varies within the rocket; usual heat transfer distribution is shown in the figure below. Here, it is shown how the minority of the heat flux is transferred by the chamber walls, normally 1 to 5% of the total energy generated.

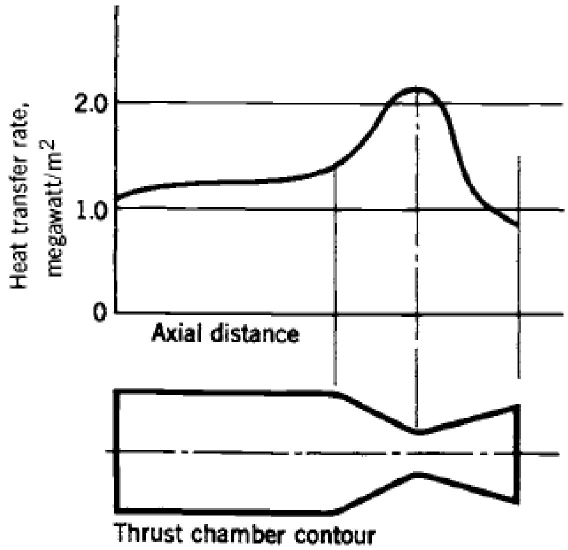


Figure 3.13: Heat flux distribution. [14]

For a small rocket of 44,820 N of thrust, the heat flux through the walls may be between 0.75 to 3.5 MW depending on the design and conditions. The amount of heat transferred by conduction is almost negligible, by far the largest part of the heat is transferred by convection and about 5 to 35 % is attributable to radiation.

High values of pressure in the chamber means higher thrust performance, but it also means higher engine inert mass. However, the increase of the heat transfer with chamber pressure often imposes design or material limits in the rocket engines.

The heat transfer intensity can vary from less than 50 W/cm2 to over 16 kW/cm2. The highest values are for the nozzle throat region and the lowest for the gas generators and nozzle exit sections.

3.4.1 Cooling techniques

The most important objective of cooling is to prevent the chamber and nozzle walls from becoming too hot because the materials used for build the walls of the nozzle lose strength and become weaker as temperature is increased.

Mainly, there are two different cooling methods, the steady state method or the transient heat transfer. Although sometimes film cooling and special insulation act like supplementary techniques to locally increase their cooling capacity.

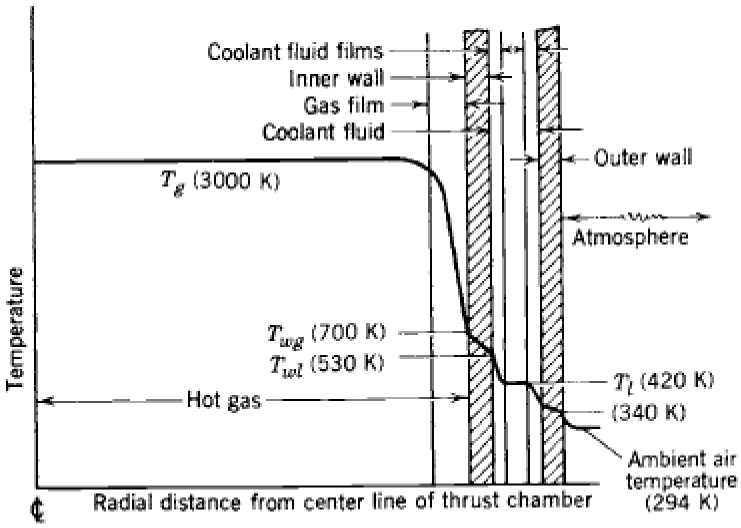


Figure 3.14: Temperature distribution in a nozzle cooled. [15]

- **Steady state method**

Here, the heat transfer rate and the temperatures of the chambers reach thermal equilibrium; and the duration is limited by the available supply of propellant. This method includes regenerative cooling and radiation cooling.

Regenerative cooling is done by building a cooling jacket around the chamber and circulating one of the liquid propellants through it before it is fed to the injector.

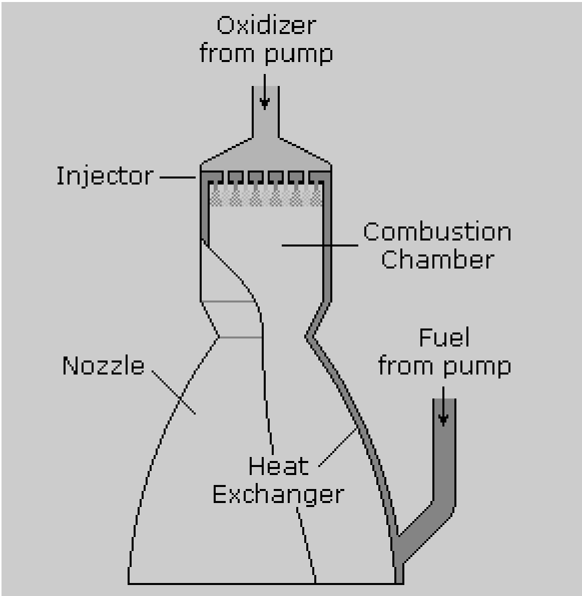


Figure 3.15: Nozzle regeneratively cooled. [16]

This cooling technique has been effective in applications with high chamber pressure and high heat transfer rates and most injectors use regenerative cooling.

In *radiation cooling*, the chamber and the nozzle have only a single wall made of high temperature material that when it reaches thermal equilibrium radiates heat away to the surroundings or to the space. This kind of cooling technique is used with monopropellant thrust chamber, bipropellant and monopropellant gas generators, and for diverging nozzle which exit's section is higher than an area ratio of 6 to 10. This cooling technique has worked well with lows chamber pressures and moderate heat transfer rates.

- **Transient heat transfer method**

In this cooling method the thrust chamber does not reach a thermal equilibrium and temperatures continue to increase with operating duration. The rocket combustion operation has to be stopped before any of the walls reaches a critical temperature at which it could fail. This technique has been used with low chamber pressures and low heat transfer rates. The cooling of the thrust chambers can be done by absorbing heat in an inner liner made of an ablative material, like for example with fiber-reinforced plastics.

Finally, the selection of the optimum cooling method depends on many factors such as type of propellant, chamber pressure, available coolant pressure, combustion chamber configuration and combustion chamber material.

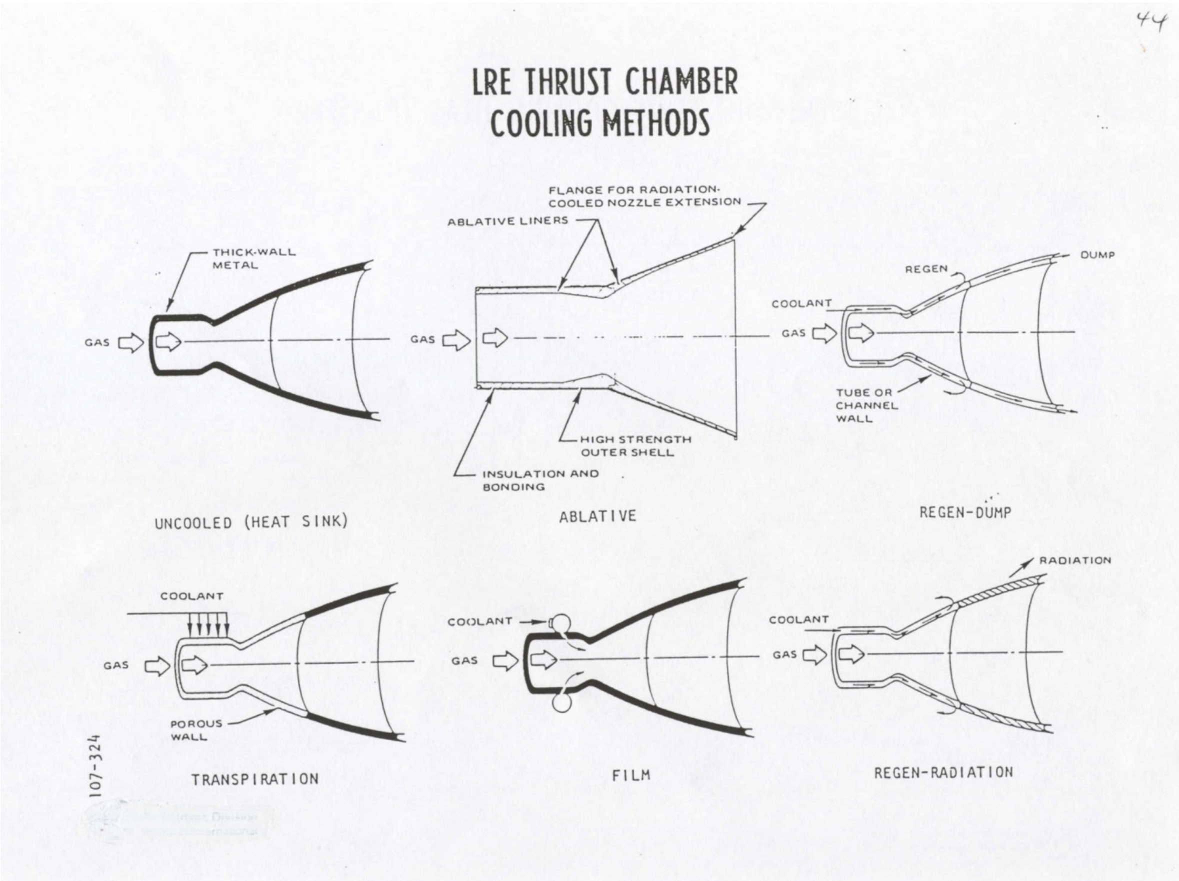


Figure 3.16: Thrust chamber cooling methods. [17]

4. DESIGN PROCESS

4.1. NOZZLE DESIGN:

In order to design a nozzle there are some factors that have been needed to bear in mind, these are the most important:

- Maximize performance: Optimize the contour of each nozzle to obtain the highest value of thrust as possible.
- Ensure structural integrity: Due to the strong loads and highs temperatures that affect the nozzle.
- High reliability.
- Safety conditions.
- Cost: Obviously all of the design factors trade with cost.

The nozzles studied in this project are made of a ceramic material because it has a very high resistance to the heat and enough mechanics characteristics to handle the loads produced over the nozzle due to the pressure.

There are several types of propellant available for rocket engines, but in this project all the designs work with liquid hydrogen, which is the liquid state of the element hydrogen. This propellant has been chose because after its combustion the only exhaust gas is water vapor, so the design is considered “zero emissions”.

During this project, different nozzle designs have been analyzed, but the experimental procedures of all of them are quite similar.

All designs have been carried out with Gambit, which allows developing the geometry of the nozzle, meshing the model and establishing the boundary conditions as well.

4.2. GAMBIT:

4.2.1 Geometry:

The first step in order to analyze the nozzles is design them. This has been done with the software Gambit:

First of all, I have set different points to guide the edges of the nozzle. In the picture below we can see an example of how was designed the Expansion-Deflection nozzle.

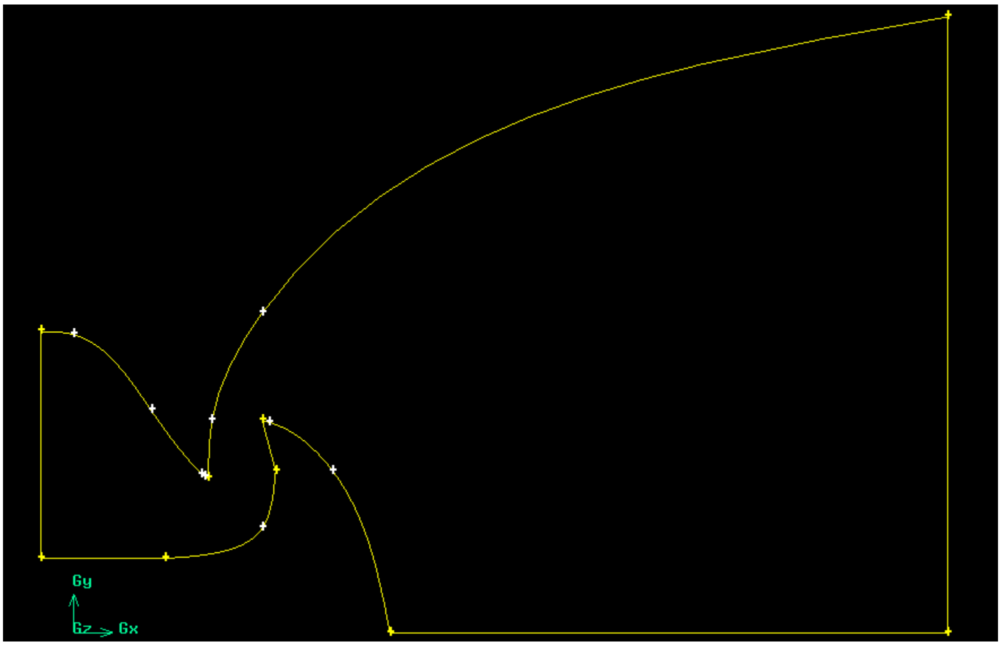


Figure 4.1: Face before being revolved of an expansion-deflection nozzle.

