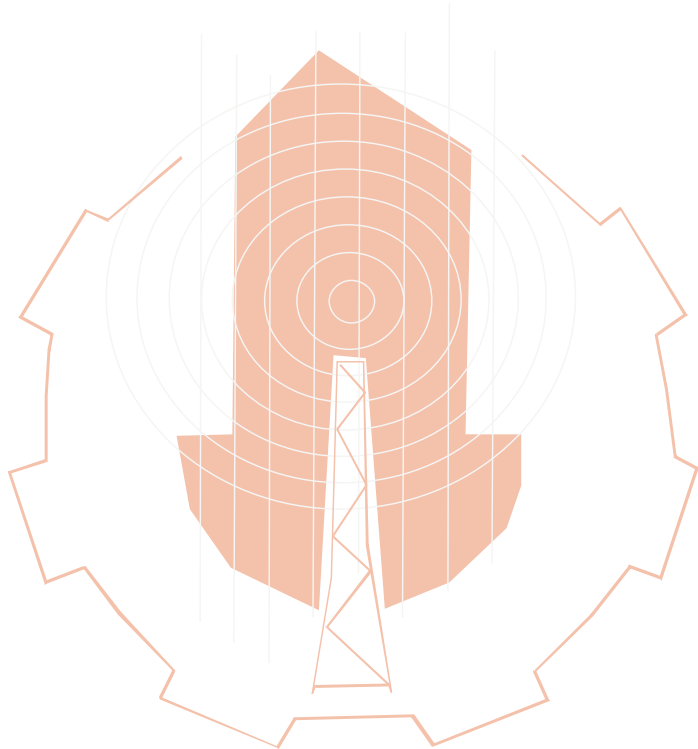




JOURNAL OF ENGINEERING RESEARCH

Refereed and issued twice annually by the
Faculty of Engineering - University of Tripoli



Issue 29 March 2020

A MOC-BASED SOFTWARE; DESIGN AND EVALUATION OF CONVERGING DIVERGING NOZZLES

Mohammed K. Elshebani and Elhadi I. Dekam *

Ministry of Industry, Tripoli, * Faculty of Engineering,
University of Tripoli, Libya.
E-mail: moh77712@yahoo.com and e.dekam@uot.edu.ly

الملخص

تصف هذه الورقة طريقة لتصميم الفوهات فوق الصوتية وتقييم سلوك سريان الهواء الداخلي خلال الفوهات بعدد 3 ماخ. وذلك للسريان التثائبي الأبعاد، والمستقر، والإيزنتروبي، وغير الدوراني خلال الفوهات المتماثلة مع الأجزاء الإضافية لتقليل الضغط التفاضلي المطلوب. تم توظيف طريقة الخواص مع استخدام الصيغة الجديدة لدالة براندتل-مير (Prandtl-Meyer). وتم تقييم النموذج الرياضي المتعلق بالتصميم، ووضع برنامج حاسوبي لتحقيق أهداف هذه الدراسة. وقد تم استخدام سبعة خطوط موجة ضغط لتحديد عقد الشبكة وتعيين خصائص التدفق لحالة التصميم. تم تحديد منحنى الجدار لعدد من مساحات العنق المختلفة، وتم الحصول على الصيغة الدنيا المربعة. تم توثيق النتائج المختلفة، بما في ذلك مجالات السرعة والكثافة، وتوزيعات الضغط ودرجة الحرارة، وتغيرات عدد ماخ داخل الفوهة المصممة. وقورنت النتائج الحالية مع مخرجات برنامج "الفلوننت"، ووجد أنهما يتفقا إلى حد كبير مع بعضهما البعض. كانت نسبة المساحة المقطعية للمخرج إلى المساحة المقطعية الحرجة 2.391 لتحقيق عدد ماخ المستهدف.

