

NUMERICAL PREDICTION OF COMPRESSIBLE HEAT FLOW WITH COMPLEX WALL TEMPERATURE IN SUPERSONIC ROCKET NOZZLES

by

Khaled BENSAYAH^{a,b*} and El-Ahcene MAHFOUDI^c

^a Interprofessional Complex Research in Aerothermochemistry (CORIA),
National Institute of Applied Science (INSA) of Rouen, Rouen, France

^b Laboratory of Mechanics, Department of Mechanical Engineering, University Teliidji Amar,
Laghouat, Algeria

^c Department of Mechanical Engineering, University Larbi Ben M'hidi, Oum El Bouagui, Algeria

Original scientific paper
<https://doi.org/10.2298/TSCI160616270B>

Wall heat transfer coefficients and static wall pressures are determined over wide ranges of stagnation pressures and stagnation temperatures under large pressure gradients in a cooled convergent-divergent nozzle. The effects of specific heat ratio, turbulent Prandtl number and wall temperature value on the heat transfer and on the position of separation flow are not yet discussed accurately. Computing correct boundary-layer under adverse pressures gradients is of a particular importance to the accurate modeling of separated flow. This numerical investigation is conducted to assess the accuracy of the SST-V turbulence model when computing boundary-layer separation in supersonic nozzle with heat transfer. It is concluded that the wall heat transfer coefficients and the position of separation point are influenced by the variation of many parameters as heat specific ratio, wall temperature, and turbulent Prandtl number.

Key words: *shock waves, compressible flow, turbulence, heat transfer*

Introduction

The structure of the rocket nozzle flow-field generated by the separation flow occurring by a strong over-expansion has attracted many experimental and numerical studies. The main features of the flow in over-expanded rocket are discussed. This numerical investigation was initiated to gain a better understanding of several parameter effects on the equilibrium flow through nozzles. Considerable effort has been devoted to predictions of gas-dynamics and convective heat transfer in supersonic nozzles [1], but comparatively little is known about real flow and convective heat transfer phenomena that can occur in nozzles of various configurations and at various operating conditions, the study originated from an interest in understanding convective heat transfer from accelerating turbulent boundary-layers, such are found over a large range of operating conditions in rocket engines. In this subject numerical heat transfer in conical supersonic nozzle is computed and compared with nozzle characterized by a truncated ideal contour (TIC) and nozzle based on optimized contour (TOC).

Complicating factors encountered therein which influenced heat transfer makes it extremely difficult to clarify the phenomena sufficiently to permit us to obtain results of general

* Corresponding author, e-mail: k.bensayah@yahoo.fr; k.bensayah@mail.lagh-univ.dz

validity. Numerical investigations of turbulent boundary-layer flow over cooled nozzle wall have usually been associated with supersonic flow, in which frictional heating effects become important and external cooling is sometimes necessary to maintain the integrity of the surface [2, 3]. The heat transfer from the hot gas to the wall should be as low that it is even not possible to decrease the amount of cooling. It is to be clarified by which parameters the compressible turbulent boundary-layer and the heat transfer through it is influenced. A few authors have been already worked on that problem [4-8], but there is no systematically investigation referring to whole phenomena and suitable methods are still missing. To full part of this gap we try in this study to investigate numerically the heat transfer associated with the turbulent flow of heated air through a converging-diverging nozzle. The aims of our project have been to clarify the effect of some important parameters, such as:

- the effect of wall temperature on the location of separation point, on the distribution of wall static pressure and on the rate of heat transfer,
- the effect of specific heat ratio on wall static pressure and heat transfer for adiabatic and isothermal wall, and
- the effect of variable turbulent Prandtl number on the heat transfer especially near the wall of the nozzle [8-10].

Flow separation in supersonic with cooled or uncooled convergent-divergent nozzles has been the subject of several numerical and experimentation studies in the past. With the renewed interest in supersonic flights and space vehicles, the subject has become increasingly important, especially for aerospace applications (missiles, supersonic aircrafts, rockets, *etc.*). One of the basic fluid dynamic phenomenon's that occurs at a certain pressure ratio of stagnation to ambient pressure in supersonic nozzles is flow separation, resulting of shock formation, and shock/turbulent-boundary-layer interaction inside the nozzle.

Several studies on supersonic nozzles [11-13] have shown that shock-wave/boundary-layer interaction occurring in highly over-expanded nozzles may exhibit strong unsteadiness that cause symmetrical or unsymmetrical flow separation. In rocket design community, shock-induced separation is considered undesirable because an asymmetry flow can yield dangerous lateral forces, which may damage the nozzle [14]. Such a situation is found in over-expanded supersonic nozzle where shock-induced flow separation occurs wherever a nozzle is operated at a shocked condition, this kind of problem was found with the comparison between TIC and TOC nozzles under the same operating conditions that a namely the free shock separation has been observed with TIC nozzle, in which the boundary-layer separates from the nozzle wall and never reattaches because the pressure difference across the nozzle is enough for complete expansion to the nozzle exit. The TOC nozzle presents a restricted shock separation characterised by a closed recirculation bubble, downstream of the separation point, with reattachment on the wall. Numerical schlieren picture is presented to clarify the major features occurring in these two nozzles.

Turbulence modeling

The $k-\omega$ SST model combines several desirable elements of existing two-equation models. The two major features of this model are a zonal weighting of model coefficients and a limitation on the growth of the eddy viscosity in rapidly strained flows. The zonal modeling uses Wilcox's $k-\omega$ model near solid walls and the standard $k-\epsilon$ model near boundary-layer edges and in free-shear layers. This switching is achieved with a blending function of the model coefficients. The SST modeling also modifies the eddy viscosity by forcing the turbulent shear stress to be bounded by a constant times the turbulent kinetic energy inside boundary-layers. This modification improves the prediction of flows with strong adverse pressure gradients and separation.