Once the shape of the nozzle is already done, these edges are connected forming a face. After this, the final step to obtain the volume we are looking for is revolving or sweeping the face in order to have the final geometry.

In the picture below, there are all the geometries designed for this project, which have been developed by the same way it has been described before. Although in the picture the whole nozzle is shown, less in the expansion-deflection nozzle, in reality in the symmetrical ones only a quarter of each nozzle were revolved in order to minimize the iteration time for each design.

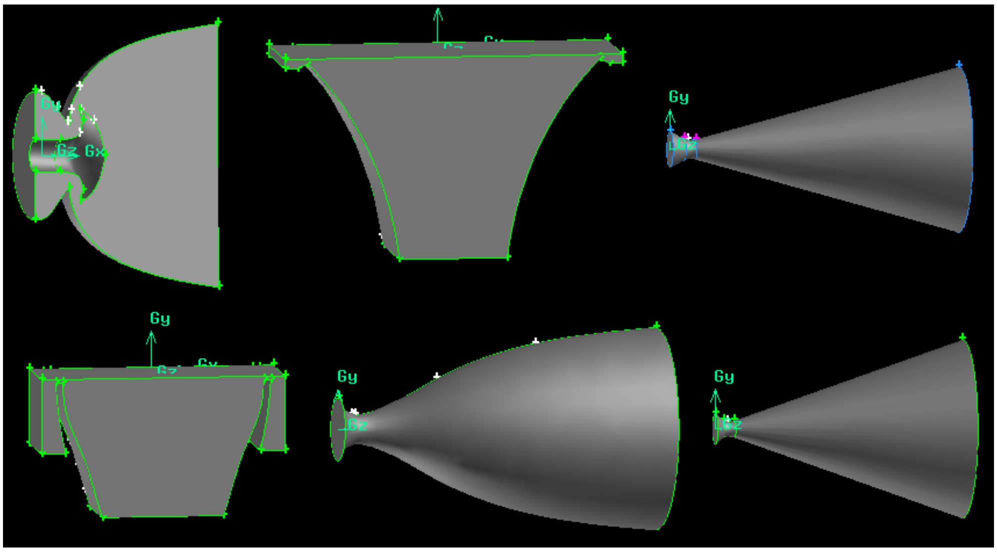


Figure 4.2: Nozzle designs.

4.2.2 Mesh:

Once the nozzle is already built, is time to mesh the model. During this stage, first of all the edges will be meshed followed by the faces and finally the volume. The more accurate the mesh would be the better results I would get later. The size of the cells depends on each design, but the elements are in all of them Tet/Hybrid and the type TGrid. This is a very important key for analyzing the model because as the design is going to be iterated as a k-epsilon viscous model, the y+ would have to be between 50 and 500, and mainly this value will depend on the mesh. In the next picture there is an example of a bell nozzle meshed.

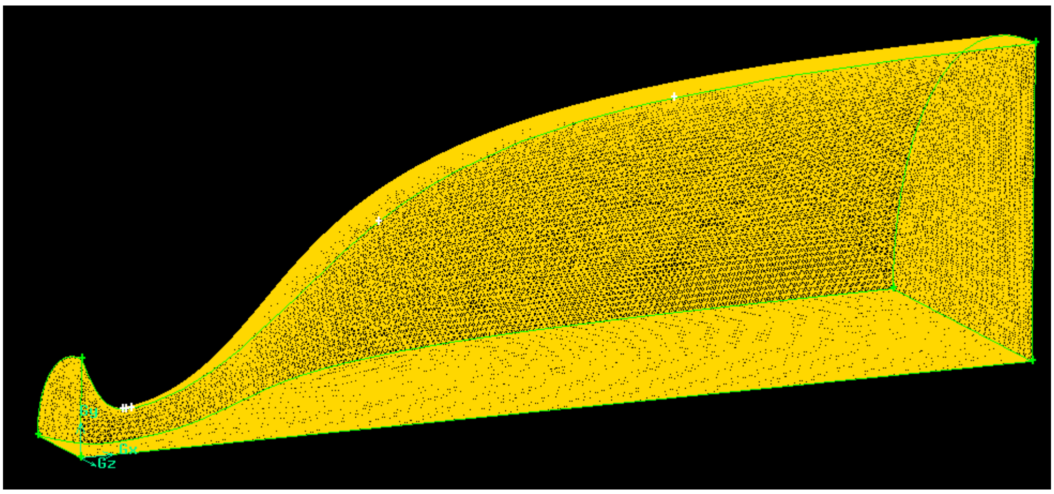


Figure 4.3: One quarter of the bell-shaped nozzle meshed.

4.2.3 Boundary conditions:

The last stage in Gambit is established the boundary conditions. As the parameters known of the model are the inlet and outlet pressures, both faces are set like pressure_inlet and pressure_outlet respectively. The symmetrical faces of the design are set like that and the faces that are supposed to be the walls of the nozzle are set like wall. And finally as the gases will pass through the nozzle, the volume is specified as fluid.

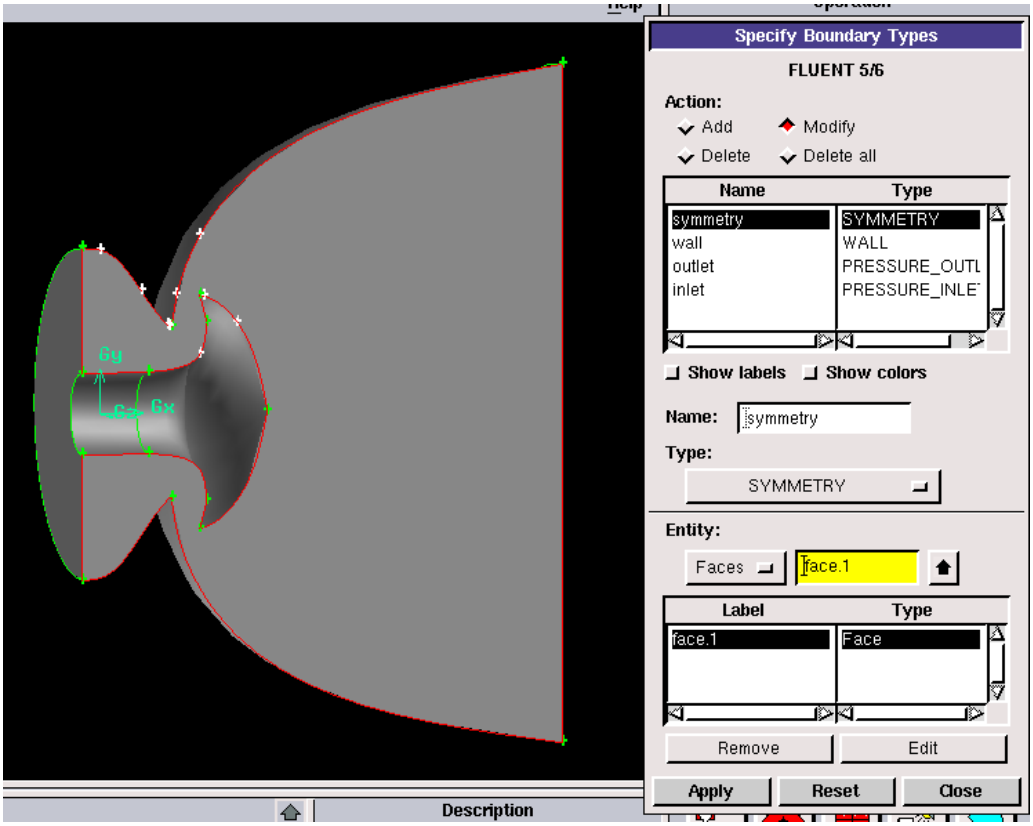


Figure 4.4: Setting boundary conditions.

Finally, the model and the mesh have to be saved and exported because further analyses will continue with Fluent.

4.3. FLUENT:

After the mesh has been exported to Fluent, it is time to set the last parameters before running the design.

First of all the grid has to be checked and scaled in order to make sure the mesh was well built.

Later, the model has to be defined. This is a 3D steady analysis as we can see in the picture below. And the viscous model has to be changed to k-epsilon.

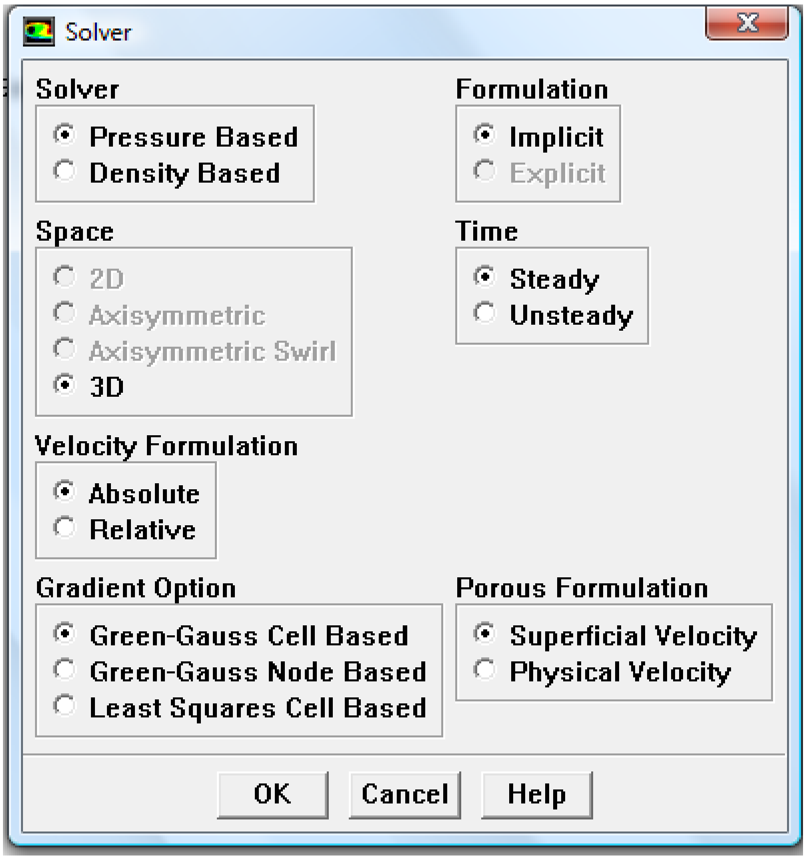


Figure 4.5: Setting solver parameters.

As it was mentioned before the gas which go through the nozzle is water vapor, so from fluent database we can copy its parameters and later change its density to gas-ideal.

In general, the combustion gasses of a hydrogen-oxygen rocket engine are appreciably in excess of the critical temperature of all of the individual species and since the rocket engine gasses expand through the nozzle in a nearly isentropic way, the descent of fluid pressure maintains thermodynamic conditions that are quite distant from the critical pressures and temperatures of all the chemical species. So, the exhaust gasses of the rocket engine are assumed to obey the ideal gas law.

The next step is set the operation conditions where the gravity is set and the operating pressure is changed to zero. And the boundary conditions where the inlet and outlet pressures are set, obviously these parameters changes from one design to another.

Before running the analysis the model has to be initialized from the boundary condition inlet. And finally the design is ready to iterate.

Once the iterations are done its time to get the results, from the menu display there are several options of obtain different graphs. The static pressure, velocity and other useful graphs can be obtained from this menu.

Due to the third law of Newton, the thrust produced by the nozzles can be found out with the report/forces menu because the force produce by the gas over the nozzles has the same quantity but in the opposite direction to the thrust.

zone name	pressure force n	viscous force n	total force n	pressure coefficient	viscous coefficient	total coefficient
wall	1902163.3	26163.646	1928326.9	3105572.7	42716.158	3148288.8
net	1902163.3	26163.646	1928326.9	3105572.7	42716.158	3148288.8

Figure 4.6: Thrust results.

5. RESULTS AND DISCUSSION

5.1. CFD

In order to develop this project and optimize the design of the nozzles, I have done the calculations with CFD, which is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. The fundamental basis of any CFD problem is the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. This software gives you the power to simulate flows, heat and mass transfer, moving bodies, etc, through computer modeling.

The CFD began in the 60's in the aerospace industry, and nowadays has become into a vital tool for many industries for the prediction of fluid flow. It has been expanded significantly to different industrial applications and industrial processes involving heat transfer, chemical reactions, two-phase flow, phase changes and mass transfer, among others.

In order to obtain an approximate solution numerically, a discretization method have to be used to approximate the differential equations by a system of algebraic equations, which can later be solved with the help of a computer. The approximations are applied to small domains in time and/or space. The accuracy of numerical solutions depends on the quality of the discretizations used as much as the accuracy of experimental data depends on the quality of the tools used.

It is important to bear in mind that numerical results are always approximate because there are reasons for differences between computed results and reality like:

- The differential equations might contain approximations or idealizations.
- The approximations made in the discretization process.
- Iterative methods are used in solving the discretizated equations. So, unless they are run for a very long time, the exact solution of the discretizated equations is not produced.