ABSTRACT

This paper describes supersonic nozzle designing procedure and evaluates the behaviour of the internal airflow through 3-Mach number nozzles. This is done for a two-dimensional, steady, isentropic, and irrotational flow through axis-symmetrical nozzles with extended parts to reduce the required differential pressure. The characteristic method technique is employed with the use of the new formula of the Prandtl-Meyer function. A design-evaluation related mathematical model is achieved. A computer program was written to achieve the objectives of the study. Seven pressure wave lines are considered where the grid points are located and the design condition flow properties are determined. Wall contour is specified in several different throat areas, where the least-square based formula is obtained. Different results are documented, including velocity and density fields, pressure and temperature distributions, and Mach number variations inside the designed nozzle. The present results are compared with the outcome of the FLUENT software. They are found to be highly agreeable with each other. The computed cross-sectional area ratio of the exit to the critical cross sectional area is found to be 2.391 for achieving the desired exit Mach number.

KEYWORDS: The Characteristic Method; Pressure Wave Lines; Supersonic Nozzle Design; Mach Number Nozzles.

INTRODUCTION

Internal compressible flow along the passages is usually affected by several different conditions; shear stress, heat exchange, cross-sectional area change, and shock

and expansion waves [1]. Most of the actual applications are associated with such conditions. One of the typical applications is employing the C-D nozzles for creating the desired supersonic gas flow in wind tunnels for testing, pipeline gas flow, aircraft engines, space vehicles, missile engines, or material coating, etc. The flow performance of such applications depends heavily upon the design of the gas exhaust nozzle [2].

Many published articles indicated that gas flow through nozzles, under the assumed conditions of an isentropic flow, is a good approximation to the actual flow situation [3,4]. Nozzles serve two primary functions. Firstly, they must control the pressure distribution to provide the correct and optimal system performance. This is accomplished through cross-sectional area variations. Secondly, they must efficiently convert the pressure energy to kinetic energy leading to achieve high exhaust velocity. This is done through efficiently expanding the exhaust gases to the available back pressure [2-4].

LITERATURE SURVEY

Meyer and Goldstein reviewed the state of the development of the method of characteristics (MOC) for the solution of problems of compressible flow [5]. In 1931, Busemann gave a construction for a nozzle to convert 2-D divergent radial supersonic flow into a high Mach number uniform supersonic flow [6]. The first high-speed wind tunnel was built by the National Advisory Committee for Aeronautics (NACA) in 1932. In the early 1940s, engineers at the Langley Memorial Aeronautical Laboratory were conducting all of their supersonic research on a very small nine-inch wind tunnel. The engineer Robert Jones, in 1945, suggested that sweptback wing geometries would minimize drag, and a larger tunnel becomes necessary for continuing technological advancement [7].

Puckett, in 1946, used this method in supersonic nozzle design and made the technique more accessible to engineers [8]. A nozzle design which does not directly employ the MOC was evolved by Friedrichs [9]. This method was expanded by Nilson and used by Baron as a basis for the design of a set of nozzles [10,11].

Puckett pointed out that smaller expansion angles than the maximum are possible. The resulting nozzle was long, but had a more uniform final flow [8]. Puckett suggested the possibility of assuming a uniform radial flow at the maximum slope cross-section. Only the flow region downstream of the inflection point need then be calculated, and a smooth curve could be faired back of the throat from the inflection point. Using this approach an analytical expression for the contour downstream of the radial flow region was found by Foelsck [12]. Atkin converted the radial flow into the uniform parallel supersonic flow at the desired Mach number with the 'Busemann' contour. This approach was adopted by Beckwith and Moore [13]. They started with a straight sonic line at the throat and improved the method by computing the contour between the throat and the inflection point using the MOC.

Practical requirements concerning nozzle handling and time delays during test periods have brought a fourth method which permits a quick change from one Mach number to another. One of the developments, that was achieved, that is the production of variable-Mach number nozzle jack-supported flexible steel sheets. However, an incompatibility between the aerodynamic and structural requirements is introduced if a point of discontinuity curvature exists at the inflection point. This has led to the development of continuous-curvature nozzles which are necessary for longer nozzle lengths.