The k - ω SST-V model uses two additional transport equations to describe the turbulence as summarized:

$$\frac{D}{Dt}(\rho k) = P_k - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (1)$$

$$\frac{D}{Dt}(\rho \omega) = \frac{\gamma \rho}{\mu_t} P_k - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_{\omega 1} \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2 \rho (1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2)$$

$$P_k = -\tau_{ij} \frac{\partial u_i}{\partial x_j} \quad (3)$$

$$\tau_{ij} = -2 \mu_t \left(S_{ij} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) + \frac{2}{3} \rho k \delta_{ij} \quad (4)$$

where β is a turbulence model constant defined as $\beta = 0.075 \cdot F_1 + 0.0828 \cdot (1 - F_1)$, σ – the turbulence model blending constant defined as $\sigma_k = 0.85 \cdot F_1 + (1 - F_1)$, τ_{ij} – the turbulent stress tensor, and S – the dimensionless strain.

A common variant of the SST model is termed SST-V, which the production term makes use of the local magnitude of vorticity Ω [15, 16]:

$$P_k = \mu_t \Omega^2 - \frac{2}{3} \rho k \delta_{ij} \frac{\partial u_i}{\partial x_j} \quad (5)$$

This vorticity source term is often a good approximation and close to the exact source term in boundary-layer [17].

The function F_1 is designed to blend the model coefficients of the original k - ω model in boundary-layer zones with the transformed k - ε model in free shear layer and free stream zones. This function is expressed in term of local variables:

$$F_1 = \tanh \left(\left\{ \min \left[\max \left(\frac{\sqrt{k}}{0.09 \omega y}, \frac{500 \mu}{\rho y^2 \omega}, \frac{4 \rho \sigma_{\omega 2} k}{CD_{k\omega} y^2} \right) \right] \right\}^4 \right) \quad (6)$$

where $CD_{k\omega}$ is a cross diffusion term added in eq. (2). According to Bradshaw's assumption the eddy viscosity is defined in the following way:

$$\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, |\Omega| F_2)} \quad (7)$$

where F_2 is a function that is one for the boundary-layer flows and zero for the free shear layers, with

$$|\Omega| = \sqrt{2 \Omega_{ij} \Omega_{ij}}, \quad \Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right), \quad F_2 = \tanh(arg_2^2), \quad arg_2 = \max \left(\frac{2 \sqrt{k}}{0.09 \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right)$$

Realizability condition in turbulence models

The two-equation turbulence models are based on the Boussinesq assumptions where the Reynolds stresses is expressed as a linear function of the mean strain tensor:

$$-\rho \widetilde{u''_i u''_j} = 2\mu_t \left(S_{ij} - \frac{1}{3} S_{kk} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (8)$$

where $C_\mu = 0.09$. As shown by Moore and Moore [18] these equations can give negative values of the normal stress S_{kk} too large. Bradshaw has noticed that in 2-D boundary-layers submitted to a strong pressure gradient the shear stress was approximately proportional to the turbulent kinetic energy with:

$$-\widetilde{u''v''} \approx \sqrt{C_\mu k} \quad (9)$$

These two remarks led to introduction of weakly non-linear turbulence models in which the factor is allowed to vary according to:

$$C_\mu = \min \left(0.09, \frac{1}{A_0 + A_S (S^a + A_\Omega \Omega^a)^{1/a}} \right) \quad (10)$$

with

$$\left\{ \begin{array}{l} S = \frac{1}{\omega \beta^*} \sqrt{2S_{ij}S_{ij} - \frac{2}{3} S_{kk}^2}, \quad S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \\ \Omega = \frac{1}{\omega \beta^*} \sqrt{2\Omega_{ij}\Omega_{ij}}, \quad \Omega_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right) \end{array} \right. \quad (11)$$

and $A_0 = 0$, $A_S = 3.23$, $A_\Omega = 0$, $a = 2$ in this case the Bradshaw coefficient (0.31) is substituted by C_μ^{V2} in the formulation of the eddy viscosity.

Numerical method

Navier-Stokes equations are solved on a computational domain of variables ζ and η (transformed co-ordinates of the physical domain), by the use of finite volumes predictor-corrector. The new system of equations is solved by using MacCormack's explicit-implicit scheme [19]. This algorithm is second-order accurate in space and time. The basic discretization for the convective fluxes is modified to account for the physical properties of information propagation, as done initially by Steger and Warming [20]. The flux splitting is made second order accurate, but in shock regions where it is lowered to first order. The viscous terms are centered and the axisymmetric source terms are integrated at the center of each control volume in both the ζ and η directional sweeps. To reach a steady-state solution with a minimum number of iterations, the explicit discretization is complemented with an implicit numerical approximation which is free from stability conditions. Thus, the block-pentadiagonal system is solved by generalized Thomas algorithm with *LU* decomposition in the η -direction, and by a line Gauss-Seidel relaxation technique in the ζ -direction. As the system is really diagonally dominant and the method is iterative, converged steady solution can be obtained in very limited number of time steps, each time step including a double sweep (backward-forward) in the flow direction. With this technique, unbounded time step values can be used. Numerical simulations have been made with CFL numbers greater than 10 [21].

We note that we performed a grid-convergence using different grids, the grid points are clustered in regions of high gradients, to ensure mesh-independent solution, all grids had a $y_1^+ = u_\tau d_w / \nu \cong 1$ (where u_τ is the friction velocity and d_w – the distance to the closet wall) for the first point. For example, with bell shaped nozzles, the grids ranged from: $150 \times 80 \times 2$, $300 \times 120 \times 2$, and $400 \times 140 \times 2$, the results from the two last grids were identical establishing

grid independence and had a $y^+ < 1$ for the first point. We note here that conical and bell shaped nozzles are not presented with the same grid.

