ANALYSIS OF PLUME IMPACT ON LAUNCH PAD DURING LIFT OFF

R. Sala, M. Galeotta
Carlo Gavazzi Space, Milano, Italy
A. Veneziani
Politecnico di Milano, Milano, Italy

ABSTRACT

The object of the work is the evaluation of a small launcher plume impinging on the launch pad and the ground segment during lift off, through the determination of the velocity, pressure and temperature fields of the exhausting gases.
The analysis was carried out initially with a theoretical model, then validated by a finite element simulation and in the end compared with experimental data.
The theoretical model used is based on the Newtonian Impact Pressure Theory, in according to have a first prediction of the domain in which we are operating.
The equation used for the finite element simulation are the Navier-Stokes equations, coupled with the energy equation and the continuity one. The turbulence model used for the simulation is a developed k-ε model. Preliminary results show that data obtained by the numerical approach can be compared to the experimental values obtained during the test of the first stage of the launcher, and the two approaches show a good correlation. Theory model can be compared just at a certain distance from the axis of the launcher because of its dependance as r^{-1} of the density field to the radial distance.
The analytical approach confirms the fact (already found with the experimental test) that at an axial distance of 5 m and a radial distance of about 6 m (the position of the launch tower) the pressure is reduced of 95% of the nozzle exhaust value while temperature is about the atmosphere temperature.
On the launch pad, directly behind the plume, numerical simulations show that pressure reaches values about 2.5 bar and temperature reaches values of 2100 K.
In conclusion it was found that pressure and temperature fields are critical during lift off just on lauch pad, while the tower is not affected by exhausting plume.

NOMENCLATURE

C_F	= thrust coefficient
C_P	= specific heat at constant pressure
FEM	= finite element method
g	= gravity acceleration
h	= altitude
I	= impulse
P	= pressure
r	= radius
V	= velocity
$\dot{W}$	= total mass flow rate
α,β	= source-flow parameters
ε	= nozzle area ratio
γ	= specific heat ratio
θ	= nozzle half-angle
ρ	= density

Subscripts

dyn	= dynamic
e	= nozzle exit plane
i	= ideal limit
sp	= specific
0	= stagnation or chamber value
∞	= far-field

1. INTRODUCTION

Supersonic jet impingement has been studied in different ways: several methods have been proposed to predict the behaviour of these plumes. Nowadays numerical approches are used to predict the jet exhausting the nozzle, because analytical models are in progress. The characteristic line method[1] can be used just when we are near the outlet of our nozzle, because it is quite impossible to predict the interaction between characteristic lines and atmosphere. So the source flow method[2] can be used to have a prediction of the boundary far from the nozzle because of the singularity of the method at zero distance from launcher axes. Moreover it has been used also the Latvala method[3], based on geometrical considerations and the shock waves relations. If we want a good prediction of the entire flow field, we have to approach the problem with numerical simulations. Several types of solvers can be used, from finite volumes method to finite elements method and they show a good approximations of the flowfield. However, the amount of required memory storage was more than double for the FEM[6]. Experimental data have been used just around ’60, when computational fluid dynamics was not affirmed yet.

During the project of a ground segment, it is always important knowing each kind of load it will impact the structures. We consider the plume impinging two main regions: the “carneau”, the structure that permits the outflow of exhausted gases, and the “mast”, the structure that is near the launcher at its lift off (launch tower). In order to design them, we have to know the load pressure and maximum temperature the structure reaches. Our object is to predict loads that will occur during the launcher lift off, in order to have a prediction of the request of resistance of both the structures

