Rebuilding of new experimental tests on a double cone at mach 9

Original
Rebuilding of new experimental tests on a double cone at mach 9 / Schettino, A.; Battista, F.; Ranuzzi, G.; D'Ambrosio, Domenic. - STAMPA. - ESA-SP-659:(2009).

Publisher:
ESA Communication Production Office ESTEC, Noordwijk, The Netherlands

Published
DOI:

Rebuilding of new experimental tests on a double cone at Mach 9

A. Schettino(1), F. Battista(1), G. Ranuzzi(1)
D. D'Ambrosio(2)

(1)*CIRA (Italian Aerospace Research Center), Via Maiorise 81043, Capua, Italy, Email: a.schettino@cira.it*
(2)*Politecnico di Torino, Dept. of Aerospace Engineering – C.so Duca degli Abruzzi, 24 - 10129 Torino, Italy, Email: domenic.dambrosio@polito.it*

ABSTRACT

This paper deals with the numerical rebuilding of an experimental test performed at Alta Spa in the frame of the italian project CAST on a double cone geometry, obtained scaling down the one already tested in the LENS facility at Calspan [1]. Also the nozzle flow was simulated, in order to get the correct non-equilibrium freestream conditions for the test article. The results are compared with the experimental data provided by Alta Spa [2].

A sensitivity analysis has been also performed in order to verify the effect of a number of parameters for which some uncertainties are still present.

1. INTRODUCTION

In the frame of the CAST project, funded by the Italian Space Agency (ASI), an advanced CFD code is being developed capable to simulate high enthalpy reacting flows on complex geometries by means of state of the art physical models; several italian research centres and University departments are involved in the project; in fact, one of the goals is to realize a strong synergy among all the italian experts in this field, creating a national consortium devoted to the study of aerothermodynamics.

The basic core of the numerical code is the H3NS code, developed at the Italian Aerospace Research Centre (CIRA) in the frame of an internal project, namely CLAE (Configuration and Local Aerothermodynamic Effects), whose goal is to improve tools and competences to be applied in the design phase of a hypersonic vehicle like the USV re-entry test bed. H3NS code, together with the bi-dimensional axisymmetric version (H2NS), was widely validated with respect to several test cases in the last years [3, 4].

In order to improve the validation of the code, many experimental tests are planned within CAST in relevant conditions. In this paper the attention is focused on the first two test cases, performed at Alta Spa, respectively consisting in an expansion nozzle (namely TC-2) and a 25-55 deg double cone probe (namely TC-1a). It must be underlined that both the nozzle and the probe were designed and built in the frame of CAST [2, 5]. In the design phase, several CFD computations were performed by CIRA and "Politecnico di Torino" (PoliTO) on different nozzle configurations to drive the design; as far as the double cone is concerned, the reference geometry is the one already tested in Calspan facilities [1, 6], to emphasize some important phenomena like shock wave boundary layer interaction (SWBLI) and shock-shock interaction.

The sharp double cone geometry was selected because the availability of previously obtained data allowed to design the experiment with a well established experience. However, the new experimental tests have been planned in the frame of CAST project in order to make direct experience on this kind of experiments and to get information not available from literature, with a direct control of the accuracy of the free-stream conditions and of the measurements over the test article.

2. TESTS REQUIREMENTS

Several computations were performed at the beginning of the project in order to define the requirements for the test; more in detail, the main goals were:

- to maximize the test article dimensions, in order to be able to include as many sensors as possible;
- to define the sensors position in order to correctly "capture" the SWBLI phenomenon;
- to define the best reservoir conditions (i.e. total pressure and enthalpy) to maximize the SWBLI phenomenon

In *Figure 1* and *Figure 2* the double cone geometry and the typical flow structure expected from the experiment are shown.

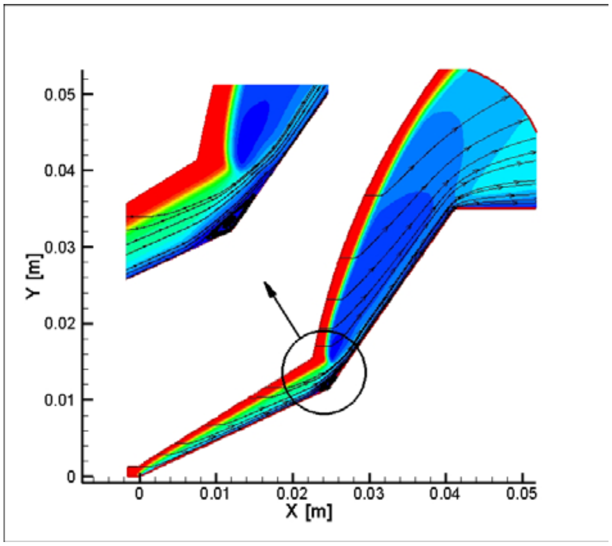

Figure 1: Mach number contour (run 3)