Discretization errors can be reduced by using more accurate interpolation or approximations or by applying the approximations to smaller regions, but this usually increases the time and cost of obtaining the solution.

Compromise is needed in solving the discretized equations. Direct solvers, which obtain accurate solutions, are not very used because they are too expensive. Otherwise iterative methods are more common but the errors produced by stopping the iteration process too soon need to be taken into account.

5.1.1 Fluent

Fluent is a package of simulation computational fluid dynamics (CFD) and the most used in the world, with a background of more than 25 years of development carried out by Fluent Inc. who are certified under the international standards of ISO 9001 and TickIT.

The structure of Fluent allows you to incorporate a lot of models for different physical and chemical processes. Not only can you perform simulations of laminar or turbulent flow, Newtonian or non-Newtonian, compressible or incompressible, single phase or multiphase, but also processes of heat transfer by radiation, conduction and convection of course, as well as melting processes and chemical reactions such as burning of gases, liquids and solid fuels.

The general capabilities of Fluent simulation are:

- 3D simulations, 2D map, 2D axisymmetric, 2D axisymmetric with swirl.
- Unstructured meshes (triangles and quadrilaterals in 2D, tetrahedra, prisms and pyramids in 3D)
- Simulations of steady or unsteady flow.
- Any speed system (subsonic, transonic, supersonic and hypersonic).
- Laminar flows, turbulent, non-viscous.
- Newtonian and non Newtonian flows.
- Wide variety of turbulence models, including k-epsilon, k-omega, RSM, DES, and LES.

- Heat transfer, including natural convection, forced or mixed; solid conjugate heat transfer / fluid, including solar radiation.
- Mixing and reaction of chemical species, including models of homogeneous and heterogeneous combustion reaction models / deposition on surfaces.
- Models multiphase and free surface, including heat transfer and chemical reactions.
- Lagrangian trajectory calculations for the dispersed phase (particles, drops, bubbles), including models for sprays and thin films.
- Models of phase change for solidification and casting applications, cavitation model and model of wet steam.
- Medium permeability porous non-isotropic, inertial resistance, heat conduction in solids, and option to calculate interstitial velocity.
- Models for fans, radiators and heat exchangers.
- Dynamic mesh capability for modeling flow around moving objects during the simulation.
- Inertial reference frames (stationary) or non-inertial (rotating or accelerating).
- Multiple frames of reference and sliding mesh options.
- Mixed model planes for rotor-stator interactions.
- Collection of tools for aeroacoustic models.
- Volumetric source terms for mass, momentum, heat and chemical species.
- Database of material properties.
- Dynamic coupling (two-way) with GT-Power and WAVE.
- Additional modules for fuel cells, magnet hydrodynamics and fiber continuum model.
- Extensive customization capability via user-defined functions (c programming).

5.1.2 Gambit

Gambit is Fluent’s software for geometry and mesh generation. The interface for geometry creation and meshing brings together most of Fluent's preprocessing technologies in one environment. Advanced tools for journals can conveniently edit and repeat the session for the construction of parametric studies.

Gambit combines CAD interoperability, geometry cleanup tools CAD, decomposition and meshing tools that result in one of the easiest ways of preprocessing, fast and direct CAD to mesh quality.

Gambit can import geometry from any CAD / CAE, Parasolid formats, ACIS, STEP or IGES. Additional modules are also available to integrate CATIA V4, Pro / E. The tolerance and repair capabilities provide geometries automatically connected during the import process.

5.2. NOZZLE ANALYSES

Several 3D analyses of different nozzles have been carried out by CFD in order to calculate and optimize their thrust.

5.2.1 Conventional nozzles

- **Conical nozzle**

A conical nozzle has been chose for the first design because is the simplest nozzle due to its geometry, its dimensions are:

Caracteristic	Dimension (cm)
Nozzle length	320
Throat diameter	26
Throat location	15
Inlet diameter	62
Exit diameter	230

Table 5.1: Dimension of the design.

This design has the same dimensions of a bell nozzle, but as it is shown in the picture, its contour is conical.

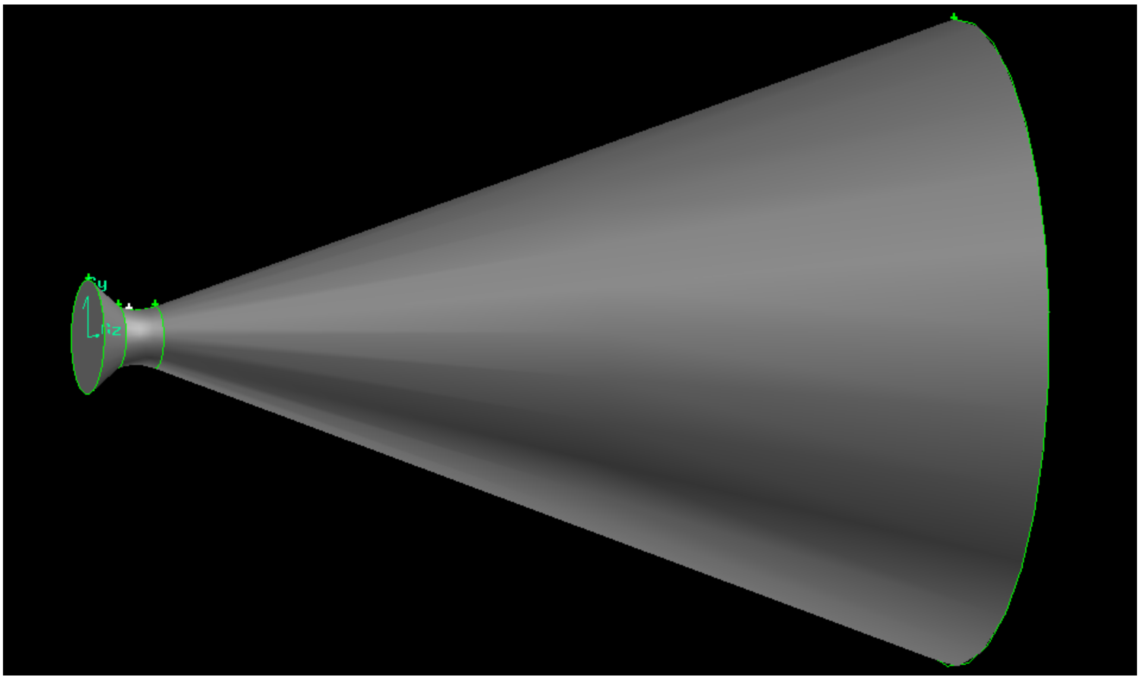


Figure 5.1: Conical nozzle design.

At the beginning, the throat has not the same shape that has in the picture. Here, the throat is curved because otherwise the flow was too concentrated in the centre line of the nozzle producing some turbulence after the throat section.

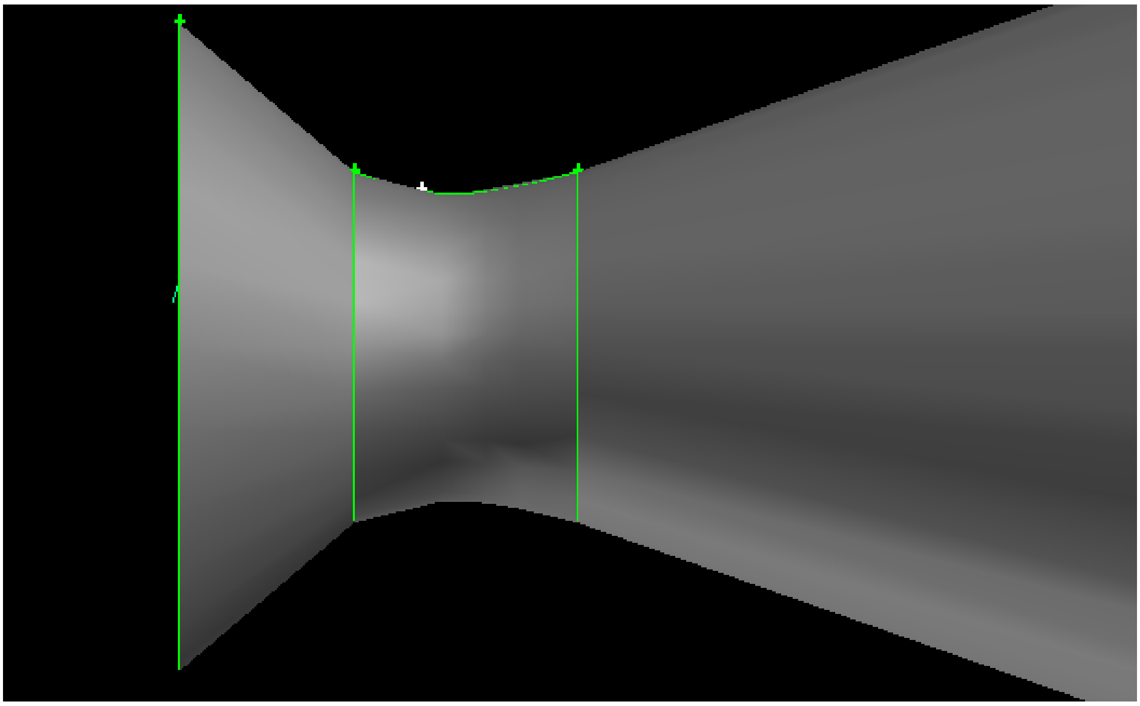


Figure 5.2: Conical nozzle's throat.

Although in the picture the nozzle has been completely revolved, in order to minimize the time of iterations and maximize the accurateness, only one quarter of the nozzle has been analyzed due to its symmetry.

In this particular case, the inlet pressure of the chamber is 20,000,000 Pa and the outlet pressure will be 101,325 Pa because it is open to the atmosphere and this analysis have been developed with the fact that the rocket was at the beginning of its trip, still under the effects of the atmospheric pressure. After doing the iterations, in the following picture, the velocity vectors coloured by velocity magnitude (m/s) of one of the symmetrical faces is shown:

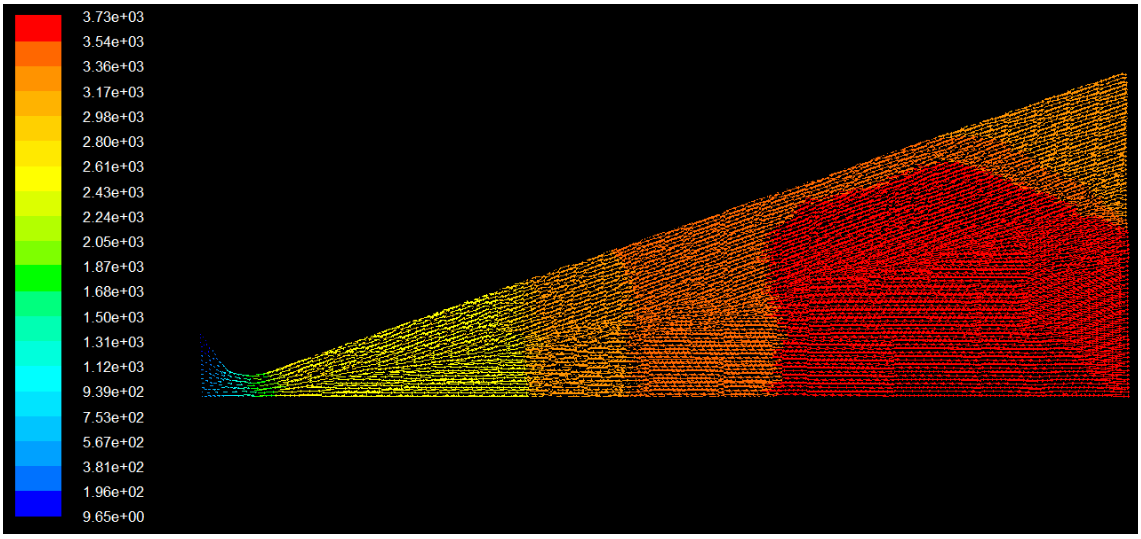


Figure 5.3: Velocity vectors colored by velocity magnitude.

Obviously the highest velocities of the gas that go through the nozzle are in the region near the exit because this is the supersonic region of the nozzle where the speed of the gases increases when the area is increased. As it is shown in the picture above, 3,730 m/s is the highest speed of the exhaust gases. This seems a good value of velocity because for bipropellants rocket engines, usual values of exhaust velocity are between 3 and 5 km/s. In the picture it is shown as well the shock wave produced at the end due to the difference pressure between the exit pressure and the ambient producing a decrease in the efficiency.

As Sir Isaac Newton expounded in his third law of motion: “Every action has a reaction equal in magnitude and opposite in direction”, (Sir Isaac Newton 1686). So, putting into practice this law, the force against the walls of the nozzle produced by the gases that pass through the nozzle has the same quantity but is opposite in direction to the thrust produced. In this particular case, the thrust obtained is 1,667,521.7 N.

zone name	pressure force n	viscous force n	total force n	pressure coefficient	viscous coefficient	total coefficient
wall	1640210.9	27310.826	1667521.7	2677895.3	44589.104	2722484.4
net	1640210.9	27310.826	1667521.7	2677895.3	44589.104	2722484.4

Figure 5.4: Thrust produced by the conical nozzle.