Riise, Rosen, and Sivells [14-16] concluded that even with solid nozzles, it is convenient to have a continuous curvature of the walls. By this means, the rate of the growth of the boundary layer is kept continuous, which makes it easier to correct the inviscid contour for boundary-layer growth and, therefore, helps to produce a more uniform flow in the test section. Optimization techniques used to design rocket nozzle contours have been utilized since 1955, as discussed by Guderley and Hantsch. Rao simplified the analysis along with a design technique that is widely used today [17]. Allman and Hoffman recognized the need to include the boundary layer loss and base pressure [18]. Beckwith and Moore presented a procedure for designing two-dimensional nozzles. The streamline coordinates were computed directly from tabulated flow parameters and appropriate equations. The MOC was used with the Foelsch equations. Any two of three parameters; length-to-height ratio, Mach number, and wall angle can determine the desired nozzle [13].

Zebbiche and Youbi gave a review of the development of the MOC used for the analysis of the compressible flow. They gave a credit to the method that represents a good approximation in a large number of practical applications [19]. Khanet et al. calculated the minimum length of the supersonic nozzle for the optimum Mach number at the nozzle exit with the uniform flow at the diverging section of the nozzle by developing a MATLAB program [20]. The design parameters of the supersonic nozzle depend directly on the stagnation parameters of the combustion chamber. Salhi et al. worked on determination of the effect of stagnation pressure on the thermodynamic, physical and geometrical supersonic flow parameters to find a general form for real gas flow [21].

Sarah worked on the design and validation of a Mach 3 wind tunnel nozzle contour. The nozzle contour was designed with the CONTOUR code developed by Sivells. The program based on a combination of analytical solutions and the MOC. The flow field of the finalized nozzle contour was studied with the use of ANSYS FLUENT [22]. Zhanget et al. designed a supersonic minimum length nozzle (MLN) which was made especially for laser-assisted oxygen cutting (LASOX). The divergent section of the MLN was designed based on the MOC. Mild steel plates in 20 mm thickness were cut to investigate the effect of cutting parameters on the LASOX cut quality of two nozzles [23]. White et al. analyzed the supersonic flow of organic fluids within 2-D C-D nozzles. Two nozzles are developed using the minimum length MOC design model, where they were designed for R245fa and Toluene as working fluids with nozzle exit Mach numbers of 1.4 and 1.7, respectively [24].

Zebbiche developed a calculation model based on the MOC to study the effect of the stagnation pressure of the combustion chamber on the 2-D and axisymmetric minimum length nozzle design giving a uniform and parallel flow at the exit section. The effect of molecular vibration was considered in the model to evaluate the behaviour of a gas at a high temperature. The comparison is made with high temperature and perfect gas models [25]. To improve the aerodynamic performances compared to the minimum length nozzle, Hamaidia et al. developed new contours of axisymmetric supersonic nozzles giving a uniform and parallel flow at the exit section. Their forms include cylindrical central bodies and external walls for the flow redress. According to the computational fluid dynamics verification made, the flow separation was reduced for the proposed nozzle compared to the minimum length nozzle [26].

In this study, a supersonic desired nozzle is designed and airflow behaviour is evaluated. The nozzle desired exit Mach number is 3 [27]. A detailed mathematical model is presented. The MOC is employed to design the nozzle and the FLUENT software is

used to study the flow. A computer subroutines are written and a software is built to accommodate different exit Mach numbers and different differential nozzle pressure ratios. The design conditions are well defined below.

OBJECTIVES

The objective of this research includes building the appropriate mathematical model for the considered case. Create a new software that deals with the derived mathematical model for the gas dynamics in supersonic nozzles. This software can design such nozzles for different gases and under different operating conditions. Comparing the results with the use of the FLUENT, and find out the main conclusions and recommendations.

