



24th COBEM - 2017



24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017- 2253

HIGH TEMPERATURE EFFECTS IN CONICAL NOZZLE FLOW

Bruno Coelho Lima

Sergio Nicolas Pachon Laiton

Instituto Tecnológico de Aeronáutica/ITA - Praça Marechal Eduardo Gomes, nº 50 Vila das Acácias CEP. 12.228-900 São José dos Campos, SP – Brasil

brunocoelho@ieav.cta.br

snpachonl@libertadores.edu.co

João Felipe de Araújo Martos

European Space Research and Technology Centre, European Space Agency, Noordwijk, Netherlands

joaodearaujo@hotmail.com

Paulo Gilberto de Paula Toro

Israel da Silveira Rêgo

Instituto de Estudos Avançados, Trevo Coronel Aviador José Alberto Albano do Amarante, no 1 Putim CEP. 12.228-001 São José dos Campos, SP, Brasil

toro@ieav.cta.br

israel.rego@ieav.cta.br

Abstract: *Comparison of theoretical-analytical analysis along with computational fluid dynamics (CFD) and experimental tests of high temperature effects on a flow over a conical nozzle are presented on this paper. For the theoretical-analytical analysis a code were developed in Python, for the CFD analysis ANSYS Fluent solver were used and experiments were performed in the Shock Tunnel T3 at IEAv. The main goal of this work is to study the effects of the high temperature flow on the nominal expected Mach number for a conical nozzle. Experiments of the, T3 shock tunnel were performed with a conical nozzle of Mach 8 nominal number, with expansion of 15 ° and output diameter of 600 mm. The shock tunnel were operated in low and medium enthalpy modes, this enthalpy difference was obtained by changing the working gas (Driver). For low enthalpy condition, effects of high temperature flow are negligible, since its total temperature were low, however, for the high enthalpy conditions effects of molecular vibration and some dissociation were presented. These effects were sufficient to change the thermodynamic properties of the fluid and therefore the flow. Several CFD simulations were performed in the Fluent starting with inviscid case and testing of the several models of turbulence available (as well as its variations) and corrections models of the thermodynamic properties present in the software were tested. A Rake with equally spaced pressure transducers positioned at nozzle exit was used to measure the obtained nozzle Mach number and compare with CFD and analytical results.*

Keywords: *conical nozzle, shock tunnel, high temperature effects.*

1. INTRODUCTION

Basically a shock tube is composed of two reservoirs: driver and driven, which consists of a high and low pressure sections, respectively (Figure 1). Between the Driver (high pressure reservoir) and the Driven (low pressure reservoir) there is a diaphragm (a physical membrane to maintain the pressure gradient). When suddenly ruptured, a moving shockwave called the Incident Shockwave (M_s) is formed (Lima, Toro, & Santos, 2013), moving towards the end of the Driven section, heating, accelerating and compressing the gas that was previously static in the Driven. Upon reaching the end of the Driven, another diaphragm installed before the nozzle reflects the incident shock wave, which in turn will generate an acceleration with the same intensity but in the opposite direction in the gas previously accelerated by the incident shock wave. This acceleration will stagnate again the gas in the Driven, and the passage of the reflected shock wave will heat again and recompress this gas, generating the stagnation condition (Nagamatsu, Geiger, & Sheer Jr, 1959).

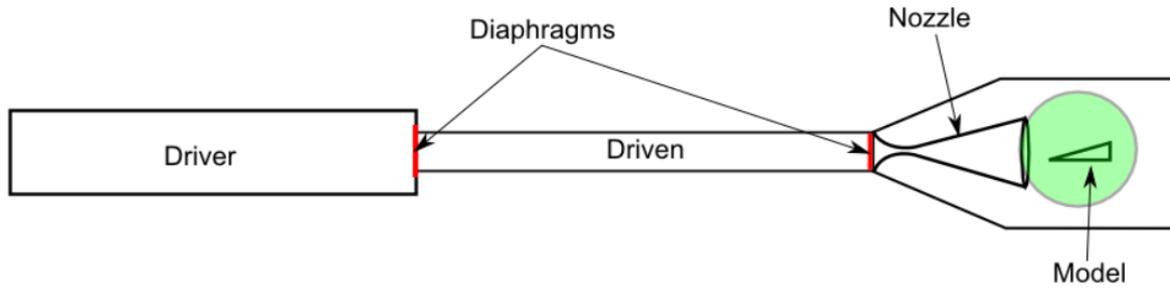


Figure 1 – Shock tunnel scheme

In order to generate hypersonic flow, shock tunnels use the thermodynamic energy of the stagnated flow in the reservoir region. This gas is at high pressures and temperatures sufficient to be expanded to the desired Mach number. The conical nozzle has two regions, convergent and divergent (figure 2), where the convergent region is used to accelerate the gases in subsonic regime while the divergent region to accelerate the gases in supersonic regime (Anderson Jr, 1985).