After these results, the next objective of the project was trying to get more thrust by optimizing the design of the nozzle.

In order to develop this idea several changes in the shape of the nozzle were done. Conical nozzles are larger than the bell type, so the optimization should be carried out mainly by increasing its length. Actually the most length has the nozzle the better thrust is obtained from it. The problem is that a designer has to keep in mind that his/her design has to fit in a structure, it has restrictions due to the properties of the material, loads that affect the structure, heat flux that needs to be transmitted to the exterior and obviously it has a budget to build it and cannot design a nozzle as large as he/she wants.

So after all this restrictions, the optimize design of the conical nozzle have these dimensions:

Caracteristic	Dimension (cm)
Nozzle length	406
Throat diameter	26
Throat location	25
Inlet diameter	51
Exit diameter	230

Table 5.2: Dimensions of the optimized design.

As it was mentioned before, the length of this design needed to be increased, so after several analyses an increase of 86 cm, which is a design approximately 20 % longer, has been chose and obviously the location of the throat has been increased as well. The inlet diameter has been reduced in 11 cm but the exit diameter keeps being the same.

In the picture below the optimize design is shown:

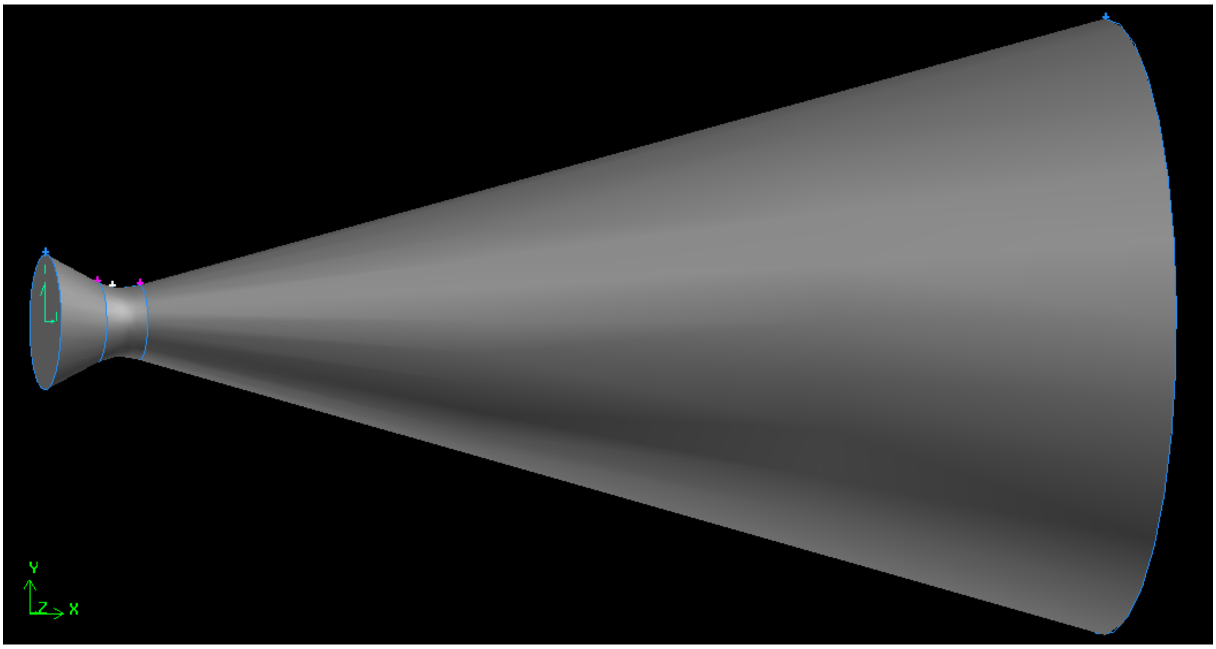


Figure 5.5: Optimized design of the conical nozzle.

In the picture the whole nozzle is shown although like in the other analysis only one quarter of it was actually revolved.

In this design the shape of the throat was curved as well in order to avoid turbulences after the throat and concentration of the flux in the centre line.

As in the previous design, the inlet pressure keeps being the same, so that way a comparison between these results could be done.

In the following picture, the velocity vectors coloured by velocity magnitude (m/s) of one of the symmetrical faces is shown:

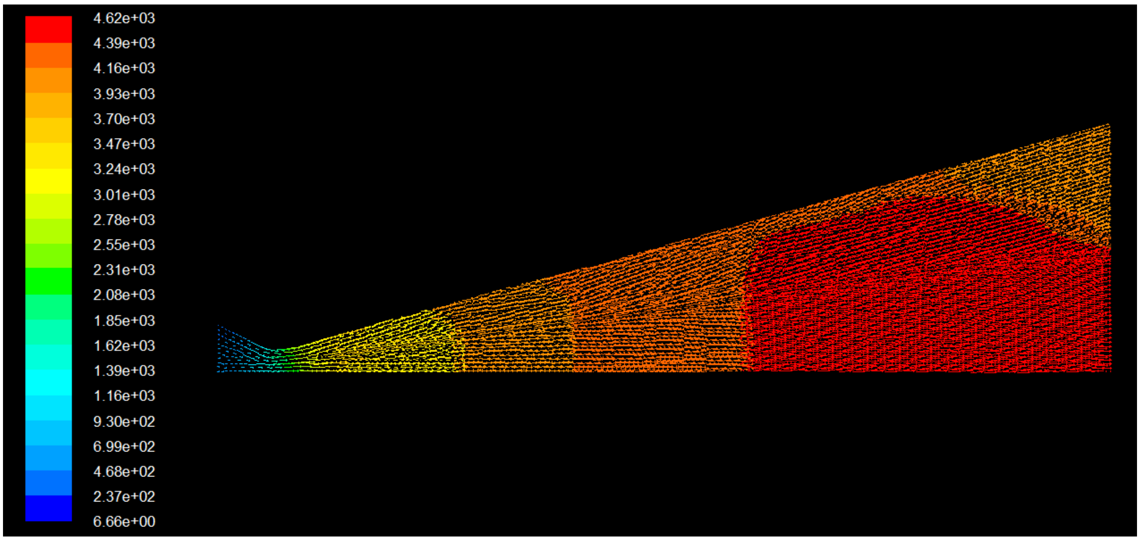


Figure 5.6: Velocity vectors colored by velocity magnitude.

The highest velocities of the gases that go through the nozzle are in the region near the exit because this is the supersonic region of the nozzle where the velocity of the gases increases when the area is increased. As we can see in the picture above, now the velocity of the exhaust gases has been increased from 3,730 m/s to 4,620 m/s due to the changes in the nozzle, so obviously the thrust produced will be increased as well. In this case, the shock waves start a little bit earlier than in the previous design.

In this particular case, the force against the walls of the nozzle produced by the gases that pass through the nozzle which has the same quantity but is opposite in direction to the thrust produced is 1,928,326.9 N.

zone name	pressure force n	viscous force n	total force n	pressure coefficient	viscous coefficient	total coefficient
wall	1902163.3	26163.646	1928326.9	3105572.7	42716.158	3148288.8
net	1902163.3	26163.646	1928326.9	3105572.7	42716.158	3148288.8

Figure 5.7: Thrust produced by the optimized design.

So, by changing the shape of the nozzle the thrust produced has been increased in:

(EQ 12) $$\Delta F = 1{,}928{,}326.9 - 1{,}667{,}521.7 = 260{,}805.2\ N$$

And the improvement ratio of the conical nozzle is:

$$Improvement\ ratio = \frac{Thrust\ of\ the\ second\ design}{Thrust\ of\ the\ first\ design} = \frac{1{,}928{,}326.9}{1{,}667{,}321.7} = 1.156 \quad \text{(EQ 13)}$$

So, the improvement percentage is 15.6%.

- **Bell nozzle**

Once the conical nozzle has been already optimized, the objective now was obtaining more thrust with a bell type nozzle than with a conical one.

In order to develop this idea, first of all I had to design the nozzle. As in the conical one, the more length the nozzle has the more thrust it is obtained. But as it has been mentioned, the designer has some restrictions that does not make possible for him/her designing a nozzle as long as he/she wants and also long nozzles has a penalty in the rocket propulsion system mass and in the design complexity as well. So, with these premises the dimensions of the design are these:

Caracteristic	Dimension (cm)
Nozzle length	320
Throat diameter	26
Throat location	15
Inlet diameter	62
Exit diameter	230

Table 5.3: Dimensions of the bell-shaped nozzle.

These dimensions are the same than the dimensions of the first one, so that way a comparison between conical and bell type nozzle can be made.

Like the dimensions of this design are the same than the first nozzle, and the inlet and outlet pressure are equal as well, the only way of improving the results and get more thrust was by modifying the contour of the nozzle. The shape of the first design of a bell nozzle has the dimensions already mentioned and also the coordinates (100,65) and (200,95).

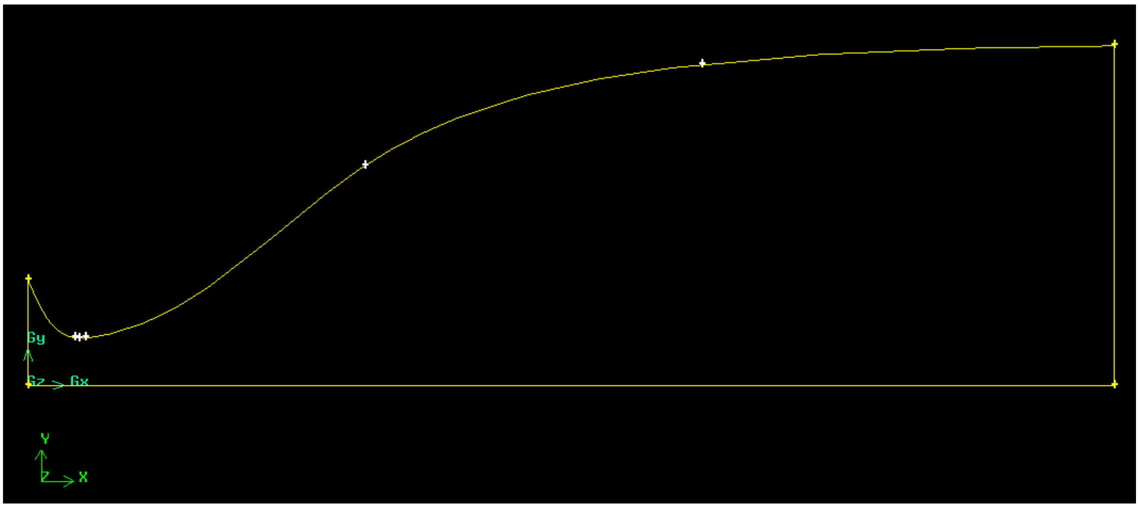


Figure 5.8: Initial face of the bell-shaped design.

So, revolving this contour the new nozzle would be built and it would seem like this:

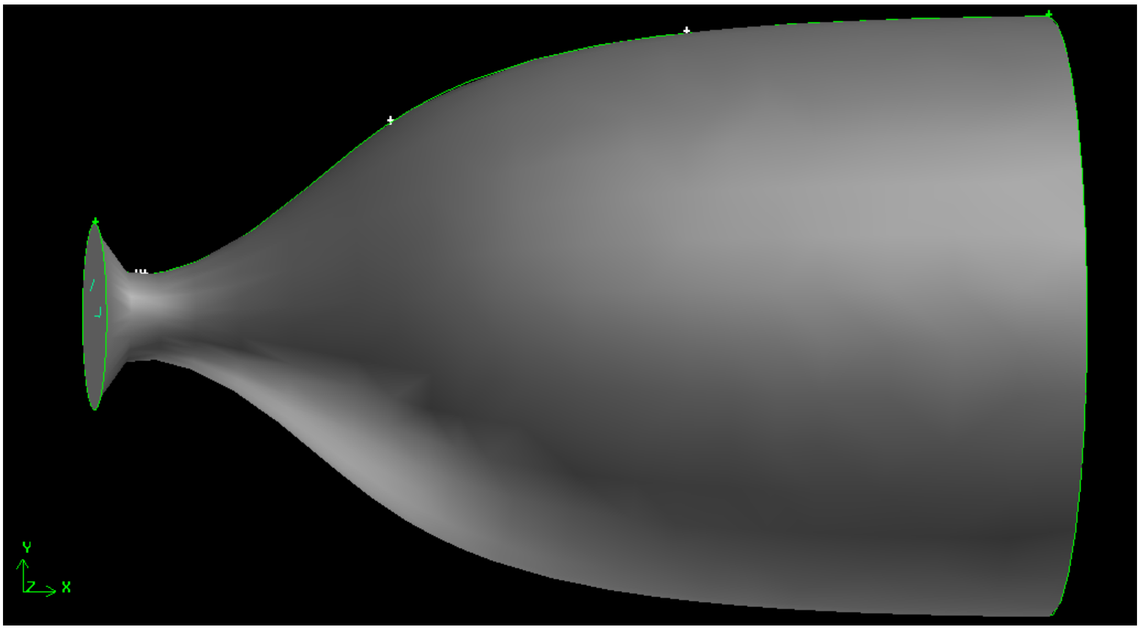


Figure 5.9: Initial bell-shaped nozzle.