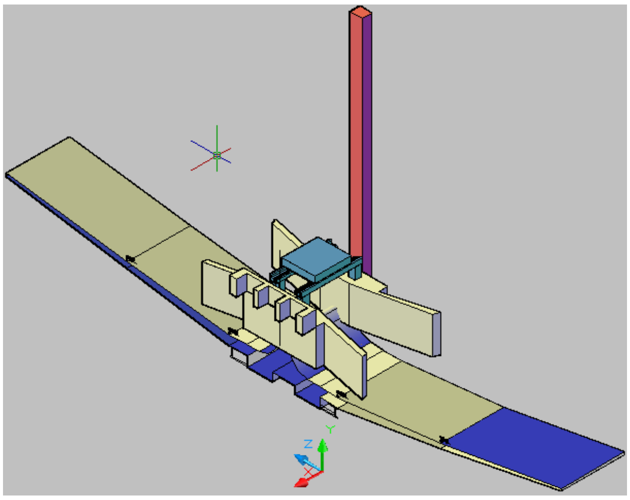

Fig. 1: Ground segment structure

There are several different approaches to obtain the result we are looking for. We will use an analytical method, more ingegneristic than mathematic to solve the Navier Stokes equations, through the Pressure Newtonian Impact Theory that evaluates the dynamic pressure assuming it derivates from a source at a fixed distance from the center of the nozzle exhaust. As we know the dynamic pressure and density, through the ideal gas law we can obtain the total temperature.

2 METHODS

Analytical Method

The analytical model is based on the source flow method. In the polar coordinate system the density is assumed to be like

$$\rho(r,\theta)=\frac{\alpha[\cos(\frac{\theta}{2})]^2}{V_\infty r^2}$$

Where V_∞ can be determinated by knowing the specific impulse I_{sp} as

$$V_\infty = g I_{sp,i}$$

The parameter α is also related to the total mass flow rate $\dot{W}$,

$$\dot{W}=\frac{8\pi\alpha}{\beta+2}$$

whether the parameter β can be calculated from supersonic exit nozzle parameters like expansion ratio ε and the gas specific ratio γ. Infact the ratio of thrust coefficient to its ideal value can be expressed as

$$\frac{\beta}{\beta+4}=\frac{C_F}{C_{F,i}}$$

where

$$C_{F,i}=\sqrt{\frac{2\gamma^2}{\gamma-1}\frac{2}{\gamma+1}^{\frac{\gamma+1}{\gamma-1}}}$$

and

$$\frac{C_F}{C_{F,i}}=\frac{1+\cos\theta_e}{2}\sqrt{1-\frac{P_e}{P_0}^{\frac{\gamma-1}{\gamma}}}+\frac{P_e\varepsilon}{P_0 C_{F,i}}$$

where P_e/P_0 is the ratio of exit to chamber pressure.
As we know the plume density, we can proceed to determine the impact pressure

$$P_{dyn}(\theta)=\rho(\theta)V_\infty^2\sin^2\theta$$

The density can also be determined as

$$\rho(r,\theta)=\rho(\theta)=\frac{\beta+2}{8\pi}\frac{\dot{W}}{V_\infty h^2}\sin^2\theta(\cos\frac{\theta}{2})^\beta=\frac{\beta+2}{8\pi}\frac{\dot{W}}{V_\infty r^2}(\cos\frac{\theta}{2})^\beta$$

By which we obtain

$$P_{dyn}(\theta)=const\sin^4\theta(\cos\frac{\theta}{2})^\beta$$

Numerical Method

Geometry and solver

The numerical computations are carried out by solving the governing equations with the boundary conditions in a two-dimensional geometry. To discretize the domain it has been used a triangular mesh more fitted where high gradients are supposed to be with skewness at least equals to 0.8. The position of the exhaust of the launcher nozzle changes form 5 m to 35 m, in order to simulate some of the most critical situations the entire ground segment has to endure. To have a preliminary information about the loads on “carneau” it has been chosen the altitudes of 5 m, 15 m and 30 m. The properties of the jet exhausting the nozzle (supersonic and compressible) means a strong dependance between continuity and momentum equations: this means the necessity to have a coupled implicit solver. Turbulence models with two equations predict with accuracy mean values of the flux, expecially near nozzle exhaust. The model we have decided to use is the k-ε realizable, a recent development of the standard model. The Courant number is set to 0.3 to avoid divergence. The criterion of convergence is set to 0.001

for all the variables. In the figure below is reported an example of the geometry used to simulate the impact on "carneau", with a detail to show how the mesh was fitted to get high gradient of variables.

