

INVESTIGATION OF THE FLOW PATTERNS ON A SOUNDING ROCKET FORE-BODY SECTION

Ana Cristina Avelar*, **Edson Basso***, **João Batista Pessoa Falcão Filho***, **Pedro Geovanny Martinez Romero****, **Jéssica Jia Ling Hsu***
Instituto de Aeronáutica e Espaço (IAE), Instituto Tecnológico de Aeronáutica (ITA)*

Keywords: *Flow Patterns, Sounding Rocket Pressure Measurements, Schlieren*

Abstract

An investigation of flow patterns in the fore-body section of a sounding rocket is presented. Wind tunnel tests on a 1:34 scaled model were carried out for Mach numbers varying from 0.4 to 1.1 using the techniques of Pressure Sensitive Paint (PSP), Schlieren and Pressure Taps. Additionally, a numerical analysis was carried out, using a commercial package, in the same range of Mach numbers. All measurements and simulations were carried out for zero angle of attack. The results obtained allowed the identification of complex physical phenomena as shock waves, boundary layer detachment, and expansion waves. Good agreements between numerical and experimental results were observed and the flow fields obtained with PSP, Schlieren and CFD were reasonably coherent.

1 Introduction

In the course of the history of near-earth space exploration, sounding rockets have providing researchers with a means of launching suborbital payloads to investigate microgravity [1]. Regarding the determination of the sounding rockets aerodynamics characteristics, estimating procedures can be of great help in several cases. However, there are configurations in which wind tunnel investigations are required, for example geometric shapes involving mismatched sections, exceptionally long bodies, unusual payload shapes, and protuberances, since very small ones can have

significant effect on sounding rockets aerodynamics.

Regarding wind tunnel analysis, surface pressure measurements are of great importance for obtaining insights on the aerodynamics of aerospace vehicles [2]. The conventional and most used method for wind tunnel surface pressure measurements on model surfaces is the technique of pressure taps, which consists of installing arrays of small orifices on a model surface and connecting them through small flexible tubes to pressure transducers. This technique, relatively simple, provides results highly reliable, but has also some drawbacks. It can be costly in price and time to manufacture and to prepare a model with the hundreds of pressure taps necessary to provide a reasonable resolution, mainly when one is dealing with industrial tests using big models. Another disadvantage is that it is not possible to install pressure taps in very thin areas of the model. An alternative for these shortcomings is the rather recent technique of Pressure Sensitive Paint (PSP) [2], which allows pressure measurements on the surface of a model without the need of lots of pressure taps, but just few pressure holes for validation or calibration. This method has been used in successfully in wind tunnels around the world for quantitatively pressure field measurement on model surfaces³ with a reasonably low cost. The PSP technique working principle is documented in the literature [3,4,5,6,7], and it is based on an oxygen-quenching process in which excited molecules are deactivated by oxygen, this phenomenon produces different degrees of

luminosity on the model surface. The PSP technique works very well in transonic and supersonic regimes, being very feasible for wind tunnel studies of sounding rockets in wind tunnel. In spite of occurring in a very short period of time, these regimes, mainly the transonic range are very important parts of sounding rocket envelop flight because of the complex flow phenomena taking place in the correspondent velocity range.

Another important and efficient tool in the understanding of the complicated flow features occurring in the flight envelop of sounding rockets is flow visualization, or the direct observation of the flow field. Visualization is an important tool in establishing flow models as a basis for mathematical models. It can be used for the direct solution of engineering problems and also as an aid to understanding the concepts of fluid motion [8]. Making use of the techniques of PSP, *Schlieren*⁹ visualization and pressure taps, a methodology for a detailed experimental investigation of the transonic flow features on space vehicles is being introduced in the Pilot Transonic Wind Tunnel, known as TTP, and located in the *Instituto de Aeronáutica e Espaço* (IAE). In the present work, a Brazilian sounding rocket, developed at IAE, the VS-40 was studied for Mach number varying between 0.4 and 1.1. In order to get more insights about the complicated flow pattern taking place in this speed range, in addition to the techniques mentioned above, numerical simulations using the CFD++ commercial software package were also conducted. The investigation was conducted for zero angle of attack. Good agreements between the results obtained with the three techniques were observed, and important flow phenomena were observed.

2 Description of the experiments

The experiments were conducted in the Pilot Transonic Wind Tunnel (TTP) located at IAE, in São José dos Campos, Fig. 1.

