Numerical Study of the Injection Process in a Transonic Wind Tunnel—Part I: The Design Point

Injectors are to be installed in a transonic wind tunnel with the ultimate objective of expanding the Reynolds number envelope. The aim of this research effort is to numerically simulate the steady mixing process involving the supersonic jets and the tunnel subsonic main stream. A three-dimensional, Reynolds-averaged Navier–Stokes numerical code was developed following the main lines of the finite-difference diagonal algorithm, and turbulence effects are accounted for through the use of the Spalart and Allmaras one-equation scheme. This paper focuses on the “design point” of the tunnel, which establishes (among other specifications) that the static pressures of both streams at the entrance of the injection chamber are equal. Three points are worth noting. The first is related to the numerical strategy that was introduced in order to mimic the real physical process in the entire circuit of the tunnel. The second corresponds to the solution per se of the three-dimensional mixing between several supersonic streams and the subsonic main flow. The third is the calculation of the “engineering” parameters, that is, the injection loss factor, gain, and efficiency. Many interesting physical aspects are discussed, and among them, the formation of three-dimensional shocks and expansions’ “dome” [DOI: 10.1115/1.2734236]

Keywords: numerical simulation, injection process, transonic wind tunnel, turbulent mixing layer

1 Introduction

Among the many ideas, processes, and capabilities that were introduced and aggregated to a wind tunnel circuit along the development history of this equipment, one of the simplest and most practical is the “injection.” The idea is in a way very simple, and corresponds, in general, to the convenient introduction of a high-speed gas jet inside the tunnel with the objective of “feeding” momentum to the main stream. Inclusively, some installations rely on injection as the only source of energy, and, as an example, some Russian facilities make use of the process [1]. Experiments by Muhlstein et al. [2] in a 6 in. transonic facility had the objective of assessing some very important aspects of the flow, e.g., energy level and flow quality, and how injection eventually affects them. If the main source of energy is injection, the operation is certainly intermittent. However, in many instances, it can be used as an auxiliary power source, as for example, in the well succeeded Calspan 8 ft transonic facility [3,4]. Originally, this tunnel operated continuously, but it was thereafter retrofitted to also run intermittently with the help of four injectors. The injected mass corresponds to 4% of the main stream flow rate and the Reynolds number range was duplicated. But, generally speaking, although the technique has been normally used, there are still serious concerns related to optimization of the running conditions, especially when injection is used in conjunction with the compressor [5,6]. The technique is being incorporated into the design of a new transonic facility in Brazil. The injectors would be installed at the entrance of the transition chamber (the element of the tunnel circuit that provides for the smooth passage between a rectangular and a circular cross section). Because of its novelty, both in position and geometrical form, a pilot wind tunnel is also being designed for the previous testing of this layout. The envelope of the pilot transonic tunnel (PTT) is shown in Fig. 1. A theoretical study of the transients in the tunnel, including also the effects of injection, was developed some years ago by the present authors [7,8], and corroborated the idea in its general terms.

The objective of the present research effort is to investigate the mixing process between the supersonic jets coming out from the injectors and the tunnel subsonic main stream. This investigation will shed light on this new application of the injection process, and will certainly help in the next steps of detailed design, construction, calibration, and operation of both facilities, pilot and industrial. The details of the problem are discussed in Sec. 2. Because, at this stage, we are mostly interested in the engineering aspects of the problem, the focus of the analysis is on the steady-state solution. The study is done on a numerical basis, and, to this end, a new Reynolds-averaged Navier–Stokes (RANS) code was developed. The basics of the code follow the diagonal algorithm of Pulliam and Chaussee [9], and turbulence effects are accounted for through the use of the Spalart and Allmaras [10] one-equation model. To keep the numerical scheme stable nonlinear spectral artificial dissipation terms are included. For every tunnel there is always an “optimized” running condition that is usually known as the “design condition,” or “design point” as many experimentalists like to call it. In relation to the injection process, this condition corresponds to both currents having the same static pressure when they first meet at the entrance of the mixing chamber. This paper focuses the study on the design point. Some off-design cases are also being investigated and the results will be published in another paper.

In the next section the problem is formulated and the main simplifying assumptions are introduced. Following, details of the numerical scheme and validation cases are briefly discussed. Finally, results of the three-dimensional calculation are presented and discussed. Besides the main physical aspects, some engineering data— injection loss factor, gain, and efficiency—are also calculated and commented on. It is important to note that the calculations and results to be reported herein correspond to the geometry and conditions of the pilot tunnel circuit.
2 Basic Formulation

2.1 Statement of the Problem. The test section of the PTT has a width of 0.30 m and a height of 0.25 m. The circuit is closed and pressurized from 0.5 bar to 1.2 bar; the range of testing Mach number is 0.2–1.3, and the standard operation is to be continuous and driven by a two-stage, 830 kW main compressor, frequency controlled. There will be a complete controlling system to maintain the desired test section Mach number, stagnation pressure, and temperature. The entire facility is being designed taking as reference the “design point” condition. In this situation, and at the test section, the Mach number is 1, and stagnation pressure and temperature are 110 kPa and 313 K, respectively. At the entrance of the injection chamber the design point corresponds to a tunnel mainstream Mach number equal to 0.51, and “asks” the same static pressure, both for the main and the supersonic streams [8,11]. Figure 1 shows the operational envelope considering the PTT test section conditions. Observe the region labeled “combined injection”—here the running can be continuous by working with the compressor alone, or on an intermittent basis by coupling the compressor, not at full power, with the injection system. A remote region can be reached only on an intermittent basis by using the main compressor at full power combined with the injector system—observe the dashed region beyond the main compressor power limit curve.