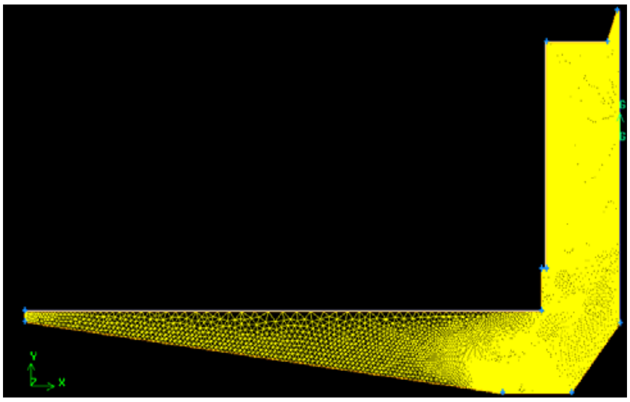

Fig. 2: Geometry mesh

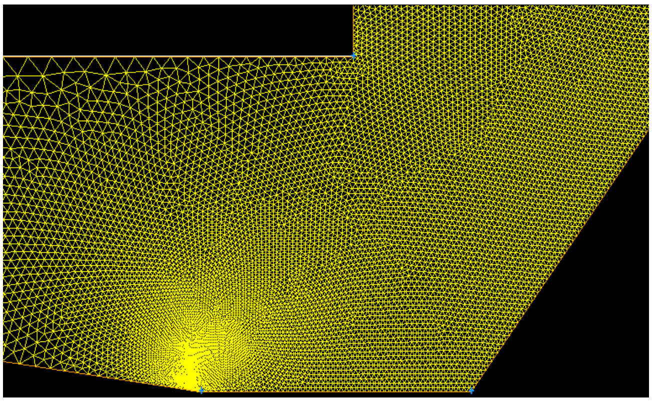

Fig. 3: Detail of the mesh

Gas properties and boundary conditions

The gas used for the simulation is the real gas flowing out the nozzle with these properties:

Cp	j/kg-k	4210.82
Thermal conductivity	W/m-k	0.39
Viscosity	kg/m-s	1.1e-4
Molecular Weight	kg/kg-mol	29.285

These data have been obtained with a specific thermo-chemical software, imposing the geometry of the nozzle and the exact composition of first stage launcher propellant. The operative pressure is set to 0 atm in order to operate only with absolute values of pressure. The total pressure doesn't change from combustion chamber to the nozzle exhaust because we assume an isoentropical expansion in the nozzle. This pressure set to 60 bar (59.215 atm) whether the temperature is set to 3543.1 K. Hence we can establish them as inlet boundary conditions, with also the hydraulic diameter set to 0.496 m and turbulence intensity of 2%.
The outlet condition is 1 atm and 300 K in order to operate in ambient condition (so our launcher is working with over-expanded jet).

All of the structure around the launcher is set as wall to have condition of impenetrability.

Experimental data

Next to numerical and analytical approach, we can compare our results to experimental data[8].
We have first stage data of the solid motor rocket in two different ways: we have total pressure, total temperature, Mach number and specific heat ratio (but we will consider frozen equilibrium while expanding in the divergent, so it will be ignored) as the axial distance increases from 0 m to 300 m. At lower altitude (0-81 m) with several centimeters step, than (82-110 m) around some meters and at higher altitudes about 10 meters (110-300 m).
Otherwise we have the same values with radial distance increasing from 0 to 41 m but also with different axial distance increasing (0, 5, 15, 30, 40, 45, 50, 60 m). As we know these data we can make a comparison among analytical, numerical and experimental values of the variables of interest.

3. RESULTS

We can now compare the results found with the numerical approach to the experimental data given for the pad structure. It is not necessary to simulate the impact on mast because loads given by exhausting jet are not critical. We have a double confirmation on it: first, the launch tower is usually designed to endure wind loads rather than lift off loads; second analytical results accord to experimental ones showing low pressure and temperature on mast.
In the figures below we can compare total pressure, total temperature and Mach number impinging on mast, 6.2 meters from launcher axis.

