Journal of Multidisciplinary Engineering Science and Technology (JMEST)
ISSN: 2458-9403
Vol. 5 Issue 10, October - 2018

Design And Optimization Of A Combustion Chamber Through The Analysis Of Flow Patterns

P. Martinez-Torres, A. Clemente-Mendoza, L. Moreno-Pacheco
Postgraduate Studies and Investigation Section
National Polytechnic Institute
Mexico City, Mexico
pamelabmtz@outlook.com, ls_moreno@hotmail.com

***Abstract*—In this investigation, the design and numerical optimization of a combustion chamber was made, for which two analysis were performed: (1) simulating only the flow behavior in order to determine the optimal combustion chamber geometry, and (2) simulating combustion considering diesel as fuel and varying both, swirlers configuration and angle of fuel injection.**

Through the results analysis it was determined that the optimal configuration of swirlers and injectors contributes to generate a pressure drop sufficiently high, which helps to achieve a flame anchoring in the primary zone, avoiding hot spots at the outlet.

Keywords*—*Angle of injection, combustion chamber, swirlers.

I. Introduction

It is known that a combustion chamber must have a design which allows the flame to self-sustain and that ensures that the combustion temperature does not surpass the maximum temperature limit from the gas turbine. However, the necessity of improving the process of combustion has attracted the attention from engineers and investigators, so that several modifications to the existing combustion chambers have been made, such as the application of swirlers in the premixed zone, angles of fuel injection optimization, variation of types of flames, utilization of weak air/fuel ratio, geometry modifications, and etcetera.

It is important to highlight that there are three important aspects to consider when a combustion chamber is being designed. The first one is to obtain adequate pressure losses, for which abrupt changes in the cross section must be avoided. The second one is to generate a good air/fuel mixture, which normally is accomplished by using swirlers. Finally, the cooling and dilution zones must be adequately located since the incoming air will help to completely oxidize the combustion residues. Besides, it controls the combustion gases temperature at the outlet of the combustion chamber.

For the control of the primary zone, most of the combustors use swirling flows, which help to anchor the flame, achieve a correct air/fuel mixture and maintain the combustion stability. Swirlers are used in order to generate the swirling flows and in Table 1, the two existing swirlers are listed.

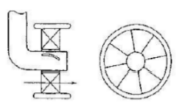
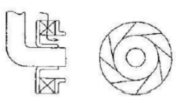
It is worth to mention that swirlers also help to reduce the velocity of the incoming flow since the diffusor is only capable of reducing it to one-third of the velocity that comes out of the compressor. This is very important because the flame can extinguish when the fluid velocity is high.

For the CFD Analysis the software ANSYS-FLUENT was employed, which uses a finite volume discretization for its simulations. The turbulence model was Standard $\kappa - \epsilon$, which is considered semi-empirical and is based on the turbulence equations, allowing the turbulent velocity to be obtained and the length scales to be determined independently. Furthermore, it has become the most implemented model in FLUENT.

II. Design Parameters

Before starting with the combustion chamber design, it was necessary to know the characteristics of the compressor, such as its operation map, from which the design point was chosen.

TABLE 1. Types Of Swirlers And Their Characteristics [1].

Type	Characteristics	Scheme
Axial	The desired amount of air must pass with a pressure loss similar to the pressure drop from the combustion chamber.	
Radial	It helps to diminish the generated emissions during combustion. The incoming air flow is determined by the effective flow area.	

The design point selection was based on its position in the operation map since it must be located in the stable zone, avoiding performance problems. The characteristics of the design point are listed in Table 2.

Other necessary parameters were also taken into account, such as maximum temperature for the turbine blades, pressure ratio, air/fuel ratio, etcetera. Some of these parameters were selected based on the desired level of technology, which in this case is level one, according to [3]. Therefore, the pressure ratio had a value equal to 0.90 and the combustion chamber efficiency was equal to 0.88.

Additionally, the maximum temperature for the turbine blades was obtained from the technical information from [2], which has a magnitude equal to 873 K. In this case, a value of 870 K was used.

Another important aspect for the combustion chamber design was the air/fuel ratio, since through this parameter the corresponding dimensions of the primary zone, the intermediate zone and the dilution zone are determined. Reference [4] recommend working with a weak mixture because it allows generating a better flame anchoring in the central recirculation zone, retaining high temperatures inside the flame and decreasing the produced heat due to the excess of air. That is the reason why the selected air/fuel ratio was equal to 17.5.