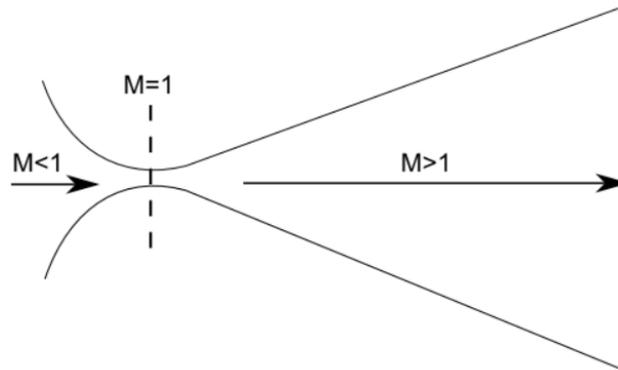


Figure 2 – Conical nozzle.

2. THEORETICAL FOUNDATION

2.1 Analytical solution

To perform the analytical solution, the flow was considered inviscid, adiabatic, reversible, calorically perfect and quasi-one-dimensional (Anderson Jr, 1985). Based on these simplifications the governing equations of a quasi-one-dimensional flow (continuity, momentum e energy) are obtained and used to obtain the area-Mach number relation.

$$\frac{A}{A^*} = \frac{1}{M^2} \left[\frac{2}{\gamma + 1} \left(1 + \frac{\gamma - 1}{2} M^2 \right) \right]^{(\gamma+1)/(\gamma-1)}$$

Pressure and temperature are given by

$$\frac{p}{p_0} = \left(1 + \frac{\gamma - 1}{2} M^2 \right)^{-\gamma/(\gamma-1)}$$

$$\frac{T}{T_0} = \left(1 + \frac{\gamma - 1}{2} M^2 \right)^{-1}$$

2.2 Experimental and computational procedures

To reduce computational costs of numerical simulations (CFD), the computational domain was simplified to a 2-D axisymmetric geometry. For this, a line of symmetry was generated on the axis of the nozzle (Figure 3). Setting the case as axisymmetric, the solver will use cylindrical coordinates.

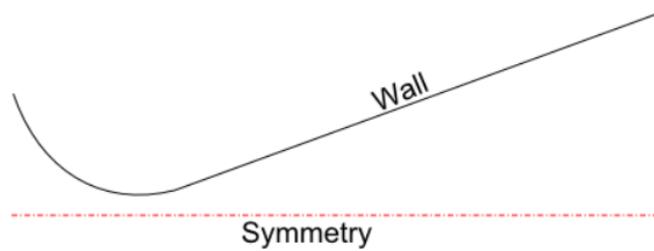


Figure 3 – Simplification of the computational domain

In numerical analysis Ansys's solver Fluent was used in stationary state, with the Implicit Roe's Averaged Method (Roe, 1981), second order in space and first order in time and finite volume method. For the viscous cases was used Reynolds Stress turbulence model, in the cases with high temperature effects Aungier-Redlich-Kwong Real Gas Model and Piecewise-Polynomial for the correction of C_p .

Several meshes were tested where the unstructured (triangular) mesh was better adapted due to the case geometry (Figure 4). The mesh convergence was obtained by gradually reducing its elements size to achieve differences lower than 3% between them. For viscous cases, it was necessary to refine the mesh in the region close to the walls, to adequately simulate the effects present in the boundary layer.