The problem of this design is that the initial pendant after the throat was too high; this produced some turbulence in the region located near the wall after the throat.

So, a new contour for the nozzle was built with the objective of avoids this problem. Now, the coordinates were (100,50) and (200,85) as we can see in the picture:

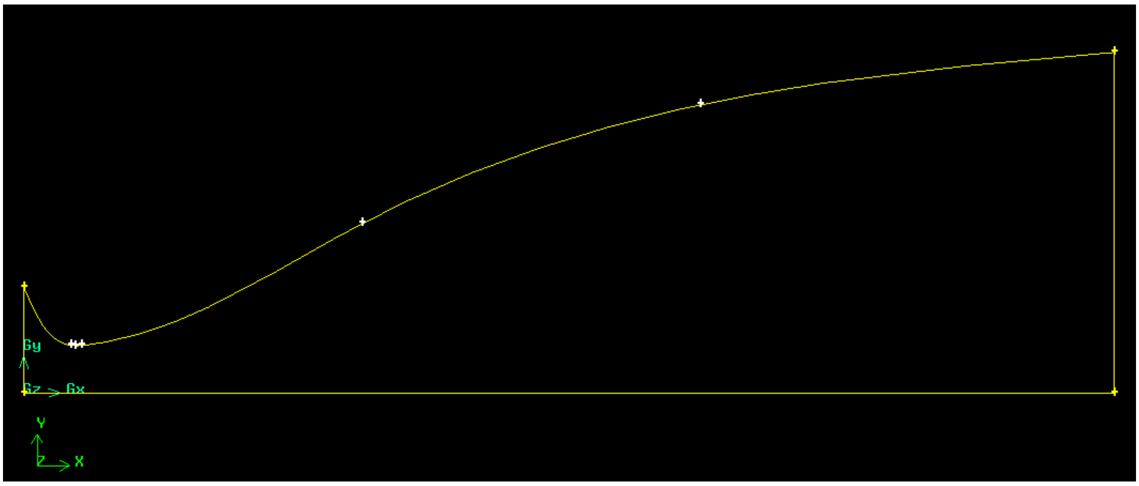


Figure 5.9: Final face of the bell-shaped nozzle.

And after revolving it, this is the final shape of the bell type nozzle:

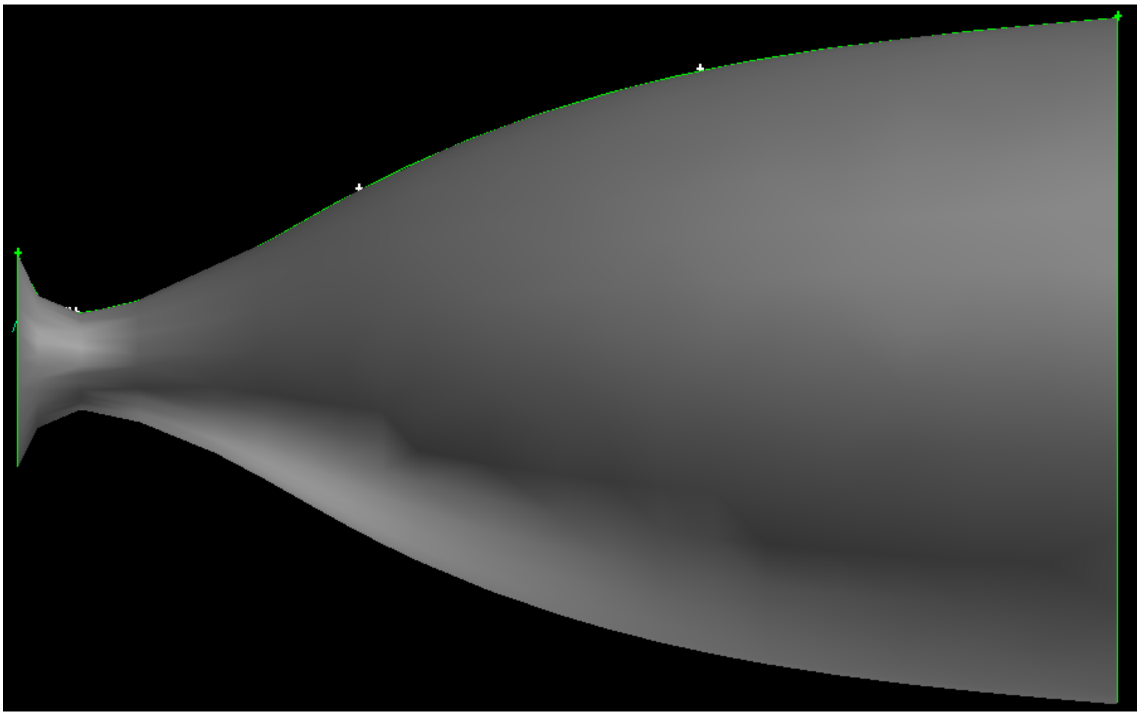


Figure 5.10: Final bell-shaped nozzle.

Like in the other designs, only one quarter of the design was actually analyzed with Fluent and the inlet and outlet pressures keep being the same.

The velocity vectors coloured by velocity magnitude (m/s) of one of the symmetrical faces is shown in the picture below:

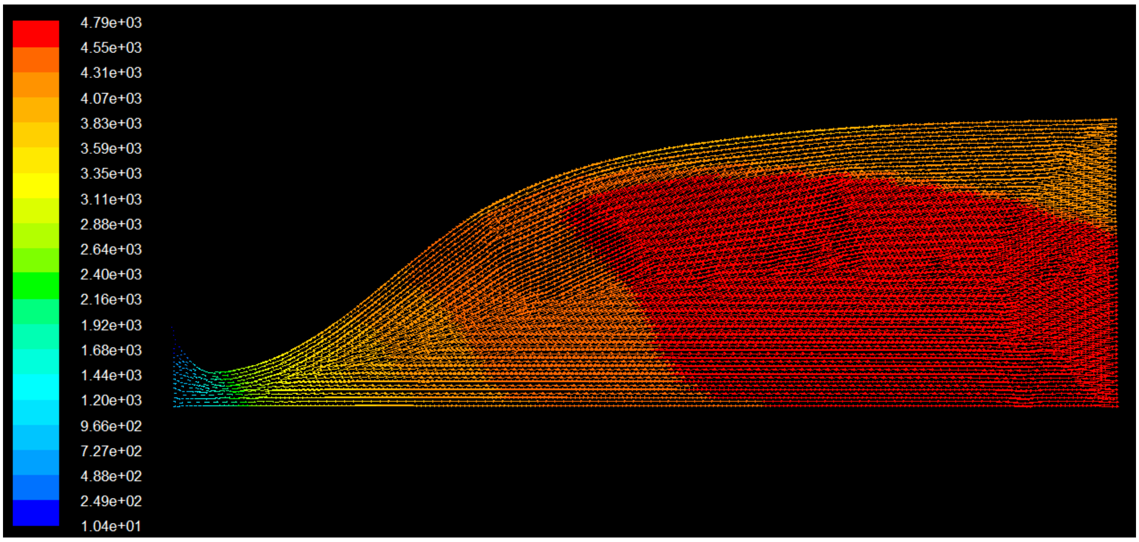


Figure 5.11: Velocity vectors colored by velocity magnitude (m/s) of the bell nozzle.

Just like in the other analysis, the highest velocity of the gases that go through the nozzle is in the region near the exit because this is the supersonic region of the nozzle where the velocity of the gases increases when the area is increased. Now, the maximum velocity of the exhaust gases are 4,790 m/s, a value quite similar to the speed of the conical design, so we can anticipate that their thrust will be similar as well. This speed keeps fitting with the usual velocities of bipropellant rocket engines, and the overexpansion produced by the difference pressure between the exit pressure and the ambient pressure starts earlier than in the previous designs.

In this particular case, the force against the walls of the nozzle produced by the gases that pass through the nozzle which has the same quantity but is opposite in direction to the thrust produced is 2,002,693.7 N. As it has been predicted, both thrust are quite similar.

Now, a comparison between the bell type nozzle and the conical one can be made.

First of all, the difference in thrust between the first design of a conical nozzle and the bell type nozzle is:

(EQ 14) $$\Delta F = 2{,}002{,}693.7 - 1{,}667{,}521.7 = 335{,}172 \text{ N}$$

And the improvement ratio is:

(EQ 15) $$Improvement\ ratio = \frac{Thrust\ of\ the\ third\ design}{Thrust\ of\ the\ first\ design} = \frac{2{,}002{,}639.7}{1{,}667{,}521.7} = 1.20$$

This means that there is an improvement of 20% between those designs. It is important to keep in mind that the dimensions of these design are the same, the only difference between them is the contour of the wall which in the case of the bell-shaped nozzle, allows a better expansion of the exhaust gases and minimize losses.

Now, a comparison between the optimized design of the conical nozzle and the bell type design is shown:

(EQ 16) $$\Delta F = 2{,}002{,}639.7 - 1{,}928{,}326.9 = 74{,}312.8\ N$$

And the improvement ratio is:

(EQ 17) $$Improvement\ ratio = \frac{Thrust\ of\ the\ third\ design}{Thrust\ of\ the\ first\ design} = \frac{2{,}002{,}639.7}{1{,}928{,}326.9} = 1.039$$

There is only an improvement of 3.9% in the bell type nozzle respect to the conical one. But the main result in this case is that the bell nozzle gets similar values of thrust respect the conical design being 86 cm less long, which means an important reduction of the costs of the nozzle.

With this design, the analyses of conventional nozzles have finished. From now on, unconventional nozzles will be analyzed.

5.2.2 Unconventional nozzles

- **Aerospike nozzle**

The first design of an unconventional nozzle is an aerospike nozzle. As it has been mentioned before, this kind of nozzle has a structure quite different from the other nozzles because the most part of it is open to the atmosphere.

The first design of this kind of nozzle has the follow dimensions:

Caracteristic	Dimension (cm)
Nozzle length	42
Throat area	100 cm2
Throat location	0
Inlet area	100cm2
Distance from end of thruster to end of ramp	33
Truncation as percent of total aerospike length	20%
sweeping length	25

Table 5.4: Dimensions of the aerospike nozzle.

This is the picture of the nozzle which also contains the follow coordinates: (±31,-3.21), (±30.57,-6.42), (±25.13,-21.38), (±22.45,-28.88), (±20.32,-36.35) and (±32.07,-21.92).

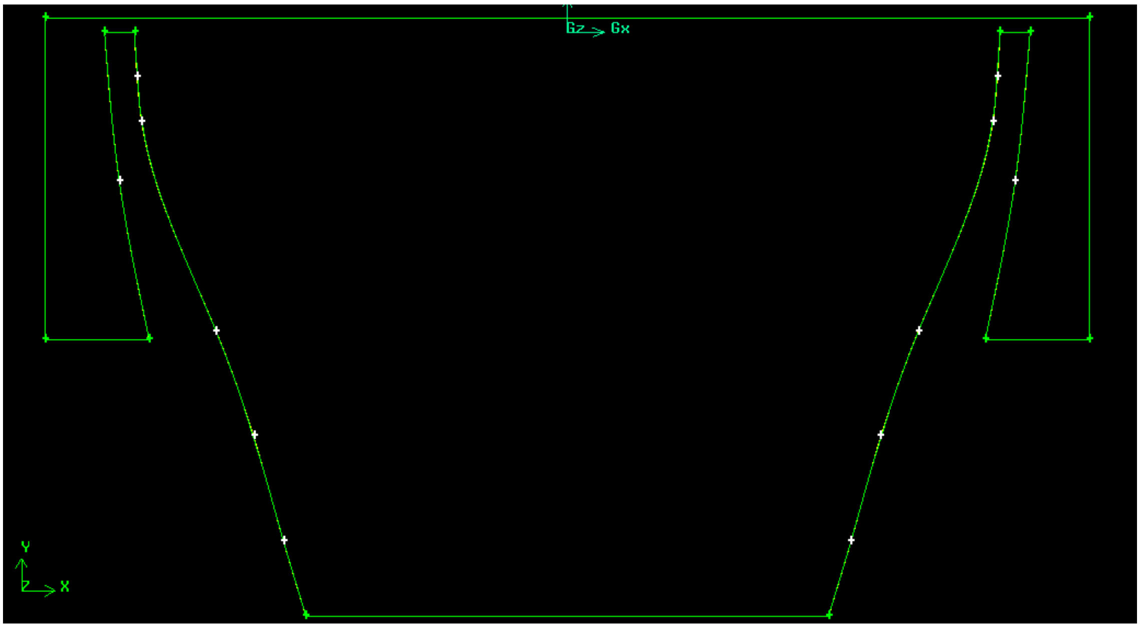


Figure 5.12: Initial linear aerospike design.