In Table 2, all the parameters considered for the combustion chamber design are listed.

TABLE 2. COMBUSTION CHAMBER DESIGN PARAMETERS.

Parameter	Value
Atmospheric pressure (P_{t1})	$78{,}185\ Pa$
Atmospheric temperature (T_{t1})	$299.05\ K$
Total pressure at the outlet of compressor (P_{t2})	$309{,}036\ Pa$
Revolutions per minute (rpm)	$90{,}000\ rpm$
Mass flow ($\dot{m}$)	$0.3459\ \frac{kg}{s}$
Isentropic efficiency (η_c)	0.70
Total temperature at the outlet of compressor (T_{t2})	$504.49\ K$
Combustion chamber efficiency (η_b)	0.88
Pressure ratio (π_b)	0.90
Maximum temperature (T_{t3})	$870\ K$
Air/fuel ratio (Φ)	17.5
Specific heat at constant pressure of air (Cp_c)	$1{,}005\ \frac{J}{kg\ K}$
Specific-heat ratio of air (γ_c)	1.4
Specific heat at constant pressure of gases (Cp_g)	$1{,}148\ \frac{J}{kg\ K}$
Specific-heat ratio of gases (γ_g)	1.33
Heating value of diesel (hPR)	$42{,}800\ \frac{J}{kg\ K}$
Specific gas constant for dry air (R_c)	$0.287\ \frac{J}{kg\ K}$
Specific gas constant for gases (R_g)	$0.284\ \frac{J}{kg\ K}$
Combustion chamber fuel/air ratio (f)	0.01341
Fuel mass flow ($\dot{m}_f$)	$0.004639\ \frac{kg}{s}$
Air mass flow inside the liner ($\dot{m}_{c_1}$)	$0.0811\ \frac{kg}{s}$
Air mass flow inside the casing ($\dot{m}_{c_2}$)	$0.2647\ \frac{kg}{s}$

III. COMBUSTION CHAMBER DESIGN

The first element of a combustion chamber is the diffusor, which controls the incoming velocity of the flow that, according to [3], it should not be higher than 60 m/s. However, in this case the flow outgoing from the compressor had a velocity equal to 54.5 m/s, which is why the implementation of a diffusor was not necessary.

In order to determine the pressure losses in the combustion chamber (ΔP_b), its values at the inlet and outlet are required and substituted in (1).

$$\Delta P_b = P_{t2} - P_{t3} \tag{1}$$

Where $P_{t3} = P_{t2}\pi_b$

$$\therefore \Delta P_b = P_{t2} - P_{t2}\pi_b = 309{,}036\ Pa - (309{,}036\ Pa)(0.9) = 30{,}903\ Pa$$

A. *Inlet Dimensioning*

An iterative process was applied in order to determine the velocity of the flow outgoing from the compressor, because this parameter cannot be obtained from the operation map and its value is essential for the inlet dimensioning. This process continued until the proposed and calculated velocities were equal. The final value of this velocity was $V_2 = 54.53\ {}^{m}/_{s}$, which is the velocity at the inlet of the combustion chamber.

Once knowing the value of the inlet velocity, the inlet dimensions were obtained through (2)-(6).

$$\rho_2 = \frac{\dot{m}_0}{V_2 A_{sal}} \tag{2}$$

$$A_{c_1} = \frac{\dot{m}_{c_1}}{\rho_2 V_2} \tag{3}$$

$$r_{c_1} = \sqrt{\frac{A_{c_1}}{\pi}} \tag{4}$$

$$A_{c_2} = \frac{\dot{m}_{c_2}}{\rho_2 V_2} \tag{5}$$

$$r_{c_2} = \sqrt{\frac{A_{c2}}{\pi} + r_{c_1}^2} \tag{6}$$

Resulting dimensions are showed in Table 3.

B. *Primary Zone Dimensioning*

In order to obtain the dimensions of the primary zone, it was necessary to calculate the combustion temperature through (7).

$$T_{tcomb} = \frac{T_{t2} Cp_c \dot{m}_{c_1} + \dot{m}_h hPR\, \eta_b}{\dot{m}_g Cp_g} \tag{7}$$

After knowing the value of the combustion temperature, the total pressure was obtained considering a total pressure drop loss equal to 60%, accordingly to [5]. Additionally, a velocity of 35 m/s was considered in order to calculate the static conditions. Finally, it was possible to determine the values of density, area and radio, which are listed in Table 4.