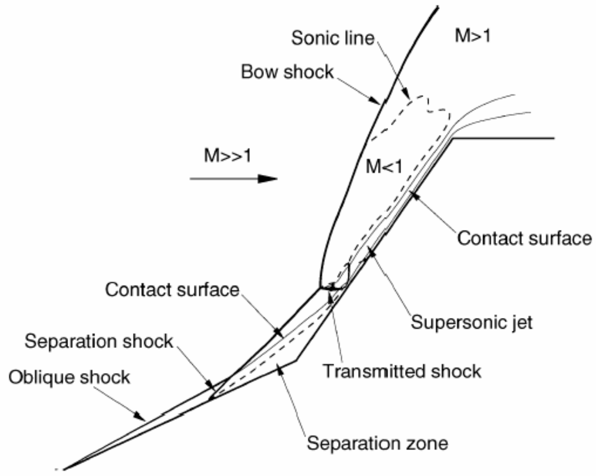

Figure 2: Shock interaction pattern [7]

At the end of the preliminary design phase, the following requirements were given to Alta concerning the nozzle exit conditions, that represent the input for the double cone tests:

- 9 ≤ Mach ≤ 10;
- core flow diameter > 80 mm
- Reynolds number (based on model base diameter) ≥ 2.000 (Re ≥ 4.000 is preferred)
- maximum flow angularity ≤ 1°
- several pressure and temperature measurements, including thermography (with heat fluxed indirectly derived)

2. NUMERICAL APPROACH

The numerical simulations have been performed with two different numerical codes, developed by CIRA and PoliTO, whose main characteristics are summarized in the following sub-sections.
The simulations have been conducted independently, with different grids and convergence strategies; however, the grid independence has been verified with both codes.
The two codes use the same numerical scheme, while some differences are present in the physical modelling; some details are given hereinafter.

2.1 CIRA code H3NS

The CFD code H3NS was developed at CIRA by the Aerothermodynamics and Space Propulsion Laboratory. Together with the bi-dimensional version H2NS it was validated with respect to several numerical and experimental data in the last years, in particular for what concerns the SWBLI phenomenon [3].

The code solves the full Reynolds Averaged Navier-Stokes (RANS) equations, taking into account chemical and vibrational non-equilibrium. Several models are available; in the present work the Park model with five species (O, N, NO, O_2, N_2) and 17 chemical reactions [9] was used. The vibrational relaxation times are obtained by the Millikan-White [10] theory modified by Park [11]. The viscosity and binary diffusion coefficients for the species are computed by means of Yun and Mason collision integrals [13], while the conductivity coefficient is obtained by using Eucken's law. The coefficients for the gas mixture are calculated using semi-empirical Wilke formulas.

From the numerical point of view, the code is based on a finite volume approach. The inviscid fluxes are computed by means of a Flux Difference Splitting scheme [14, 16], while second order approximation is obtained with an Essentially Non Oscillatory (ENO) reconstruction of interface values [17]. Time evolution is performed by an explicit multistage Runge-Kutta algorithm coupled with an implicit evaluation of the source terms.

To take into account the effects of rarefaction, the slip boundary conditions proposed by Kogan [15] have been used. These boundary conditions have been obtained by matching the solution of Boltzmann equation in the Knudsen layer to the solution of the macroscopic Navier-Stokes equations, thus yielding:

$$V_s = 1.012\lambda\left(\frac{\partial V_\tau}{\partial n}\right)_w \quad (1)$$

$$T_s - T_w = 1.73\frac{\gamma}{\gamma-1}\frac{\sqrt{\pi}}{4}\lambda\left(\frac{\partial T}{\partial n}\right)_w \quad (2)$$

where V_S is the "slip velocity" and T_S the "slip temperature".

2.2 PoliTO code

The CFD simulation tool used by the Aerothermodynamics group based at the Department of Aerospace Engineering of the Politecnico di Torino (PoliTO) is the result of a continuous collaborative activity carried out in the last years with the Aerothermodynamics and Propulsion Group of Thales Alenia Space Italia – Torino (TAS-I-TO)

The code is capable of simulating 3D, 2D axisymmetric and 2D plane high-temperature flowfields using multi-block domain decomposition and a finite volumes discretization. The integration in time is carried out adopting an explicit scheme, except for chemical and internal energy source terms, which are treated implicitly. As for H3NS, the convective fluxes at cells interfaces are evaluated using an upwind flux-difference splitting method [16], with ENO-type second order reconstruction [17]. Viscous fluxes are evaluated using centered schemes. The code can run on parallel architectures both in OPEN MP and in MPI mode.

