17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-684

RANS HYPERSONIC FLOW SIMULATIONS OVER REENTRY CAPSULES

Ruan Ramon P. P. Pereira
Rodrigo C. Palharini
João Luiz F. Azevedo
Instituto de Aeronáutica e Espaço, 12228-904 – São José dos Campos – SP – Brazil
passos.ruan@gmail.com
palharini.rc@gmail.com
joaoluiz.azevedo@gmail.com

Abstract. *The present work presents preliminary results obtained with a Reynolds-averaged Navier-Stokes computational fluid dynamics code, called LeMans, capable of solving weakly ionized hypersonic flow over reentry capsules. This code has been modified to increase its uses for other flow conditions and a continuous effort has been employed to verify and validate each feature that is added to it. First analysis was made to validate the results for cold gas flow, which brings plausible results with one-temperature model code. The obtained results have good agreement with validated CFD code hy2Foam. Next step is to study and validate other capabilities of the code such as two-temperature model and chemical reactions.*

Keywords: *Aerothermodynamics, Atmospheric Reentry, Reynolds Averaged Navier-Stokes.*

1. GENERAL INTRODUCTION

Space vehicles deal with extreme conditions during the atmospheric reentry phase. Severe aerothermodynamics loads are imposed to external surface of the spacecraft during the reentry phase, demanding reliable heat shields to protect the vehicle's surface from those extremely hostile conditions (Palharini et al, 2015).

In this scenario, a computational code has been used by Aerodynamics Division of the Instituto de Aeronáutica e Espaço: a CFD code based on a Reynolds-averaged Navier-Stokes (RANS) formulation called LeMans. This code was originally developed at the University of Michigan for the simulation of weakly ionized hypersonic flow-fields (Scalabrin, 2007), and later modified at IAE. With strong capabilities such as code parallelization and unstructured meshes, this CFD code solves the RANS equations using a finite volume formulation and treats inviscid fluxes using a modified Steger-Warming flux vector splitting (FVS) approach, which is less dissipative near the wall inside boundary layers, but with additional numerical dissipation close to shock waves.

The goal of this work is to use LeMans code to simulate flight reentry conditions of IAE's sub-orbital platform called SARA, acronym of Satélite de Reentrada Atmosférica. This space vehicle has been developed to perform experiments in microgravity environment. Relevant efforts has been made to determine surface parameters of SARA, especially in the descendent trajectory, passing through rarefied regions where continuum hypothesis is not valid and low atmosphere where CFD approach is more adequate (Bird, 1994). In altitudes higher than 80 km, a molecular dynamics approach can been used. IAE researchers are investing in a DSMC approach using dsmcFoam, an open source CFD toolbox developed by OpenFOAM, freely available under the GNU general public license. An important current trend is to comprise a CFD approach for low altitudes and a DSMC approach for high altitudes so that entire trajectory of a space vehicle can be studied.

2. THEORETICAL AND NUMERICAL FORMULATION

2.1 Mathematical Formulation

The RANS conservation equations for the three-dimensional system can be written as (Scalabrin, 2007),

$$\frac{\partial Q}{\partial t} + \frac{\partial (E - E_v)}{\partial x} + \frac{\partial (F - F_v)}{\partial y} + \frac{\partial (G - G_v)}{\partial z} = 0. \tag{1}$$

The vector Q of conserved variables, the inviscid flux vectors $E, F\ and\ G$and the viscous flux vectors $E_v, F_v\ and\ G_v$, are given by,

$$Q = \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ e \end{Bmatrix}, E = \begin{Bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uw \\ (e+p)u \end{Bmatrix}, F = \begin{Bmatrix} \rho v \\ \rho vu \\ \rho v^2 + p \\ \rho vw \\ (e+p)v \end{Bmatrix}, G = \begin{Bmatrix} \rho w \\ \rho wu \\ \rho wv \\ \rho w^2 + p \\ (e+p)w \end{Bmatrix}, E_v = \begin{Bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{xx}u + \tau_{xy}v + \tau_{xz}w - q_x \end{Bmatrix},$$

$$F_v = \begin{Bmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \tau_{yz} \\ \tau_{yx}u + \tau_{yy}v + \tau_{yz}w - q_y \end{Bmatrix} \; e G_v = \begin{Bmatrix} 0 \\ \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \\ \tau_{zx}u + \tau_{zy}v + \tau_{zz}w - q_z \end{Bmatrix}. \quad (2)$$