The tunnel has a test section that is 30 cm wide, 25 cm high and 80 cm length, with slotted walls to diminish shock reflection from the walls. The tunnel has a conventional closed, continuous, circuit driven by a main compressor



Fig. 1 The Pilot Transonic Wind Tunnel [10].

of 830kW of power, and with an intermittent injection system which operates in a combined mode, for at least 30 seconds. TTP has automatic pressure controls from 0.5 bars to 1.25 bars, with Mach number varying between 0.2 and 1.3 as well as control of temperature and humidity in its test section. This facility is very feasible for academic researches, to carry out tests with simplified geometry vehicles, as rocket models and airfoils, airplane basic configuration as well as experiments aimed at the development of new aerodynamic transonic profiles [10].

The model analyzed in the present study is shown in Fig. 2. It is a 1:34 scale model of the VS-40 sounding rocket [9]. In the present study, just the fore-body section was considered, as shown in Fig. 3. In this figure it is possible to see the nine pressure holes installed in the fore-body section.



Fig. 2 The VS-40 model.



Fig. 3 The VS-40 model fore-body section.

From Fig. 3 it can be noticed the occurrence of a small backward step between the frustum cone and the cylindrical region. The reason is that this model was fabricated in 2009 and the rocket nose is a preliminary version of the Sub-orbital SARA, a Brazilian platform for microgravity experiments. In Portuguese, SARA means *Satélite de Reentrada Atmosférica*.

A launching of the VS-40 sounding vehicle carrying the SARA Sub-orbital is expected for soon, and an updated version of the platform will be used, the backward step observed in Fig 2 and Fig does not occur in the final configuration. However, in the current analysis, the small step is important to illustrate the impact that this type of protuberance can have in the flow pattern around a sounding rocketed model.

For the PSP measurements a commercial PSP system from Innovative Scientific Solutions, Inc. (ISSI) was used. A tiny layer of FIB basecoat (ISSI FB-200) was sprayed on the model surface followed by the application of the top coat (ISSI UF-400). In order to avoid blockage of the pressure taps, air was softly blew through the holes during the painting procedure. Once painted, the model was dried up in an oven at 60 °C for 90min. Then it was installed in the wind tunnel test section, as shown in Fig. 4.



Fig 4 The VS-40 model installed in the TTP test section.

Visualization using a simplified *schlieren* system was conducted in order to get insights about the shock occurrences and other flow features typical of the transonic regime since the *schlieren* technique is a very appropriate tool for studying non-homogeneities in a flow, as the

geometry of shock waves [9]. The experiments with *schlieren* S-40 was conducted later than the PSP measurements.

The *schlieren* work principle is well documented in the literature and it is based on the flow visualization due to illumination intensity variation, which occurs as an effect of the light beam refractions caused by local density gradients in the flow [9].

The main components of the *schlieren* system used in the present research are two 6” diameter parabolic mirrors, two 8” diameter flat mirrors, one knife edge, and a CCD camera PCO 1600. The light source used consists of a point source of white light with a diverging beam. This light source has basic adjustments, which allow the center of the beam to be directed to the first of two parabolic mirrors and a linear adjustment, allowing the source to be positioned at the focal point of the first parabolic mirror. This parabolic mirror focuses the light beam into parallel rays, which are directed toward the first flat mirror positioned at an angle of 45° to these rays, and which directs them to the TTP test section and chamber window, making in this way the parallel light beam to pass through these windows. The second flat mirror is positioned on the opposite side of the tunnel and directs the parallel light beam toward the second parabolic mirror, which collects and directs the light beam toward the knife-edge, where they are focused to a point, and goes on through the CCD camera [10]. The Schlieren system used in TTP is represented in Fig. 5.

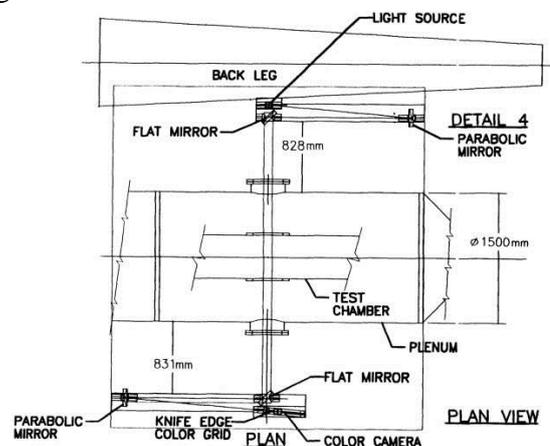


Fig 5 Schlieren system used in TTP.

3 Numerical Analysis

The present study gives continuation to a previous one, by the same authors [10], in which preliminary results were presented for few Mach number values, 0.6, 0.8 and 1.0. So, the description of the numerical simulation is basically the same presented in the previous article.