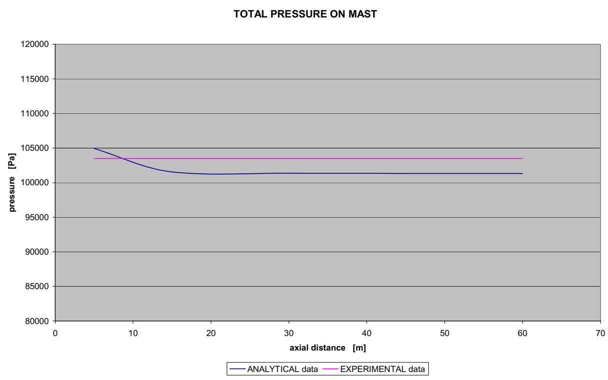

Fig. 4: Total pressure on mast

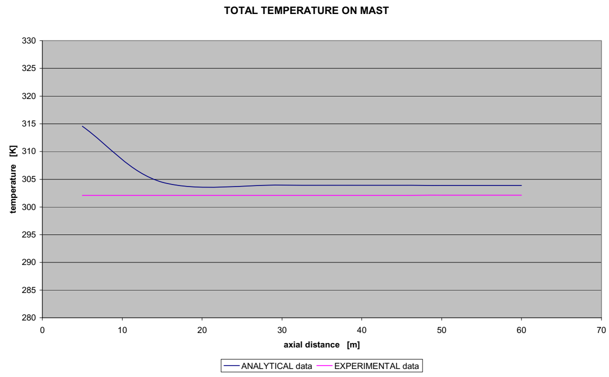

Fig. 5: Total temperature on mast

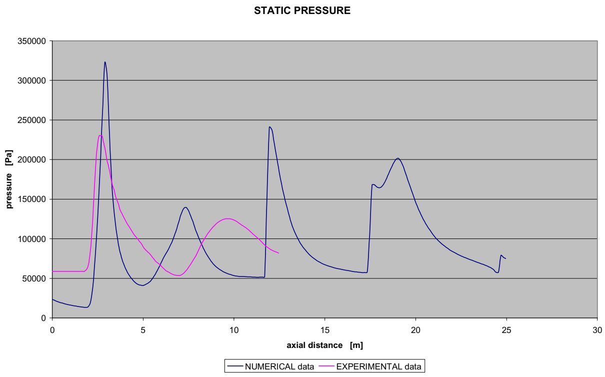

Fig. 7: Static pressure on “carneau” from nozzle to the outflow at 5 meters

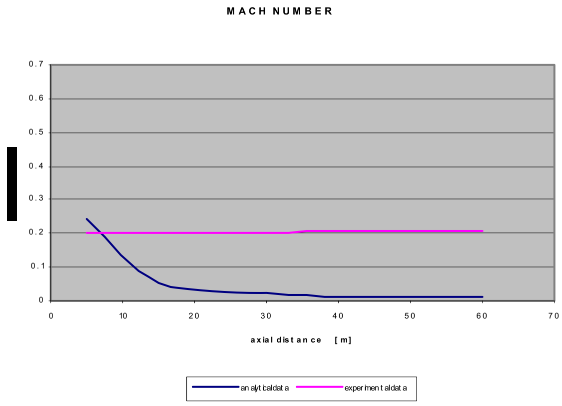

Fig. 6: Mach number on mast

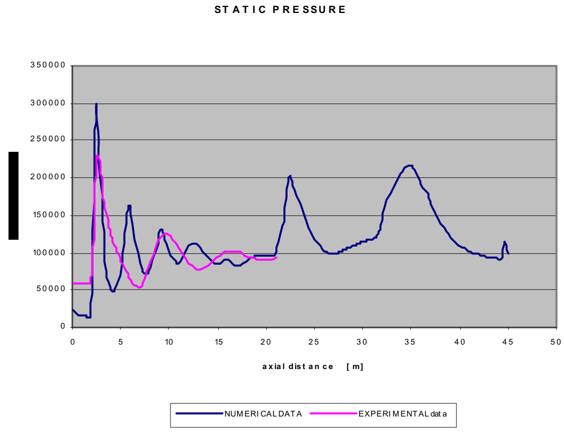

Fig. 8: Static pressure on “carneau” from nozzle to the outflow at 15 meters

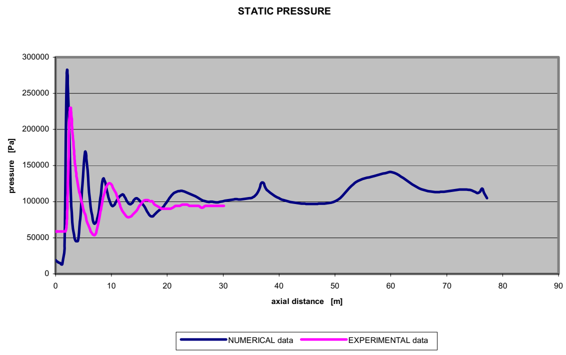

Fig. 9: Static pressure on “carneau” from nozzle to the outflow at 30 meters

We can easily see how analytical data are similar to experimental ones: for pressure we have an offset of about 2000 Pa, that is 2% of the nominal pressure. Moreover maximum impact pressure is of 103500 Pa, too low to be a dimensional value. For the temperature we have an error of 1.18% on nominal temperature, and its maximum value is 302.11 K, just a few degrees over the ambient temperature. In the end, for the Mach number we have an error of 25%, but if we relate it to the Mach number of 0.25, that is a completely uncompressible flowfield, we can infer that Mach number has no important effects on structure.
Concerning the “carneau” the situation is different.