MATHEMATICAL FORMULATION

The mathematical model deals with the behaviour of the steady axis-symmetrical frictionless compressible flow along the nozzle, to determine the required wall-contour specifically along the diverging section of the desired nozzle. This process is done considering a fully-insulated C-D nozzle. The employed MOC and characteristic curves are first introduced.

The Method of Characteristics

The physical conditions of a 2-D, steady, isentropic, irrotational flow can be expressed mathematically by the Nonlinear Differential Equation of the velocity potential [1,27]. The MOC is a mathematical formulation that can be used to find solutions to the aforementioned velocity potential, satisfying given boundary conditions for which the governing Partial Differential Equations (PDE) become Ordinary Differential Equations (ODE). The ODE only holds along a set of curves known as characteristic curves, which will be discussed in the next section. As consequences of the special properties of the characteristic curves, the original problem of finding a solution to the velocity potential is replaced by the problem of constructing these characteristic curves in the physical plane.

The method is founded on the fact that changes in fluid properties in supersonic flows occur across these characteristics, and are brought about by pressure waves propagating along the Mach lines of the flow, which are inclined at the Mach angle to the local velocity vectors. The MOC is a numerical procedure appropriate for solving the 2-D compressible flow problems. By using this technique, flow properties can be calculated at distinct points throughout a flow field. The MOC, that is implemented in the computer algorithms, is an important element of supersonic computational fluid dynamics software. These calculations can be executed with the aid of spreadsheet programming or technical computing software.

Characteristics

Characteristics are 'lines' in a supersonic flow oriented in specific directions along which disturbances or pressure waves are propagated [1]. Anderson and others describe three properties of characteristics; firstly, a characteristic in a 2-D supersonic flow is a curve or line along which physical disturbances propagate at the local speed of sound relative to the gas [1,3,4]. Secondly, a characteristic is a curve across which flow properties are continuous, although they may have discontinuous first derivatives, and along which the derivatives are indeterminate. Thirdly, a characteristic is a curve along which the governing PDE may be manipulated into an ODE.

While the ratios of duct areas are relatively straightforward to determine, based on desired test-section Mach numbers and tunnel run times, determining an optimum channel contour is slightly more complicated. It is in the region immediately after the sonic throat where the flow is turned away from itself that the air expands into supersonic velocity. This expansion happens rather gradually over the initial expansion region. For a Prandtl-Meyer expansion scenario, it is assumed that the expansion takes place across a centered fan originating from an abrupt corner. This phenomenon is typically modelled as a continuous series of expansion waves, each of which is turning the airflow by an infinitesimal amount. The CFD is one of the techniques used to obtain the numerical solution to fluid flow and heat transfer problems [2].

The Basics of the Mathematical Formulation

The flow at the nozzle throat and exit sections are considered to have the same direction. The ratios of the flow properties at any section along the nozzle, to those properties in the critical condition, are to have remained always valid. They are taken into account to evaluate the numerical calculations and to compare the results based on the derived present model with those results achieved through the available literature theory. The flow-related calculations along the nozzle are rather delicate since the nozzle shape is unknown a priori.

The main required contour of the wall along the nozzle, or what so-called the central body, is that concerning the diverging section, in which the flow accelerates from Mach number $M = 1$ at the throat section up to the design Mach number M_E at the exit section. The flow has an angle that is defined as θ^* at the throat section. The flow through the central body keeps redress itself in the diverging section, along the passage from $\theta = \theta^*$ at the throat to $\theta = 0$ at the exit section of the nozzle. The flow-based calculations and the contour determination of the central body for a supersonic flow are done here for a steady, non-rotational, 2-D of a calorically perfect gas. According to these assumptions, the MOC gives the following characteristic equations [1];

$$\frac{dy}{dx} = \tan(\theta - \mu) \quad (1)$$

$$\frac{dy}{dx} = \tan(\theta + \mu) \quad (2)$$

Where x and y are the spatial coordinates, θ is the angle of the flow streamline measured from x , and μ is the Mach angle measured from the flow direction. A graphical interpretation of equations 1 and 2 is given in Figure (1), where the streamline makes an angle θ with the x -axis.