The injection system is composed of high-pressure reservoirs that supply air to ten fixed-geometry convergent-divergent nozzles, through a pressure regulator valve that maintains an exit Mach number of 1.9 during about 45 s (at the design condition).

---

Fig. 1 PTT operational envelope; test section conditions

Fig. 2 Tunnel plenum chamber showing the injectors section and feeding system: (a) transition/injection chamber; (b) lateral view; and (c) inlet section
Figure 2(a) sketches the details of the injectors' installation into the tunnel main circuit. The supersonic jets are discharged at the entrance of the transition chamber, which is the longitudinal section of the circuit that provides for the passing from a square to a circular cross section. In Fig. 2(b) some details of the transition chamber are given—the exit section corresponds to the entrance of the first diffuser—and in Fig. 2(c) the transition chamber entrance cross section is shown, with the positioning of the injectors: five at the floor and five at the ceiling. The height and width of the nozzles at their exit section are 0.0225 m and 0.0157 m, respectively.

The domain of calculation corresponds to the volume embraced by the transition section (in Fig. 2(b), it is represented by the volume between planes S1 and S2), and its longitudinal length in the PTT is 60 cm. Let us imagine that the main compressor is running, and therefore there is already a steady main stream in the tunnel. Our goal is to calculate the flow that is established in the transition chamber after the nozzles feeding is opened. At the beginning and at the end of the injection operation there are quick transitions, but the reservoirs are able to sustain 45 s of injection operation (at the design condition, see Fig. 12). We shall focus on the steady state of the injection operation.

2.2 Simplifying Assumptions. In this section we shall discuss the main simplifying assumptions that were introduced in order to model the above physical situation. First of all, a simplified geometry was adopted. The main conceptual point in the transition element is to keep the cross-sectional area constant, in spite of the geometrical form variation along the axis of the tunnel. Hence, the cross-sectional form was then admitted as being square and equal in shape and area to the entrance section. The simplified geometry is shown in Fig. 3.

A second point refers to the treatment of the flow about, and inside, the injectors. Observe Fig. 2(b). The flow that reaches the nozzles, coming from the main stream and at this position mostly dominated by the tunnel wall boundary layer, has a very complicated three-dimensional topology around the pieces. Besides, and because the injectors protrude from the tunnel wall, (most probably) a horseshoe vortex will be formed in front of each of the nozzles. We did not tackle this topology, but rather, admitted the wall of the nozzles as flat plates, and calculated the outside boundary layers along them, considering as the starting point the conditions of the flow at the tunnel main stream. For the inner boundary layers, the reference was taken as the average between the states of the gas inside the nozzle's stagnation chamber and at the exit section. But, what exactly is the argument here? Well, one has to be concerned with the fate of the flow around and inside the nozzles, because this flow is going, ultimately, to compose part of the boundary conditions at the domain of calculation entrance plane (see Fig. 3).

We understand that the real geometric form of the transition chamber is an important issue for this problem, as well as the exact flow about and inside the injectors. But, we also understand that a problem of such magnitude has to accommodate some simplifications at the first approach, otherwise the challenge turns out to be "almost impossible." This is the standard way of treating difficult research problems. After having coped with all the basic issues, it is our plan to extend the study further and possibly eliminate some of these assumptions.

At last, observe now Fig. 2(c). It represents the inlet section of the mixing region. There are two symmetry planes: one horizontal and one vertical (Planes 6 and 1). We have therefore considered as the computational domain one quarter of the whole mixing region. See also Fig. 3. However, a fine grid representation of this smaller volume, considering the resolution that has to be provided at all the viscous gradient regions, would still result in about 8 x 10^6 node points, a situation completely out of reach of the computational power available to the authors. Therefore, we proceeded as follows. We ran the code for the one-quarter region but considering a coarse grid (about 1 x 10^6 points). Then we transferred the resulting solution to each of the individual subregions, A, B, C, D, and E, obtained by dividing the original domain through planes 2, 3, 4, and 5 (Fig. 2(c)). We then reran the code for all of these subcells but considering a finer grid for each of them (about 1.5 x 10^6 points for each subregion).

2.3 The Engineering Parameters. Once the solution of the problem is obtained in converged form, one has the basic data for the evaluation of some "integral" parameters that are characteristic of the process. The injection "engineering" parameters are: (i) the loss factor; (ii) the injection gain; and (iii) the injection efficiency. In this subsection we shall derive the proper expressions for the calculation of these factors.