The PoliTO/TAS-I-TO code can handle any reacting gas mixture in chemical and vibrational non-equilibrium, provided that data concerning species properties, chemical equilibrium constants, chemical reaction rates and vibrational relaxation parameters are provided in an appropriate format through input files.

Transport models go from the simplified Fick's law and Wilke's formulas to first order Chapman-Enskog theory in the first non-vanishing approximation in terms of Sonine polynomials. Any set of collision integrals can be used to evaluate transport coefficients, provided their functional dependence from temperature is described according to a specific fitting formula.

For the simulations presented in this paper, the code was run in the 2D-axisymmetric mode. Since ionization is not likely to occur due to the relatively low energy levels, a 5 species air model was selected. Chemical reaction rates are the one proposed by Park in [11]. Vibrational non-equilibrium is modelled accounting for the V-T transfer mechanism only and using the Millikan and White relaxation rates with the correction suggested by Park in [12]. Transport coefficients are obtained using the collision integrals by Gupta et al. [18]. Mixture viscosity, thermal conductivity and mass diffusion fluxes are obtained solving the algebraic systems arising from the Chapman-Enskog theory in the first non-vanishing approximation in terms of Sonine polynomials.

3. NOZZLE REBUILDING

An extensive test campaign has been performed to characterize the nozzle exit conditions, that are the input for the double cone tests [5].

The nozzle flow was numerically rebuilt to calculate the exit conditions and to verify if they fulfilled the requirements described in the previous section.

Hereinafter the nominal conditions used for the first CFD simulations are listed, together with the hypothesis made for the boundary conditions:

- $P_0 = 2.95$ bar
- $H_0 = 4.98$ MJoule/kg
- Wall temperature = 300 K
- Catalytic wall

These boundary conditions are justified by the fact that it is a pulsed wind tunnel, and therefore the wall temperature does not significantly increases during the run; furthermore the throat region of the nozzle is mostly metallic, and therefore can be assumed fully catalytic.

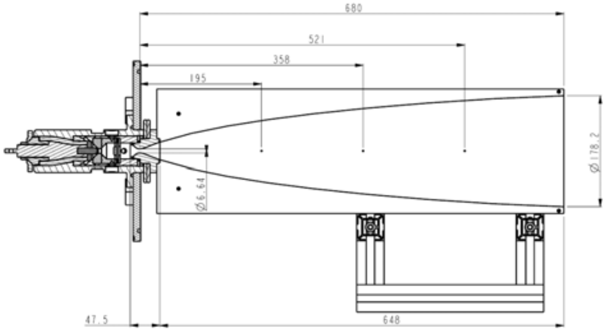

Figure 3: Scheme of Alta Spa nozzle

In Figure 4 and Figure 5 the numerical results are compared with the available experimental data. From Figure 4 it can be seen that the pitot pressure computed by CIRA at the nozzle exit is in good agreement with the experiments, while the PoliTO computation provides a higher pitot pressure with respect to the

experimental one. However it must be underlined that in general the numerical solutions were found very sensitive to small differences of the physical modelling and of the geometry. In *Figure 4* also the effect of the uncertainty in the nozzle diameter is shown. In fact, the nominal throat diameter used for the computations is 6.46 mm; however, an accurate measurement of the nozzle throat is not easy, due to its very small dimensions. According to the ALTA measurements, the real throat diameter could range between 6.46 and 6.48 mm. The effect of this uncertainty is quite small, but not negligible, as can be seen in *Figure 4*, being the difference approximately 50 Pa.
As far as the core flow diameter is concerned, it can be seen that it is approximately 80 mm, as required.

In *Figure 5* also the wall pressure is plotted; it can be seen that for this variable the agreement is good by using both codes, being the numerical results within the experimental error bar.

On the basis of the results of this preliminary phase, it was concluded that the nozzle fulfilled the requirements and that the CIRA code well reproduces the experimental data, while with the PoliTO code some discrepancy in the pitot pressure is still present.

It must be added that some simulations have been performed also with cold gas (T_0=300 K), and in this case the agreement with experimental data was not satisfactory; this was probably due to a separation of the flow inside the nozzle, due to the stronger expansion. However, the cold case results are still under analysis.

