European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2006
P. Wesseling, E. Oñate, J. Périaux (Eds)

CFD ANALYSIS OF FLOW IN THE START SYSTEM OF A LIQUID PROPELLANT ENGINE

H. Karimi* and E. E. Taheri†

* K.N.T University of Technology, Faculty of Aerospace Engineering
Tehran, Iran
e-mail: karimi@kntu.ac.ir
Web page: http://www.kntu.ac.ir/

† K.N.T University of Technology, Faculty of Aerospace Engineering
Tehran, Iran
e-mail: mit512000@yahoo.com

Key words: Liquid Propellant Engine, Solid Propellant, Start System, CFD Analysis

Abstract. *Starters are mechanisms that are used for generating the working fluid for initial rotation of LPE (Liquid Propellant Engines) turbopumps. The function of the start system is preparation of initial power for starting a turbine. This turbine is attached to the fuel and oxidizer pumps. When these pumps work in a suitable case, the required power of turbine, is prepared from another source (gas generator) and then there isn't need to starter operation.*
A solid propellant starter is a solid propellant motor that instead of thrust, momentum of it's exit gases is used for rotating the turbine. The values of power and operating time of starter, are specified from downstream conditions of the start system. Generally, the type and efficiency of turbine and pumps, propellant rate and density, control mechanisms in the path of propellant and etc , are determinant of these values.
In this type of start system, gases due to combustion, pass through a convergent-divergent nozzle and enter the turbine and rotate it.
In this paper, with due consideration to the role and importance of start system in the process of LPE operation, in order to better recognition of present phenomena in this system, the flow due to combustion in a start system that uses solid propellant, has been simulated via solving the navier-stokes equations.
These equations have been expressed in the form of time-dependent, axisymmetric, compressible and viscous and have been solved by means of finite volume methods in company with continuity and energy equations.
For finite volume methods the domain is divided up into a number of control volumes, with the value at the centre of the control volume being held to be representative for the value over the entire control volume. By integrating the PDEs over the control volume the equations are cast into a form that ensure conservation.

1 INTRODUCTION

As was mentioned, starters are mechanisms that are used for generating the working fluid for initial rotation of LPE turbopumps (figure 1). There are different types of penumatic and hydraulic starters that use the following methods for generating the working fluid :

a. Gas (Helium, Nitrogen, Air or Hydrogen) that is located in a ballon or a high pressure tank.

b. Combustion products due to main propellant ingredients that enter from propellant tanks.

c. Combustion products due to solid propellant grain, that is located in the start chamber (figure 2).

The most important advantage of solid propellant start system in comparison with other above mentioned start systems, in addition to simplicity, is non-necessity to existance of high pressure tanks, because the sealing of these tanks, particularly for long-time flight launch vehicles or LV that shall be stored for long time in stock, is very difficult.

In this paper, the flow due to combustion in a start system that uses solid propellant has been simulated. The function of starter is preparation of required power for rotating a

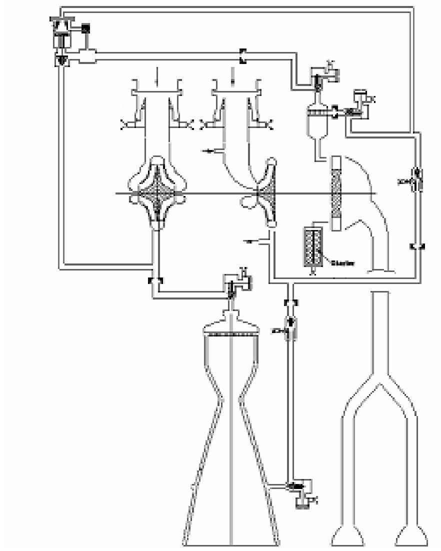

Figure 1 : A schematic view of a liquid propellant engine

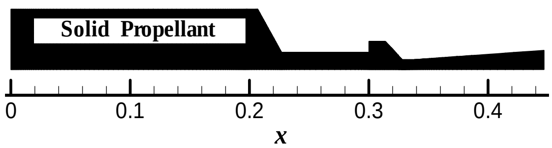

Figure 2 : The start system of a specific liquid propellant engine

turbine that is attached to the fuel and oxidizer pumps. This function must be attended with several following characteristics :

a. Starter must generate a specified power for turbine in a given revolution.

b. Operation time of starter must be in a suitable range and must be certain.
c. Starter must be have a specified treatement alongside of the other main components of motor.

Finally the starter must be compatible with mechanical constraints due to geometric limitations and mechanical strength.

In fact, solid propellant starter is a solid propellant motor that instead of thrust, momentum of it's exit gases is used for rotating the turbine. The values of power and operating time of starter, are specified from downstream conditions of the start system. Generally, the type and efficiency of turbine and pumps, propellant rate and density, control mechanisms in the path of propellant and etc , are determinant of these values.