2.3.1 The Loss Factor. As the injection process is considered adiabatic, the loss in the injector chamber is directly related to the stagnation pressure drop, Δp0, that is, we can define the total pressure loss factor, K, by

\[ K = \frac{\Delta p_0}{p_{0r}} = \frac{(\Delta p_{02l} - p_{0r})}{p_{0r}} \]  

(1)

where \( p_{0r} \) is the cross-sectional mean stagnation pressure. Therefore, for obtaining K, one needs to calculate the mean stagnation pressure variation between the entrance and exit sections of the mixing chamber. We, now, will describe how this calculation is performed. The relation between the mean stagnation pressures can be obtained once the values of mean entropy at the inlet and exit planes are known

\[ \frac{p_{0l}}{p_{0e}} = e^{-(\gamma - 1) \frac{s_{in}}{R}} \]  

(2)

The mean value of the entropy, \( s_{in} \), at a determined cross section can be evaluated by referring to Fig. 4. The area of the section is already divided by the grid nodes in cells. Because we already know the entropy values at the nodes, the following definition of the entropy at cell n is adopted

\[ s_n = \frac{1}{3} (s_1 + s_2 + s_3 + s_3) \]  

(3)

For the calculation of entropy values, one needs a reference state. This state can be anyone of convenience, but once defined, the reference must be kept the same for all entropy calculations, even for the case of different sections. Now the mean value of entropy for the section can be established as

\[ \bar{s} = \frac{\sum_{n=1}^{m} n \cdot s_n}{\sum_{n=1}^{m} n} \]  

(4)

What we need now is to devise an approach for the evaluation of \( \bar{p}_{0l} \) or \( \bar{p}_{0e} \). As already mentioned these symbols represent mean
values of the stagnation pressures at a certain cross section. But one has to be careful with this concept. For example, whatever the evaluation strategy, it is more difficult to “visualize” a “mean stagnation pressure” at the injection chamber entrance plane because of the great differences in properties between the two streams (including the mass flow). This is not so for the exit section, where the mixing process is almost done, and the idea of a mean stagnation pressure is therefore more natural. We have then proceeded as follows. Let us suppose that \( \rho_{0,e} \), \( T_{0,e} \), and \( \bar{s}_e \) are the mean values of stagnation pressure, stagnation temperature, and entropy, respectively, at the exit section. For a generic cell \( n \) (at the same exit section), we can write that

\[
\Delta s_n = s_e - \bar{s}_e = -R \ln \left( \frac{p_{0,n}}{p_{0,e}} \right) + c_p \ln \left( \frac{T_{0,n}}{T_{0,e}} \right) \tag{5}
\]

If we multiply this equation by \( m_n \) and sum up for all cells, the first member will be equal to zero in accordance with Eq. (4), and observing that \( \bar{s}_e \) is the mean value. There results then the following expression for the evaluation of \( \bar{p}_{0,e} \).

\[
\bar{p}_{0,e} = \exp \left\{ \sum_{n=1}^{n_e} m_n \ln \frac{p_{0,n}}{p_{0,e}} - \frac{\gamma - 1}{\gamma} \sum_{n=1}^{n_e} m_n \ln \left( \frac{T_{0,n}}{T_{0,e}} \right) \right\} \tag{6}
\]

The mean stagnation temperature, \( T_{0,e} \), is a consequence of the calculation of \( \bar{p}_{0,e} \), the mean stagnation enthalpy, by an expression similar to Eq. (4). Because the gas is supposed to be calorically perfect, one has immediately

\[
T_{0,e} = \frac{\bar{h}_{0,e}}{c_p} \tag{7}
\]

Now, \( \bar{p}_{0,e} \) can be evaluated by Eq. (2), and the total loss factor \( K \) follows immediately from Eq. (1).

2.3.2 The Injection Gain. Another important parameter that is used to quantify the injection process is its gain, normally indicated by the symbol \( \lambda \). There are different ways to define the gain depending on the specific engineering application. For this kind of problem, the best definition is [12]

\[
\lambda = \frac{\bar{p}_{0,e} - \Delta \bar{p}_0}{\bar{p}_{0,2}} \tag{8}
\]

where \( \Delta \bar{p}_0 \) is the variation of mean stagnation pressure in the mixing process; and \( \bar{p}_{0,2} \) is the main flow mean stagnation pressure at the entrance plane, before the injection operation is started.

2.3.3 The Injection Efficiency. The efficiency is expressed in terms of the entropy variation between the inlet and exit sections. With this definition, all the irreversibilities are accounted for. Following Nogueira et al. [13], one writes

\[
\eta = -\left( \frac{\bar{s}_2 - \bar{s}_1}{\bar{s}_1 - \bar{s}_e} \right) \left( \frac{\bar{m}_0}{\bar{m}_1} \right) \tag{9}
\]

where subscripts "1" and "2" refer to the two separate inlet streams.

3 Computational Model

3.1 The Code. The mathematical model is represented by the Reynolds-averaged Navier-Stokes equations, written in generalized coordinates and conservation-law form. By “Navier-Stokes equations” we recognize the collection of the continuity, momentum, energy, and any constitutive equation necessary to represent the medium. The medium is air considered as an isotropic and Newtonian fluid, and as a thermally and calorically perfect gas. The numerical algorithm follows closely the main lines of the finite-difference, diagonal scheme due to Pulliam and Chaussee [9], complemented by a nonlinear, spectral-radius-based artificial dissipation strategy due to Pulliam [14]. A one-equation turbulence model as suggested by Spallart and Allmaras [10] is also aggregated to the system of equations.