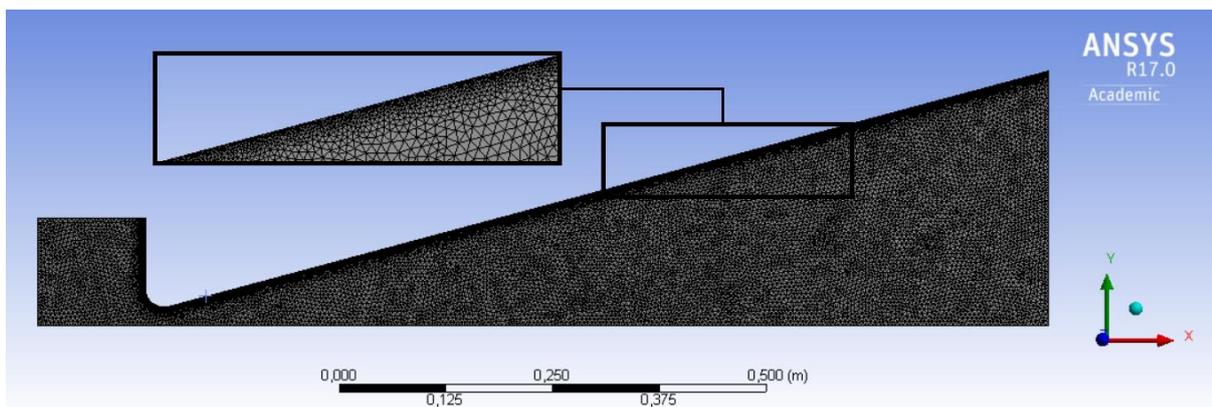


Figure 4 – Mesh for viscous cases

Changing between low and high enthalpy modes, it is necessary to change the gas used in the driver section. Using a lighter gas, with higher gas constant (R) faster incident shock waves are obtained, consequently, higher pressures and temperatures in the stagnation region (Ferri, 1961). Helium and air were used for the high and low enthalpy experiments, respectively. To obtain the pressure distribution at the outlet of the nozzle, equally spaced pressure transducers were distributed along the nozzle in a rake (Figure 5).

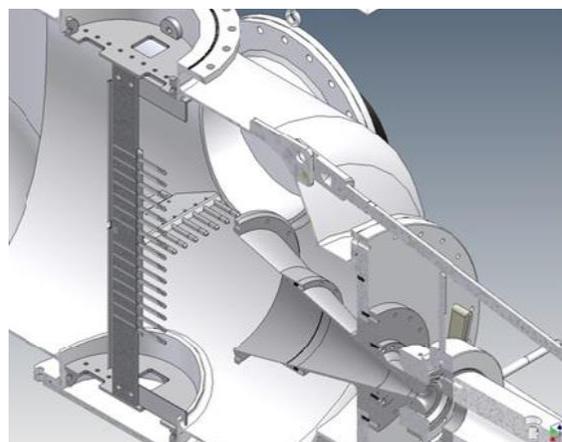


Figure 5 – Rake installed in Shock Tunnel T3.

Fonte: Laboratório de Aerodinâmica e Hipersônica Prof. Henry T. Nagamatsu [JM1].

In the experiments performed in the T3 shock tunnel, the pressure transducers are used as Pitot (Figure 6). With the impact pressure and enthalpy of stagnation it is possible to discover the Mach number in an experimental way (Pinto, 2016).

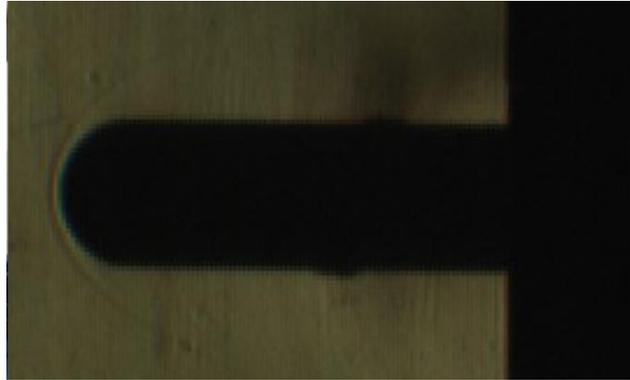


Figure 6 - Schlieren photography of a rake pitot

3. RESULTS AND DISCUSSION

The flow properties used in this study were taken from laboratory data (Pinto, 2016), in the low enthalpy case was used the stagnation pressure of 395 PSI and stagnation temperature 975 K. For high enthalpy case was found 1468 PSI and stagnation temperature 1932 K was used.

Table 1. Experimental and CFD results for conical nozzle at Mach 8 low enthalpy.

Property	CFD	Experimental
Mach	7.71	7.81
Velocity (m/s)	1345.28	1384.45
Temperature (K)	84.8	78.58
Pressure (Pa)	176.13	174.85

For low enthalpy case, the results were satisfactory, although the Mach number of the simulation is slightly below the one found in the experiment, this could have occurred due to a measurement error in the experiment or numerical or truncation error in the CFD simulation.

The simulated nozzle has nominal Mach number 8 and a Mach number less than 8 is expected due to the effects of viscous forces. In cases of high enthalpy, we also have high temperature effects. As possible to see that the real Mach number is considerably below the nominal Mach (ideal case) (Figure 7). This difference is caused by two reasons: viscous forces and high temperature effects.