In this type of start system, gases due to combustion, pass through a convergent-divergent nozzle and enter the turbine and rotate it. This function must be done according to specified principles.

2 ASSUMPTIONS

In order to recognizing the present phenomena in the start system, the gas flow due to solid propellant combustion, shall be simulated. In this direction, there are several complexities. These complexities are due to path geometry of gases and solid propellant. The accurate modelling of these geometric complexities, is requiring fully 3-D view, that this subject, has several problems like as long execution time. Anyhow, in this paper a simplified model of starter has been presented. This simplifying is according to the basic following assumptions[1] :
a. Combustion products are chemically neutral and their compositions are not changed.
b. General scheme of gas flow is the same at different times.
c. Initial geometric form of propellant does not change during the burning.
d. There is not erosive burning.

3 THE RESULTS INVESTIGATION

Considering the above mentioned assumptions and by using of a computational code, the gas flow in the starter has been simulated via solving Navier-Stokes equations. These equations have been expressed in the form of time-dependent, axisymmetric, compressible and viscous and have been solved by means of finite volume methods in company with continuity and energy equations [2,3,4,5,6,7] .

Figure 3 shows the grid of flow field (Figures 3.a , 3.b and 3.c). This flow field has been formed from 13000 elements. These elements are used for capturing important details of flow.

At table 1, some properties of solid propellant and gas inside the starter have been presented. The resulted answers from the numerical simulation, define flow field. For obtaining these answers, it has been supposed that the changes of burning areas are negligible. With due consideration to solid propellant burning rate and mean velocity of gas inside the starter, this assumption is reasonable.

Figure 4 shows the velocity vectors at different areas of the starter. In figure 4.a details of velocity variations at the beginning part of flow field has been presented. It is observed that, at the upper areas of solid propellant, there is rotation of velocity vectors. Naturally, the

Conduction Factor	Viscousity Factor	Solid Propellant Density	Solid Propellant Burning Rate	Specific Heats Ratio (γ)	Gas Constant
W/m.K	*N.s/m²*	*kg/m³*	*mm/s*	-	*Kj/kg-K*
0.2	6.66e-5	1520	13	1.25	352.4

Table 1: Some properties of solid propellant and gas inside the starter

direction of flow is from left to right, but at a distance about 6 centimeter from the beginning edge of solid propellant, the direction of flow has been reversed.

This rotation is because of smallness of flow passage area between solid propellant and start chamber wall. Furthermore, at the midsection of flow field, the gas velocity is more than the upper part. Moreover, the generated gases at the beginning forehead of the propellant, are flow towards the interior of flow field.

In figure 4.b the details of velocity vectors at the end part of solid propellant, have been shown. The gas flow at the solid propellant forehead, by rotation towards the main axis of flow field, completes the rotation of generated gases between propellant and start chamber wall. Because of momentum injection in this region, the wake formation behind the propellant has been prevented.

In figure 4.c the velocity vectors between nozzle and convergent part of starter body has been presented. In this region, it doesn't observe an unexpected phenomenon.

In figure 4.d , the velocity vectors at the nozzle and it's expansion area have been shown. From the viewpoint of fluid dynamics, the existence of expanding area, hasn't effect on the upstream flow. At the inside of nozzle, the flow is fully natural. We shall pay attention to the difference between the scale of velocity vectors in figures 4.a , 4.b , 4.c and 4.d . Due to expanding the velocity variations throughout the flow field, the representation of velocity vectors by using one scale is not suitable.

In figure 5 the streamlines inside the start chamber has been shown. The rotation of streamlines at the beginning part of propellant (figure 5.b) and the weakness of flow inside the expansion area of nozzle (figure 5.e) are observed.

In figure 6 the pressure contours and its distribution inside the start chamber have been presented (figures 6.a and 6.b). According this figure, The pressure decreases from the starter head towards the end of nozzle. The rate of pressure variations inside the region that solid propellant exist, is very insignificant. The cause of pressure reduction in this region, is velocity augmentation due to gas generation. In the convergent part of starter body, due to area reduction of flow passage, the pressure decreases violently. In the region between the convergent part of body and the beginning part of expanding region, the pressure variations is insignificant. The insignificant reduction of pressure is due to the friction. The pressure reduction in the expanding area and nozzle is very significant.

In figure 7 the temperature contours and its distribution inside the start chamber has been shown (figures 7.a and 7.b). The general trend of temperature variations is similar to pressure variations.

In figure 8 the mach number contours and its distribution inside the start chamber have been presented (figures 8.a and 8.b). The alteration trend of this parameter is completely natural and conforming to reality.