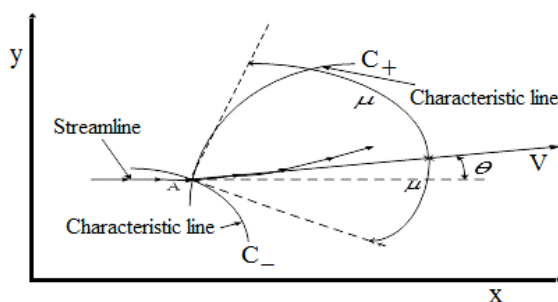


Figure 1: Left and right-running characteristic lines through point A.

Two characteristic lines are passing through point A, one at the angle μ above the streamline, and the other at the angle μ below the streamline. In the real case, the characteristic lines are curved. However, when the grid is fine, so that the points are close one to the other, one can represent the curves by straight lines. Hence, this work is based on the straight lines of Mach flow.

The characteristic line which is given by the angle $(\theta-\mu)$ is called a C^- characteristic; where it is a right-running characteristic analogous to the C^- . The characteristic that is given by the angle $(\theta+\mu)$ is called a C^+ characteristic; where it is a left-running characteristic analogous to the C^+ . Along the C^- and C^+ characteristics, the Prandtl-Meyer functions are given as [1];

$$\theta + v(M) = \text{const} = K_- \quad (3)$$

$$\theta - v(M) = \text{const} = K_+ \quad (4)$$

Where $v(M)$ is the Prandtl-Meyer expansion function, and K is a constant. These two equations, 3 and 4, are called the compatibility equations, where each relates velocity magnitude to the flow direction along the characteristic lines. They have no terms involving the spatial coordinates x and y . Hence, both can be solved without requiring knowledge of the geometric location of the characteristic lines. This geometrical independence of the compatibility equations is unique only to the present case of the 2-D irrotational flow.

The angle v is measured compared to the direction of the velocity vector at the nozzle throat. Referring to Figure (2), the lines AC and AB represent the local Mach waves at the throat and exit sections, respectively. These lines are inclined by angles of μ_A and μ_E , respectively. These angles take the values of $\mu_A = 90^\circ$ at the throat, and $\mu_E = \text{arcsine}(1/M_E)$, which is less than 90° at the exit. Between these two Mach lines, there is an infinity of centered diverging Mach waves exiting from point A at the lip as indicated in Figure (2). Each line gives the local Mach number, from which one can easily deduce the point of the central body contour. Consequently, point A is the point of discontinuity in parameters and in particular in M and θ [27].