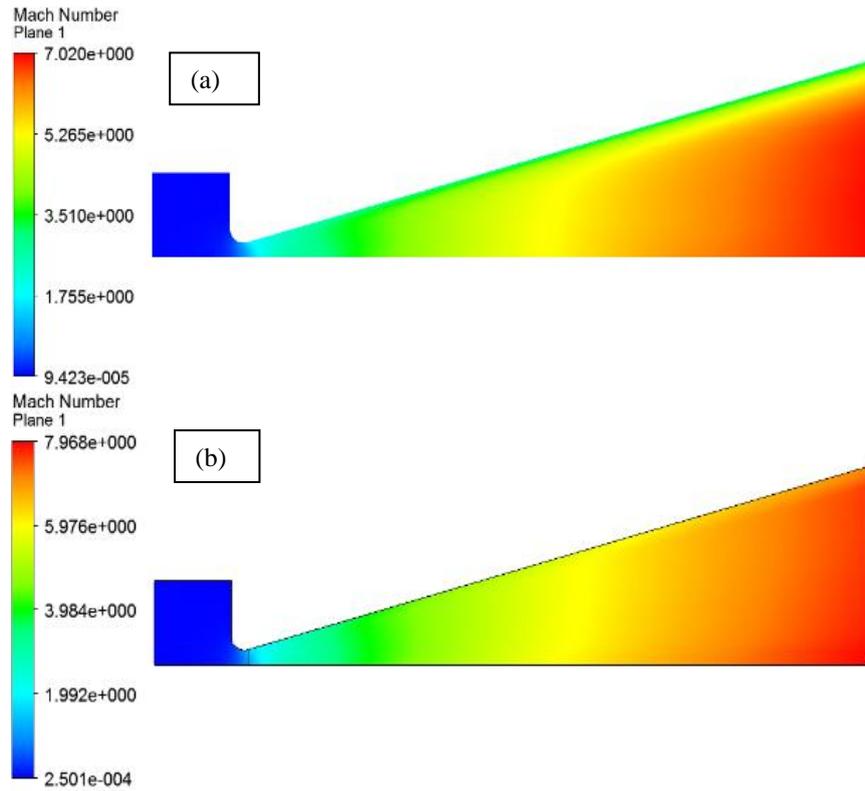


Figure 7- Contour of the Mach number showing the difference between nominal and actual Mach number, (a) Viscous and Real gas case; (b) Ideal gas case.

Effects of high temperature do not influence the flow during its whole course in the nozzle. This effect is more influent on the convergent section and at the beginning of the divergent section.(Nagamatsu et al., 1959).

The stagnation properties of the flow obtained at low enthalpy tests (pressure and temperature) were not high enough to cause dissociation of air molecules, the constant R of the gas was maintained throughout the experiment. However, C_p and C_v varied greatly in the convergent region of the nozzle, following the same temperature behavior (Figure 8 and 9).

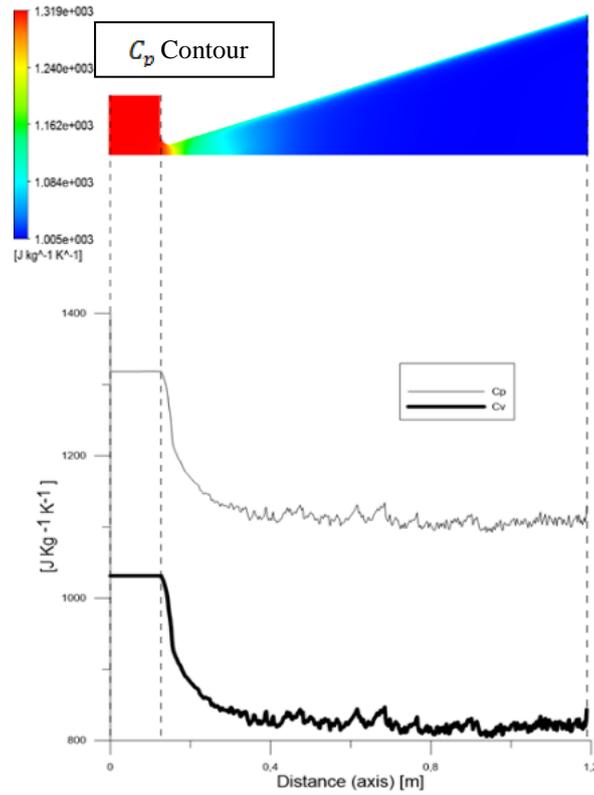


Figure 8 - C_p and C_v variation through the nozzle in Mach 8 high enthalpy simulation