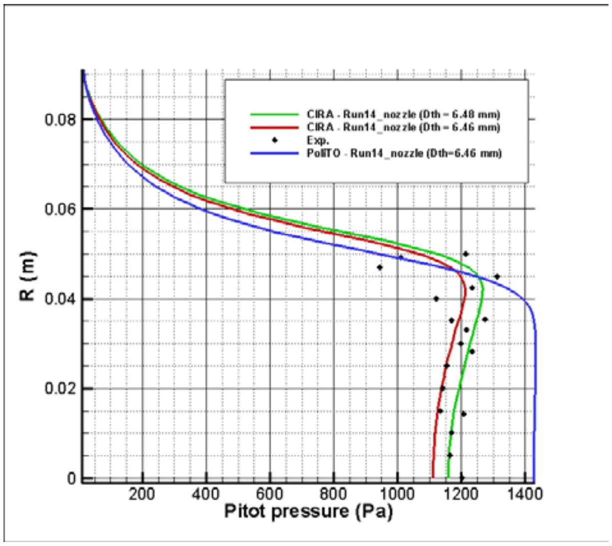

Figure 4: P_0=2.95 bar. H_0=4.98 MJoule/kg. Pitot pressure

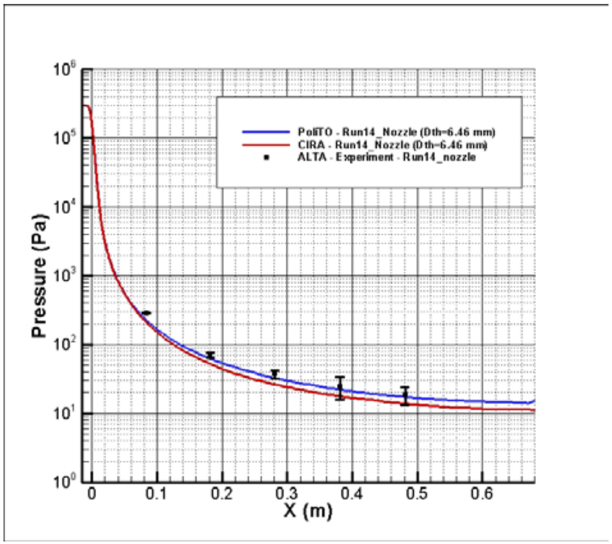

Figure 5: P_0=2.95 bar. H_0=4.98 MJoule/kg. Wall pressure

.

Once the nozzle characterization has been completed, the tests with the double cone (*Figure 6*) probe have been performed [2].
Two nominal conditions have been selected:

ID	P_0 [bar]	H_0 [MJoule/kg]
Run 8 (12-03-2008)	3.11	3.61
Run 3 (13-03-2008)	3.17	4.64

In *Figure 7* and *Figure 8* the Mach number and pressure profiles at the nozzle exit are respectively shown for both cases. It can be noted an inversion of the trend of the nozzle expansion, probably due to the different transport models.

Figure 6: Double cone probe installed in the test chamber.

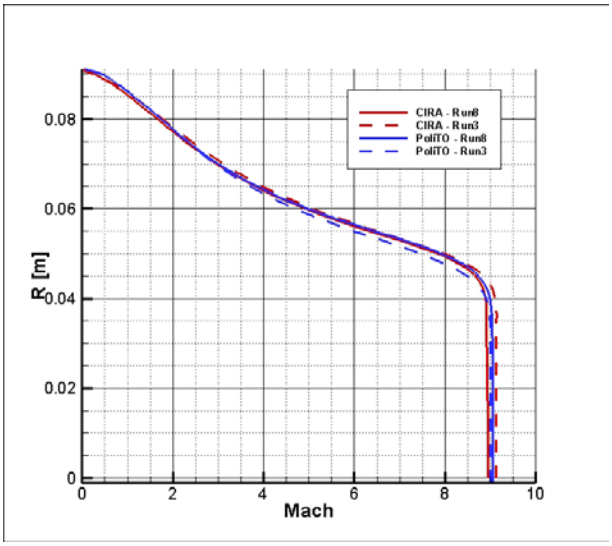

Figure 7: Inlet conditions for double cone. Mach number

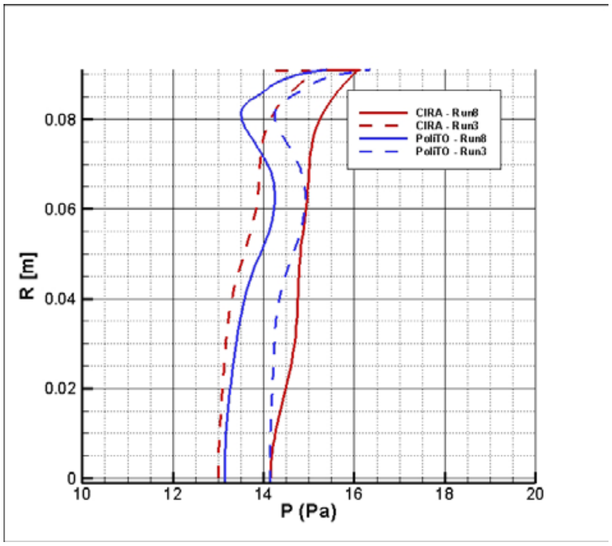

Figure 8: Inlet conditions for double cone. Exit pressure

It must be underlined that it is important to simulate the nozzle in order to get the input conditions for the double cone, because, as shown also in [7], the non-equilibrium conditions have a significant influence on the solution. In *Figure 9* the vibrational temperatures along the symmetry axis are plotted; it can be seen that the two codes are in a very good agreement.