As it is shown in the picture in both sides of the nozzle there are two elements which function is drive out the gas flow near the walls in order to produce more thrust. In next optimizations these elements will be removed.

Sweeping this face, the design would be like the one in the picture above.

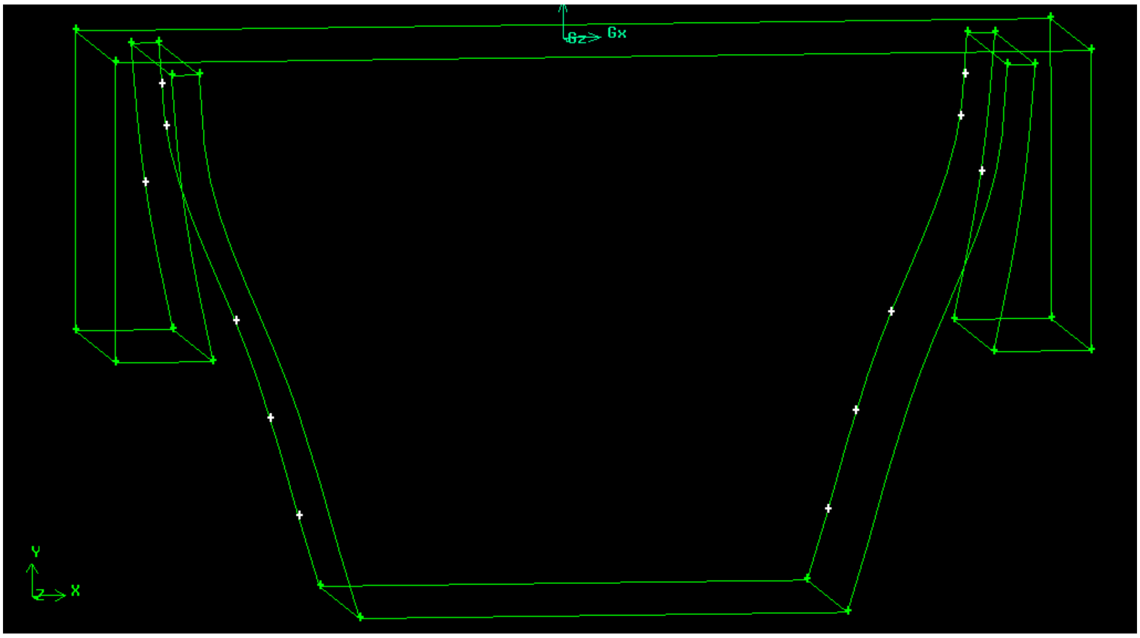


Figure 5.13: Initial linear aerospike design.

Like this design is a small aerospike nozzle, where the pressure in the chamber is 506800 Pa, the thrust produced will be low as well and a rocket with this kind of propulsion system would need several nozzles like this in order to be able to flight. Its thrust is 386,123.59 N.

So, like the thrust produced was not enough, the optimized design should be bigger and with a higher pressure chamber. With these objectives the second design of an aerospike nozzle has these dimensions:

Caracteristic	Dimension (cm)
Nozzle length	332
Throat area	10573 cm2
Throat location	0
Inlet area	10573 cm2
sweeping length	236
width of the nozzle at the end	236
width of the nozzle at the beginning	640

Table 5.4: Dimensions for the final aerospike design.

This is the picture of the nozzle which also contains the follow coordinates: (±31,-10), (±12.1,-41) and (±34,-10).

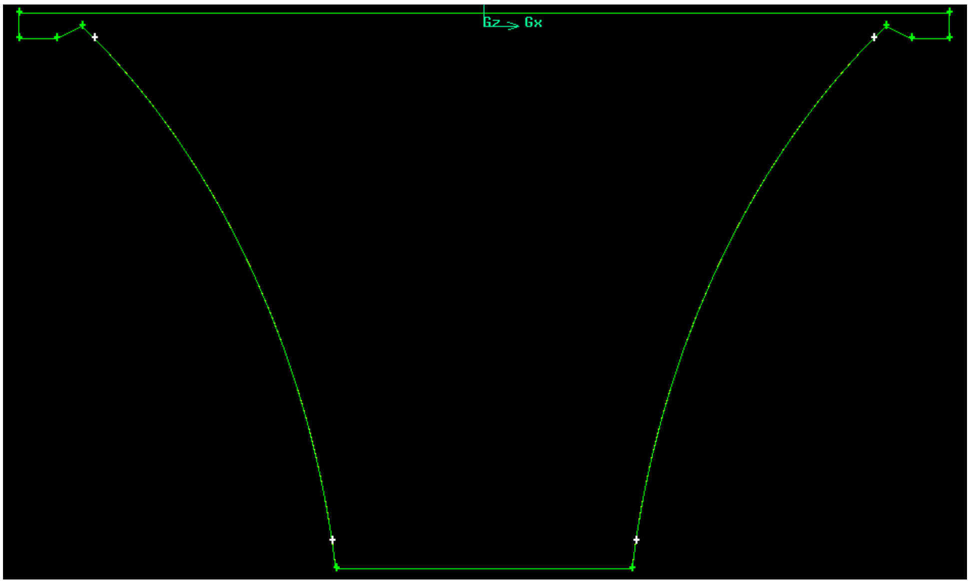


Figure 5.14: Face of the final aerospike design.

As it was mentioned before, due to the shape of the nozzle and the orientation of the pressure inlet region in this design the elements that were in both sides of the nozzle have been removed.

Like in the previous design, sweeping this face, the design would be like the one in the picture above.

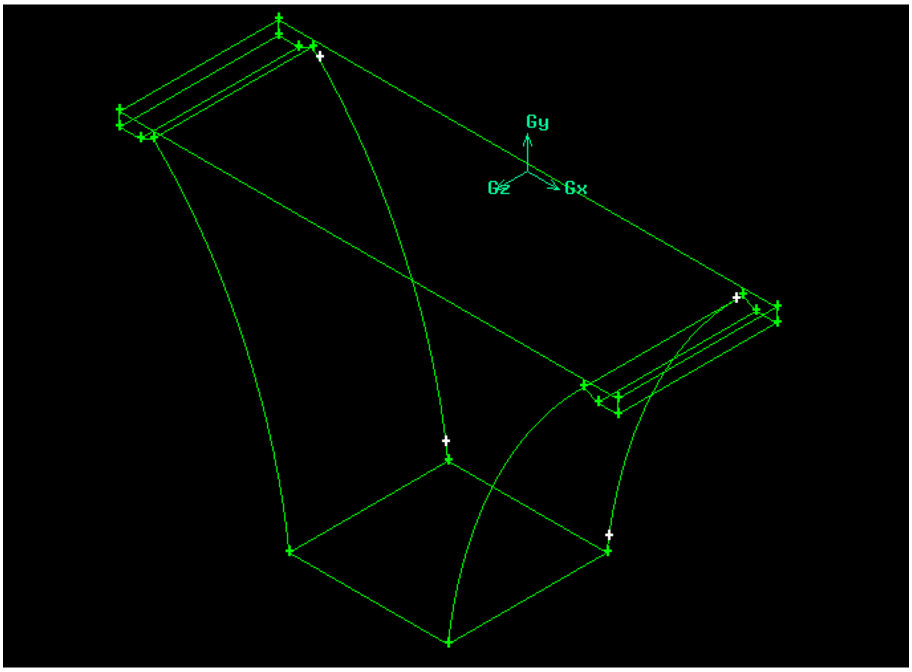


Figure 5.15: Final aerospike nozzle design.

In order to calculate the thrust produced, two different regions with different kind of mesh were created. One region with a very accurate mesh with a small size of cell located near both sides of the wall of the nozzle and another one much bigger surrounding the nozzle with a less accurate mesh as it is shown in the following pictures:

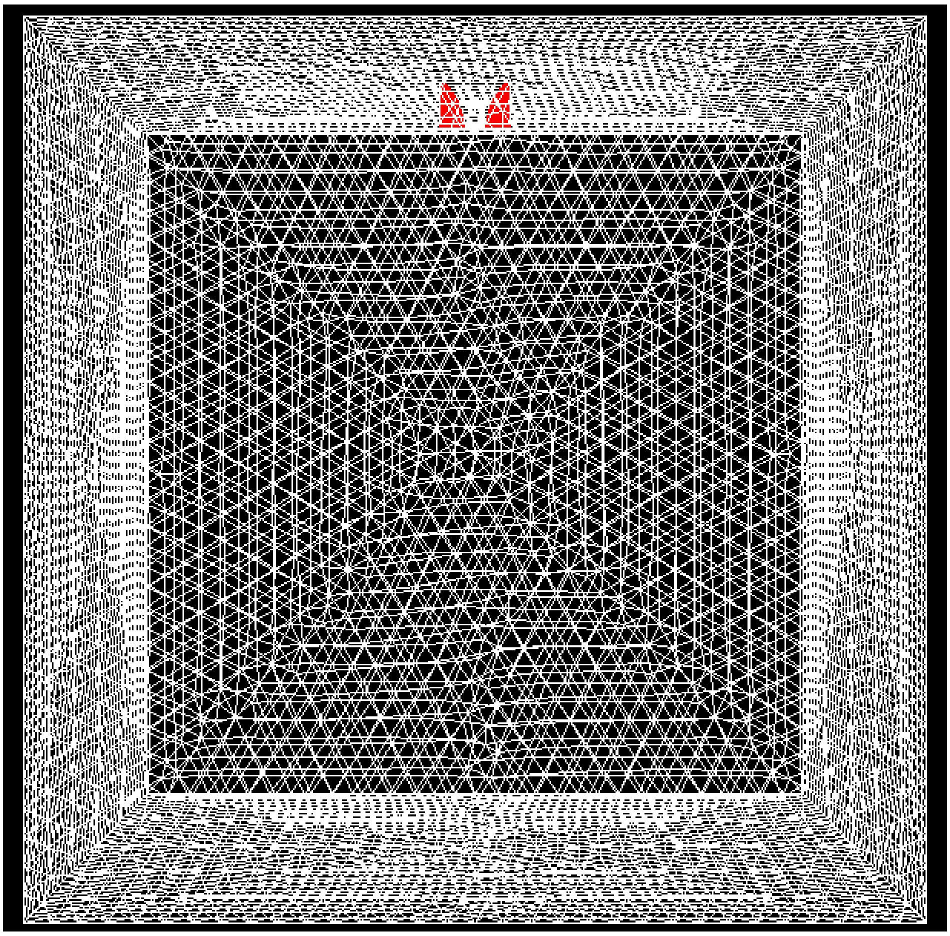


Figure 5.16: Surroundings of the nozzle.

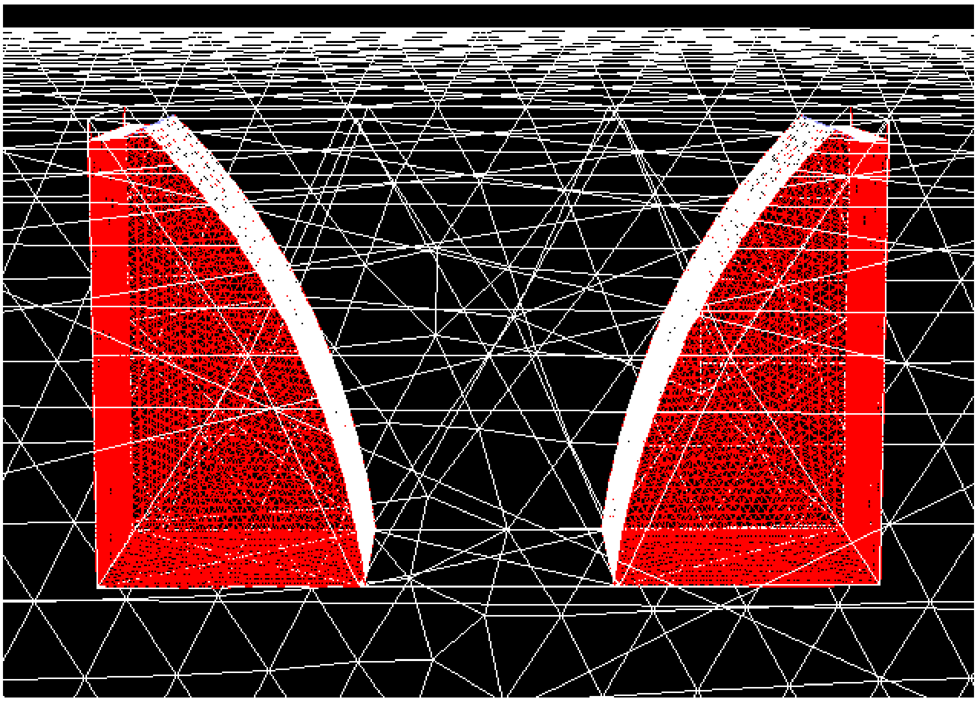


Figure 5.17: Proximities of the nozzle.