In the case of the casing, total conditions at the inlet were considered because there is no energy addition. The dimensions of the casing are showed in Table 4 as well.

C. *Outlet Dimensioning*

The next step of the combustion chamber design process was to obtain the dimensions at the outlet. This was performed by considering that the total mass flow would be the sum of the air mass flow at the inlet plus the fuel mass flow. Due to the previous, it was necessary to calculate the total temperature at the first dilution zone with an energy balance, assuming that 50% of the cooling air would enter through it. This temperature has a value of 1129.85 K.

As can be seen, the temperature at this point was too high so it was necessary to add a second dilution zone. Thus, a second energy balance was accomplished and a total temperature equal to 870 K at the outlet of the combustion chamber was obtained. After knowing the temperature at the outlet, it was possible to calculate the total pressure at the same point, obtaining a pressure value equal to 278,132 Pa.

TABLE 3. INLET DIMENSIONS.

Liner		Casing	
Area (A_{c_1})	$0.0007026\ m^2$	Area (A_{c_2})	$0.00229\ m^2$
Radio (r_{c_1})	$0.01495\ m$	Radio (r_{c_2})	$0.0308\ m$

TABLE 4. PRIMARY ZONE DIMENSIONS.

LINER		CASING	
Area ($\boldsymbol{A_{c_{1comb}}}$)	$0.00525\ m^2$	Area ($A_{c_{2comb}}$)	$0.00355\ m^2$
Radio ($\boldsymbol{r_{c_{1comb}}}$)	$0.04090\ m$	Radio ($r_{c_{2comb}}$)	$0.0529\ m$

In order to determine the dimensions at the outlet, another iterative process was performed because flow velocity needed to be calculated. It was found that the velocity was equal to 65.36 m/s, with which the static conditions were obtained and, with (2)-(6), the values of area and radio were estimated and listed in Table 5.

D. *Combustion Chamber Length*

To determine the length of the combustion chamber it was necessary to calculate its volume, which was done through (8), giving a value equal to 0.001589 m^3.

$$V_{cc} = \frac{\eta_b \dot{m}_f hPR}{P * q_{cc}} \tag{8}$$

Once knowing the total volume of the combustion chamber, it had to be divided into sections. Finally, with the truncated cone volume equation, the length of each section could be obtained, which are showed in Table 6.

E. *Cooling Holes Design*

The following methodology was applied in order to calculate the number of cooling holes for each stage:

1. Determine the average density of the air between the liner and the casing.
2. Establish an inlet velocity for each stage.
3. Establish the amount of incoming air for each stage.

TABLE 5. OUTLET DIMENSIONS.

Area ($\boldsymbol{A_3}$)	$\mathbf{0.00479\ m^2}$
Radio ($\boldsymbol{r_3}$)	$0.03904\ m$

TABLE 6. COMBUSTION CHAMBER SECTIONS LENGTH.

Primary zone	$L_1 = 0.05630\ m$
Intermediate zone	$L_2 = 0.1255\ m$
Dilution zone	$L_3 = 0.1068\ m$

4. Calculate the necessary area for the air to enter to the liner.
5. Determine the diameter of the holes.
6. Divide the total area by the area of one hole.

Through the precious methodology it was estimated that the intermediate zone and the dilution zone would count with 58 and 60 cooling holes, respectively. In Fig. 1, the geometry of the combustion chamber can be observed.

IV. CFD Analysis

The CFD Analysis was performed with the software ANSYS-FLUENT. In the case of the meshing, the tetrahedral method was used, with an element size equal to 0.001 m. Besides, the boundary conditions for the first analysis are listed in Table 7, considering atmospheric conditions as $P_{atm} = 78{,}185\ Pa$ and $T_{atm} = 300\ K$.

It is important to mention that two different analyses were performed: in the first one, only the flow behavior was studied, without considering the combustion process; in the second one, combustion was taken into account with diesel as fuel. Furthermore, for the second analysis, it was necessary to activate the module 'Species', with which it is possible to verify the quality of the mixture.

The boundary conditions stablished for the combustion analysis are showed in Table 8.