REFERENCES

[1] Sutton G. P., *Rocket Propulsion Elements, An Introduction to The Engineering of Rockets* , John Wiley and Sons, (1992).
[2] Chorin A. J., "Numerical solution of the Navier–Stokes equations", *Mathematics of omputations* , **22** : 745–762 , (1968).
[3] Ferziger J. H. and Peri´c M., *Computational Methods for Fluid Dynamics*, Springer–Verlag, Berlin, (1996).
[4] Patankar S. V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere, NewYork,(1980).
[5] Jones I. P., *CFDS-Flow3D User Guide*, (1994).
[6] Bertran V. and Jensen G., "Recent applications of computational fluid mechanics", *Schiffstechnik/Ship Technology Research*, **41**:131–134, (1994).
[7] Lilly D. K., "On the computational stability of numerical solutions of time–dependent non-linear geophysical fluid dynamics problems". *Monthly Weather Review*, **93**(1):11–26, (1965).

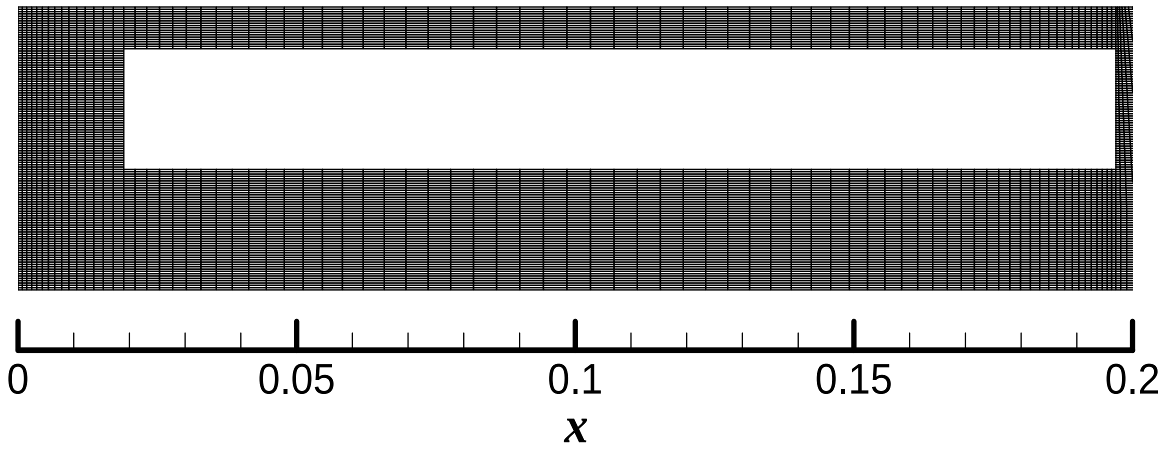

Figure 3.a

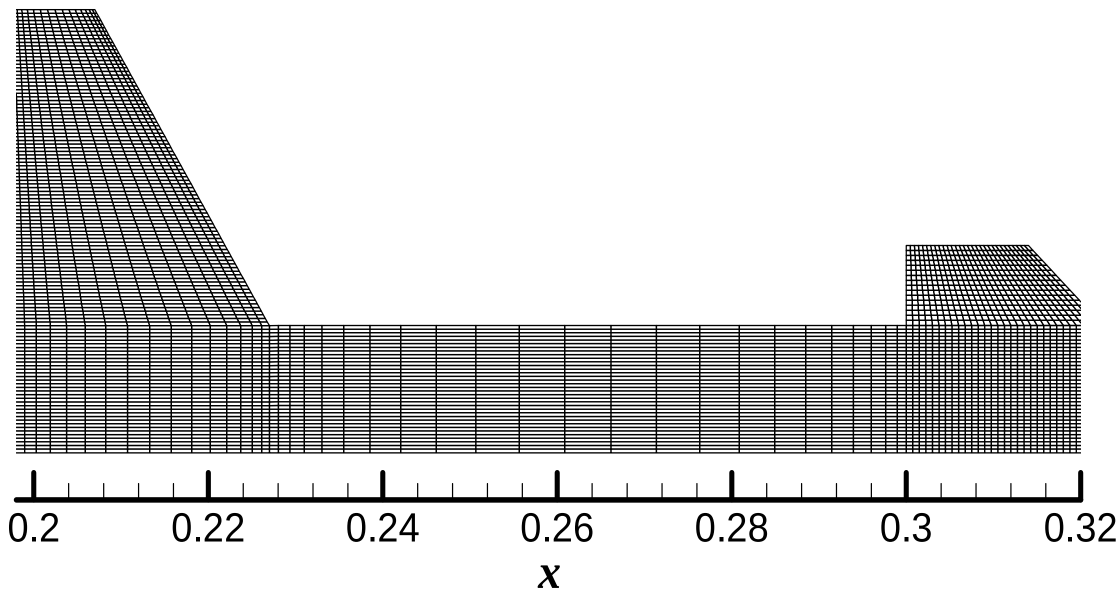

Figure 3.b

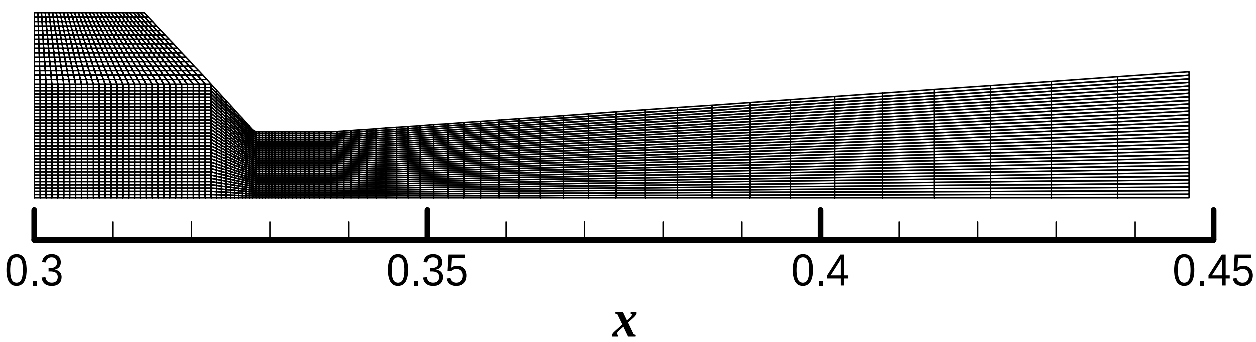

Figure 3.c

Figure 3 : Grid of flow field inside the start chamber at different views

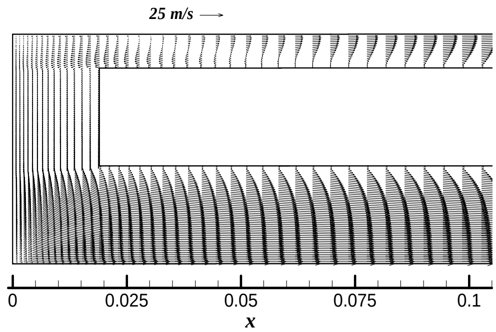

Figure 4.a

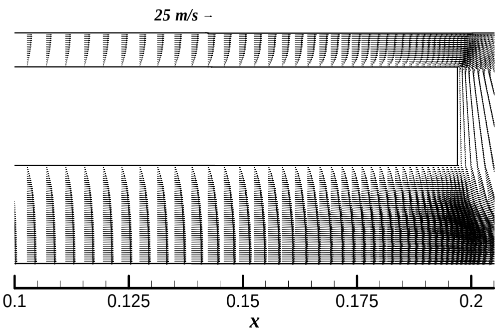

Figure 4.b

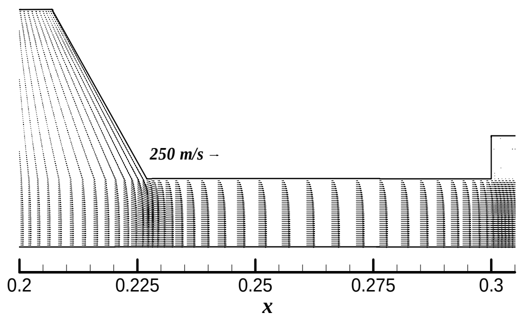

Figure 4.c