In these expressions, it has been used the standard gas-dynamics nomenclature for the variables: ρ for density, u, v and w for Cartesian velocity components in the x, y and z directions, p for pressure, e for the total energy per unit volume, τ_{ij} for the viscous stress components and q_i for the translation-rotational heat flux. Assuming Newtonian fluid with Stokes' hypothesis, the viscous stress components can be expressed by,

$$\tau_{ij} = \mu\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) + \delta_{ij}\lambda(\vec{\nabla}.\vec{u}), \quad \lambda = -\frac{2}{3}\mu, \quad (3)$$

where μ is the viscosity coefficient, i and j refer to Cartesian direction components x, y or z, and $u_x = u, u_y = v$ and $u_z = w$. The heat flux components are modeled by Fourier's law,

$$\vec{q} = q_x\hat{x} + q_y\hat{y} + q_z\hat{z} = -\kappa\vec{\nabla}T, \quad (4)$$

where κ is the thermal conductivity coefficient. The system closure is done by the equation of state for a perfect gas,

$$p = (\gamma - 1)(e - \frac{1}{2}\rho(u^2 + v^2), \quad (5)$$

where γ is ratio of specific heats.

2.2 Numerical Method

In order to solve numerically the set of differential equations in Eq. 1, a spatial integration using the finite-volume method is required, calculating fluxes across each cell faces. A Flux Vector Splitting (FVS) scheme is used to calculate the inviscid fluxes and a centered scheme to calculate viscous fluxes. An implicit method has been performed for the time discretization. The finite-volume method applied for a generic mesh is obtained by integrating the set of equation in each cell, given by

$$V_{cl}\frac{\partial Q}{\partial t} = -\sum_{j \in cl}\left(\vec{F}_j - \vec{F}_{v,j}\right) \cdot \vec{n}_j s_j, \quad (6)$$

where V_{cl} is the volume of the *cl*-th cell, $\vec{F}_j = E\hat{x} + F\hat{y} + G\hat{z}$ and $\vec{F}_{v,j} = E_v\hat{x} + F_v\hat{y} + G_v\hat{z}$ are, respectively, the inviscid and the viscous flux at *j*-th face of the *cl*-th cell, $\vec{n}_j$ is the normal vector to the *j*-th face pointing outwards of the *cl*-th cell and s_j is the area of the *j*-th face. By the homogeneous property of the inviscid flux vector, it can be rewritten as,

$$\vec{F} \cdot \vec{n} = F_n = \frac{dF_n}{dQ}Q = AQ, \quad (7)$$

where F_n is the normal flux at the *j*-th face and A is the Jacobian of the inviscid flux. The matrix A can be diagonalized by its eigenvectors' matrices R^{-1} and R and its eigenvalues diagonal matrix Λ, and separated into positive and negative components,

$$A = R^{-1}\Lambda R = R^{-1}(\Lambda_+ + \Lambda_-)R = R^{-1}\Lambda_+ R + R^{-1}\Lambda_- R = A_+ + A_-, \quad (8)$$