A Numerical investigation of the flow patterns on the V-40 fore-body section were conducted using the CFD++ commercial software package. The flow was modeled using RANS (Reynolds-Averaged Navier-Stokes) equations, together with the Spallart-Almaras turbulence model for turbulent viscosity calculations. The computational grid used was build of hexahedrons in most part of the domain, with a small number of prisms. Since the model is axisymmetric and only zero angle of attack was considered, the flow field was simplified and restricted to only 10° in the circumferential direction with 10 equally spaced planes, and the central plane was selected for getting the results for comparison purpose. The model was represented without the fins at its base with its geometry finishing at the end of the mesh. These considerations do not have an effect on the flow field at the region of interest in the present study, which is the model fore-body section. Undisturbed flow conditions were initially imposed at all field calculation volumes. Satisfactory boundary conditions were applied to the far field boundary, and adiabatic wall and boundary layer conditions at the solid walls. Figure 6 shows the CFD grid used in the simulations. It can be observed the mesh refinement on the transition between the frustum cone and cylindrical part of the model.

The numerical simulations main flow parameters are shown in Tab. 1

Table 1. Numerical Simulations flow parameters.

Mach Number	0.4	0.6	0.7	0.8	0.9	1	1.1
Pressure (Pa)	83919	73461	67551	61469	55401	49500	44029
Temperature (K)	297	286	281	276	268	262	252
Reynolds ($\times 10^6$)	2.03	2.72	2.94	3.08	3.17	3.19	3.18

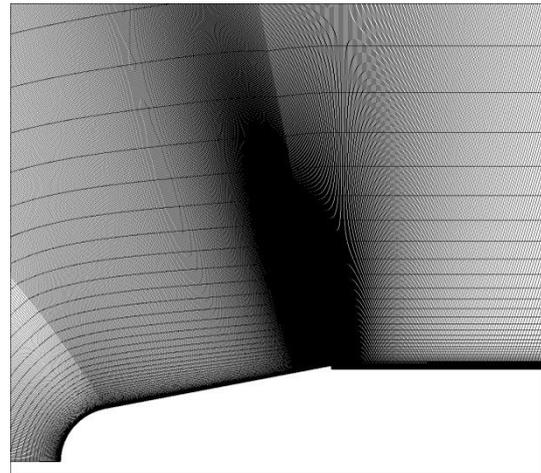
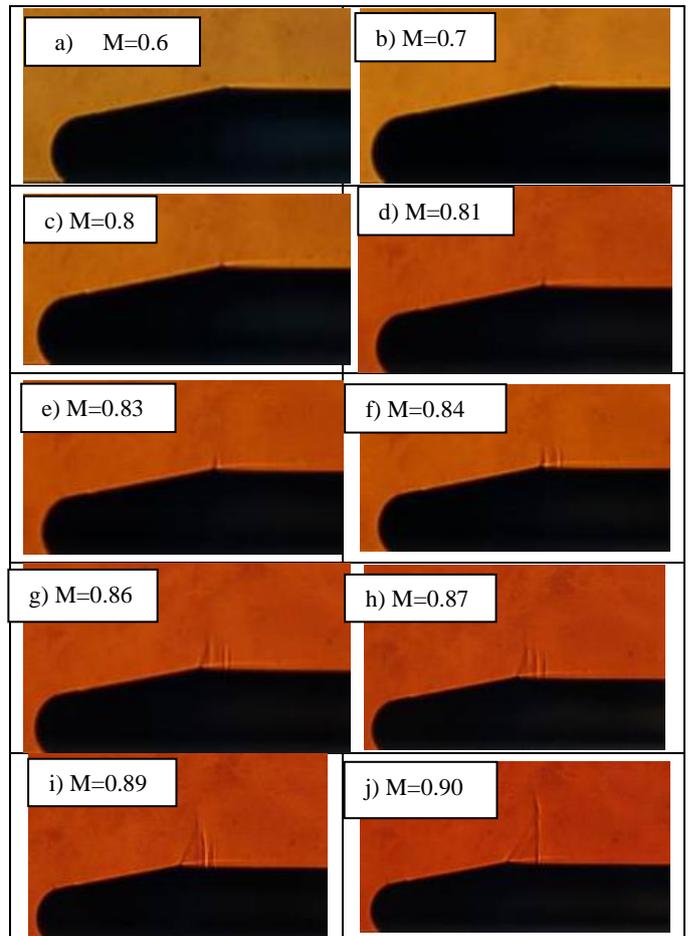


Fig 6 Numerical Grid.

3 Results and Discussions

Schlieren imagens starting with Mach Number of 0.6 up to Mach Number equal to 1.08 are presented in Fig 7.