Results and discussion

Validation of SST-V turbulence model

To validate the turbulence model for strongly favorable pressure gradient, we select experimental data provided by Cuffel *et al.* [5], the conical nozzle of fig. 2(a) used in this computation had half-angles of convergence and divergence of 45° and 15° , respectively, with $Ru = r_c/r_{th} = 0.625$. The static pressure in the flow field was computed in different region, the computation is conducted with air at a stagnation pressure of 4.82 bar and a stagnation temperature of 300 K. Important parameter of isentropic flow field in the transonic region of nozzle with circular arc throat is the ratio of radius arc and the radius of the throat Ru . For values of Ru considerably greater than unity, *i. e.*, the flow is nearly 1-D with the gradual throat contour. With the decrease of the ratio Ru 2-D flow effects become important. Provided that Ru is not less than about 2 (nozzle of fig. 1), the computation and existing 2-D flow theories [22-24] adequately predict the transonic flow field as indicated by the wall static pressure calculation and measurements [4]. However, for nozzles with tighter throats ($Ru < 1$), such as are found in some rocket engines, these theories do not apply. This configuration presents some internal flow in the transonic region of a nozzle with a small ratio of $Ru = 0.625$.

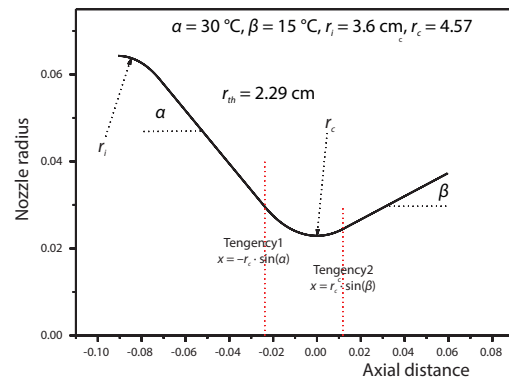


Figure 1. Nozzle geometry

Axial distributions of internal static pressure computations and measurements of Cuffel *et al.* [5] are shown at various radial locations in fig. 2(b). The radial static pressure variation is large, with values of the static pressure along the centerline as much as three times those at the wall. In fact, radial pressure variation is so large that near the wall at the throat, the radial static-pressure gradient is about the same as the axial static pressure gradient. The gas expands more rapidly along the wall than along the centerline, and there are correspondingly large radial

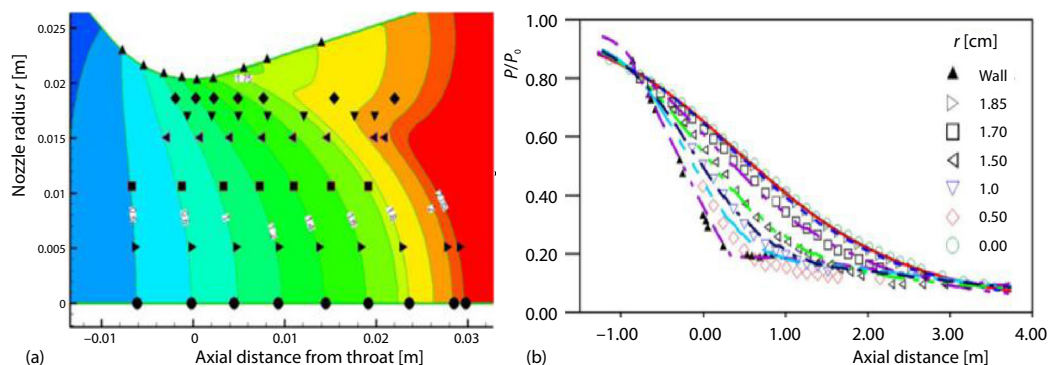


Figure 2. (a) Contour of Mach number in the transonic region (symbols represent experimental data), (b) comparison of measured and calculated static pressure in the transonic region
(for color image see journal web site)

variations in the Mach number, fig. 2(a), calculated for isentropic flow ($\gamma = 1.4$). In the inviscid flow field in the throat plane, the Mach number is 0.8 at the axis and 1.4 at the edge of the boundary-layer. Computed static-pressure rise along the wall just downstream of the tangency between the circular-arc throat and the conical divergent section shown in fig. 2 is believed to be associated with a compressive turning of the flow.

The persistence of the strong angular motion acquired by the flow in the small radius-of-curvature throat region can lead to an overturning of the flow so that streamlines near the wall become inclined to the downstream conical wall, as indicated by calculations.

Table 1. Different test cases

Case	Stagnation pressure	Stagnation temperature
Case 1	$P_0 = 5.18$ bars	$T_0 = 843.33$ K
Case 2	$P_0 = 17.49$ bars	$T_0 = 572.22$ K
Case 3	$P_0 = 3.08$ bars	$T_0 = 835$ K
Case 4	$P_0 = 5.18$ bars	$T_0 = 1105$ K
Case 5	$P_0 = 17.51$ bars	$T_0 = 840.55$ K

Table 2. Physical properties of different gases

Gas	γ	R [$\text{m}^2\text{s}^{-2}\text{K}^{-1}$]	C_v [$\text{Jkg}^{-1}\text{K}^{-1}$]	n
Air	1.4	287	717.5	0.67
Cl_2	1.34	117	344.1	1
CH_4	1.32	518	1618.7	0.87
N_2O	1.31	189	609.6	0.89

Effect of specific heat ratio on the distribution of wall static pressure

In order to analyze the effect of γ on the position of separation, in fact it is difficult to take into account only the effect of the heat specific ratio, because many variables involved, two gases with the same γ may have different specific heats, molar masses, and different chemistries. To approximate the effect of γ , we made our calculations with cold temperature $T_0 = 300$ K to avoid the dependence of specific heats on temperature. The tab. 2 summarizes the different parameters of gases selected.

In fig. 3(a), the position of separation point indicated by the report $A_{\text{sep}}/A_{\text{th}}$ according to the ratio of stagnation pressure to outlet nozzle pressure P_0/P_a . To prevent the effect of the viscosity power law is used for each gas:

$$\frac{\mu}{\mu_0} = \left(\frac{T}{T_0} \right)^n \quad (12)$$

It is clear from the results shown to observe the combined effect of and specific heats, and make a meaningful comparison, one can see the difference between air and N_2O because these two gases have similar specific heats. One observes that, when γ decreases, the separation point moves downstream with a slightly increase for the separated pressure and this observation is more pronounced with increasing stagnation pressure, P_0 . These results confirm dependency

The region where Mach number lines begin to converge, which is the onset of shock formation associated with the compressive turning of the flow to become parallel to the wall, is evident in fig. 2(a) from the shape of the Mach number contours. The weak oblique shock wave forms subsequently and extends downstream, intersecting the centerline at $x = 8$ cm. It is reasonable to assume that the flow is isentropic throughout this region.