3.2 Boundary Conditions. First, we focus on the problem of the larger domain. This volume, whose entrance face is the rectangle \( PQRS \) (see Figs. 2(c) and 3), has six faces. Boundary conditions for the upper and left faces are established considering symmetry relative to Planes 6 and 1, respectively. The lower and right faces are solid walls, and the nonslip condition is to be enforced. We have also considered solid walls as adiabatic surfaces. At the exit plane extrapolation was done by means of a simplified convection equation. But, by far, the most difficult situation we have faced was the establishment of the boundary conditions at the entrance plane (rectangle \( PQRS \)). The difficulty comes from the various viscous regions that are present and that correspond to boundary layers that reach (and cross) the entrance plane. These layers are formed along the bottom edge, ceiling, and lateral walls, and along the external and internal surfaces of the injectors. But, even more difficult was the tackling of the viscous corners formed at each interaction of two of those boundary layers. Outside the viscous regions, there exist two “potential” cores: the supersonic, inside the nozzle, and here all conditions are fixed, and the subsonic, outside the nozzle, where we have relied on the one-dimensional longitudinal form of the nonviscous gas-dynamic characteristic equations.

Second, there were the cases of the smaller domains, i.e., subregions, or cells, \( A, B, C, D, \) and \( E \). For all of them, the upper, lower, inlet, and exit faces require exactly the same corresponding conditions discussed above. The differences are related to the lateral surfaces. The boundary condition at the left face of cell \( A \) can be established considering symmetry relative to Plane 1. The conditions at the right face of cell \( A \) are fixed from the solution at the larger domain. The right face of cell \( E \) is a solid wall, therefore, the nonslip condition applies. For the left face of cell \( E \) the conditions are fixed and are obtained from the solution at the larger domain. Internal cells like \( B, C, \) and \( D \) all have lateral faces whose boundary conditions are also fixed by the solution at the larger domain. Here, an important observation is due. Because the smaller domains have grids that are finer when compared to the corresponding regions of the larger volume, when passing values we have relied on a bilinear interpolation procedure [15].

3.3 Grids. A typical mesh needs to have several refinements to properly calculate regions of large viscous gradients. Close to solid walls and at the mixing interfaces, nodes' clustering was a necessity. Figure 5 shows the grid topology at the inlet plane of subregion \( E \), the one subregion that has as the right frontier the
right lateral wall of the tunnel. The inner subgrids have topologies similar to grid E, but without refinements at the right lateral faces, evidently. But, almost always, clustering brings with it code stiffness. In order to relieve this effect and achieve higher CFL number values, a radial "diffusion" of nodes along the vertical and lateral directions was adopted at the streams mixing regions. In other words, the "radial" positioning of the nodes followed the radial enlargement of the mixing jet as it grew along the tunnel.

3.4 Interim Comment. In the above, we presented a quick and very general overview of the many challenges that appeared in the development of this work. But, there is a whole myriad of details which are very important, and there is no room to describe them here. This same argument applies to the validation cases; we will only touch this matter in the next section. Anyway, details of the code development and other numerical information are not the focus of this paper, and all these subjects are being published elsewhere (see Ref. [16], to where the interested reader is referred).

4 Results and Discussion

4.1 Validation Cases. Before attempting to calculate the flow at the injectors' mixing chamber, the new code was extensively validated. Several well-established physical situations of increasing complexity were simulated. These were: (i) Laminar and turbulent subsonic and supersonic flows along a two-dimensional flat plate (2D); (ii) nonviscous, laminar, and turbulent flows in a transonic convergent-divergent nozzle (2D); (iii) interaction of a shock wave and a laminar boundary layer (2D, 3D); (iv) interaction of a shock wave and a turbulent boundary layer (2D, 3D); (v) turbulent mixing of two parallel jets, one supersonic and the other subsonic (2D, 3D); and (vi) turbulent mixing of two parallel supersonic jets (2D, 3D). 2D and 3D mean, respectively, two-dimensional and three-dimensional simulations. For the sake of conciseness we shall briefly discuss only cases (iv) and (vi). These test cases were run using the two-dimensional and the three-dimensional version of the new code. The results to be presented in the sequel come all from the three-dimensional simulations. The reader should observe that much attention was dedicated to the validation of the code for the turbulent jets mixing cases. This had to be so, because the nature of the problem to be tackled, namely the injection mixing process, is connected to the physics of the mixing jets.

With the objective of working with a smaller and, consequently, finer mesh, we followed Wilcox [17] and adopted the calculation domain represented in Fig. 6, for the simulation of the interaction of a shock wave and a turbulent boundary layer, where $\delta$ represents the boundary layer thickness just before the recirculation region. But before focusing on this grid, the flow starting at the leading edge of the plate had to be calculated with the basic aim of obtaining the boundary layer profile at the entrance of the smaller mesh.

Figure 7 is the resulting pressure field over which we have "marked" the limits of the recirculation region. The structure of the system of shock waves is quite apparent and the separation and reattachment waves are well represented. This case corresponds to $Re_s=2.5 \times 10^7$, $Ma_s=2.96$, and $\theta=12.75$ deg, where $Re_s$ is the Reynolds number based on the boundary layer thickness at the entrance of the domain. The length of the separation bubble calculated by this method is equal to $3.54\delta$, a value that compares well with the result of Wilcoxx [17] for the same case, $3.87\delta$. Another pertinent comparison is made in Fig. 8, where experi-
Fig. 6 Domain of calculation for the shock-wave/boundary-layer interaction

Fig. 7 Static pressure field for the shock-wave/turbulent boundary-layer problem. Values are made dimensionless by the inlet pressure. Coordinates are given in meters.