Additionally, swirlers were designed for the analysis of combustion, which would help to achieve the desired performance. For the design of the swirlers, it was considered that fuel injection would be longitudinally, i.e., axial, which is the reason why simple geometries were proposed, such as square and streamline, which can be visualized in Fig. 2a and 2b. However, by analyzing the results it could be observed that the maximum temperature at the outlet of the combustion chamber was higher than 1200 K, which is why an annular flow swirler (Fig. 2c) was proposed, but the results showed that the temperature at the outlet was higher than 1200 K too. Finally, it was decided that a coupling of two different swirlers would be used. This coupling can be observed in Fig. 2d and is composed by an annular flow swirler and a central flow swirler.

TABLE 7. BOUNDARY CONDITIONS FOR THE FIRST ANALYSIS.

Boundary conditions at the inlet	Boundary conditions at the outlet
$P_t = 309{,}036\ Pa$	$P = 302{,}190\ Pa$
$P = 305{,}876\ Pa$	$T_t = 504.5\ K$
$T_t = 504.5\ K$	

TABLE 8. BOUNDARY CONDITIONS FOR THE ANALYSIS OF COMBUSTION.

Boundary conditions at the inlet	Boundary conditions at the outlet
$\dot{m}_{c_1} = 0.0811 \frac{kg}{s}$	$P = 275{,}742\ Pa$
$\dot{m}_{c_2} = 0.2647 \frac{kg}{s}$	$T_t = 870\ K$
$P = 305876\ Pa$	
$T_t = 504.5\ K$	
$\dot{m}_f = 0.004639 \frac{kg}{s}$	

V. Results

A. Flow Analysis Results

Through the flow analysis a decrease in velocity near the wall of the liner due to the abrupt dilation between cross sections could be observed, which generated turbulence and boundary layer separation. Additionally, total pressure losses presented along the entire combustion chamber, visualized in Fig. 4a, where it can be observed also that the cooling zones did not accomplish their function correctly since there was no homogenization of the flow at the outlet.

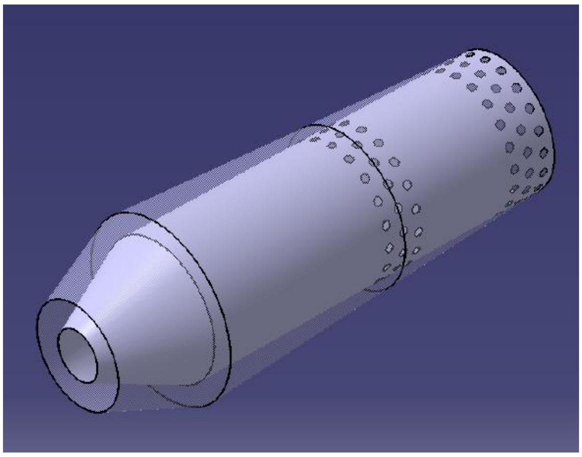


Fig. 1. *Combustion chamber geometry.*

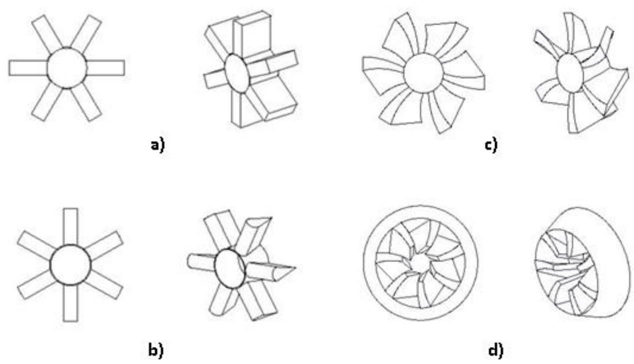


Fig. 2. *a) Square swirlers, b) streamline swirlers, c) annular flow swirlers and d) annular-central flow swirlers.*

B. Redesign

Due to the mentioned phenomena, it was necessary to redesign the combustion chamber, for which the studies made in [6] were consulted. They stablished that there is an interval of opening cone angle for a greater efficiency, avoiding a kinetic energy loss excess in the section. This interval is between 5° and 10°. In this case, a value equal to 10° was used.

The cooling holes geometry was modified as well, having a diameter increment to 7 mm. Furthermore, the cooling zones were displaced, moving them away from the outlet in order to accomplish a complete homogenization of the flow. The final design can be observed in Fig. 3.