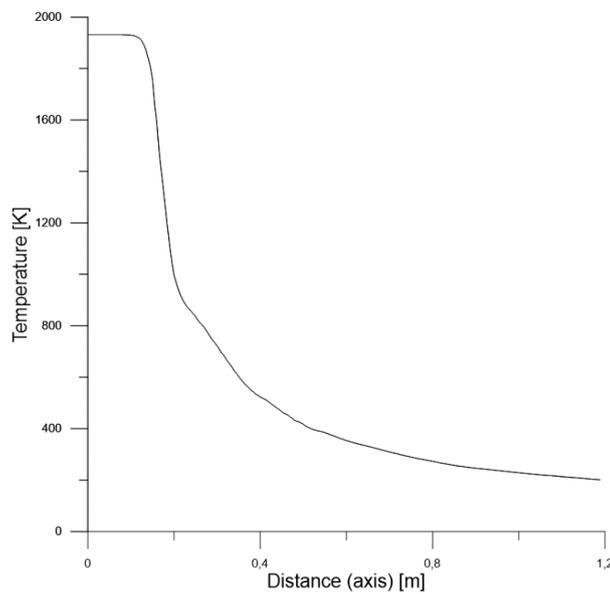


Figure 9 - Temperature behavior through the nozzle in Mach 8 high enthalpy simulation

Despite the correction of high temperature effects, the CFD results were still far from experimental. This happened because the viscous effects were not being considered correctly, adding roughness to the wall the results of the CFD approached. It is not part of the scope of this paper to study the viscous effects on the nozzle wall, in this way a mean roughness of a machined part that is 0.00005 m has been added.

Table 2. Experimental, analytical and CFD results for conical nozzle at Mach 8 high enthalpy.

Property	Analytical	Experimental	CFD – Real Gas	CFD – Real Gas Roughness Height
Mach	8	7.02	7.26	7.06
Velocity (m/s)	1897.4	1993.11	2004.12	1997.58
Temperature (K)	140	200.32	189.22	199.49
Pressure (Pa)	1041.11	1704.38	1345.13	1569.49

Analyzing the above table, we can see that the effects of high temperature have more impact than the viscous effects generated by the boundary layer. It is also possible to note that the analytical case presented discrepant values, due to the Mach number being considerably larger. In the experiment was found a flow with Mach number of 7.02, while the nominal Mach was 8, error about 12.25%. Considering the effects of high temperature, the error found was 3.3%. For the most complete case (with temperature and roughness effects), the approximation with the experiment was good, but the pressure found at the outlet of the nozzle had an error of about 8%.

4. CONCLUSIONS

The primary objective of this paper is to show the importance of considering effects of high temperature on conical nozzles flow in hypersonic shock tunnels. Analytical, numerical (CFD) and experimental results were presented.

As expected for the low enthalpy experiment, the error between the actual and nominal Mach was moderated because the temperature was below 1000 K. The difference may have been influenced by the viscous effects (eg boundary layer).

The effects of the high temperatures on the flow are clear when comparing the Mach number of the three cases: analytical (8), high experimental enthalpy (7.02) and low experimental enthalpy (7.81). Where the difference between the nominal Mach number (analytical case) and the real Mach number for the case with high enthalpy (hence high temperature) was 12.25%. While this difference for the case of low enthalpy was only 2.37%.

CFD proved to be versatile and reliable in discovering the real Mach number before each set of experiments, eliminating wasted experiments. It can also be used as a tool to aid in the development of new nozzles. It was also possible to observe that the viscous effects cannot be disregarded during the design of new devices

5. REFERENCES

- Anderson Jr, J. D., 1985. *Fundamentals of Aerodynamics. Fundamentals of Aerodynamics*. McGraw-Hill, Marilandy, 3rd edition.
- Ferri, A., 1961. *Fundamental Data Obtained from Shock-Tube Experiments*. Pergamon Press.
- Lima, B. C., Toro, P. G. P., & Santos, A. M. dos. (2013). Analytic Theoretical Analysis of the Incident and the. In *22nd International Congress of Mechanical Engineering*. Ribeirão Preto, SP.
- Nagamatsu, H. T., Geiger, R. E., & Sheer Jr, R. E., 1959. Hypersonic Shock Tunnel. *ARS Journal*.
- Pinto, D. R. (2016). *Caracterização do escoamento hipersônico na seção de teste do túnel de choque hipersônico t3 hipersônico na seção de teste do túnel de choque hipersônico T3*. Monograph, ETEP Faculdades, Brazil.
- Roe, P. L. (1981). Approximate Riemann Solvers, Parameter and Difference Schemes. *Journal Of Computational Physics*.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.