The following figures compare numerical and experimental pressure, temperature and Mach along all the “carneau” structure for all the altitudes considered.

We can see how numerical simulation are qualitatively similar to first stage data, but numerical approach shows stronger shock waves closer to each other. This is due to the presence of the structure below the nozzle. Numerical variables are plotted on all the “carneau” length, as shown in the figure below (it has not been possible for experimental data). Experimental data instead are used just for the axis line, because they come from free jet and they don’t consider any structure below.

For the temperature we have the same qualitatively result, we can see a little increment of temperature near the stagnation point, that is not captured by experimental data.

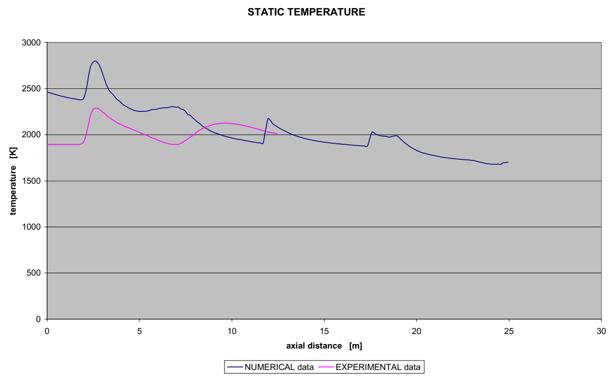

Fig. 10: Static temperature on "carneau" from nozzle to the outflow at 5 meters

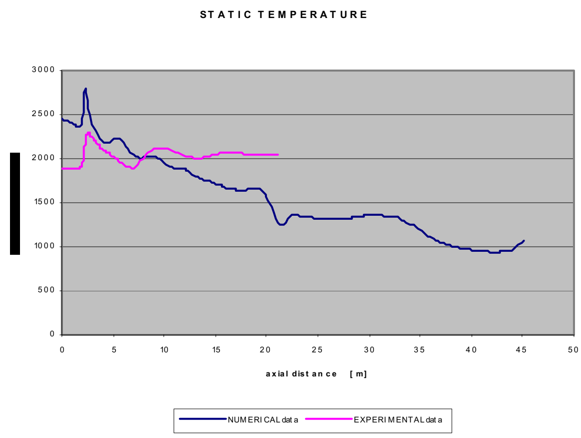

Fig. 11: Static temperature on "carneau" from nozzle to the outflow at 15 meters

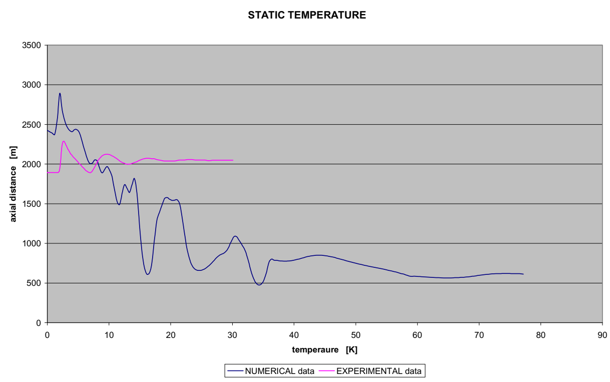

Fig. 12: Static temperature on "carneau" from nozzle to the outflow at 30 meters

For Mach number we can see the oblique shock wave effect, even if Mach number decreases because it reaches the value zero at stagnation point. Along the pad we can also see the supersonic flow.

.