The diagonal eigenvalues matrix is separated in two different matrices, according to the signal of the eigenvalue,

$$\Lambda_+ = diag(\lambda^+), \qquad \Lambda_- = diag(\lambda^-),$$
$$\lambda^{\pm} = \frac{1}{2}\left(\lambda \pm \sqrt{\lambda^2 + \epsilon^2}\right), \quad (9)$$

where ϵ is a small number included to add numerical dissipation and correct sonic glitch phenomena. The Steger-Warming scheme uses the separation of fluxes into positive and negative parts, based on the signal of eigenvalues Λ_+ and Λ_-. Decomposing the fluxes on a downstream and on an upstream flux in relation to the face orientation,

$$\vec{F} \cdot \vec{n} = F_j^+ + F_j^- = (A_{cl}^+ Q_{cl} + A_{cr}^- Q_{cr}), \tag{10}$$

where *cl* and *cr* is the cell on the left and right sides of the face.

The original Steger-Warming scheme adds much dissipation and deteriorates boundary layer profiles [60]. LeMANS has implemented a modified Steger-Warming FVS scheme (MacCormack and Candler,1989), that uses average states for evaluation of separated Jacobians. This modification yields good boundary layer resolution, but it is necessary to switch back to the unmodified form of Steger-Warming scheme close to shock waves.

$$\begin{gathered} \vec{F} \cdot \vec{n} = F_j^+ + F_j^- = \left(A_{j+}^+ Q_{cl} + A_{j-}^- Q_{cr}\right), \\ A_{j+}^+ = A^+\left(Q_{j+}\right), A_{j-}^- = A^-\left(Q_{j-}\right), \\ Q_{j+} = (1-w)Q_{cl} + wQ_{cr} \\ Q_{j-} = wQ_{cl} + (1-w)Q_{cr} \\ w = \frac{1}{2}\frac{1}{(\alpha\nabla p)^2 + 1}, \nabla p = \frac{|p_{cl} - p_{cr}|}{min(p_{cl}, p_{cr})} \end{gathered} \tag{11}$$

where α is a parameter set to 6 for conventional problems or larger values for problems that require more dissipation and *w* is a pressure switch, that assumes the value of 0.5 changing to the modified Steger-Warming scheme in regions with small pressure gradient and approximately 0.0 close to shock waves changing to the original Steger-Warming scheme. The coefficient ϵ is chosen to guarantee no deterioration inside de boundary layer, Scalabrin (2007) modeled it as,

$$\epsilon_j = \begin{Bmatrix} 0.3\left(|u_j| + a_j\right) & , \ d_j > d_0 \\ 0.3\left(1 - |\vec{w}_j \cdot \vec{n}_j|\right)\left(|u_j| + a_j\right), & d_j \le d_0 \end{Bmatrix} \tag{12}$$

where d_j is the distance of *j-th* face to the closest wall and should be larger than the boundary layer thickness but smaller than the shock stand-off distance (Scalabrin, 2007), $\vec{w}_j$ is the normal vector of the closest wall, therefore the term $\left(1 - |\vec{w}_j \cdot \vec{n}_j|\right)$ turns to zero the value of ϵ at faces that are parallel to wall inside the boundary layer. All simulations have set d_0 equal to $5.0 \times 10^{-4}\ m$. Viscous terms are evaluated using properties at the cell centers and at the nodes, calculated using simple average of cells that shares that node. To solve the time integration of Eq. (6), LeMANS has the implementation of the Euler backward implicit method,

$$V_{cl}\frac{\Delta Q^n}{\Delta t} = \left(-\sum_{j \in cl}\left(F_{n,j} - F_{nv,j}\right)s_j\right)^{n+1} = R_{cl}^{n+1}, \tag{13}$$

where R_{cl}^{n+1} is the residue at the *cl*-th cell. Doing the linearization at time $n+1$ and using the FSV approach, it results in the set of equations below

$$V_{cl}\frac{\Delta Q}{\Delta t} = R_{cl}^n + \left(-\sum_{j \in cl}\left(\frac{\partial F_{n,j}^+}{\partial Q} + \frac{\partial F_{n,j}^-}{\partial Q} - \frac{\partial F_{nv,j}^+}{\partial Q} - \frac{\partial F_{nv,j}^-}{\partial Q}\right)\Delta Q s_j\right)^{n+1}, \tag{14}$$