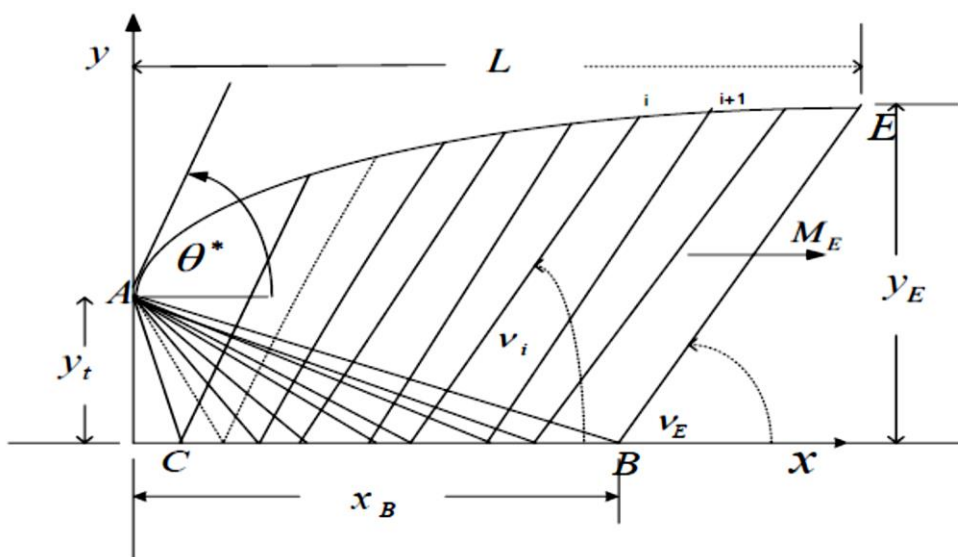


Figure 2: Parameters of an intermediate Mach line connecting the points.

Various flow properties; Mach number, flow deviation angle, and the thermodynamic ratios; pressure, temperature, and density, are all fixed along each Mach line exiting from point A. This characteristic gives a chance for determining quickly and explicitly the position of the wall points. Each Mach line that starts from point A is absorbed by the wall considering the 2-D flow.

Figure (2) presents the flow field, including the properties and parameters of the intermediate Mach lines connecting point A with a corresponding i^{th} point along the wall. There are four characteristic flow regions including; a uniform flow region, 2-simple flow regions, and a non-simple flow region, which all are illustrated in Figure (3). The calculation procedure is composed of a series of fundamental computational processes that execute the defined characteristic Equations in each of the four named regions.

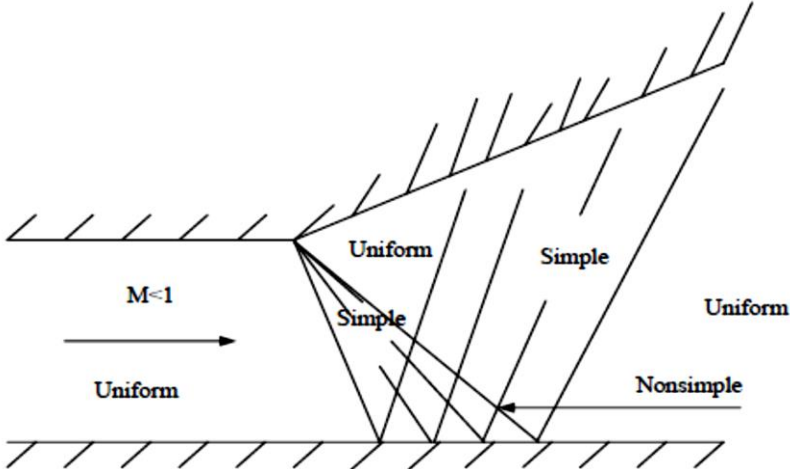


Figure 3: The four characteristic regions

Determination of the Contour

Now, referring to Figure (3), the seven considered wave lines are drawn, where the number equals the famous number of lines used in literature. Employing the characteristic Equations along each wave line, one can trace the different points in the grid. As indicated in Figure (4), we may see points 1 to 8 are all traced along the first inclined line, while the second line carrying points 9 to 15, and so on.