In this work to study the effect of some important parameter on the wall static pressure and the wall heat transfer, such as the specific heat ratio, the wall temperature and the turbulent Prandtl number (tab. 1).

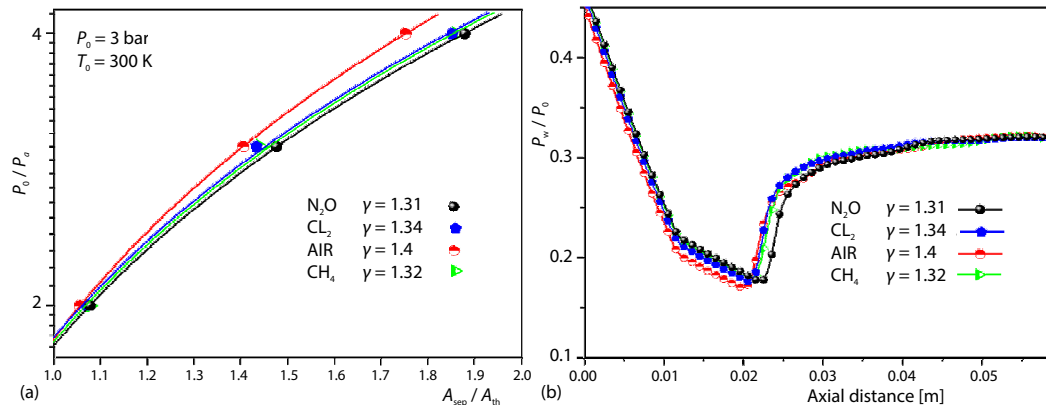


Figure 3. (a) Effect of γ on the location of separation for different gases for nozzle of fig. 1, (b) effect of γ on the distribution of wall static to stagnation pressure for different gases for nozzle of fig. 1
(for color image see journal web site)

of separation on γ as noted in [25]. Figure 3(b) shows the wall static pressure that increases with the decrease of the specific heat ratios γ .

Effect of wall temperature on wall static pressure ratio

The results of static-to-stagnation pressure ratios along the nozzle are shown in fig. 4 for a range of different parameters (stagnation temperature, stagnation pressure, specific heat ratio, and wall temperature). Figure 4 shows the computed pressures ratios, P/P_0 , along the wall of the nozzle. The SST-V turbulence model provides a good agreement with the experimental data. Figure 4 shows an enlargement in the regions of separation point in order to clarify the effect of γ and T_w on the behavior of the pressure ratio level and the position of separation point.

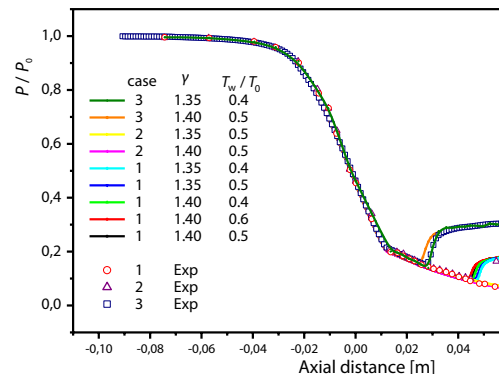


Figure 4. Wall static pressure distribution
(for color image see journal web site)

The three values ($T_w/T_0 = 40\%$, 50% , and 60%) are selected corresponding to measures of Back *et al.* [4] and others who have demonstrated a variable profile of the wall temperature. In most cases found in the literature, this rate ranged from 0.3 to about 0.7.

Over the wall temperature is high. The separation point is upstream relative to a lower temperature, and away from the experimental position of separation point, and its observed that a constant wall temperature of about 40-50% of present the better approximation, because experimentally it is found that the ratio T_w/T_0 for the three case 1, 2, and 3 is about 0.42 in the supersonic flow near the separation region.

The influence of wall temperature on the variation of separation pressure ratio is not clearly highlighted according to the various results of the literature. However, that the coldest walls give a larger wall Mach number at initial point of separation and thus a less wide separation. This tendency is in agreement with other experimental results [26], and in contrast with [27], who has observed no effect of the wall temperature. Even cryogenically cooled nozzles

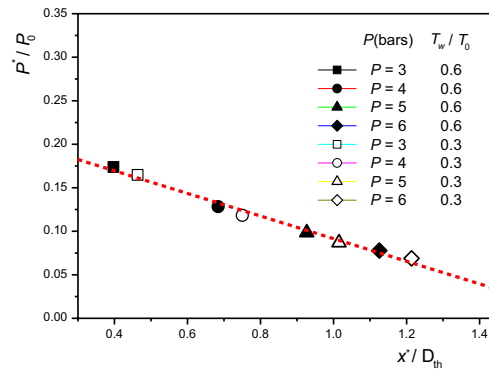


Figure 5. Pressure at the initial point of the interaction region

(for color image see journal see web site)

It is well observed that the location x^* of the pressure increase with cold wall. Quasi-similarity of the distribution of P^* is well observed, this distribution can be described by the following relation:

$$\frac{P^*}{P_0} = A \left(\frac{x^*}{D_{th}} \right) + B \quad (13)$$

It seems that this line had a slope of approximately ($A = -0.13$), this linear distribution is well marked, more P_0 increases, the difference between the two successive axial positions decreases, we can see that the position x^* do not behave exactly linearly with the stagnation pressure, *i. e.* if in fact we increase the pressure of 1 bar the position of the separation should not follow the linear expression (13) so we cannot say that this relation releases a consistent similarity.

The same tendency of fig. 5 can be obtained, when the stagnation pressure P_0 increases, the position of separation increases. At the same stagnation pressure the position of separation increase with the decrease of the wall temperature, T_w . The slope of about (-0.17) is observed for both temperature of wall.