The details of the performed test campaigns are reported in references [5] and [2].

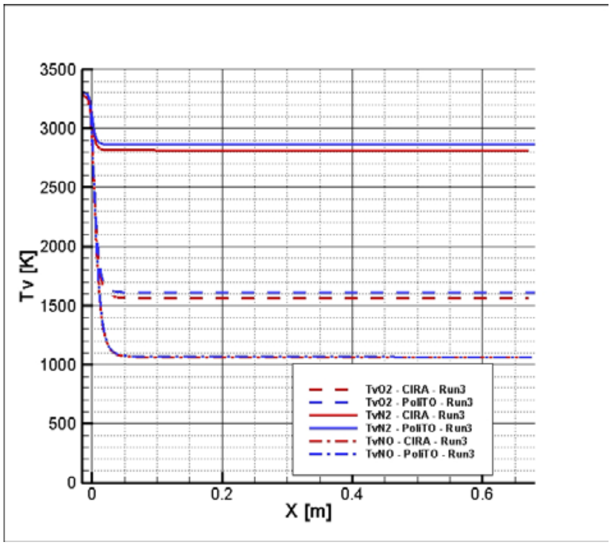

Figure 9: Run3 and Run 8. Vibrational temperatures along the axis

4. DOUBLE CONE RESULTS

In this section the numerical results obtained with both H3NS and PoliTO codes are shown and compared with the available experimental results [2]. However, it must be underlined that further experimental results are still under analysis and could be very useful to improve the interpretation and the accuracy of the experimental data, especially for what concerns the heat fluxes, that are obtained with a reverse method from the temperature measurements; therefore these comparisons will be updated in a future work.

As already written, the input conditions for the computations shown in this section are the ones shown in *Figure 7* and *Figure 8*; more in detail, the axis values of the PoliTO solutions, reported in the next table, were used.

ID	Mach	Pres [Pa]	Temp [K]
Run 8	9.058	13.14	168.9
Run 3	9	14.14	200

In *Figure 10* and *Figure 11* the resulting wall pressure and heat fluxes along the double cone are respectively plotted.

From *Figure 10* it can be seen that, with respect to the experimental data, both CFD codes seem to show a stronger interaction, with a much higher pressure peak and a well visible effect of the transmitted shock with a consequent second smaller peak of pressure over the 55 deg cone. From the measured pressures and heat fluxes, instead, the separation length seems smaller. More in detail, from the thermography data shown in [2], it seems that a clear separation is visible in run 8, but not in the run 3.

As far as the pressure levels are concerned, it must be also noted that the difference between Run 3 and Run 8 conditions is most significant in the CFD results, especially the ones from CIRA, while is almost negligible in the experimental data.