where the terms derivative fluxes terms are the true Jacobians of the split fluxes. So, the final form of the system of equations to be solved by LeMANS is,

$$\begin{gathered} M_{cl}\Delta Q_{cl}^n + \sum_{j \in cl} N_j^- \Delta Q_{cr,j}^n = R_{cl}^n, \qquad N_j^- = \left(\frac{\partial F_{n,j}^-}{\partial Q}(Q_{j-}) - \frac{\partial F_{nv,j}^-}{\partial Q}(Q_j)\right)s_j, \\ N_j^+ = \left(\frac{\partial F_{n,j}^+}{\partial Q}(Q_{j+}) - \frac{\partial F_{nv,j}^+}{\partial Q}(Q_j)\right)s_j, \qquad M_{cl} = \frac{V_{cl}}{\Delta t} + \sum_{j \in cl} N_j^+. \end{gathered} \tag{15}$$

3. SIMULATION TEST CASE

In the verification and validation of computational codes for hypersonic aerothermodynamics, one of the most useful test cases is the cold flow over a 70° blunted cone. In order to validate the computational tools available at IAE, the computed results are compared with experimental results generated at the SR3 wind tunnel in Meudon, France (Moss et al, 1993 and Allegre et al, 1997). The experimental setup and geometry main parameters are shown in Fig. 1. Experimental data for the 70° blunted cone was obtained using freestream conditions shown in Tab. 1. Nitrogen gas flow at high Mach number and low Knudsen numbers were considered in the experiments.

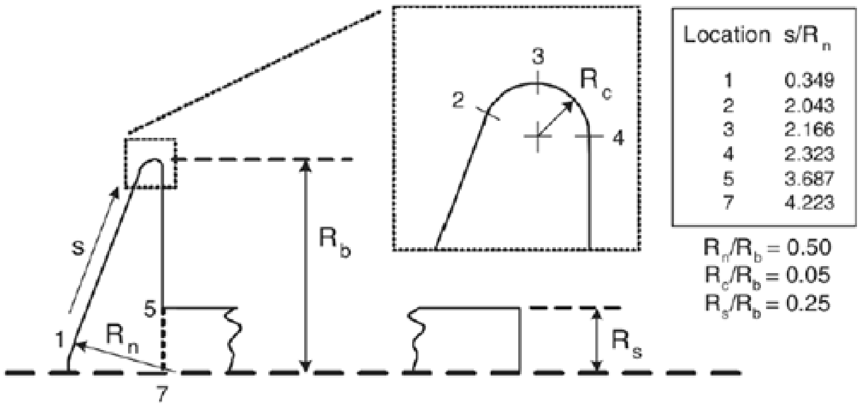

Figure 1. Dimension in millimeters of the 70° blunted cone geometry (Allegre et al, 1997).

The simulation was made using an axis-symmetric mesh with squared volumes in a structured-like grid (Fig. 2) in order to capture the most important phenomena through the shock wave. Preliminary simulations did not provide good enough results in the wake regions, thus, the domain was truncated at the probe shoulder at the corner of the probe. Since the flow is supersonic, any downstream perturbations do not reach flow characteristics upstream near the nose. The mesh is composed by 40000 squared volumes with a refinement such that to capture more accurately gradients of properties close to the wall.