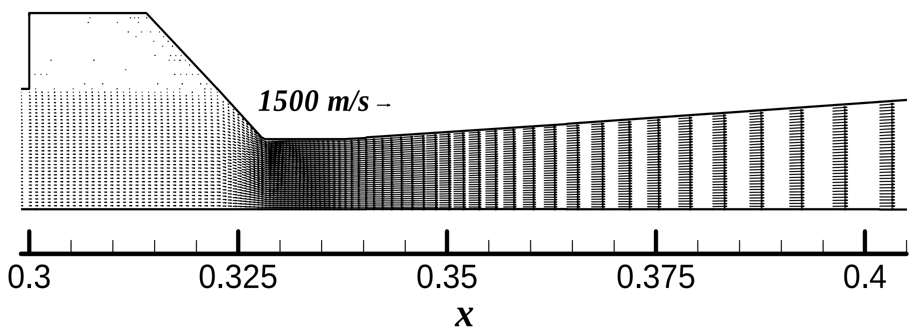

Figure 4.d

Figure 4 : The velocity vectors at different areas of the starter

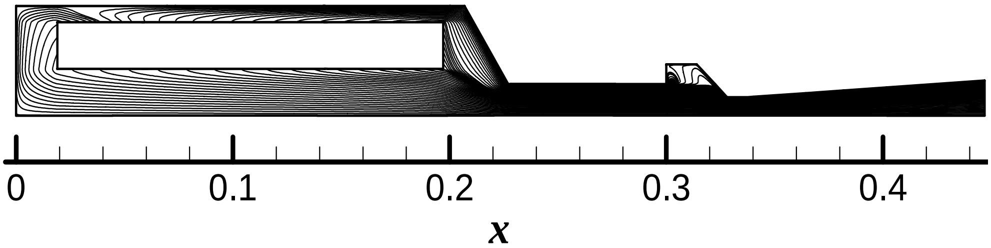

Figure 5.a

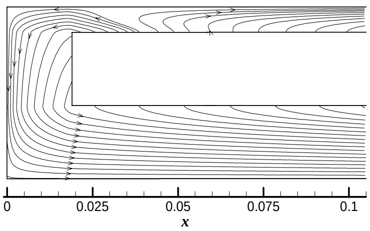

Figure 5.b

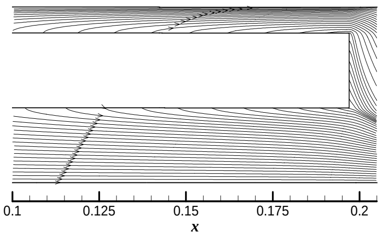

Figure 5.c

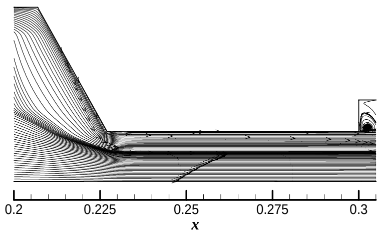

Figure 5.d

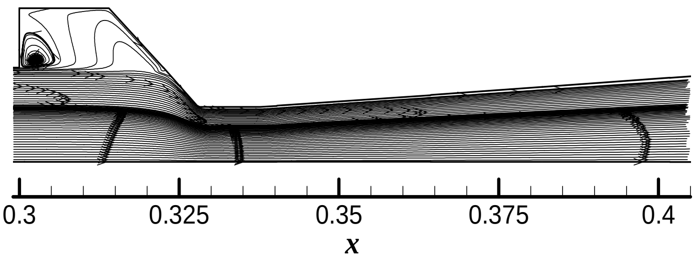

Figure 5.e