Wall heat transfer coefficient

The wall heat transfer coefficient was computed by:

$$h = \frac{q_w}{T_{aw} - T_w} \quad (14)$$

The adiabatic wall temperature, T_{aw} , is the wall temperature for zero heat flux and is related to Mach number by [2]:

$$\frac{T_{aw}}{T_0} = \frac{1 + r \frac{\gamma - 1}{2} M^2}{1 + \frac{\gamma - 1}{2} M^2} \quad (15)$$

The adiabatic temperature was calculated by taking the recovery factor 0.89 and with a calculated recovery in adiabatic wall nozzle as shown in fig. 6(a).

The recovery factor, r , is known to be depending on many parameters [3]. In the subsonic part of the nozzle the recovery factor slightly decreases and displays a minimum just

deviate slightly from the uncooled and normally cooled walls. The result of reference [27] seems to be in contrast to theoretic considerations of the wall temperature effect, since a cooler wall is normally believed to lead to a lower separation pressure ratio. We can noted that the results with $\gamma = 1.35$ are clearly better, this value is in agreement with the value proposed by Vieser *et al.* [28] and Tong *et al.* [29].

Figure 5 present the pressure ratio, P^*/P_0 , vs. the dimensionless axial locations of P^* at different temperature of wall, P^* is the pressure of initial point of compression region, which present the first deviation from the vacuum pressure profile, at this pressure the flow has not yet separated.

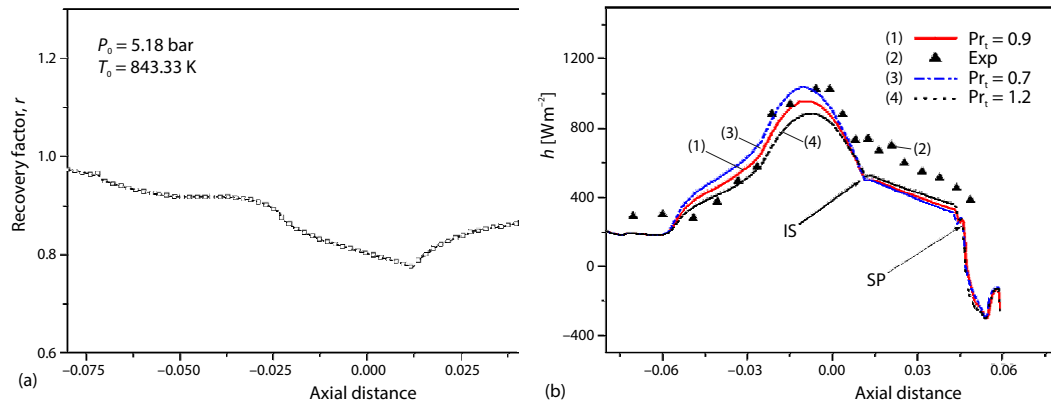


Figure 6. (a) Distribution of the recovery factor along the nozzle, (b) heat transfer coefficient with different turbulent Prandtl number (IS – internal shock, SP – separation point)

after the throat, this position correspondent to the position of the internal shock, and increasing rapidly to average value which is in line with the well-known correlation.

The fig. 6(b) shows the axial evolution of the heat transfer coefficients, one can see clearly that the turbulent Prandtl number had a significant effect on the heat transfer, especially on the subsonic region, can be observed with a Pr_t equal to 0.7 the maximum coefficient of heat transfer is better achieved and this is consistent with the numerical study of Xiao *et al.* [8] and of supersonic flow over a flat plate made by Sommer *et al.* [9, 10] which reached a Pr_t close to 0.5 on the wall and increases to 1.6 and then an asymptotic value of about 0.9. The turbulent Prandtl number is seen to vary rapidly in the region very near the wall. It increases from a wall value of about 0.5 to a maximum of approximately 1.6 and then decreases to about 0.75 before settling back to a value of 0.9 at $y^+ = 200$. Thereafter, Pr_t remains fairly constant at 0.9.

As shown in the eq. (14) the heat transfer coefficient is related with three parameters such as the adiabatic wall temperature, T_{aw} , the wall temperature, T_w , and the wall heat flux, q_w . The fig. 7(a) shows that the calculated adiabatic wall temperature against the experiment of Back *et al.* [4] is in excellent agreement and this result explains that the difference found in the prediction of the heat transfer coefficient comes from the wall temperature distribution and the calculated wall heat flux. In fact for this calculated Case 1 the measured wall temperature varies

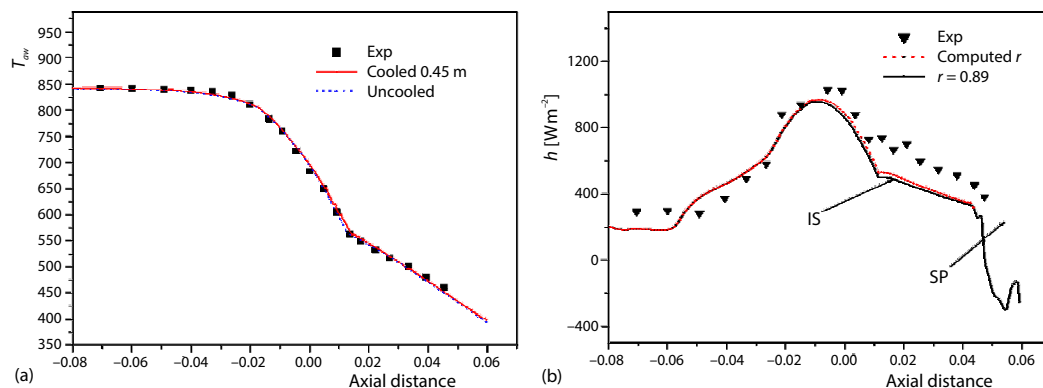


Figure 7. (a) Adiabatic temperature comparisons, (b) heat transfer coefficient with constant and variable recovery number, r (IS – internal shock, SP – separation point) (for color image see journal web site)

over a range that exceeds 100 K, particularly in the critical throat region of the nozzle. As found by Tong and Luke [29] that the use of experimental wall temperature lowers the peak computed heat transfer coefficient by about 30%, quantifying the effect of using detailed experimental wall temperature compared to prescribing a fixed temperature.