Fig. 8 Pressure distribution on the flat plate wall

Fig. 9 Sketch of the experimental setup of Goebel and Dutton [23]

As can be seen our prediction follows closely both the two different numerical results of Wilcoxon [17] (Re = 2.5 × 10^5 and 1.0 × 10^6, for 1 and 2, respectively) and the experiments of Reda and Murphy (without and with sidewalls treatment, for 1 and 2, respectively) [18,19]. The present simulation corresponds to Re = 2.5 × 10^5.

The second validation test corresponds to the turbulent mixing of two high-speed streams. This research topic has received much attention, especially in recent years. Ou and Goldberg [20] have treated numerically the case of the layer between two supersonic streams, with a finite volume technique and the k-ε turbulent model, for a mixing condition very similar to the one depicted in Fig. 9. Freund et al. [21] investigated the evolution of a supersonic jet and its acoustic field by means of direct simulation. A hybrid approach, RANS and LES, is applied by Georgiadis et al. [22], in the prediction of the interaction between two supersonic streams. This simulation corresponds to Case 2 of Goebel and Dutton [23], the details of which will be discussed below. Chinzei et al. [24] obtained, based on their experimental measurements, a correlation between the growth rate of the mixing layer and the streams velocity ratio. Samimy and Addy [25] presented results relative to a supersonic/supersonic mixing, and discussed the influence of the splitter plate (that separates the two jets) in the resulting flow. The turbulent structure of the layer in a supersonic/subsonic mixing is studied by Clemens and Mungal [26] in three different experiences. The relative Mach number, M_r, was made to vary and the effects of the compressibility in the mixing process were duly investigated. The authors found that a higher relative Mach number induces a three-dimensional character in the mixing layer. But, by far, the best collection of data for comparison purposes is that of Goebel and Dutton [23], who examined the total of seven cases. The authors investigated the turbulent mixing of two high-speed streams in the range of relative Mach numbers from 0.40 to 1.97, which covers a region of significant compressibility effects. Several important aspects were investigated and discussed: the similarity region, the growth rate, and some turbulent correlations, among other studies, for a number of combinations of supersonic and subsonic streams. The tunnel, a sketch of which is shown in Fig. 9, was duly prepared for the two-dimensional mixing of the streams. The reader should observe that, initially, the currents are not parallel but instead have a relative angularity of 2.5 deg. There is also a splitter plate separating the oncoming flows, whose trailing edge is 0.5 mm thick at the entrance of the mixing region. We shall present here the numerical simulation of Goebel and Dutton's Case 2. This physical case corresponds to the mixing of two
supersonic flows with Mach numbers 1.96 and 1.36 ($M_s = 0.91$). When entering the mixing chamber the conditions are set such that the two jets have practically the same static pressure. Because the flows are supersonic, the boundary conditions at the entrance of the computational domain are all fixed. The floor and the ceiling of the tunnel are simulated as solid walls, and at the exit section extrapolation was applied by means of a simplified convection equation. One of the most important parameters in the study of a mixing layer is its thickness, $b$. At the start of the mixing process there is a transition region, after which the mixing layer velocity profiles develop into a similar pattern. This pattern is lost when the layer starts to feel the presence of the walls. The growth of the layer at the similar stretch, characterized by the rate $db/dx$, is a basic parameter of the mixing process. Figure 10 shows the result of the numerical calculation, and the reader should note that the length of the similar stretch—between 0.10 and 0.45—was taken as equal to the experimental observation. The value of $db/dx$, obtained by linear regression, is equal to 0.039, while the experimental value is 0.038, a difference of about 2%. 

To further illustrate the case in Fig. 11 we show the cross-sectional velocity profile in the middle of the growth region, plotted together with experimental values [23] and with another numerical simulation based on a hybrid approach, RANS and LES [22]. The reader can observe that the comparison of the present solution with the experimental data is very good.

As already stated, the validation results that were presented above come all from the three-dimensional version of the code, which confirms the overall accuracy of the numerical code. Nevertheless, we would like to comment on the 3D grid. In the streamwise and vertical directions the grid was kept exactly equal to the two-dimensional case. To construct the 3D grid we introduced a certain length along the spanwise direction $z$, and established symmetrical boundary conditions at both lateral faces. With the objective of checking the possible influences of the spanwise length and discretization upon final values, we have investigated them with three different lengths, and for each length with three different resolutions. For example, in the case of the jets mixing, the lengths were defined as 0.05 m, 0.10 m, and 0.20 m, while 21, 41, and 81, were the adopted spanwise number of nodes (the number of points in the streamwise and vertical directions were, respectively, 81 and 105). Considering the level of precision of the algorithm, no significant differences were detected between different configurations.

For completeness, a verification of the code was also done. A grid refinement study was performed together with the determination of the order of the numerical method [27]. The physical case of reference was the two-dimensional turbulent boundary layer along a flat plate, which was calculated with grids $481 \times 481$, $241 \times 241$, $121 \times 121$, $61 \times 61$, and $31 \times 31$. Solutions for the two finer grids were basically coincident, and therefore they served as reference for the calculation of the errors on the other grids. The mean value obtained for the order of the method was 2.25, very close to the theoretical value (that is equal to 2).