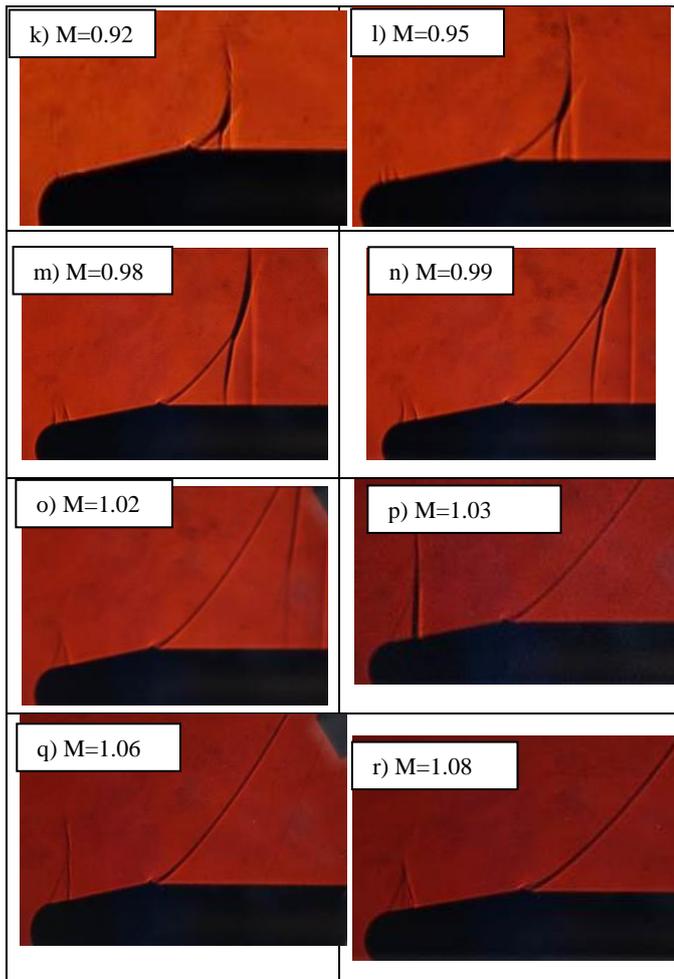


Fig 7 Schlieren images of flow patterns around VS-40 fore-body section for $M=0.7$ to $M=1.08$.

At Mach numbers of 0.6 and 0.7 the flow seems completely subsonic, but a separation region can be noticed at Mach number of 0.7 and 0.8. At Mach number 0.81 it is not possible to perceive significant expansion or shock formation, only a tiny region at the frustum-cone end indicating some density gradient. Apparently this region progress until showing some appreciable contrast at Mach number 0.83 where one can see a small dark region followed by a higher one at Mach number of 0.84. Then, it is possible to conclude that the critical is about Mach number 0.83. Although the images are not of high definition, it is possible observe the boundary layer detachment caused by the back step which connects the frontal part to the cylindrical region of the model. At Mach number 0.86 the expansion effect is stronger and a mild multiple shock

wave formation is observed. This kind of formation is provoked by the interaction between laminar boundary layer and shock wave [11]. Stronger formation can be perceived at Mach number 0.87 and a very familiar expansion and shock wave pattern can be initially seen followed by two shock waves. As the speed is increased this first formation is expanded and the two shocks merge in one at Mach number 0.89. The following image (Mach number 0.91) shows that the shocks are merged in one normal shock wave in a very familiar pattern of expansion followed by shock wave. However, when the speed is still increased this pattern is modified probably due to a strong boundary layer detachment at the frustum cone end (see Mach number 0.92). In this case the outer flow coming from the frustum cone does not feel the corner but a distorted geometry lead by the detached boundary layer and the expansion region has an other formation. Also the shock wave has a more complex look. At Mach number 0.95 one can see that this effect is amplified. At this same Mach number condition it is possible to observe that the front region of the model starts to reach supercritical condition shown by tiny multiple normal shock pattern. It is interesting to note that in this region the number of tiny shocks is very high because of the very laminar condition of the boundary layer. It is known that laminar boundary layer are weaker and any adverse gradient of density can provoke local thickening of boundary layer and consequent expansion/shock wave group. In the subsequent pictures, one can see that the increase of the speed will provoke the merge of these tiny expansion/shock wave in fewer and stronger ones. At Mach number 0.98 one can see only two of these groups and at Mach number 1.01 only one group with a very familiar shape can be observed. In the same speed range (from Mach number 0.92 to 1.06) the expansion / shock wave formation after the frustum cone evolves to stronger shock wave with single formation, after experiencing boundary layer / shock interference.

It is worth noting that for Mach number 1.08 it is not possible to observe anymore the shock wave after the frustum cone and the expansion/shock wave formation at the initial

part of the model is well defined. It is important also to observe the presence of a mild shock wave in front of the model because of the supersonic characteristic of the flowfield. For Mach number between 1.01 and 1.06 it is not possible to observe the detached shock wave in front of the model because it is out of the visualization window.