In this particular case, the inlet pressure is 15,300,000 Pa which is higher than the first design of an aerospike nozzle, but it is lower than the chamber pressure of the conventional nozzles that have been already analyzed. After the iterations, this is the contours of velocity of the nozzle:

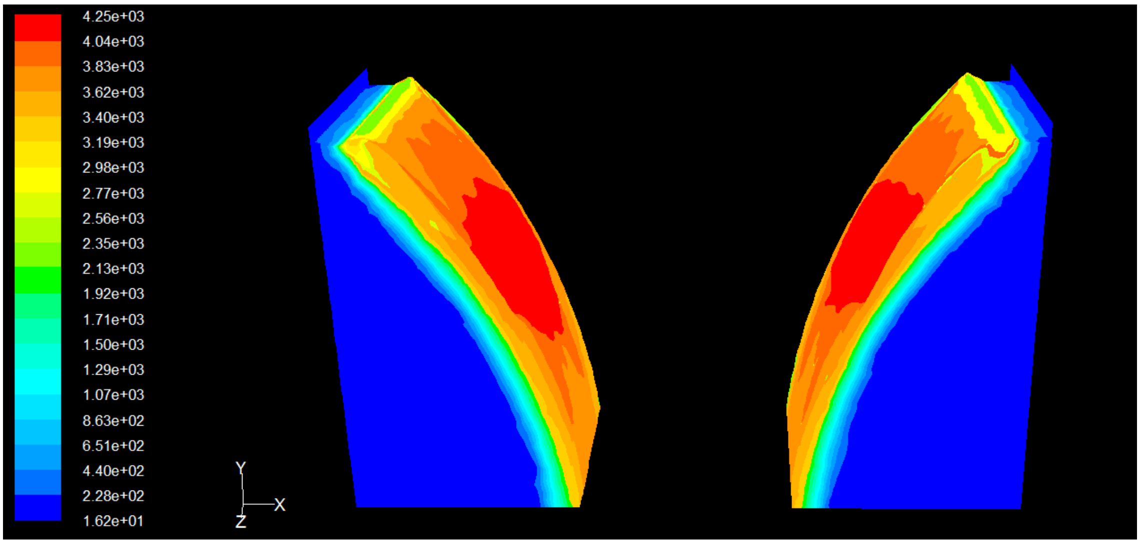


Figure 5.18: Contours of velocity (m/s) of the nozzle.

In this case, the thrust produced by this nozzle is 2,079,775.5 N. So, it is been proved that these kind of design are as good propulsion systems as the conventional nozzles with the benefit of reaching this amount of thrust with a smaller nozzle.

- **E-D nozzle**

The expansion-deflection nozzle is the last of the nozzles analyzed in this project and as well as the aerospike nozzle, it is one of the most important unconventional designs.

As it is been mentioned before, this kind of nozzles are similar to the normal convergent-divergent nozzle, but it has a solid surface along the centreline of the nozzle which provides the benefit that this kind of nozzle can match exit pressure over a larger range of flight conditions but obviously they are heavier.

The first design of this kind of nozzle has the follow dimensions:

Caracteristic	Dimension (cm)
Nozzle length	273
Throat diameter	92
Throat location	50
Inlet diameter	177
Exit diameter	373

Table 5.6: Dimensions of expansion-deflection nozzle.

This is a picture of half nozzle where other coordinates of the contour of the wall are shown as well, the coordinates are: (0,0.885), (0.23,0.66), (0.38,0.47), (0.39,0.465), (0.41,0.63), (0.27,0.22), (0.56,0.31), (0.56,0.63), (0.58,0.62), (0.56,0.95), (0.77,0.48) and (0.94,0) being these coordinates in meters.

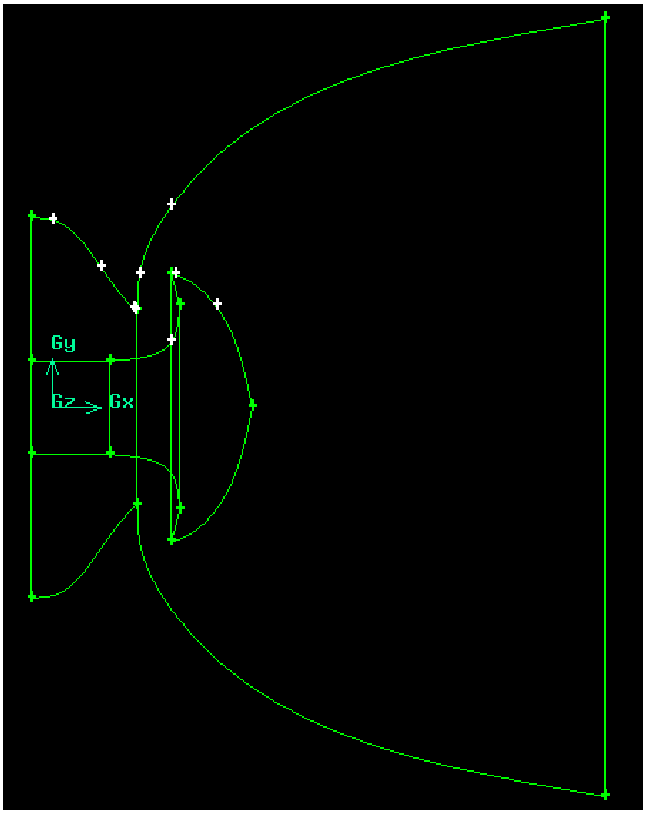


Figure 5.19: Face of the initial expansion-deflection nozzle.

The problem with this design was that the pendant after the throat was too high so some changes in the contour of the nozzle were done in order to avoid the turbulences after the throat.

In the next pictures the new contour of the nozzle is shown, most of the previous coordinates keep being the same, but instead of (0.41,0.63) there is (0.43,0.63).

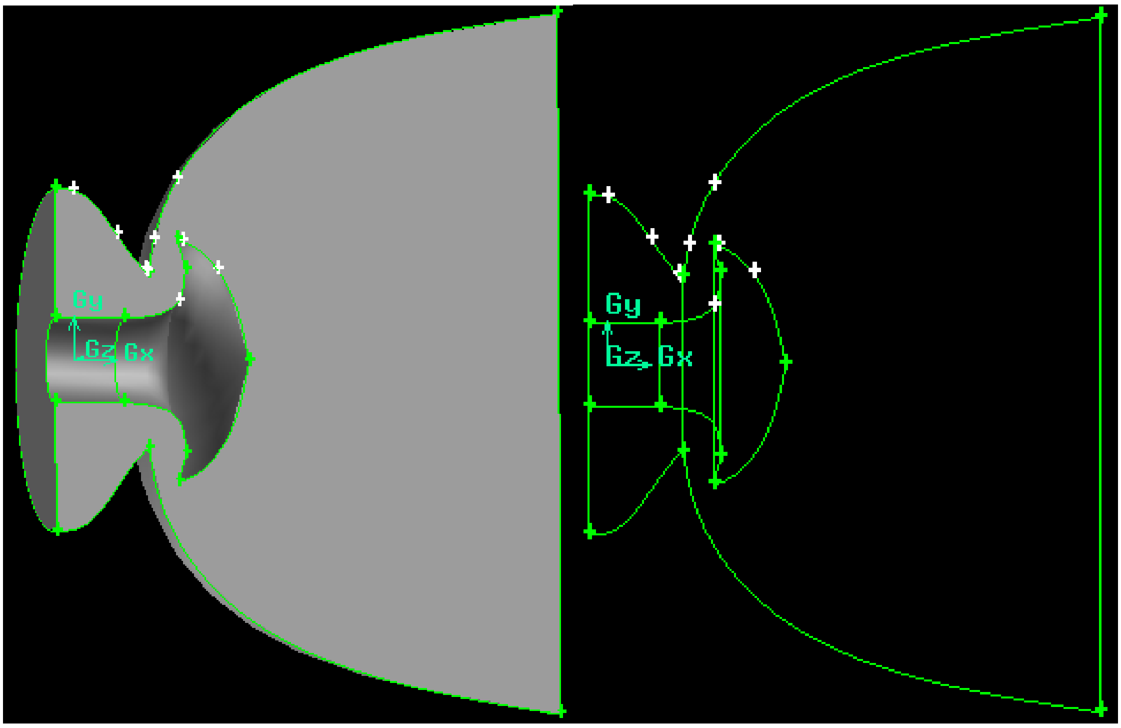


Figure 5.20: Final expansion-deflection design.

Although in the picture half of the nozzle is shown, only one quarter of it was actually built and analyzed in Fluent.

For this particular nozzle, the pressure in the chamber is 11,800,000 Pa and like the exit is open to the atmosphere; its pressure is 101,325 Pa. So, due to the inlet pressure in this case is lower than the pressure of the other designs developed in this project; its thrust is going to be lower as well.

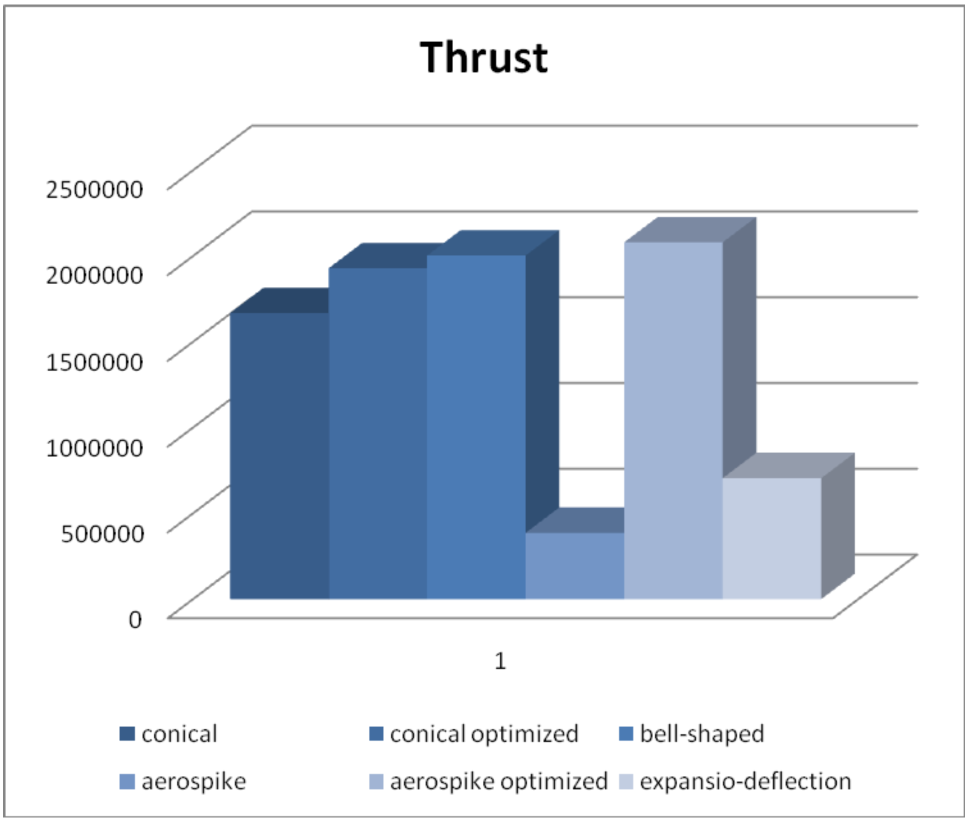
After the iterations, the force against the walls of the nozzle produced by the gases that pass through the nozzle which has the same quantity but is opposite in direction to the thrust produced is 704,960.5 N. As it was presumed before, the thrust produced by this nozzle is quite lower than the results from the other designs due to its lower chamber pressure, but it is remarkable that this amount of thrust has been reached with less length than a conventional nozzle.

5.3. DISCUSSION:

After running all the analyses, these are the results of thrust for each design:

NOZZLE	THRUST (N)
First conical nozzle	**1,667,521.7**
Optimized conical nozzle	**1,928,326.9**
Bell-shaped nozzle	**2,002,693.7**
First aerospike design	**386,123.59**
Optimized aerospike design	**2,079,775.5**
Expansion-Deflection nozzle	**704,960.5**

Table 5.7: Thrust produced by each nozzle.



Graph 5.1: Thrust produced by the nozzles.

As it is shown, both conventional and unconventional nozzles are valid for rocket engines due to its similar values of thrust, mainly between aerospike and conventional nozzles. But the main advantage of the unconventional nozzles is that they achieve this thrust with shorter lengths, resulting in an easier structure.

The main reason for achieving such goods values of thrust with unconventional nozzles is due to the altitude compensation because conventional nozzles are optimum for only one altitude while unconventional nozzles has optimum performance from sea level to the space.

But the main disadvantage of unconventional nozzles is that its design and manufacturing process are more complex than in conventional nozzles, resulting in more expensive kind of configurations.