Fig 5 : The streamlines distribution inside the start chamber

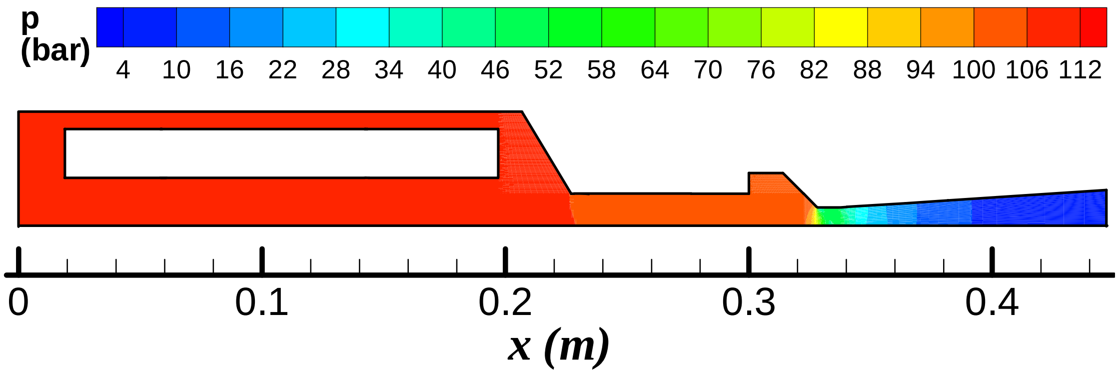

Figure 6.a

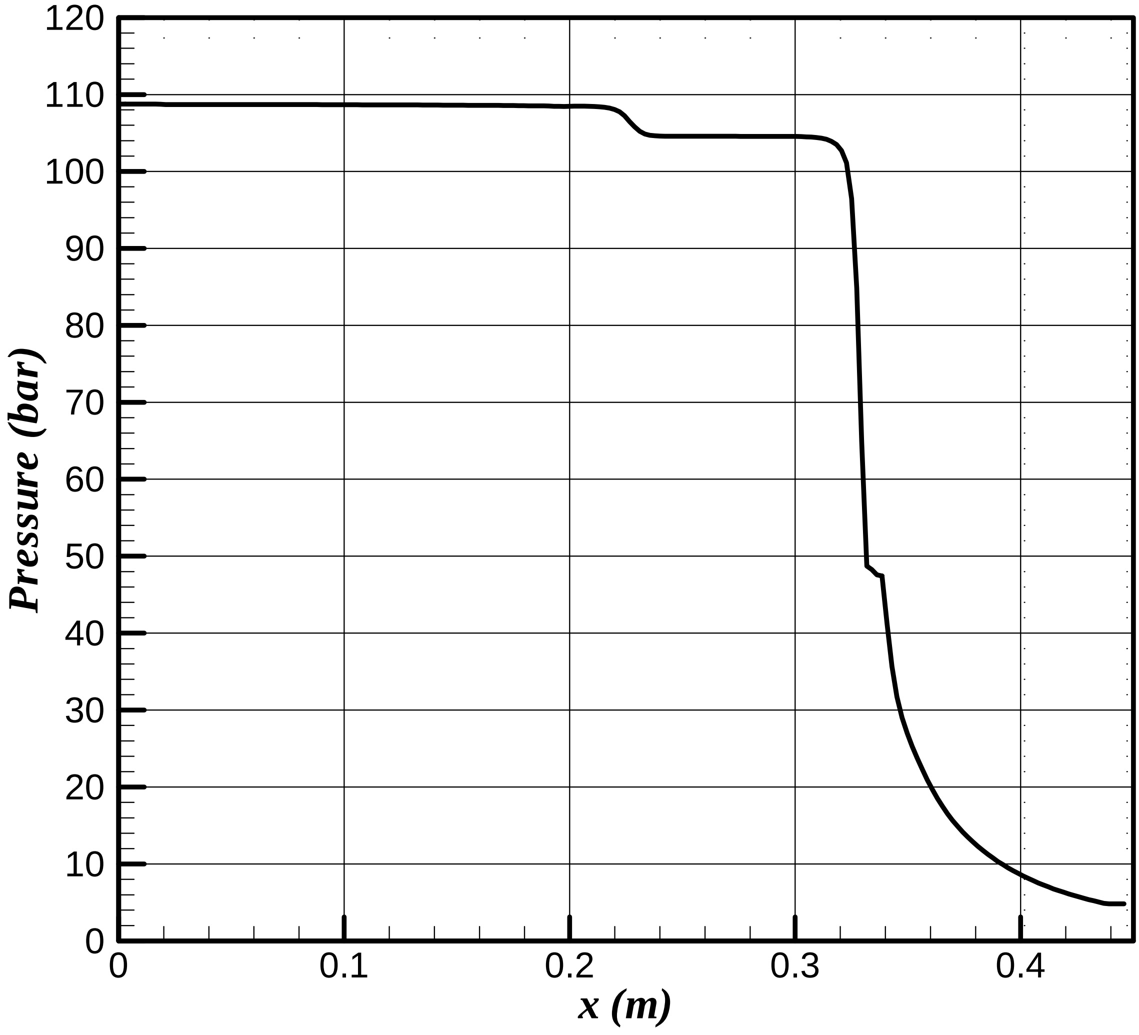

Figure 6.b

Figure 6 : The pressure contours and its distribution inside the start chamber

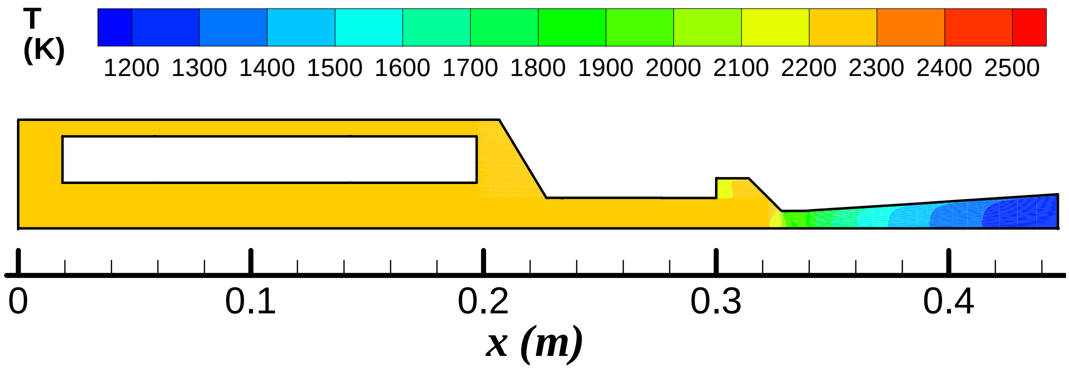

Figure 7.a

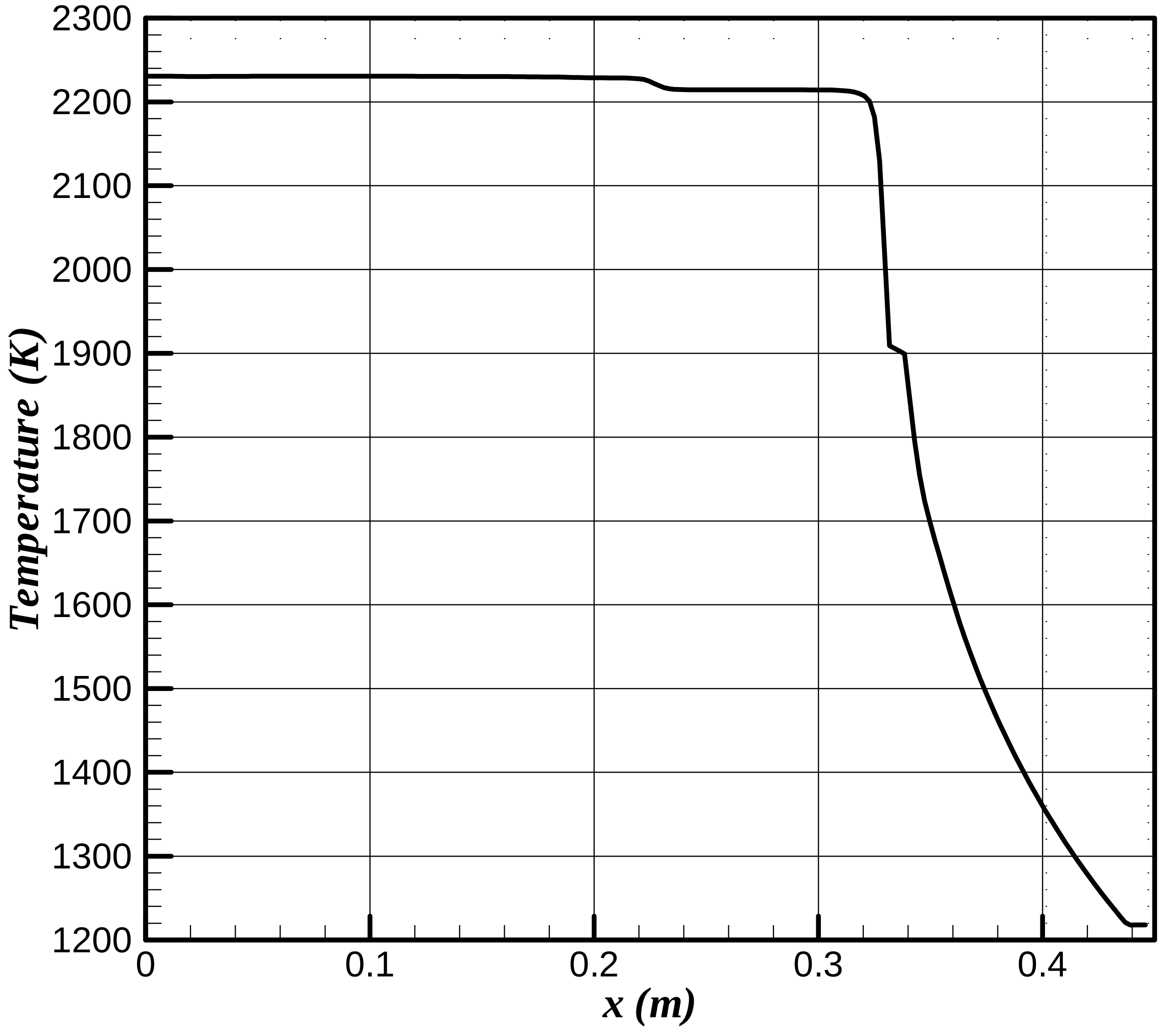

Figure 7.b

Figure 7 : The temperature contours and its distribution inside the start chamber