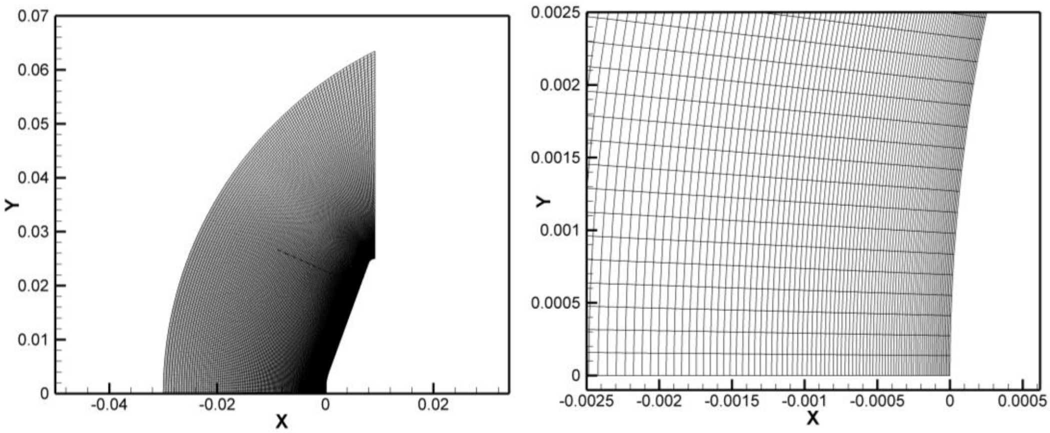

Figure 2. Details of the computational grid.

Table 1. Experimental flow condition for the planetary probe configurations (Moss et al, 1993)

$\boldsymbol{\rho}_\infty [kg/m^3]$	$\boldsymbol{V}_\infty [m/s]$	$\boldsymbol{T}_\infty [K]$	$\boldsymbol{M}_\infty$	$\boldsymbol{Kn}_\infty$	$\boldsymbol{Re}$
5.19×10^{-5}	1502	14.0	19.7	0.011	2221

4. RESULTS AND DISCUSSION

The properties along stagnation line is shown in Figure 3. It is possible to see that LeMans results have good agreement when compared with hy2Foam results. The position of shock wave for LeMans simulation is located upstream than the predicted for hy2Foam. It is plausible to attribute that difference to the model of redistribution of energy between modes. LeMans uses a simplified modeling with one single temperature, while hy2Foam uses a two-temperature model that combines translational and rotational modes in one variable T_{tr} and vibrational temperature T_v. Temperature distribution predicted by LeMans is higher than predicted by hy2Foam because of this lack of distribution of energy between other modes. Figure 4 shows Mach number and pressure and Figure 5 shows density and translational temperature distribution near the nose of the probe. Again, LeMans results have more compression than hy2Foam, because of the difference of treatment of exchange energy between different modes.

Future simulations can be made enabling the non-equilibrium model of LeMans, with a cost of higher computational time simulation, because one more equation is added of Eq. (2), relative with vibrational mode. The modeling of back part of the probe has a basic problem: continuum hypothesis fails in rarified regions with high Knudsen numbers. Viotti et al (2016) shows large high Knudsen regions in the wake of this probe, so that CFD based simulations does not have good results at the wake of the probe, while DSMC or hybrid methods show more agreement. However, for reentry applications, the main merit figures are related to the properties near the nose of a capsule, where Knudsen number is usually low due the high compression in this region, and CFD and DSMC results agree with experimental data.