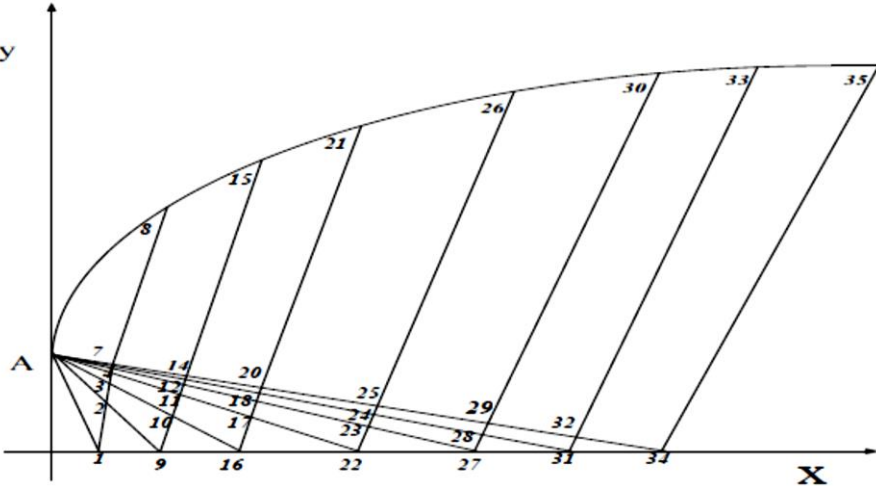


Figure 4: Wave grid-lines index

Points 8, 15, 21, 26, 30, 33 and 35 represent points on the wall contour, while points 1, 9, 16, 22, 27, 32 and 34 are traced along the nozzle x-centerline. As mentioned above, each characteristic line has a constant value either K_+ or K_- .

Kernel's Region and the Initial Point

The location of point A is defined as the last point on the first expansion region defined by θ^* . Since this is a 2-D solution, the nozzle centerline acts as the axis of symmetry. The resultant characteristic mesh, or Kernel's region represented by the line ACB in Figure (2), and also it is shown in Figure (5).

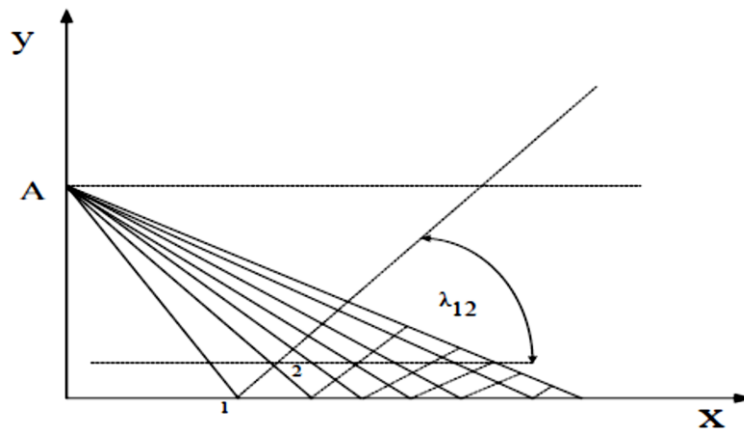


Figure 5: Calculation processes of the characteristics in the Kernel's region.

The flow-related calculations in the Kernel's region begin at point A in Figures (2) and 5. The initial expansion angle θ^* is related to the desired M_E through the following relations [1];

$$\theta^* = \frac{v(M_E)}{2} \quad (5)$$

$$v(M) = \sqrt{\frac{\gamma+1}{\gamma-1}} \tan^{-1} \sqrt{\frac{\gamma-1}{\gamma+1} (M^2 - 1)} - \tan^{-1} \sqrt{M^2 - 1} \quad (6)$$

Now, dividing the flow expansion region, $0 \leq \theta \leq \theta^*$, we have;

$$\Delta\theta = \Delta v = \frac{\theta^*}{N-1} \quad (7)$$

The right wall contour downstream of the throat will be roughly presented. The calculation in the Kernel's region is presented in Figures (2 and 5). For the first point 1 in Figure (5), $y_1 = 0$ and θ_1 and v_1 are calculated from this relation;

$$\theta_1 = v_1 = \theta^* - \text{INT}[\theta^*] \quad (8)$$

Where $\text{INT}[\theta^*]$ is the integer value of the maximum angle of the throat nozzle. According to the grid, we go forward to determine the position x of point 1 as follows;

$$x_1 = \frac{y_t}{\tan(\mu_1 - \theta_1)} \quad (9)$$

$$\mu_1 = \sin^{-1}\left(\frac{1}{M}\right) \quad (10)$$

Internal Points

Let us start with the left-running characteristic line. Referring to Figure (5), begin to calculate the points 2-7 on the first left-running characteristic wave line. Now, we are going to determine the position of the points by using the following relations [1,2];