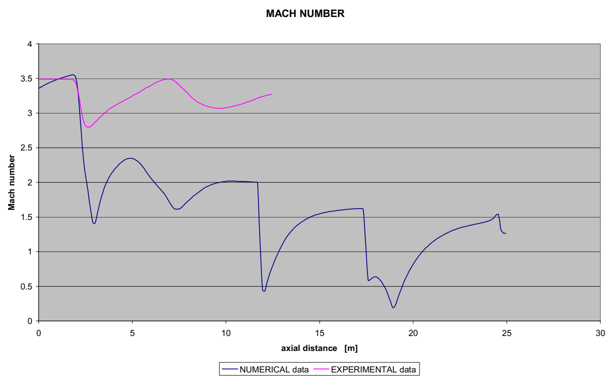

Fig. 13: Mach number on "carneau" from nozzle to the outflow at 5 meters

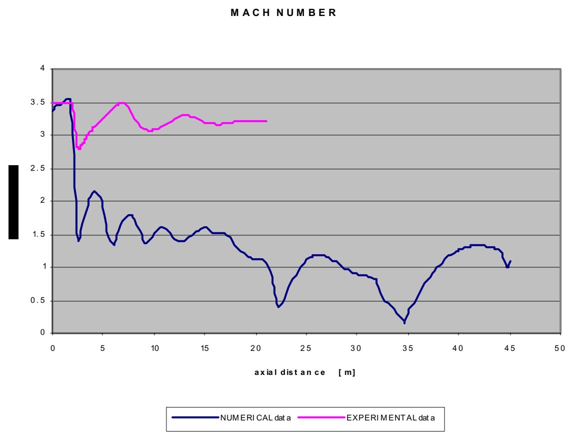

Fig. 14: Mach number on "carneau" from nozzle to the outflow at 15 meters

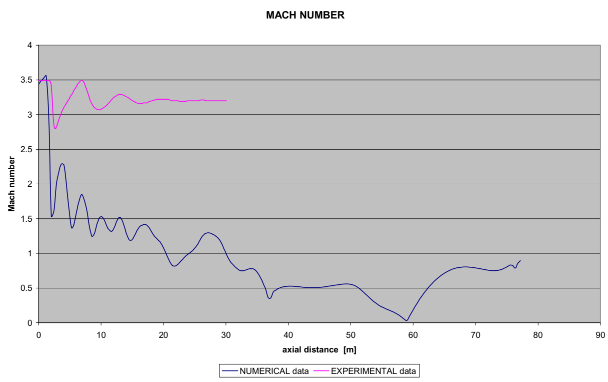

Fig. 15: Mach number on "carneau" from nozzle to the outflow at 30 meters

Now we can compare all the results found for the distance along the launcher's height and compare themselves.

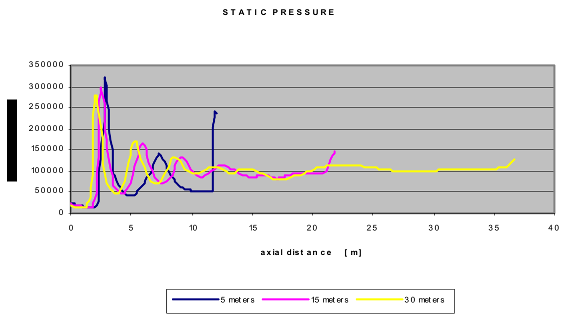

Fig. 16: Static pressure on "carneau" from nozzle to the outflow

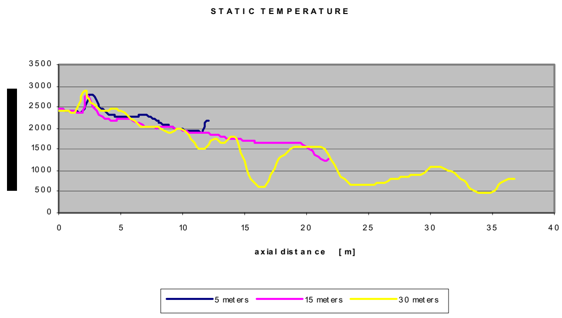

Fig. 17: Static temperature on "carneau" from nozzle to the outflow

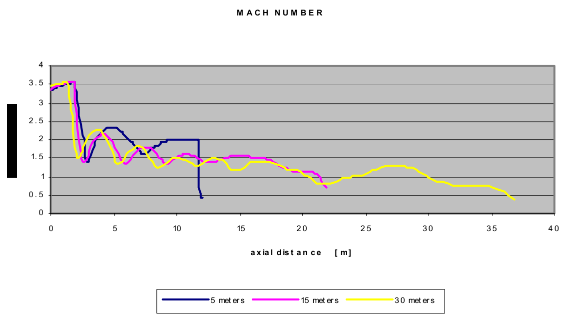

Fig. 18: Mach number on "carneau" from nozzle to the outflow

And if we compare the evolution of the three variable fields along the pad we see