Figure 7(b) shows the heat transfer coefficient along the nozzle for Case 1 with calculated and prescribed recovery factor, the heat transfer coefficient is a maximum at the location upstream of the throat, in fact the peak heat transfer coefficient is located upstream of where the mass flux is largest, a good agreement is found in the subsonic region, and the difference become important downstream the internal shock wave, we can note here that this difference in computing wall heat transfer come from: firstly from the wall temperature because in the supersonic region the wall temperature is lower than the fixed temperature and secondly from the calculation of the wall heat flux. That mean the combination of many parameter such as the specific heat ratio, this parameter is not constant in the flow field. The accuracy of turbulence model in calculating the turbulent boundary-layer growth downstream the throat in the supersonic region, this one is strongly influenced by the internal shock wave for this case and in general by all kind of shock waves interactions, that caused the boundary-layer to become thinner or thicker to depend on the operating conditions and the configuration of the nozzle. The internal shock shown in fig. 7(b) downstream the throat caused by the rise of pressure, this rise is not clear in fig. 7, the schlieren picture and the derivative wall static pressure can show the increase of pressure (figure not presented here).

Figure 8(a) shows the effect of wall temperature and heat specific ratio on the rate of heat transfer along the wall of the nozzle. We can see clearly that the heat transfer is influenced by the wall temperature specially in the throat region. The wall heat transfer increase with the decrease of wall temperature. This tendency is the result of the fact that the heat transfer is proportional to the difference of temperature between the wall temperature and the temperature of flow field.

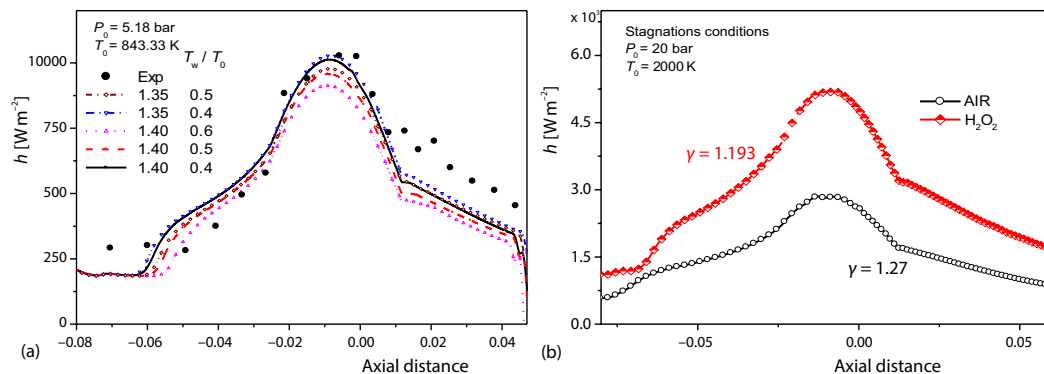


Figure 8. (a) Effect of wall temperature and specific heat ratio on the distribution of heat transfer along the nozzle of fig. 1, (b) effect of specific heat ratio on the rate of heat transfer (for color image see journal web site)

The effect of specific heat ratio is better presented in fig. 8(b), (for air with $\gamma = 1.27$ and H_2O_2 with $\gamma = 1.19$). The rate of heat transfer increase with the decrease heat ratio. This result indicate that there is a combined effect of specific heat ratio and the specific heat C_p or C_v . In fact, when the pressure at the vicinity of wall increase and both temperature and density decrease, the rate of heat transfer is proportional to the specific heat C_p or C_v , and γ . So, the rate of heat transfer is dependent on the properties of the gas used.

In figure 9(a) the effect of the stagnation pressure is presented, the heat transfer coefficient increase with increasing stagnation pressures as a result of a large mass fluxes, the variation with stagnation temperatures is less clear at low stagnation pressure, and the heat transfer coefficient decrease with an increase of stagnation temperature.

The effect of divergence profile between TOC and TIC nozzles on heat transfer has been presented in fig. 10(a). The geometry of these two nozzles until the throat is the same, fig. 9(b), it was observed no change in the rate of heat transfer in the subsonic region, a simple difference in heat transfer coefficient occurred in sonic throat of nozzles.

The major difference in heat occurred in the supersonic region, as seen in fig.10(a), for this situation, the heat transfer coefficient is vastly different, since it is high in separation region for TOC nozzle. Values of heat transfer coefficient increase to a peak value in the vicinity where the wall pressure rises to a second peak, after decreasing downstream of the separation shock wave.

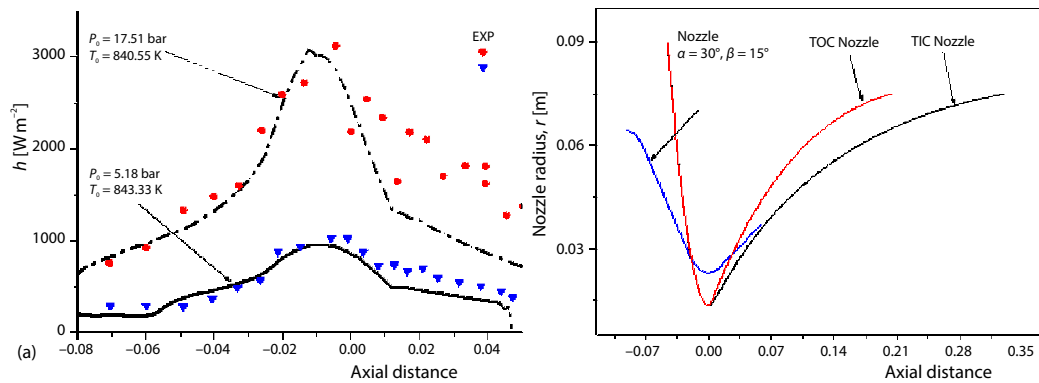


Figure 9. (a) Effect of stagnation pressure on heat transfer along the nozzle, (b) geometry of studied nozzles