Numerical results of the flow patterns around the flow field around the VS-40 fore-body section is shown for the Mach numbers values of 0.4, 0.6, 0.7, 0.8, 0.9, 1.0 and 1.1. The parameters used in the simulations are presented in Tab 1.

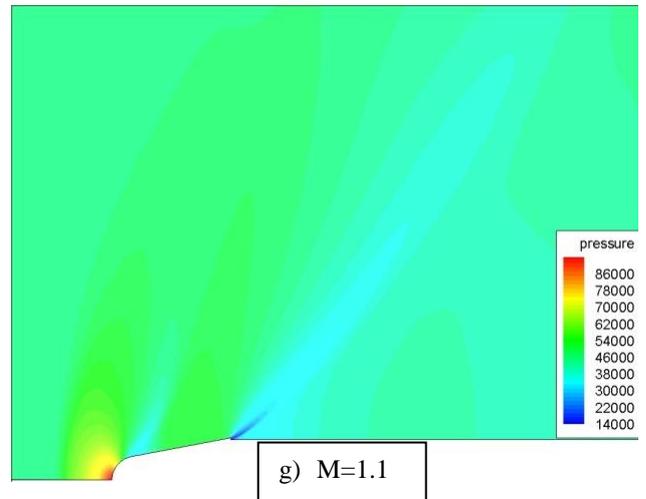
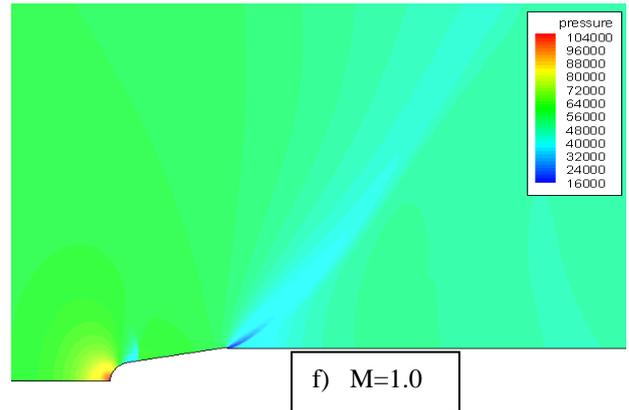
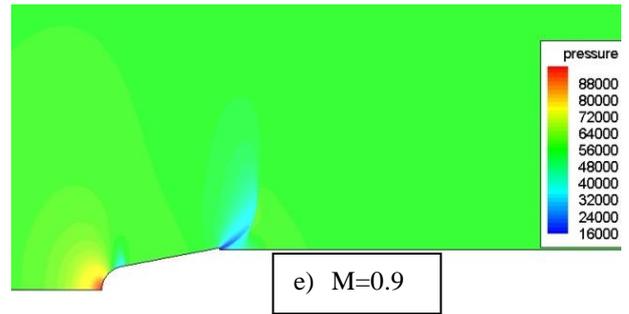
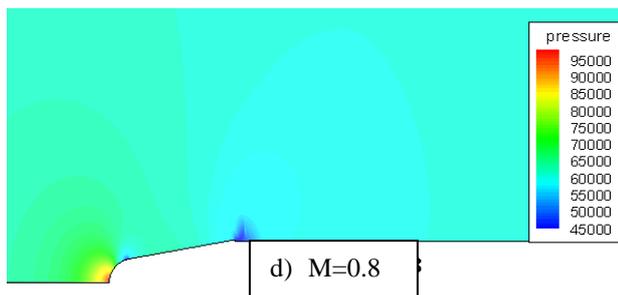
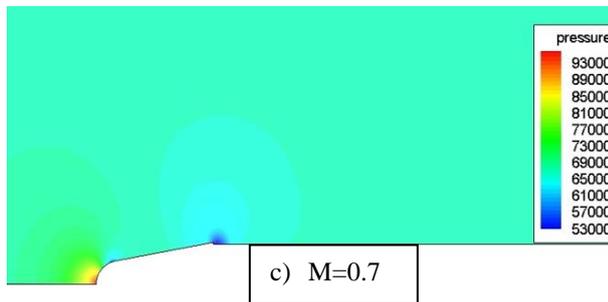
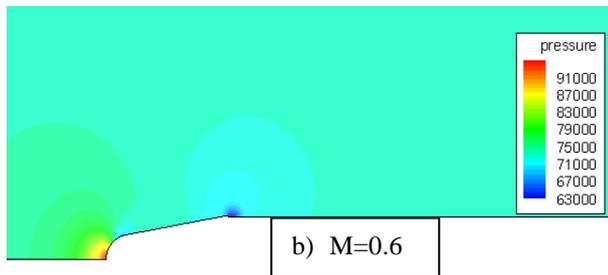
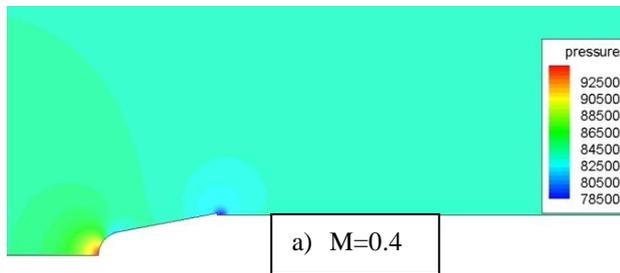


Fig 8 Numerical pressure fields obtained for M=0.4, 0.6, 0.7, 0.8, 0.9, 1.0 and 1.1.