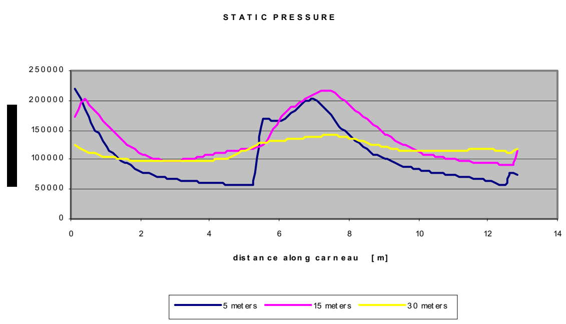

Fig. 19: Static pressure on "carneau"

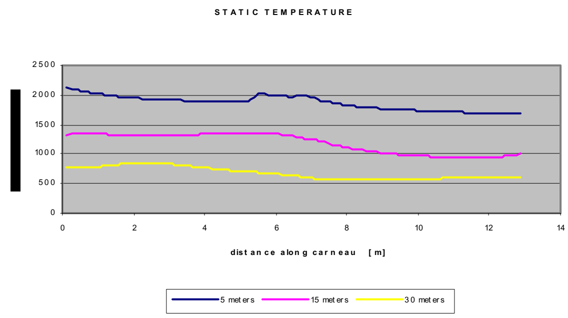

Fig. 20: Static temperature on "carneau"

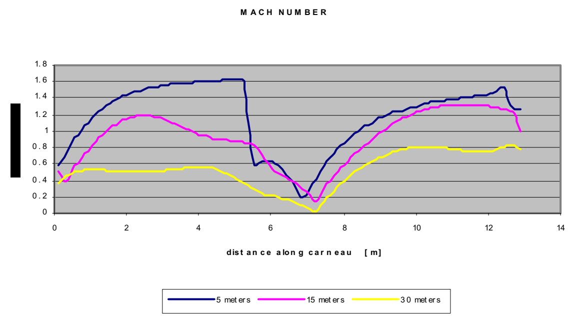

Fig. 21: Mach number on "carneau"

As long as the launcher is near to the structure, it "feels" the presence of a wall. This induces the Mach number to decrease and the pressure to increase; the plume is "less over-expanded" and the shock wave system farer to the nozzle exhaust. From the graphs reported above, the critical point is the upper point of the pad and the point where the plate becomes plane. Moreover, if we compare the pressure, temperature and Mach fields for the three axial distance from the pad, we can see how 5 meters altitude is dimensioning for the pad. At this level, pressure reaches about 2.4 bar for the highest point and temperature of 2200 K. From the evolution of fields along the "carneau" we can observe the second critical point. Here the pressure reaches 2 bar and 2000 K.

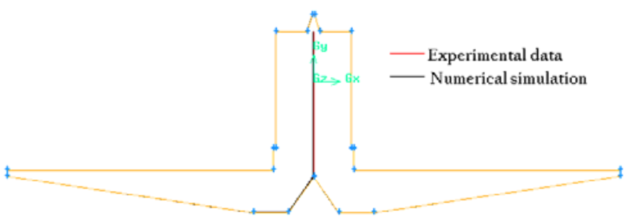

Fig. 22: Scheme of references coordinates for experimental data and numerical simulations

4. CONCLUSIONS

We have determined two critical points for the "carneau": the upper point and the point where geometry becomes plane. In figure below it is possible to se a qualitatively report of the flux along the pad, where it is possible to identify the two most critical points:

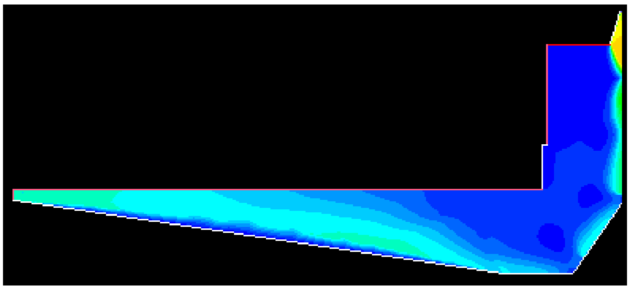

Fig. 23: Flowfield on the “carneau”