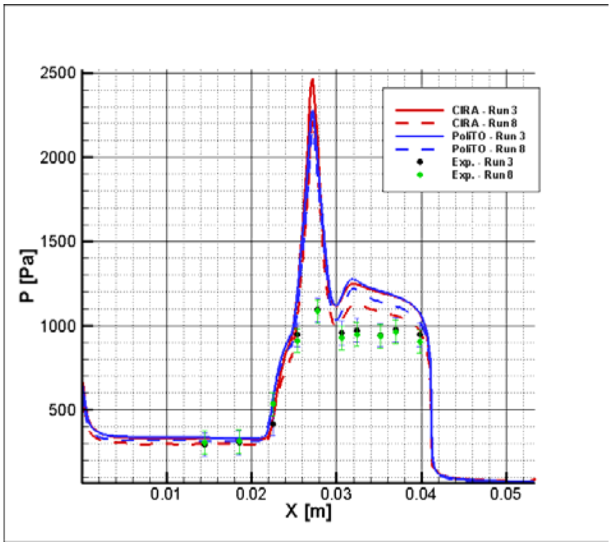

Figure 10: Double cone. Wall pressure comparison for Run 3 and Run 8

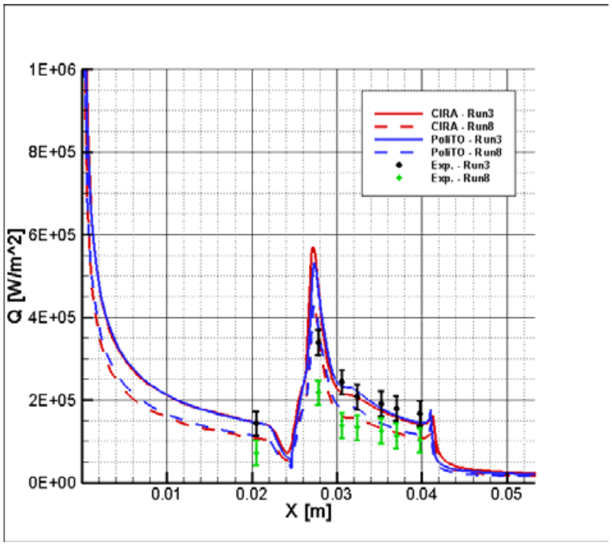

Figure 11: Double cone. Heat flux comparison for Run 3 and run 8

Based on this preliminary analysis, it seems in general that in the experiments the SWBLI phenomenon has been weaker than expected; the reasons for this behaviour are still under analysis; in [2] a preliminary hypothesis is made that the difference is due to the fact that the runs characterized by a higher enthalpy, as Run 3, are more unstable and therefore exhibit a worse heat transfer to the fluid and a less uniform flow. However the data are still under analysis, also taking into account the preliminary numerical rebuilding shown in the present paper.

It must be also noted that, in spite of the previous considerations, the comparison between the numerical and experimental heat fluxes is very good, especially for run 3. This fact is quite strange, considering that, in general, by means of CFD it is much easier to well reproduce the pressure rather than the heat flux; a possible explanation for this aspect is discussed in the following section.

5. SENSITIVITY ANALYSIS

In order to support the analysis of the experimental data and verify the possible effects due to a number of parameters for which some uncertainty is present, a sensitivity analysis with respect to these parameters has been performed with H3NS code.

As previously stressed, a first aspect that was verified is the influence of the real geometry of the nozzle (*Figure 4*); as already written, the effect is small but not negligible, being the difference in the pitot pressure nearly 50 Pa.

Another point that was verified is the effect of the wall temperature on the solution; in fact, it is well known that the wall temperature can have a significant influence on the separation length. In all the computations described in the previous sections, an average wall temperature of 350 K was assumed, but the experimentally measured temperature typically ranges from 300 K to 380 K, also depending on the location along the test article; therefore, some computations were performed imposing a wall temperature of 300 K, in order to see the sensitivity of the solution in the range of interest. The result of this analysis was that no significant influence of the wall temperature was found in this range.

Another possible uncertainty is due to the wall catalysis; the test article is made of Macor, that is a material typically considered as non catalytic; however, depending on the test case, it can happen that also a very low catalicity can have a not negligible influence on the solution, especially for what concerns the heat flux. Therefore, an additional computation has been performed assuming a partially catalytic wall with $\gamma_O = \gamma_N = 0.01$. Also in this case, however, no significant effect was found neither on the wall pressure nor in the heat flux (*Figure 12*).

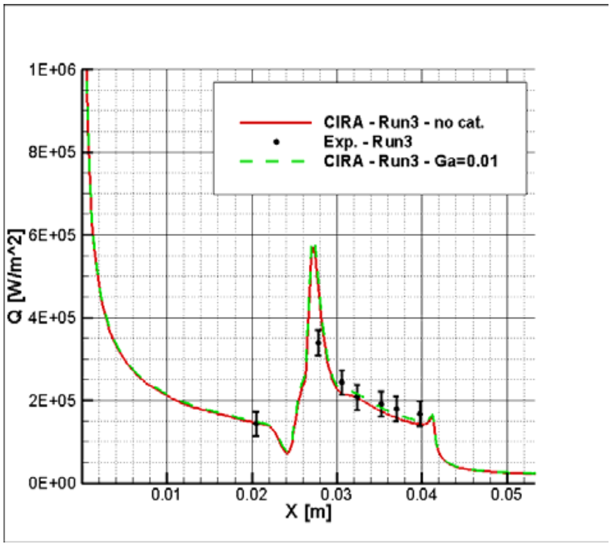

Figure 12: Double cone. Run 3. Effect of wall catalysis

Another aspect that was taken into account in the present work is the possibility of incipient rarefaction, that could reduce the separation length [7, 8].

The Run 3 was recomputed imposing a slip conditions on both velocity and temperature; actually a small effect on the separation was obtained, but no significant effect was found on the pressure levels, except over the nose tip (*Figure 13*).