### 4.2 The Injection Process

When the control valve of the compressed air is opened (Fig. 2(a)), the supersonic streams start transferring momentum to the tunnel primary flow at the injection chamber. This induces an acceleration of the main stream until a final equilibrium condition is established, and this final condition is a function of the actual situation of the complete tunnel circuit. In other words, every element of the circuit has an influence on this final equilibrium state. A study of this dynamic process was undertaken some years ago by the authors of this work [7, 8]. In this study a simplified mathematical model, based on the concept of "lumped parameters" [28], was used, with the basic aim of simulating the aerodynamic responses to various inputs to the tunnel stream, and among them, the injection of a supersonic stream. The analysis also included the many control devices that exist in a typical transonic circuit. The results of this simulation revealed that injection causes a new distribution of parameters along the circuit. In particular, there is a shift in the main compressor point of operation, with consequent variations in its performance and compression rate. Besides, other modifications appear and the tunnel circuit finally adapts to the new conditions imposed by the injection process. At the test section, a new upper level of stagnation pressure and Mach number, and a slight decrease in stagnation temperature, are finally established. Figure 12 illustrates the resulting stagnation pressure variation at the test section for the design point [7, 8]—the reservoirs are able to sustain about 30 s of steady flow at the test section. Locally, at the injection chamber, there also happens to be a raise in stagnation pressure and Mach number.

For a running of the PTT at the design point the flow at the entrance of the mixing section is always subsonic, and the value of the Mach number is 0.51 [11]. But, what mostly characterizes the design condition (in terms of the injection process) is the equality of the static pressures between the main stream and the supersonic jets, exactly at the entrance of the injection chamber (for more details, see Sec. 2.1). This, in principle, is the optimum operation condition, because it will avoid the appearance of strong compression and/or expansion waves and, consequently, losses will be minimized. Due to the strong acceleration provided by the supersonic jet, the injection process will induce an increase in the Mach number of the subsonic stream, and this effect will be transmitted upstream to the entrance of the mixing chamber and also further up to the test section. Because the inlet static pressure is maintained, an increase in the Mach number will correspond to an increase in the stagnation pressure. The Mach number and the stagnation pressure simply do not rise even further, because of the
counterbalancing effect of the pressure losses along the circuit. By the way, the ultimate rise of the stagnation pressure is the result of the net balance between "new energy" introduced by the supersonic stream and the losses along the tunnel circuit.

The point now is the following. This rising of the stagnation pressure at the entrance of the injection chamber has, in some way, to be "informed" to the numerical code; otherwise the simulation will not mimic properly the real physics of the process. How to achieve this goal? The following procedure was adopted. After calculation is started, one allows for the rise of the stagnation pressure at the mixing chamber entrance plane, until the plateau value given in Fig. 12 is reached. Let us note that the stagnation pressure rises due to the rise of the Mach number and the constancy of the static pressure at the entrance plane. After that point is reached, there is a shift of boundary conditions at the inlet section: from this moment on, the stagnation pressure is kept constant and the static pressure is left to change. The code is then run until final convergence. The practical mechanism that permits the augmentation of the stagnation pressure value is the "extrapolation from inside." The tunnel stream is supersonic at the entrance of the mixing section, therefore information is transported upward because one of the longitudinal characteristic velocities, \( a = a \), is negative. Therefore, extrapolation of values of \( a \) from plane \( i = 2 \) to 1 (the entrance plane) is done using the corresponding characteristic equation. This procedure is performed at the "potential" core of the main supersonic stream. The point of concern now is the fate of the static pressure at the supersonic entrance. It should not "drift away" because this would mean that the design condition is no longer being satisfied. But, as we pointed out before, the ending point of this maneuver—the plateau value of the stagnation pressure in Fig. 12—is an equilibrium condition, and, most probably, the static pressure will not vary significantly. We show that this is so in Fig. 14.

4.3 The Physics of the Injection Process. We pass now to the discussion of the physical results. Before that, let us observe Fig. 13. Lines 1, 3, and 5 are the traces of vertical planes on the entrance plane. Lines 1, 3, and 5 contain the geometrical centers of the exit sections of Injectors 1, 2, and 3, respectively. Lines 6 and 7 are the traces of horizontal planes on the entrance plane. Line 6 contains the geometrical centers of Injectors 1, 2, and 3. The distance of Line 7 to the floor of the tunnel is \( y = 0.1 \) m. Plotted on Fig. 14 are distributions of static pressure along these lines. The distributions correspond to the final converged solution. As the reader can observe, the static pressures of the main stream and the supersonic jet, at the entrance plane, are basically the same at the end of the converging process, which is a guarantee that the design condition "was not lost" during the maneuvering strategy proposed above. The results to be presented, therefore, correspond all to the tunnel operating at the design condition.