From Fig 8a to Fig 8c, no significant pressure gradient is observed, which is coherent with the flow patterns shown in the *schlieren* images shown in Fig 7a and Fig 7b. Comparing Fig 7c, M=0.8, or Fig 7d, (M=0.81) with Fig 8d it, and Fig 7i and Fig 7j, (M=0.89 and M=0.9) with Fig 8e, it can be noticed that in the related *schlieren* images, as well as in the CFD pressure field, for M=0.8, no significant expansion or shock

formation occurs, just a small pressure reduction region at the frustum-cone. However, for $M=0.9$ shock wave occurrence are clearly noted from the *schlieren* images, Fig 7i and Fig 7j, and also from the CFD pressure field, Fig 8e. For Mach numbers 1.0 and 1.1, the shock wave geometry shown in the *schlieren* images were reasonably well captured by the CFD analysis. In the initial part of the model, the expansion/shock wave patterns seems well captured by the numerical simulations.

Figure 9 shows experimental results obtained with the PSP technique, pressure taps and CFD simulations, for three different Mach number values. The blue squares in Fig. 9 show the relative differences between pressure taps and CFD results. The red squares show the relative differences from pressure taps to the PSP curve. Regarding to the PSP results, the pressure profiles shown in Fig. 9 were obtained along the dashed lines on the PSP field shown in Fig. 10, which was drawn over the tap holes, for each Mach number value considered. The PSP software used for data processing, OMS 3.1 ProImage, allows showing pressure profile along any interval on the pressure field.