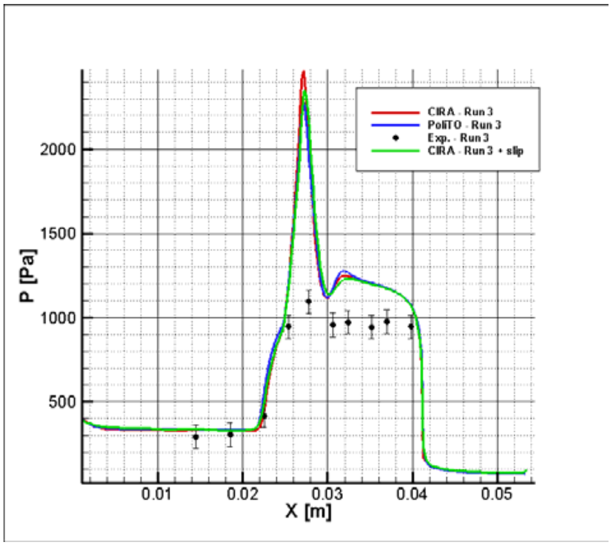

Figure 13: Double cone. Run 3. Effect of slip condition

An important effect was instead found due to the uncertainty in the freestream conditions. As already shown in previous section, comparing the results of the nozzle simulations obtained through CIRA and PoliTO codes, some small differences were found (*Figure 7* and *Figure 8*); therefore, Run 3 was repeated by using exactly the nozzle exit conditions computed by CIRA, that are

ID	Mach	Pres [Pa]	Temp [K]
Run 3	9.13	13	191

This very small difference had a significant effect on the pressure level after the reattachment, as can be seen in *Figure 14*. The solution still does not match the experimental data, but this effect indicated a good direction for future investigations; in fact, since the nozzle solution is very sensitive to small differences in geometry and physical modelling, this aspect must be carefully taken into account. In particular, it can be very interesting to verify the effect of different transport models, that can affect the nozzle exit solution.

Moreover, in *Figure 14* it can be seen that this small change in the freestream conditions does not significantly affect the heat flux, that remains within the experimental error bar; this can justify the fact that the comparison was better in the heat flux with respect to the pressure.

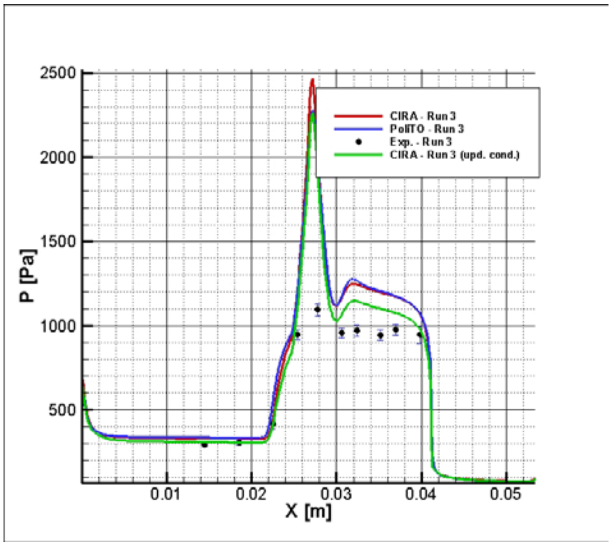

Figure 14: Double cone. Run 3. Effect of different freestream conditions

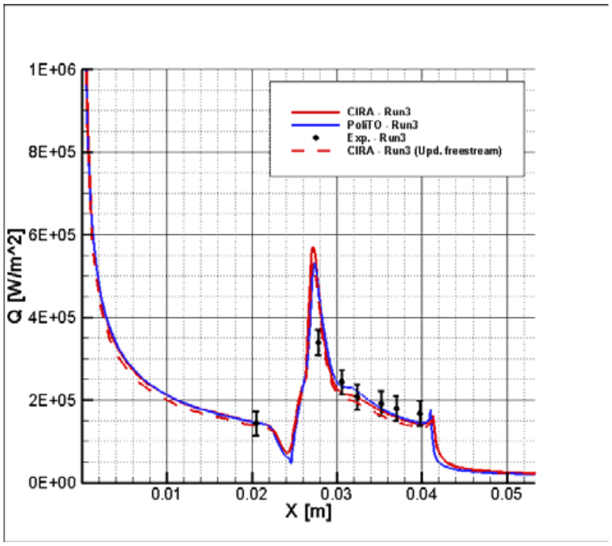

Figure 15: Double cone. Run 3. Effect of different freestream conditions on

5. FUTURE WORK

Several steps are foreseen in the future, in order to complete the current analysis; the most important are described hereinafter.

First of all, as soon as all the experimental data will be available, additional computations will be performed with different inlet conditions, in order to have a more extensive envelope of solutions as function of total pressure and enthalpy.

Furthermore, the new models developed in the frame of CAST project will be used in order to verify their influence on the solution; an improvement of the accuracy is expected, both for the nozzle and the double cone. The attention will be especially focused on the effect of different transport models in the nozzle simulation.

Another test that could be interesting to perform, based on the experience made in [7], is the use of DSMC simulation to confirm the possible effect of an incipient rarefaction.