The points are the ones where velocity becomes null, that are stagnation points.
The analytical approach shows to be acceptable enough only at a certain distance from the axes, because it presents a singularity among the simmetry axes of the launcher. The numerical approach is more indicated for a prediction of the flowfield in the entire domain, even if it presents an high computational cost.
A possible natural development of this work is a three-dimensional geometry, in order to have the complete flow field simulation. Moreover we have used a frozen equilibrium expansion in the divergent of the nozzle. It is possible to simulate a situation in which the chemical reactions take place not only in the combustion chamber, but also they are still in act during expansion to have less conservative approach.
In the end it is also possible to consider a two-phase efflux, in which there are also solid particles of aluminum that contributes to a decrease of the specific total impulse.
Through the picture reported below, it is possible to compare our results with what really happens during a launcher lift off. In this case an Ariane 4 lift off has been taken as an example, we can see the over-expanded exhausting jet and it can be compared to our simulation with a free jet.

Fig. 24: Ariane 4 lift off

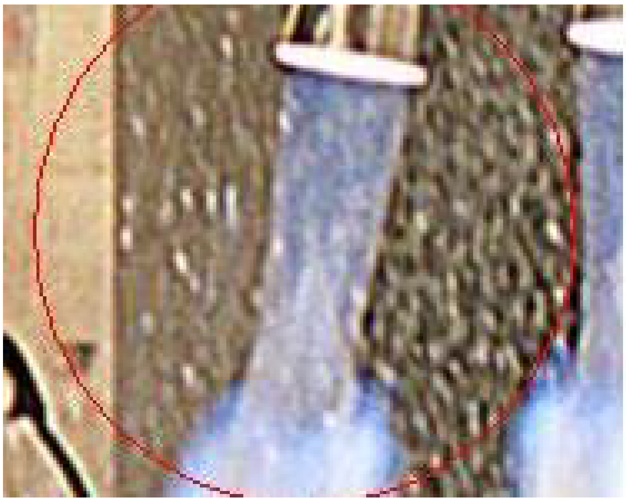

Fig. 25: Detail of over-expanded jet of Ariane 4

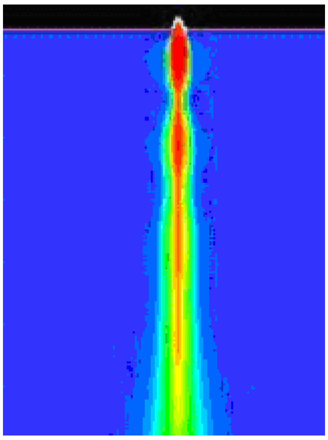

Fig. 26: Free jet of our simulations

5. BIBLIOGRAPHY

1 W. C. Rochelle, D. E. Kooker, Heat-Transfer and Pressure Analysis of Rocket Exhaust Impingement on Flat Plates And Curved Panels, Journal of Spacecraft and Rockets, Vol. 6, No. 3

2 E. Mayer, R. Prickett, Rocket Plume Impingement Heat Transfer on Plane Surfaces, Journal of Spacecraft and Rockets, Vol. 24, No. 4

3 K. L. Nash, K. W. Whitaker, L. M. Freeman, Predicting Exhaust Plume Boundaries with Supersonic External Flows, Journal of Spacecraft and Rockets, Vol. 31, No. 5

4 C. Y. Loh, Computation of Tone Noise From Supersonic Jet Impinging on Flat Plates, NASA/CR-2005-213426, Mar. 2005

5 F. F. Alvi, J. A. Ladd, W. W. Bower, Experimental and Computational Investigation of Supersonic Impinging Jets, AIAA Journal, Vol. 40, No. 4

6 K. Nakajima and Y. Kallinderist, Comparison of Finite Element and Finite Volume Methods for Incompressible Viscous Flows, AIAA Journal, Vol. 32, No. 5

7 E. T. Piesik, R. R. Koppang, D. J. Simkin, Rocket-Exhaust Impingement on a Flat Plate at High Vacuum, Journal of Spacecraft and Rockets, Vol. 3, No. 11

8 VG-SI-1/2-C-001-IPT