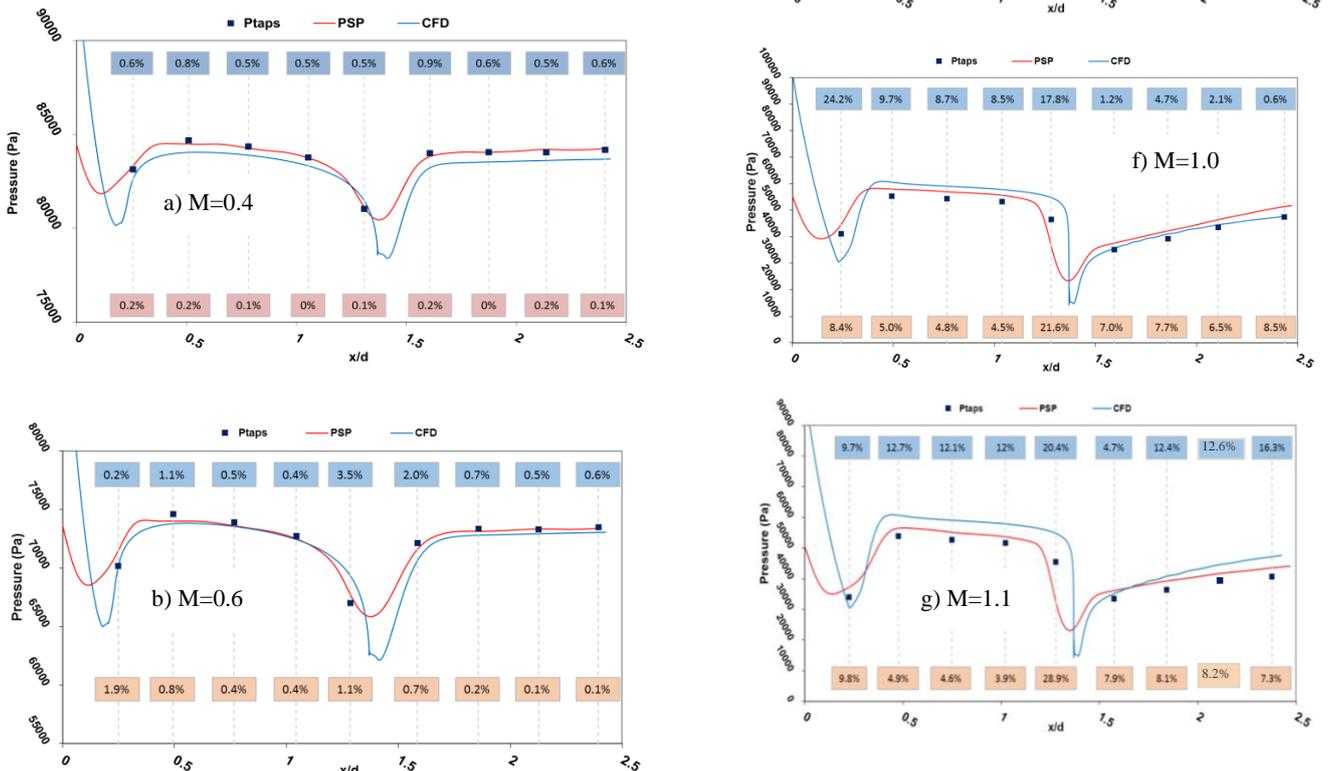


Fig 9 Pressure profiles – CFD, PSP, Ptaps.

In Fig. 10 PSP results of pressure field over the model surface are presented for the Mach number values of 0.6, 0.7, 0.8, 0.9, 1.0 and 1.1, *i.e.*, all Mach numbers considered except 0.4, which was not added because the pressure gradient in this velocity range is very low, being hard to perceive with the same palette of colors. Even though the fact that CFD simulations were carried out without taking into account the wind tunnel walls, in general the agreements between numerical and experimental results are reasonable, especially for the Mach number values of 0.4, 0.6, 0.7 and 0.8. For $M=0.9$, $M=1$ and $M=1.1$, the differences between CFD and experimental results are more significant, mainly regarding to the 5th pressure taps, which is located at the end of the frustum cone, where the back step region aggravates the expansion region, which is more evident in CFD results. The experiments show pressure profiles much smoother than those from CFD, probably because the highly refined mesh applied in the back step region allowed more detailed information regarding the pressure field over the model, mainly for the higher values of Mach number, when complicated flow patterns with shock waves occurrence and shock wave boundary layer interactions take place. For $M=0.9$, the disagreement between numerical and experimental results were higher than expected, which is due probably to the complicated flow patterns shown by the *schlieren* visualization for this Mach number. The highest difference is observed in the back step region. In general, the best agreements were observed between the PSP and pressure taps results.

It is worth noting that the partially spherical geometry at the model tip probably can be resulting in bad pressure determination with PSP because only one CCD camera was used, and the limit of the reflection wave angle from a surface is about 50° . This explains the discrepancies found close to the model tip for all study cases.

A comparative observation in Fig 10a to Fig 10f, and observing also the palette of colors, the decrease of pressure in the regions of shock and expansion waves can be easily noted. The pressure gradient in these regions intensifies as

the Mach number increase, as shown in the pressure profiles in Fig. 9.

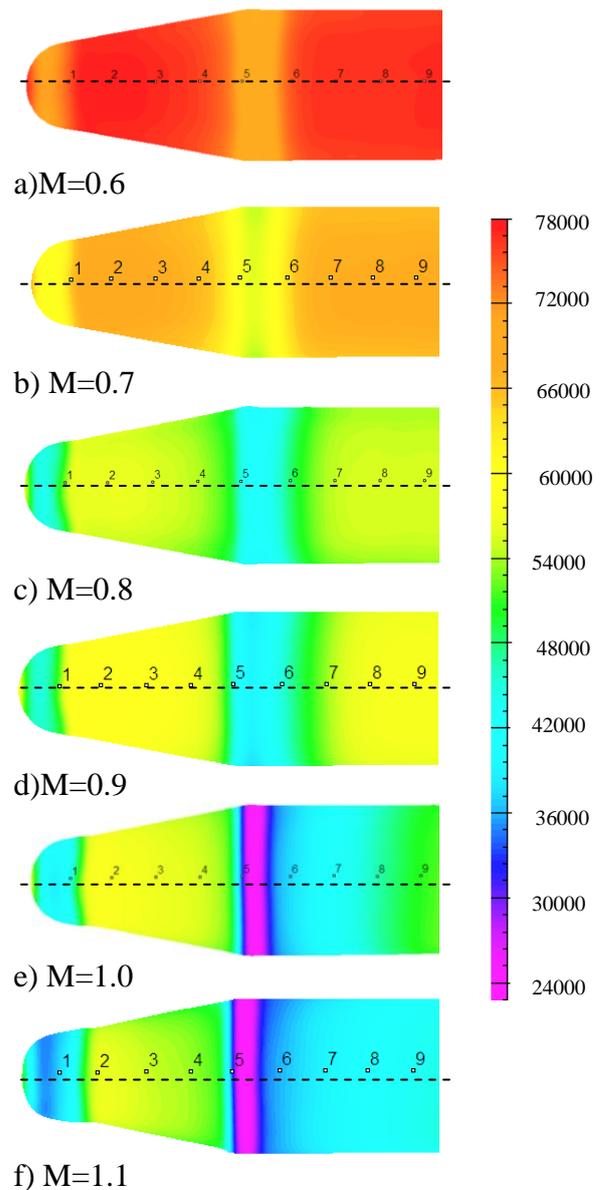


Fig 10 PSP pressure field on the model surface.