Finally, the effect of non uniform freestream will be checked more in detail, performing at least one simulation of the complete geometry, including both nozzle and test chamber; however, some preliminary computations seem to show that this should not affect the separation length, and therefore the hypothesis of constant profile should be a good approximation for this test.

5. ACKNOWLEDGEMENTS

This work was performed within the CAST project, funded by the Italian Space Agency (ASI), aimed to develop an advanced numerical tool for the simulation of high enthalpy reacting flows and to the creation of an experimental database for the code validation. The experimental data were provided by Alta Spa.

6. REFERENCES

1. Wadhams T.P., Holden M.S., "Summary of experimental studies for code validation in the LENS facility and comparisons with recent NS and DSMC solutions for 2D-3D separated regions in hypervelocity flows", *42nd AIAA Aerospace Sciences Meeting and Exhibit*, 5-8 January 2004, Reno, AIAA 2004-0917.
2. P. Caredda et al. "Experimental Results on a Sharp Double Cone in a Mach 9 Non-equilibrium Flow", *Sixth European Symposium on Aerothermodynamics for Space Vehicles, Versailles* (France), 3-6 november 2008.
3. Marini M., Schettino A., "MSTP WORKSHOP 1996-Contributions to Hyperboloid Flare Test-Cases I.c and III.b", *Workshop on Reentry Aerothermodynamics and Ground to Flight Extrapolation*, ESA/ESTEC Noordwijk, 25-27 March 1996.
4. Borrelli S., Schettino A. "MSTP Workshop 1996 - Reentry Aerothermodynamics and Ground-to-Flight Extrapolation. A Contribution to TC 1.a

ONERA-F4 Nozzle Problem", *Workshop on Reentry Aerothermodynamics and Ground to Flight Extrapolation*, ESA/ESTEC Noordwijk, 25-27 March 1996.

5. D. Baccarella et al. "Characterization of a New Mach 9 Nozzle for the HEAT Hypersonic Wind Tunnel", *Sixth European Symposium on Aerothermodynamics for Space Vehicles, Versailles* (France), 3-6 november 2008.
6. Druguet M.C., Candler G.V, Nompelis I. "Effect of Numerics on Navier-Stokes Computations of Hypersonic Double-Cone Flows", *AIAA Journal*, Vol.43, No.3, March 2005.
7. Nompelis I., Candler G.V, Holden M.S. "Effect of Vibrational Non-equilibrium on Hypersonic Double-Cone Experiments", *AIAA Journal*, Vol.41, No.11, November 2003.
8. R. Votta, G. Ranuzzi, M. Di Clemente, A. Schettino, M. Marini "Evaluation of Local Effects of Transitional Knudsen Number on Shock Wave Bounday Layer Interactions", *Miami …*
9. C. Park, "A Review of Reaction Rates in High Temperature Air", *AIAA paper 89-1740*, June 1989.
10. R.C. Millikan, D.R. White "Systematic of Vibrational Relaxation", *The Journal of Chemical Physics*, Vol. 39 No.12, pp. 3209-3213
11. Park, C., Jaffe, R., and Partridge, H., "Chemical-Kinetic Parameters of Hyperbolic Earth Entry," *Journal of Thermophysics and Heat Transfer*, Vol. 15, No. 1, 2001, pp. 76–90.
12. Park, C., "Review of chemical-kinetic problems of future NASA missions - I: Earth entries," *Journal of Thermophysics and Heat Transfer*, Vol. 7, No. 3, 1993, pp. 385–398.
13. K.S. Yun, E. A. Mason, "Collision Integrals for the Transport Properties of Dissociating Air at High Temperatures", *The Physics of Fluids*, Vol. 39 No.12, pp. 3209-3213
14. Borrelli S., Pandolfi M., "An Upwind Formulation for the Numerical Prediction of Non Equilibrium Hypersonic Flows", *12th International Conference on Numerical Methods in Fluid Dynamics*, Oxford, United Kingdom, 1990
15. Kogan N. M., *Rarefied Gas Dynamics,* Plenum, New York, 1969
16. Pandolfi, M., "A Contribution to the Numerical Prediction of Unsteady Flows," AIAA Journal, Vol. 22, No. 5, 1983, pp. 602–610
17. Harten A., Engquist, B., Osher, S., and Chakravarthy, S., "Uniformly high order essentially non_oscillatory schemes, III", *Journal of Computational Physics*, vol. 71, 1987, pp. 231-303.
18. Gupta, R., Yos, J., Thompson, R., and Lee, K.-P., "A Review of Rates and Thermodynamic and Transport Properties for an 11-Species Air Model for Chemical and Thermal Non-equilibrium Calculations to 30000 K," NASA-RP-1232, Aug. 1990.