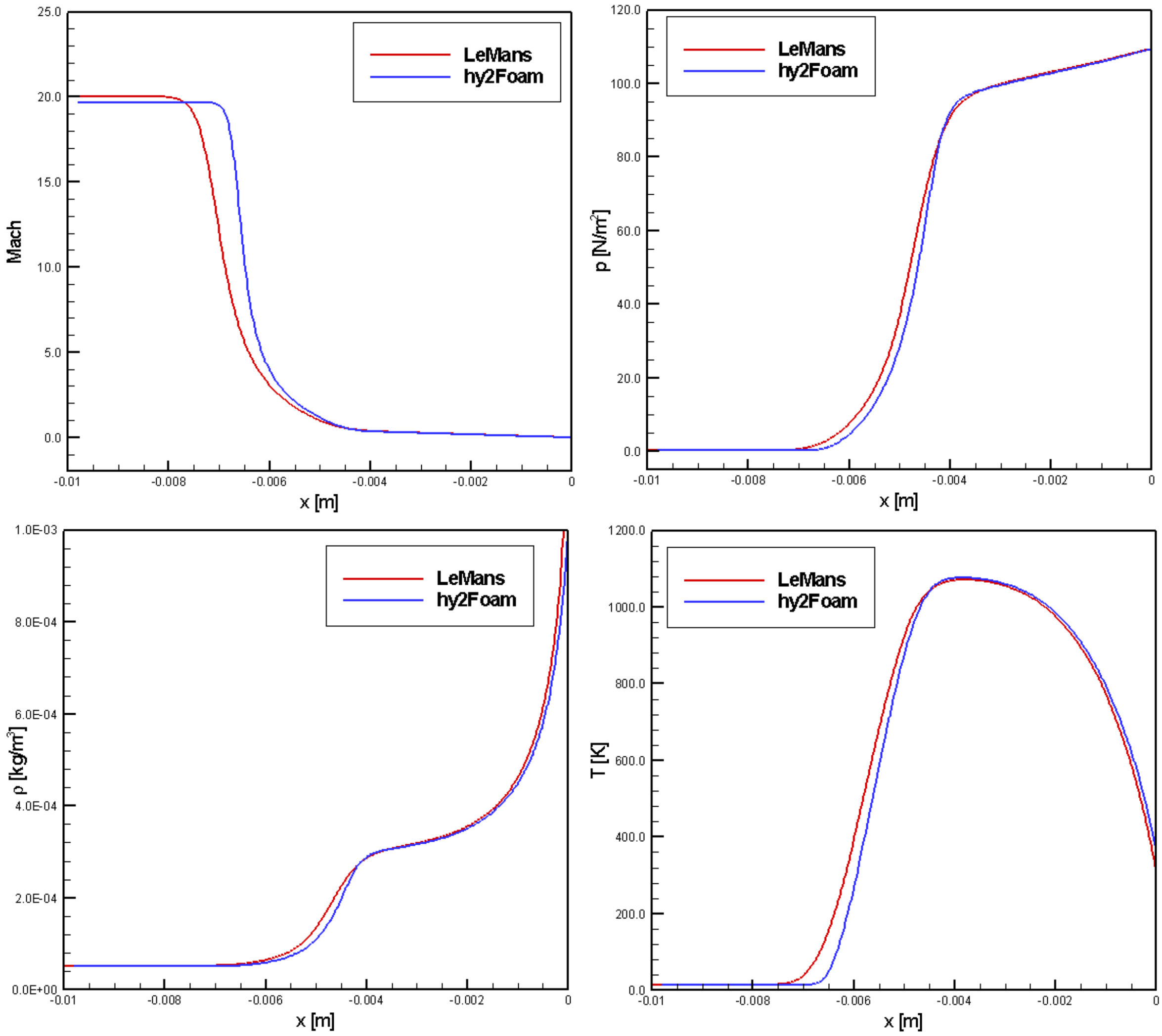

Figure 3. Simulation results along stagnation line.