C. Results Obtained With The Redesigned Combustion Chamber

Thanks to the analysis realized on the redesign combustion chamber it could be visualized that the boundary layer separation phenomenon was reduced. Furthermore, it was proved that the modifications to the dilution holes had positive effects, since the secondary flow and the primary flow were completely homogenized.

In Fig. 4b it can be observed that in spite of the increase in pressure losses near the inlet liner wall, the zone where the boundary layer phenomenon occurred decreased, diminishing the total pressure losses and flow instabilities.

D. Results From The Combustion Analysis

With the swirlers coupling, a higher pressure drop inside the combustion chamber was obtained. Nonetheless, the temperature at the outlet was higher than 1200 K, which caused that the position and angle of injection were modified. Two injectors were used instead of one and the angle of injection was changed to 15°.

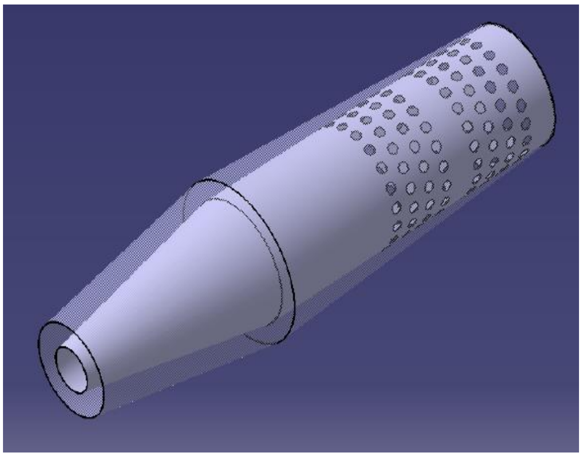


Fig. 3. *Redesigned combustion chamber.*

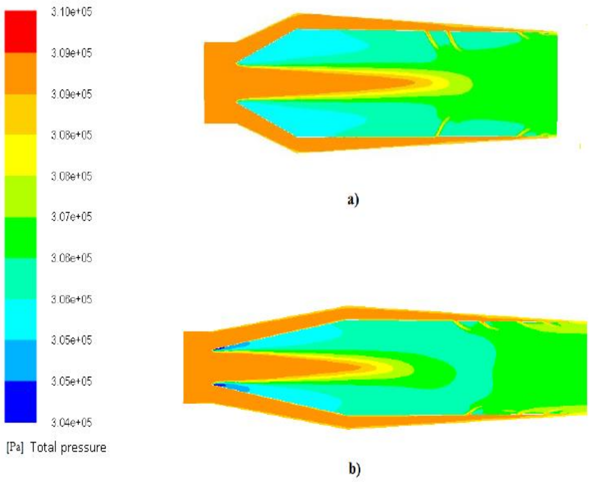


Fig. 4. *Total pressure contours from the a) initial and b) redesigned combustion chamber.*

However, even with these modifications, the obtained outlet temperature was hotter than 1200 K. Thus, the final modification was to alter the angle of injection to 90°, which caused a pressure drop equal to 258,073.92 Pa and a maximum outlet temperature of 1185.32 K.

The temperature behavior at the outlet can be visualized in Fig. 5, where a minimum temperature gradient is presented, indicating that there are not any hot spots, which guarantees a correct coalescence between secondary and primary flows. It can be noticed also in the pressure losses along the combustion chamber showed in Fig. 6.

In Fig. 6 it can be seen that the maximum pressure loss is located between 0.03-0.15 m, which is enough to achieve the desired flame anchoring, and that the dilution zone can be found between 0.23-0.32 m, indicating a complete incorporation of the secondary flow.

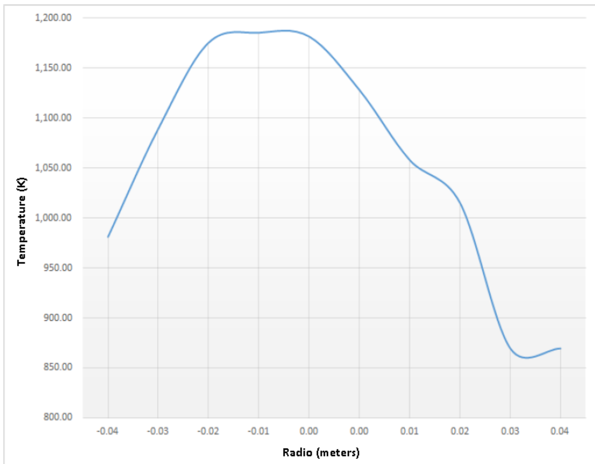


Fig. 5. *Temperature behavior at the outlet with an angle of injection equal to 90°.*