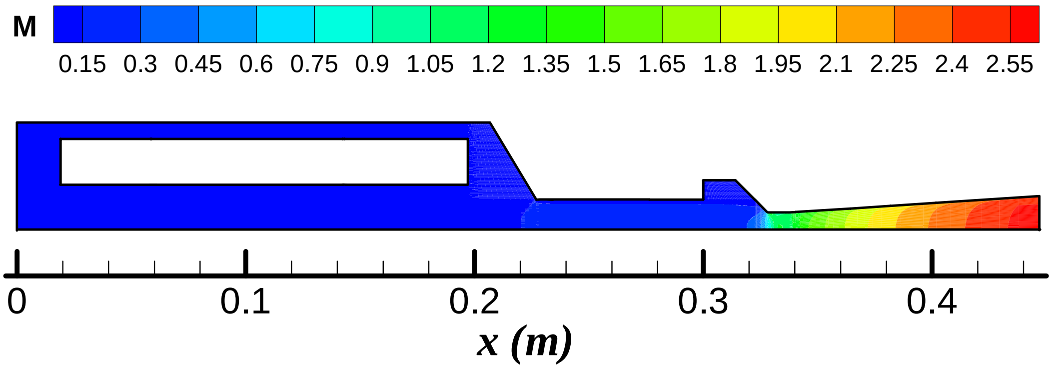

Figure 8.a

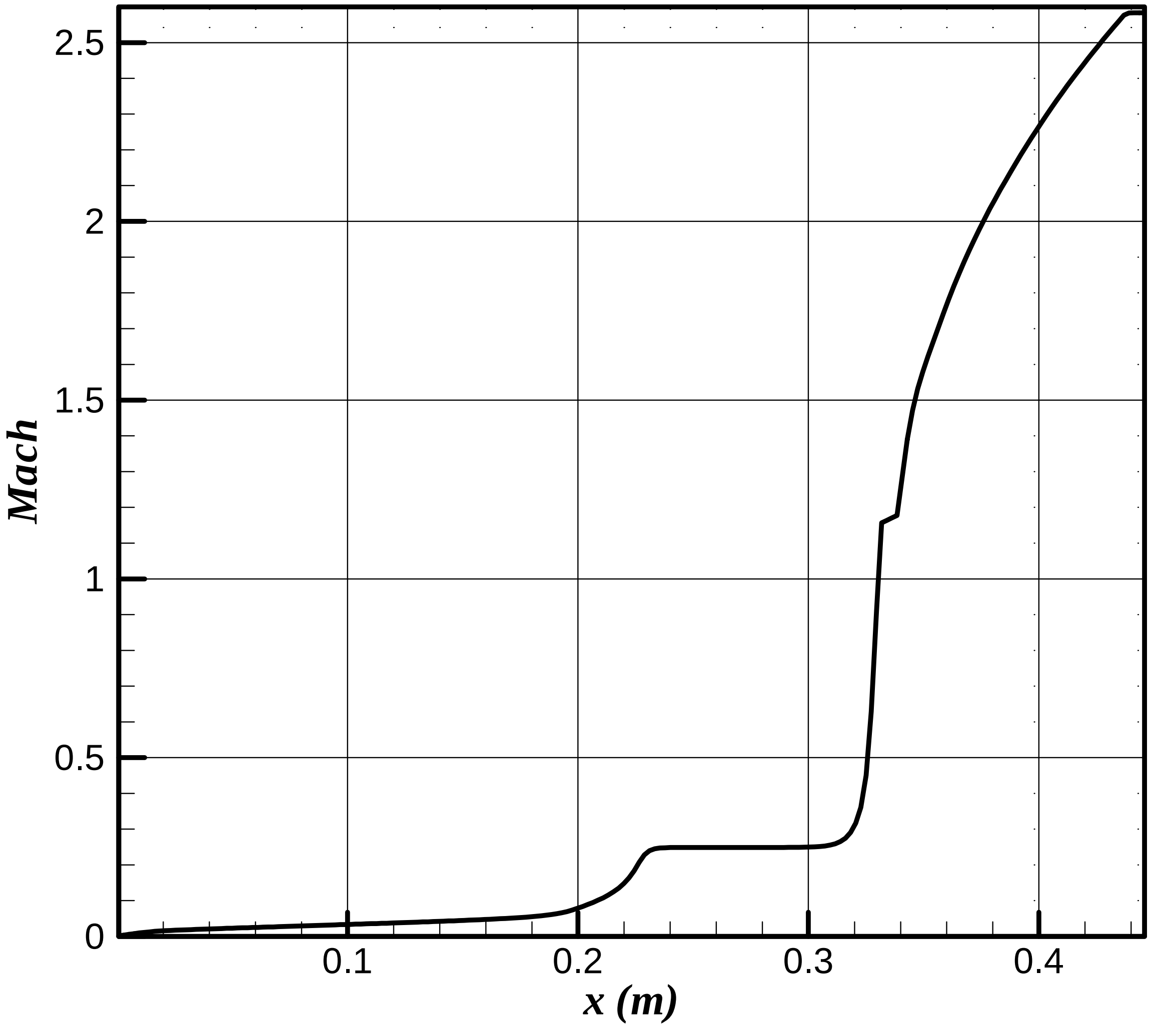

Figure 8.b

Figure 8 : The mach number contours and its distribution inside the start chamber