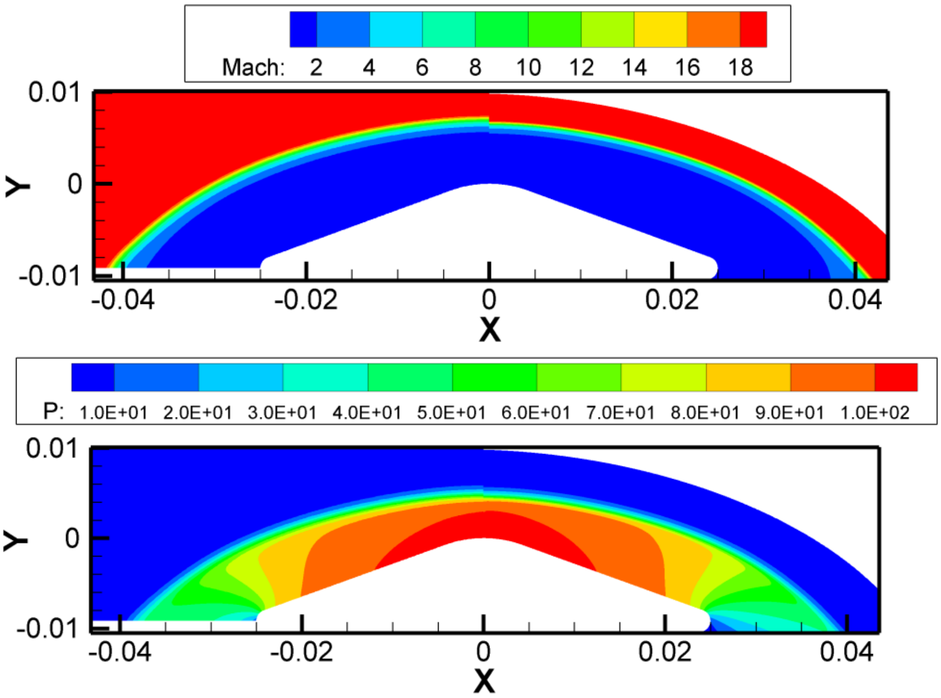

Figure 4. Simulation results for Mach number and pressure (in pascals) for LeMANS (left) and hy2Foam (right) simulations.

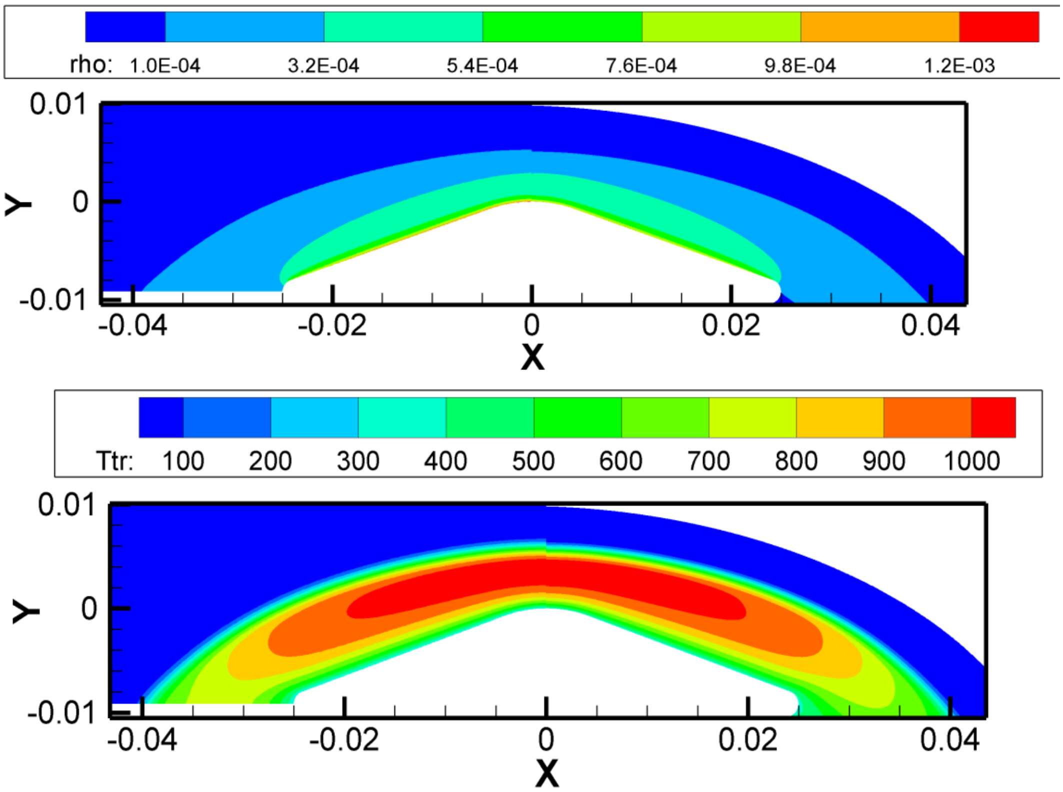

Figure 5. Simulation results for density (kilograms per cubic meters) and translational temperature (in kelvins) for LeMANS (left) and hy2Foam (right) simulations.