Figure 15 shows the dimensionless static pressure fields on the vertical plane containing Line 1 (Fig. 13) and on the horizontal plane containing Line 6 (Fig. 13), respectively. In this last instance only the vicinity of Injector 1 is shown. The pressure fields are basically homogeneous but, nevertheless, a very small difference between the subsonic and supersonic values at the entrance to the injection chamber (in the vicinity of the injectors—see Fig. 14) is sufficient to give rise to compression and expansion regions, in spite of the fact that the strength of these disturbances are very mild in this situation. At the moment of writing this paper we are running some off-design cases and, in this instance, the strength of these waves is really very high. But, by far, the most interesting physical effect to be learned from these figures is the appearance of alternating expansion and compression regions. These have, in this three-dimensional situation, the form of domes, whose main dimensions are on the same order of magnitude as those of the injectors. But a quick physical reasoning

![Fig. 13 Special lines definition at the entrance plane of the mixing chamber. Static pressure plots along these lines will illustrate the condition at the design point.](image-url)
would confirm this result: the supersonic jets coming out of the nozzles have to develop between the floor of the tunnel and a kind of “conical subsonic envelope” that limits them.

Figure 16(b) is a general view (from above) of the Mach number field on the horizontal plane that contains Line 6. One can observe the influence of the supersonic streams in the overall acceleration at the end of the chamber. But as one moves away from the high-speed stream the acceleration falls accordingly. This can

---

Fig. 14 Static pressure distributions along Lines 1, 3, 5, 6, and 7. Values are made dimensionless by the static pressure of the main stream at the entrance plane. Thin lines mark injectors’ walls positions.

Fig. 15 Static pressure fields in regions close to injector 1: (a) vertical plane; and (b) horizontal plane. The solid black rectangles at the left of the plots mark the position of the injector. Coordinates are given in meters.
be checked by observing Fig. 16(a), where the Mach lines are shown for a vertical plane passing by the center of Injector 2. The topology of the mixing layers can also be appreciated by inspecting Fig. 17, where isolines of turbulent viscosity are shown. Values in Fig. 17 are made dimensionless by the molecular viscosity at the exit section of the nozzles. The production is so high in the mixing layers that the level of turbulence at the exit of the chamber is about 3000 times that at the inlet of the supersonic stream. In spite of that, there is no complete mixing at the exit of the chamber, what means that the process will finally end inside the first diffuser. Another very interesting aspect can be observed in Fig. 18, where the map of dimensionless stagnation pressure at the entrance plane is shown. At the start of the calculation process, the main (subsonic) stream stagnation pressure at this plane was equal to 1.193—a value made dimensionless by the static pressure at the exit of the injectors. Observing the field one can see that the overall stagnation pressure was raised to 1.249 as a result of injection. But close to the right lateral plane, which happens to be the lateral wall of the tunnel (see Fig. 3), there appeared to be a difficulty in this process. But the explanation here is simple, because the wall with its boundary layer constitute what can be recognized as an important "loss region," and therefore it represents a difficulty for raising the stagnation pressure.

Important views of the mixing layers are shown in Figs. 19 and 20. Figure 19 corresponds to the first stations, while Fig. 20 illustrates the last sections of the injection chamber. In Fig. 19 one can see the beginning and the evolution of the mixing process, and recognize clearly the presence of the "potential" cores. Here, one can observe the radial diffusion of nodes as mentioned earlier in Sec. 3.3, an expedient that was used in order to relieve grid stiffness. In Fig. 20 the reader can confirm the fact that the streams are not completely mixed when the end of the chamber is reached. The evolution of the mixing layers along all the longitudinal
Fig. 19 Streamwise velocity profiles on vertical (a) and horizontal (b) planes containing the geometrical center of the injector 1. Coordinates in meters.

Fig. 20 Streamwise velocity profiles on a vertical plane containing the geometrical center of injector 1. Representation at the exit of the injection chamber. The dashed line represents the injector height. Coordinates in meters.

Fig. 21 Turbulent viscosity field on a horizontal plane passing by the geometrical center of the injectors' exit sections. Coordinates in meters.

Fig. 22 Three-dimensional view of the jets development along the length of the mixing chamber.
length of the chamber can be appreciated in Fig. 21, which is a flooded version of Fig. 17. Figure 22 is a three-dimensional perspective of the mixing layers represented by the radial limits of the potential cores. Here one can appreciate the very important fact that the supersonic jets exit the nozzles rectangularly, and ultimately evolve to an almost circular cross-section form (see also Ref. [29]). The flattening at the bottom of the jets is an influence of the tunnel floor.

An important aspect that should be stressed in our findings is the fact that the mixing process was not complete up to the end of the mixing chamber, which is attested to especially by Figs. 16 and 20. This might have an impact on the functioning of the first diffuser, because part of the flow at its inlet would be supersonic. Two extra lines of work present themselves now in this research project. The first is to investigate the flow at the entering region of the diffuser considering the mixed supersonic/subsonic inlet conditions. The second is an effort in the direction of eliminating some of the simplifying assumptions in order to assess whether they have any influence on the retarding of the mixing process. We are very confident in the accuracy of the code developed, and believe at this point that the length of the mixing chamber is too short. One should not forget that, as we have stressed above, the injection system was adapted to an already existent (the basic conceptual design of the tunnel was already frozen) transition chamber. But with the available numerical tool we now have on hand, it is not a difficult task to predict what the minimum chamber length for a complete mixing should be.