From the analyses of the nozzles, it can be presumed that the geometry of the convergent section and the throat are not very important in order to achieve high performance. That is because the subsonic flow can be easily turned into very low pressure drop with any shape, instead the contour of the diverging section is quite important to reach an appropriate expand of the exhaust gases.

6. CONCLUSIONS

This project has been carried out with some software problems that have slowed the analysis of the nozzles because heat transmission through the wall of the nozzles was supposed to be done as well as the thrust. But due to the enormous difficulty and time required, in designing and analyzing several conventional and unconventional nozzles, only the thrust produced was analyzed with CFD and the heat transfer in the nozzle have been analyzed under a theoretical point of view.

The first part of the project was the design of each nozzle, where several factors had to be taken into account like: maximize their performance, ensure structural integrity, high reliability, safety conditions and cost. Although during the development of this project the most important has been optimize the performance.

In the picture below all the designs built in Gambit are shown:

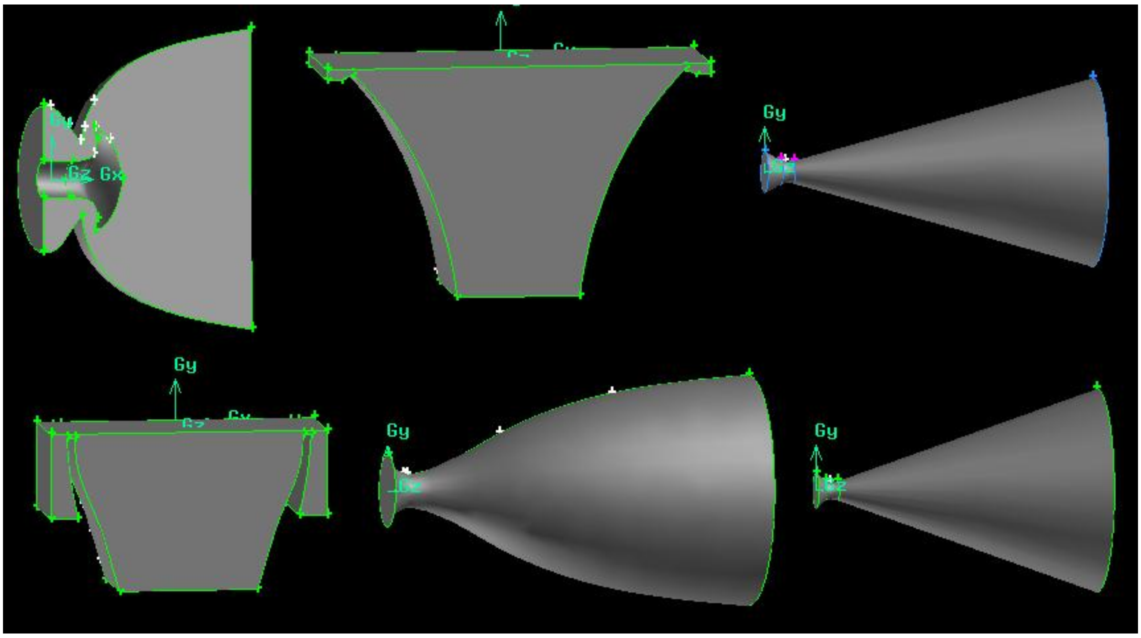


Figure 6.1: Nozzles design.

Once the design was built, it was time to analyze them with CFD. The thrust produced by each design is shown in the table below:

NOZZLE	THRUST (N)
First conical nozzle	**1,667,521.7**
Optimized conical nozzle	**1,928,326.9**
Bell-shaped nozzle	**2,002,693.7**
First aerospike design	**386,123.59**
Optimized aerospike design	**2,079,775.5**
Expansion-Deflection nozzle	**704,960.5**

Table 6.1: Thrust produced by each nozzle.

The three first designs are conventional nozzle and the last three are unconventional configurations.

Nowadays, conical nozzles are rarely used as rocket engines because long nozzles have a penalty in the rocket propulsion system mass and also in design complexity. The improvement between the optimized and non-optimized design of this project was of 15.6%, an important upturn mainly by increasing its length 86 cm.

The expansion produced in bell-shaped nozzles is more efficient than a conical one of similar area ratio and length because the wall contour is designed to minimize losses. Engineers developed this wall contour that changes gradually enough to avoid oblique shocks in order to obtain higher performance and shorter length designs. The improvement between this nozzle and the first conical nozzle was of 20 % and between the bell nozzle and the second design was of 3.9 %.

The needed of some form of altitude compensation in the nozzles that were able to operate from sea level to orbit is the reason of the development of unconventional nozzles. Similar values of thrust respect a conventional nozzle can be achieved at high altitudes with only one quarter of their length, and with better performance in low altitudes with an aerospike nozzle. Other advantages of this configuration are: lower vehicle drag, thrust vectoring and lower vehicle weight, although it has some other disadvantages like more difficult cooling or manufacturing and designing process.

Expansion-deflection nozzles have long been considered for rockets due to its potential increments in performance offered through altitude compensation. But there are several difficulties with it because a more complex combustion chamber and throat design is needed, and the non-isentropic flow phenomena such as viscosity and shock waves on flow field composition.

As it is shown, unconventional nozzles can reach values of thrust as higher as the conventional configurations but with a shorter designs which means less structural problems.

The main reason for achieving such goods values of thrust with unconventional nozzles lies in the altitude compensation because conventional nozzles are designed for one optimum altitude while unconventional nozzles has optimum performance from sea level to the orbit.

The heat created during the combustion process in a rocket is contained within the exit gases. Most of this heat is expelled along with the gas flow that contains it, although a significant portion (1 to 5 %) of this heat is transferred to the walls of the engine. This heats transmitted through the walls is a serious structural problem for the nozzle and needs to be solved.

The heat transfer intensity can vary from less than 50 W/cm2 to over 16 kW/cm2 and the highest values are for the nozzle throat region and the lowest for the gas generators and nozzle exit sections.

Several cooling techniques like regenerative cooling or radiation cooling were invented with the objective of prevent the chamber and nozzle walls from becoming too hot because the materials used for build the walls of the nozzle lose strength and become weaker as temperature is increased.

7. RECOMENDATIONS

First of all, I would like to remark that this project has provided the basis and foundations of an analysis in performance of 3 dimensional nozzles.

Analyze other kinds of unconventional configurations like annular bell nozzles will be a recommendation for further work as well as analyze two-step nozzles that would provide altitude compensation to conventional nozzles.

8. REFERENCES

[1]. Marty Curry. 2002, *NASA* (online), retrived via Mozilla. http://www1.dfrc.nasa.gov/Gallery/Graphics/X-33/index.html

[2]. 199?, *Universidad Pública de Valencia* (online), retrived via Mozilla. http://www.upv.es/satelite/trabajos/Grupo5_b99.00/Propulsi.htm

[3], [16]. Robert A. Braeunig. 1997, *Rocket & Space Technology* (online), retrived via Mozilla. http://www.braeunig.us/space/propuls.htm

[4]. George P.Sutton.2000, *Rocket propulsion elements* (New York, Wiley-Interscience Publication). John Wiley 2000. Page 58.

[5], [6]. Rocketdyne. 1999, *Aerospaceweb* (online), retrived via Mozilla. http://www.aerospaceweb.org/design/aerospike/compensation.shtml

[7], [10]. R.A. O'Leary and J. E. Beck. 1992, Pratt & Whitney Rocketdyne Technical Consulting (online), retrived via Mozilla. http://www.pwrengineering.com/articles/nozzledesign.htm

[8]. George P.Sutton.2000, *Rocket propulsion elements* (New York, Wiley-Interscience Publication). John Wiley 2000. Page 76.

[9]. George P.Sutton.2000, *Rocket propulsion elements* (New York, Wiley-Interscience Publication). John Wiley 2000. Page 90.

[11]. Huzel and Huang.1967, *Aerospaceweb* (online), retrived via Mozilla. http://www.aerospaceweb.org/design/aerospike/compensation.shtml

[12]. Rocketdyne. 1999, *Aerospaceweb* (online), retrived via Mozilla. http://www.aerospaceweb.org/design/aerospike/x33.shtml

[13]. Dressler, Gordon A; Mueller, Thomas J.; Rotenberger, Scott J. 2003, *free patents online* (online), retrived via Mozilla. http://www.freepatentsonline.com/6591603.html

[14]. George P.Sutton.2000, *Rocket propulsion elements* (New York, Wiley-Interscience Publication). John Wiley 2000. Page 285.

[15]. George P.Sutton.2000, *Rocket propulsion elements* (New York, Wiley-Interscience Publication). John Wiley 2000. Page 311.

[17]. Lynne Haas. 2002, Pratt & Whitney Rocketdyne Technical Consulting, retrived via Mozilla.
http://209.85.135.132/search?q=cache:Pp31a84SXxEJ:www.engineeringatboeing.com/dataresources/LiquidRocketEngineNozzles.ppt+Boeing,+Rocketdyne+Propulsion+and+Power+Liquid+Rocket+Engine+Nozzles&cd=1&hl=es&ct=clnk&gl=es

George P.Sutton.2000, *Rocket propulsion elements* (New York, Wiley-Interscience Publication). John Wiley 2000.

J.H.Ferzinger, M.Peric. 2002, *Computational methods for fluid dynamics* (Springer, New York). BertelsmannSpringer.

T.D.Eastop, D.R.Croft, W.E.Watson.1963, *Applied thermodynamics for engineering technologist* (Longman, Singapore).

Jack D. Mattingly. 1996, *Elements of gas turbine propulsion* (Amer Inst Of Aeronautics, New York)

Barnes W. McCormick. 1979, *Aerodynamics, aeronautics, and flight mechanics* (Wiley, 1995).

J.Spence,A.S.Tooth. 1994, *Pressure vessel design: concepts and principles* (E&F N Spoon, Oxford)

Angel Franco Garcia. 2006, *Fisica con ordenador* (online),retrived via Mozilla.
http://www.sc.ehu.es/sbweb/fisica/dinamica/cohete1/cohete1.htm

Robert A. Braeunig. 1997, *Rocket & Space Technology* (online), retrived via Mozilla.
http://www.braeunig.us/space/propuls.htm

199?, *Universidad Pública de Valencia* (online), retrived via Mozilla.
http://www.upv.es/satelite/trabajos/Grupo5_b99.00/Propulsi.htm

N.V. Taylor C.M.Hempsell J.MacFarlane, R. Osborne, R. Varvill, A.Bond, S.Feast, 2008, *Project Stern* (online), retrived via Mozilla.
http://www.projectstern.co.uk/information.html

J.C. Hyde and G.S. Gill. 1976, *NASA* (online), retrived via Mozilla. http://trs.nis.nasa.gov/archive/00000186/01/sp8120.pdf

R.A. O'Leary and J. E. Beck. 1992, Pratt & Whitney Rocketdyne Technical Consulting (online), retrived via Mozilla. http://www.pwrengineering.com/articles/nozzledesign.htm

Huzel and Huang.1967, *Aerospaceweb* (online), retrived via Mozilla. http://www.aerospaceweb.org/design/aerospike/compensation.shtml

Rocketdyne. 1999, *Aerospaceweb* (online), retrived via Mozilla. http://www.aerospaceweb.org/design/aerospike/x33.shtml

200?, *Airbone engineering* (online), retrived via Mozilla. http://www.airborneengineering.co.uk/stern.html

Departamento de Ingenieria Mecanica. 200?, *Universidad de la Rioja* (online), retrived via Mozilla. http://www.unirioja.es/cu/manuruiz/simulacion%20que%20es%20cfd.shtml

2006, *Cavendish CFD* (online), retrived via Mozilla. http://www.cavendishcfd.com/index.htm

Lt Gen Jay W. Kelley. 1994, *Spacecast 2020* (online), retrived via Mozilla. http://www.fas.org/spp/military/docops/usaf/2020/app-i.htm

Dressler, Gordon A; Mueller, Thomas J.; Rotenberger, Scott J. 2003, *free patents online* (online), retrived via Mozilla. http://www.freepatentsonline.com/6591603.html

Lynne Haas. 2002, Pratt & Whitney Rocketdyne Technical Consulting, retrived via Mozilla. http://209.85.135.132/search?q=cache:Pp31a84SXxEJ:www.engineeringatboeing.com/dataresources/LiquidRocketEngineNozzles.ppt+Boeing,+Rocketdyne+Propulsion+and+Power+Liquid+Rocket+Engine+Nozzles&cd=1&hl=es&ct=clnk&gl=es

200?. *Air-Attack.com* (online), retrived via Mozilla. http://www.air-attack.com/page/5/NASA-SR-71-LASRE-Project.html

Tom Benson. 200?, *NASA* (online), retrived via Mozilla.

http://exploration.grc.nasa.gov/education/rocket/nozzle.html

Tom Benson. 200?, *NASA* (online), retrived via Mozilla.

http://exploration.grc.nasa.gov/education/rocket/rktthsum.html