5. CONCLUSION

In this research project, the CFD method was used to simulate hypersonic flow over a 70° blunted cone. The main goal was to validate the LeMans code by comparing the results between this code and hy2Foam. The case simulated shows great agreement between two codes. This is expected because both modeling applied continuum hypothesis in a finite volume approach. However, it is important to mention that Case 1 of Moss (1993) cannot be simulated because of the breakdown of continuum hypothesis, a statistical approach DSMC based will lead better results. The slight differences observed can be attributed of distinct ways to handle temperature, specifically in terms of energy redistribution between rotational and vibrational modes. It is expected that a simulation of a gas with atmosphere freestream conditions lead not good enough results, because of the need of modeling the exchange of energy between translational, rotational and vibrational modes.

The next step is improve the capability of LeMans, validating its current method, and adding other features that solves more physics phenomena of the problem, like adding the influence of energy exchange modes with a two-temperature model and chemical reactions, including effects of ionization. Other idea is to make a tool that comprises a hypersonic continuum flow approach and a rarified molecular dynamics approach. IAE researchers have been studied a hybrid CFD-DSMC code using LeMans and dsmcFoam, with an adaptive switching, employing the DSMC code at the regions where local Knudsen number is higher than 0.1 and the CFD code for the other regions.

6. ACKNOWLEDGEMENTS

The authors gratefully acknowledge the support provided by CAPES via the PNPD program which allowed the development of this investigation. The authors also gratefully acknowledge the partial support for this research provided by Conselho Nacional de Desenvolvimento Científico e Tecnológico, CNPq, under the Research Grants No. 309985/2013-7, No. 400844/2014-1 and No. 443839/2014-0. The authors are also indebted to the partial financial support received from Fundação de Amparo à Pesquisa do Estado de São Paulo, FAPESP, under the Research Grants No. 2013/07375-0 and No. 2014/25438-1. The results were obtained using the hy2Foam code developed by the James Weir Fluids Laboratory based at the University of Strathclyde.

7. REFERENCES

Allegre, J., Bisch, D. and Lengrand, J. C, “Experimental Rarefied Heat Transfer at Hypersonic Conditions over 70-Degree Blunted Cone”, Journal of Spacecraft and Rockets, Vol. 34, No. 6, 1997.

Bird, G.A., Molecular Gas Dynamics and the Direct Simulation of Gas Flows, Oxford Science Publications, 1994.

MacCormack, R. W., and Candler, G. V. The solution of the Navier-Stokes equations using Gauss-Seidel line relaxation. Computers and Fluids 17 (1989), 135–150.

Moss, J.N., Dogra, V.K. and Wilmoth, R.G., “DSMC Simulations of Mach 20 Nitrogen Flows About a 70 deg. Blunted Cone and Its Wake” NASA Technical Memorandum, National Aeronautics and Space Administration, Hampton, Virginia, 1993. Harbor.

Palharini, R.C., White, C., Scanlon, T.J., Brown, R.E., Borg, M.K. and Reese, J.M., “Benchmark Numerical Simulations of Rarefied Non-reacting Gas Flows Using an Open-source DSMC Code”. Computers & Fluids, 2015.

Scalabrin, L.C., “Numerical Simulation of Weakly Ionized Hypersonic Flow Over Reentry Capsules”. PhD thesis, The University of Michigan, 2007.

8. RESPONSIBILITY NOTICE

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