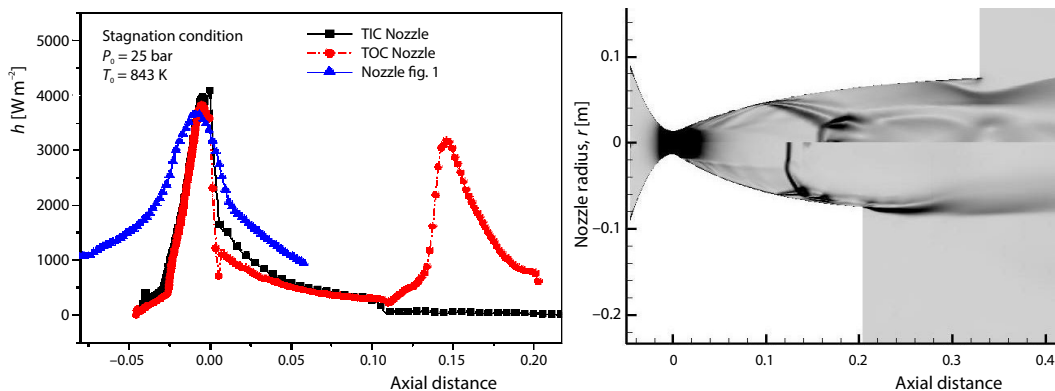


Figure 10. (a) Comparison of heat transfer rate along nozzles presented in (b) Schlieren picture, top half TIC nozzle and bottom half TOC nozzle

This pressure is associated with the incidence of shock wave, which reflected off the axis through a normal shock. The heat transfer coefficient remains relatively high at locations beyond the peak value, and heat transfer is to the wall over the entire separation region.

This occurs because of the interaction between the shock waves located downstream of the separation shock wave and the flow in the region near the wall where mixing and lateral transport of heat are appreciably increased. The important point to discuss here is the location of separation point, if this location is far from throat, the flow field in the separation region consisted of cool ambient air drawn into the nozzle and which then owed along the wall in a reverse direction to the heated mainstream flow. Provided that reflected shock waves were not incident on the wall in the separation region, the heat transfer in the separation region was relatively low. This was primarily because of the cool ambient air flow along the wall.

However, when separation occurred well within the nozzle just downstream of the throat, when the reflected shock waves were incident on the wall, fig. 10(b), heat transfer in the separation region exceeded sometimes the throat value when the boundary-layer was laminar at the throat. This is believed to be caused by the interaction between multiple shock waves (pseudo-shock) and the flow near the wall. This interaction increased mixing between the cool ambient air and the heated mainstream, and increased the lateral transport of heat.

Conclusions

A parallel implicit finite volume code based on the full Navier-Stokes unsteady equations is developed to solve axisymmetric nozzle flows using MacCormack's scheme. The present results treat the effects of several parameters on the dynamic and thermal characteristics of flow through a cooled convergent-divergent nozzle. The computational results indicated the following.

- Internal flow computational in a nozzle with a small ratio of r_c/r_{th} of 0.625 revealed radial variations in the flow. At the throat the Mack number was 0.8 at the axis and 1.4 near the wall.
- Interesting effects resulting from a variation in the Reynolds number have been observed in the nozzle heat transfer studies. For a fixed stagnation temperature and geometry, the Reynolds number can be controlled by adjusting the inlet stagnation pressure, and the heat transfer coefficients increased with increasing stagnation pressure as a result of a larger mass fluxes.
- The primary effect of varying the stagnation temperature is to change the mass flux, heat transfer coefficients are expected to decrease with an increase in stagnation temperature at higher stagnation pressure, and this trend is not clear at low stagnation pressure. With the combined effect of specific heat ratio and specific heats, it is found that the wall static pressure increases with the decrease of, and the separation flow location is earlier with increasing of specific heat ratio. The heat transfer distribution in the divergent section, just downstream of the adverse pressure gradient, is influenced in a way that is dependent on boundary-layer structure and the prescribed wall temperature.
- The heat-transfer coefficient is a maximum upstream of the throat, where the mass flux, deduced from wall static pressure, is largest. The wall temperature has an influence on the rate of heat transfer, the wall static pressure, and on the position of separation flow location. Furthermore work is required to gain some knowledge of the flow and thermal boundary-layers within a convergent-divergent nozzle.
- One of most significant result obtained in this paper is probably the well detection of separation shock patterns (in TOC nozzle); internal shocks were purposely used in the past in the design of over-expanded nozzles as a means of increasing the wall pressure. More intention is needed in the future to clarify the real origin and effects of this kind of structure patters.

Acknowledgment

The research reported herein was performed by the CRIHAN (Centre de Resource Informatique de Haute Normandy, France).

Nomenclature

a_1	– Bradshaw constant
C_p	– specific heat at constant pressure, [Jkg ⁻¹ K ⁻¹]
C_v	– specific heat at constant volume, [Jkg ⁻¹ K ⁻¹]
D	– diameter, [m]
F_1, F_2	– auxiliary functions in turbulence model
h	– heat transfer coefficient, [Wm ⁻² K]
k	– turbulent kinetic energy, [m ² s ⁻²]
M	– Mach number
P	– pressure, [Nm ⁻²]
Pr	– Prandtl number
Pr_t	– turbulent Prandtl number
R	– gas constant, [m ² s ⁻² K ⁻¹]
r	– radius, radial co-ordinate, recovery factor
T	– temperature, [K]
t	– time, [s]
u_i	– mean velocities, [ms ⁻¹]
x	– axial co-ordinate, [m]

Greek symbols

γ	– specific heat ratio
μ	– dynamic viscosity, [Pa·s]
μ_t	– turbulent viscosity
ρ	– density, [kgm ⁻³]
Ω	– scalar measure of the vorticity tensor
Ω_{ij}	– vorticity tensor
ω	– specific turbulent dissipation rate, [s ⁻¹]