4 Conclusions

Relevant physical phenomena along a sounding rocket, as compression, shock and expansion wave formations, as well as interactions between shock-wave and boundary-layer were investigated in the present work. The CFD analysis was very useful in providing more detailed information about the flow field in terms of parameters not easily determined experimentally as well as in regions of hard

access for measurements, as in the backward step occurrence in the geometry analyzed.

The combination of the three experimental techniques used demonstrated as well a very powerful methodology of pressure measuring over surfaces. The Pressure Sensitive Paint (PSP) is a very effective approach for obtaining pressure distribution on model whole surface and together with *schlieren* images allows to obtaining a great amount of information about very complex flow phenomena occurring mainly in the transonic region.

For each study case errors were determined between different approaches, *i. e.*, between PSP and pressure taps and CFD and pressure taps and good agreements were observed.

Acknowledgments

The authors thanks FAPESP through the process 2013/23690-2, aimed at repairing the TTP compressor blades, CAPES trough Pró-estratégia project number 20, 2011 and CNPq for the financial support through the processes: 560200/2010-2, PIBIC/IAE 800031/2013-0 and 310646/2012-0 (first author)

References

- [1] Wessling F. C. and Maybee G. W. Sounding Rocket Lessons Learned. *Proc of the AIAA 29th Aerospace Sciences Meeting*, January 1991.
- [2] Liu, T., Sullivan, J. P. *Pressure and Temperature Sensitive Paints*, Springer-Verlag, New York, 2005.
- [3] Engler R. H., Klein C. and Trinks O. Pressure sensitive paint systems for pressure distribution measurements in wind tunnels and turbomachines, *Meas. Sci. Technol.*, Vol. 11, 1077, doi:10.1088/0957-233/11/7/320, 2000.
- [4] Gregory J. W., Asai K., Kameda M., Liu T. and Sullivan J. P. A Review of Pressure Sensitive Paint for High Speed and Unsteady Aerodynamics. *Proc of the Institution of Mechanical Engineers, Part G, Journal of Aerospace Engineering*, Vol. 222, No. 2, pp. 249-980, 2008.
- [5] Bell J. H., Schrairer E. T., Hand L. A. and Mehta R. D. Surface Pressure Measurements Using Luminescent Coatings, *Annu. Rev. Fluid Mech.*, vol. 33, pp.155–206, 2001.
- [6] Vardaki E., Stokes N. P., Patel S. and Gustafsson P. Pressure Sensitive Paint Measurements on the Gripen Model at the ARA Transonic Wind Tunnel. *Proc of the 50th AIAA Aerospace Sciences Meeting including*

the New Horizons Forum and Aerospace Exposition, 2012, Nashville, Tennessee 4 Copyright Issues.

- [7] Quinn M. K., Gongora-Orozco N., Kontis K. and Ireland, P. Application of Pressure Sensitive Paint to Low-speed Flow around a U-bend of Strong Curvature. *Experimental Thermal and Fluid Science*, Vol. 48, pp. 58-66, 2013.
- [8] Kline S. J., *Flow Visualization*, internet movie from the National Committee for Fluid Mechanics Films under a grant from the National Science Foundation, <http://web.mit.edu/hml/ncfmf.html>, “Flow Visualization”, 1943.
- [9] Merzkirch W., Density-Based Techniques, *from The Springer Handbook of Experimental Fluid Mechanics*, 2007.
- [10] Avelar A. C., Basso, E., Falcão Filho, J. B. P., Romero, P. G. M. and Hsu, J. J. L. Experimental and Numerical Analysis of the Flow Patterns around a Sounding Rocket in the Transonic Regime. *Proc of 32nd AIAA Applied Aerodynamics Conference, AIAA Aviation and Aeronautics Forum and Exposition*, Atlanta, USA, 2014.
- [11] Houghton E. L. and Carpenter P. W. *Aerodynamics for Engineering Students*, 5th ed. – Oxford, Boston: Butterworth/Heinemann, 2003.

Copyright Statement

The authors confirm that they, and/or their company or organization, hold copyright on all of the original material included in this paper. The authors also confirm that they have obtained permission, from the copyright holder of any third party material included in this paper, to publish it as part of their paper. The authors confirm that they give permission, or have obtained permission from the copyright holder of this paper, for the publication and distribution of this paper as part of the ICAS 2014 proceedings or as individual off-prints from the proceedings.