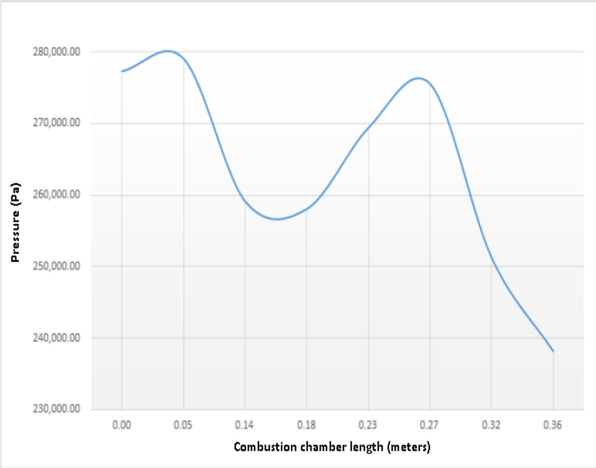


Fig. 6. *Pressure behavior at the outlet with an angle of injection equal to 90°.*

From Fig. 5 and 6 it can be noted that the zone where the highest pressure loss is located, is also where the highest temperature exists. This is because the pressure losses are the ones that cause the correct flame anchoring.

In Fig. 7 it can be visualized that there is not a flame lengthening inside the combustion chamber, avoiding hot sport at the outlet.

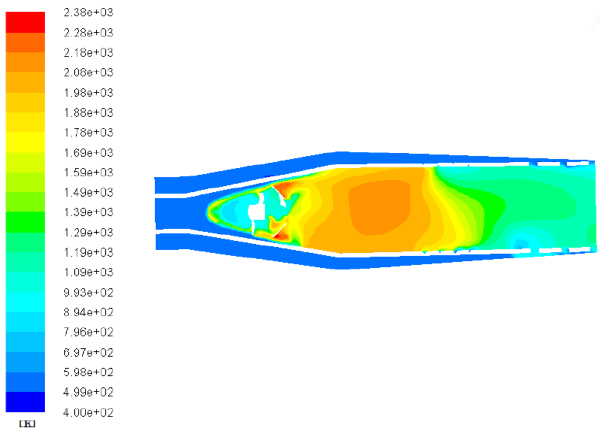


Fig. 7. *Temperature contours inside the combustion chamber.*

Conclusion

Taking into account the requirements that a combustion chamber must accomplish, it can be concluded that the selected design is adequate. Furthermore, through the analysis it was possible to determine the importance of the different elements that make up the combustion chamber.

Different phenomena that are important for the correct combustion chamber performance could be observed:

- Swirlers help to create a correct air/fuel mixture by generating turbulence.
- Pressure losses are linked to the turbulence generation, which helps to achieve a complete incorporation of the secondary flow into the primary flow.
- In order to prevent any damage to the turbine blades, it is necessary to avoid the flame lengthening, which is accomplished by implementing a proper configuration of swirlers and angle of injection.

Acknowledgment

The authors want to thank the National Polytechnic Institute, especially to all the professors and investigators from the Thermal and Applied Hydraulics Engineering Laboratory, because without them, this investigation could have not been performed.

References

[1] A. Lefebvre, Gas turbine Combustion, Second edition, *Taylor and Francis Group*, 1999.

[2] Borg Warner, www:turbos:bwauto:com=products=turbochargerTurbine:aspx.

[3] J.D. Mattingly, Elements of gas turbine propulsion, *McGraw Hill*, 1996.

[4] L. Mixquititla-Casbis, J.C. García Castrejón, F.Z. Sierra Espinoza y A. Valera-Medina, Análisis numérico del campo de flujo en cámara de combustión para aplicaciones en microturbinas de gas.

[5] B.S. Steckin, P.K. Kazandzan, L.P. Alekseev, A.N. Govorov, N.E. Konovalov, Ju. N. Necaev. y R.M. Federov , Teoría de los Motores a Reacción, 1962.

[6] J.A. Spurk y Nuri Aksel, Fluid Mechanics, Second edition, Springer, 2008.