Subscripts and superscripts

0	– nozzle entrance condition
aw	– adiabatic wall
e	– free stream condition
sep	– separation position
th	– throat position
w	– parameters on the wall surface
*	– initial point at the interaction region

References

- [1] Back, L. H., *et al.*, Convective Heat Transfer in a Convergent-Divergent Nozzle, *Int. J. Heat Mass Transfer*, 7 (1964), 5, pp. 549-568
- [2] Lutum, E., *et al.*, An Experimental Investigation of Film Cooling on a Convex Surface Subjected to Favourable Pressure Gradient Flow, *Int. J. Heat Mass Transfer*, 44 (2001), 5, pp. 939-951
- [3] Lebedev, V. P., Film-Cooling Efficiency in a Laval Nozzle under Conditions of high Freestream Turbulence, *Journal of Heat Transfer*, 128 (2006), 6, pp. 571-579
- [4] Back, L. H., *et al.*, Comparison of Measured and Predicted Flows through Conical Supersonic Nozzles, with Emphasis on the Transonic Region, *AIAA Journal*, 3 (1965), 9, pp. 1606-1614
- [5] Cuffel, R. F., *et al.*, Transonic Flow field in a Supersonic Nozzle with Small Throat Radius of Curvature, *AIAA Paper*, 7 (1968), N7, pp. 1364-1366
- [6] Delise, J. C., Naraghi, M. H. N., Comparative Studies of Convective Heat Transfer Models for Rocket Engines, *Proceedings*, 31st Joint Propulsion Conference and Exhibit, San Diego, Cal., USA, 1995
- [7] Xu, J., Zhao, C., Two-Dimensional Numerical Simulations of Shock Waves in Micro Convergent-Divergent Nozzles, *Int. J. Heat Mass Transfer*, 50 (2007), 11-12, pp. 2434-2438
- [8] Xiao, X., *et al.*, Role of Turbulent Prandtl Numbers on Heat Flux at Hypersonic Mach Numbers, *AIAA*, 45 (2007), 4, pp. 806-813
- [9] Sommer, T. P., *et al.*, A Near Wall Four-Equation Turbulence Model for Compressible Boundary- Layers, NASA Contractor Report (NASA-CR-4436), Arizona State University, Phoenix, Ariz., USA, 1992
- [10] Sommer, T. P., *et al.*, A Near Wall Variable Turbulent-Prandtl-Number Turbulence Model for Compressible Flows, *AIAA*, 31 (1993), N1, 27-35
- [11] Hadjadj, A., Onofri, M., Nozzle Flow Separation, *Shock Waves*, 19 (2009), 3, pp. 163-169
- [12] Lawrence, R. A., Symmetrical and Unsymmetrical Flow Separation in Supersonic Nozzles, Research Report Number 67-1, Southern Methodist University, Dallas, Tex., USA, 1967
- [13] Verma, S. B., Study of Flow Separation in Truncated Ideal Contour Nozzle, *Journal Propuls. Power*, 18 (2002), 5, pp. 1112-1121
- [14] Nave, L. H., Coffey, G. A., Sea-Level Side Loads in High-Area-Ratio Rocket Engines, *Proceedings*, 9th Propulsion Conference, Las Vegas, Nev., USA, 1973
- [15] C. L. Rumsey, Compressibility Considerations for k - ω , Turbulence Models in Hypersonic Boundary-Layer Applications, NASA, Washington DC, TM 215705, 2009

- [16] Rumsey C. L., Consistency, Verification, and Validation of Turbulence Models for Reynolds-Averaged Navier-Stokes Applications, *Proceedings*, 3rd European Conference for Aerospace Sciences, Versailles, France, Paper EUCASS2009-7
- [17] Menter, F. R., Improved Two-Equation k - ω Turbulence Models for Aerodynamic Flows, NASA, Washington DC, TM 103975, 1992
- [18] Moore, J. G., Moore, J., Realizability in Two-Equation Turbulence Models, *Proceedings*, 30th Fluid Dynamics Conference, Norfolk, Va., USA, 1999
- [19] MacCormack, R. W., Current Status of Numerical Solution of the Navier-Stokes Equations, *Proceedings*, 23rd Aerospace Sciences Meeting, Reno, Nev., USA, 1985
- [20] Steger, J., Warming, R. F., Flux Vector Splitting of the Inviscid Gas Dynamics Equations with Application to Finite Difference Methods, NASA, Washington DC, TM-78605, 1979
- [21] Campbell, C., Farley, J., Performance of Several Conical Convergent-Divergent Rocket Type Exhaust Nozzles, NASA, Washington DC, TN D-467, 1960
- [22] Sauer, R., General Characteristics of the Flow through Nozzles at Near Critical Speeds, NASA, Washington DC, TM-1147, 1947
- [23] Oswatitsch, K., Rothstein, W., Flow Pattern in a Converging-Diverging Nozzle, NASA, Washington DC, TM-1215, 1949
- [24] Hall, I. M., Transonic Flow in Two-Dimensional and Axially-Symmetric Nozzles, *Quarterly Journal of Mechanics and Applied Mathematics*, 15 (1962), 4, pp. 487-508
- [25] Herbert, M. V., Herd, R. J., Boundary-Layer Separation in Supersonic Propelling Nozzles, NGTE Report No. 3421, 1964
- [26] Delery, J., Marvin, J., Shock-Wave Boundary-Layer Interactions, AGARD Report No. 28, NATO, Neuilly sur Seine, France, 1986
- [27] Schmucker, R. H., Status of Flow Separation Prediction in Liquid Propellant Rocket Nozzles, Technical Memorandum TM X-64890. NASA, Washington DC, 1974
- [28] Vieser, W., *et al.*, Heat Transfer Predictions Using Advanced Two-Equation Turbulence Models, CFX/ANSYS, Tech. Rept. CFX-VAL 10/0602, Otterfing, Germany, 2002
- [29] Tong, X. L., Luke, E., Turbulence Models and Heat Transfer in Nozzle Flows, *AIAA*, 42 (2004), 11, pp. 2391-2393