4.4 Performance of the Mixing Process. We now present the engineering parameters' numerical values for the injection operating at the design condition. Tables 1 and 2 summarize the main data for both flows at the start and end of the numerical simulation. The calculated value of $K$ turned out to be 0.40. This is a high figure, especially when compared to 0.26, the value of the loss factor at the test section [11,12] (considering a standard installed model). But one should remember that the mixing process is extremely turbulent. Notwithstanding this, at the design point, the gain resulted in $\lambda = 1.085$, i.e., there is a real gain, and, therefore, a definite advantage in using the injection concept. The calculated injection efficiency resulted is $\eta = 0.67$.

5 Conclusions

The flow in the injection chamber of a transonic wind tunnel, for conditions designated as the design point, was successfully simulated. A new finite-difference computer code, that incorporates an assortment of very powerful numerical tools, was developed, and proved to be extremely robust, reliable, and accurate. The physics of the three-dimensional mixing process, a very involved situation, was properly investigated. This is attested by the finding of the compression/expansion domes, the verification of the maps of the mixing layers, and the calculation of the parameters that characterize the injection process, among many other important results. In spite of the pressure losses, a consequence of the high turbulence activity at the mixing between high-speed streams, the operation presented a positive gain. An important aspect that should be stressed in our findings is the fact that the mixing process was not complete up to the end of the mixing chamber, which is confirmed especially by Figs. 16 and 20. This might have an impact in the functioning of the first diffuser.

Acknowledgment

The authors would like to express their gratitude to CNPq, the Brazilian National Council of Research and Development, for the partial funding of this research, under Grant No. 302863/2004-4.

Table 2 Results of the numerical simulation at entrance and exit sections

<table>
<thead>
<tr>
<th></th>
<th>Entrance</th>
<th>Subsonic</th>
<th>Supersonic</th>
<th>Exit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stagnation pressure (kPa)</td>
<td>547.2</td>
<td>101.4</td>
<td>113.2</td>
<td></td>
</tr>
<tr>
<td>Specific entropy (kJ/kg K)</td>
<td>150</td>
<td>676</td>
<td>633.0</td>
<td></td>
</tr>
<tr>
<td>Mass flow (kg/s)</td>
<td>2.76</td>
<td>20.00</td>
<td>22.90</td>
<td></td>
</tr>
<tr>
<td>Stagnation pressure ratio$^a$</td>
<td>5.40</td>
<td>1</td>
<td>1.12</td>
<td></td>
</tr>
<tr>
<td>Mass flow ratio$^a$</td>
<td>1</td>
<td>7.27</td>
<td>8.30</td>
<td></td>
</tr>
</tbody>
</table>

$^a$Values at the subsonic stream taken as reference.

Nomenclature

- $a =$ sound speed (m/s)
- $\bar{a} =$ average of freestream speeds of sound, $(a_1 + a_2)/2$ (m/s)
- $A =$ cross sectional area (m$^2$)
- $b =$ mixing layer thickness between transverse locations where $x = (u_1 - 0.1 \Delta u)$ and $\bar{x} = (u_2 + 0.1 \Delta u)$ (m)
- $CFL =$ number of Courant, Friedrichs, and Lewy
- $c_p =$ specific heat at constant pressure (kJ/kg K)
- $h =$ specific enthalpy (kJ/kg)
- $H =$ height of tunnel section (m)
- $i =$ computational field node counter
- $K =$ pressure loss coefficient
- LES = large eddy simulation
- $M =$ Mach number
- $M_r =$ relative Mach number, $= \Delta u/\bar{a}$
- $m =$ mass flow (kg/s)$^a$
- $n =$ total number of cells in a cross section
- PTT = pilot transonic tunnel
- $p =$ static pressure (Pa)
\[ q = \text{dynamic pressure (Pa)} \]
\[ R = \text{gas constant (J/kg K)} \]
\[ \text{Re} = \text{Reynolds number} \]
\[ s = \text{specific entropy (J/kg K)} \]
\[ T = \text{static temperature (K)} \]
\[ u = \text{local mean streamwise velocity (m/s)} \]
\[ \Delta u = \text{freestream velocity difference, } u_1 - u_2 \text{ (m/s)} \]
\[ x = \text{streamwise coordinate (m)} \]
\[ y = \text{vertical coordinate (m)} \]
\[ z = \text{lateral, or spanwise, coordinate (m)} \]
\[ \alpha = \text{rotation flow angle (deg)} \]
\[ \gamma = \text{specific heat ratio} \]
\[ \delta = \text{boundary layer thickness (m)} \]
\[ \eta = \text{injection efficiency} \]
\[ \theta = \text{flow deflection angle due to shock wave (deg)} \]
\[ \lambda = \text{injection gain} \]
\[ \phi = \text{mixing chamber wall angle (deg)} \]

Subscripts
\[ 0 = \text{stagnation condition} \]
\[ 1,2 = \text{high and low speed, respectively} \]
\[ i,e = \text{inlet and exit cross section stations, respectively} \]
\[ n = \text{generic cell} \]
\[ sh = \text{shock impinging position at the flat plate} \]
\[ w = \text{wall conditions} \]
\[ \delta = \text{boundary layer thickness} \]
\[ \infty = \text{free stream value} \]

References


[12] Sverdrup Technology Inc., 1989, "Brazilian Transonic Wind Tunnel Concept Definition Study," Contractor Report for TTS and TTP Projects, CTA-IAT, Aeronautics and Space Institute, São José dos Campos, SP, Brazil.