$$x_i = \frac{y_t - y_j + (x_i - x_j)\tan\beta_{ij}}{\tan\alpha_{ij} + \tan\beta_{ij}} \quad (11)$$

$$y_i = y_j + (x_i - x_j)\tan\beta_{ij} \quad (12)$$

$$\beta_{ij} = \frac{1}{2}[(\mu_i + \mu_j) + (\theta_i + \theta_j)] \quad (13)$$

$$\alpha_{ij} = \frac{1}{2}[(\mu_i + \mu_j) - (\theta_i + \theta_j)] \quad (14)$$

$$\alpha_i = \mu_i - \theta_i \quad (15)$$

$$\theta_i = \theta_{i-1} + \Delta\theta \quad (16)$$

$$v_i = v_{i-1} + \Delta v \quad (17)$$

$$\mu_i = \sin^{-1}\left(\frac{1}{M_i}\right) \quad (18)$$

Where $i=2,3,\dots,N$, and $j=i-1$.

Equations and Procedure for Internal Points

As a sample of the analysis of the internal grid points in the non-simple region, consider the three points 3, 9 and 10. These points lie on the same line of the C⁻ characteristic, where they have the same characteristic value, that can be presented in the following manner [1,2];

$$(K_-)_{10} = (K_-)_3 = \theta_3 + v_3 \quad (19)$$

$$(K_+)_{10} = (K_+)_9 = \theta_9 - v_9 \quad (20)$$

Where point 10 represents the intersection of the C^- with the C^+ characteristic lines. Now, at point 10, both characteristic values should be satisfied according to the following relations [1,2];

$$\theta_{10} + v_{10} = (K_-)_{10} \quad (21)$$

$$\theta_{10} - v_{10} = (K_+)_{10} \quad (22)$$

Solving both Equations, the properties, θ , and v , can be obtained as follows;

$$\theta_{10} = \frac{1}{2} [(K_-)_{10} + (K_+)_{10}] \quad (23)$$

$$v_{10} = \frac{1}{2} [(K_-)_{10} - (K_+)_{10}] \quad (24)$$

Now, all other flow properties at point 10 can be obtained [1]. According to the Prandtl-Meyer function, calculate the associated M_{10} , by which the static values, P_{10} and T_{10} , can be found. The speed of sound and thus the flow velocity can be computed. Refer to Figures (4 and 5), consider the two other important points, 14 and 15, where both points lie on the same C^+ characteristic line with the constant K_+ value. Thus;

$$(K_+)_{14} = (K_+)_{15} \quad (25)$$

This value can be obtained from the known properties at point 14 as follows;

$$(K_+)_{14} = (K_+)_{15} = \theta_{14} - v_{14} \quad (26)$$

Now, the flow properties at the wall point, point 15, can be obtained as follows;

$$(K_+)_{15} = \theta_{15} - v_{15} \quad (27)$$

In this Equation, $(K_+)_{15}$ and θ_{15} are known, thus v_{15} can be directly determined. Hence, all other flow properties at this point can be obtained, in the same manner, that was followed above for point 10. The characteristic line between points 14 and 15 is assumed to be a straight-line segment with an average slope of $\frac{1}{2}(\theta_{14} + \theta_{15}) + \frac{1}{2}(\mu_{14} + \mu_{15})$. The expansion section of the specified slopes; $\theta_8, \theta_{15}, \theta_{21}, \theta_{26}$, etc. is introduced in Figure (4). The characteristic mesh and flow properties at the associated grid points are then calculated [1,2], more details on the solution can be found in Elshebani [27]. Point 35 is a wall point on the straightening section of the contour. The purpose of the straightening section is to cancel the expansion waves generated by the expansion section. Therefore, there are no waves that would be reflected in the straightening section.

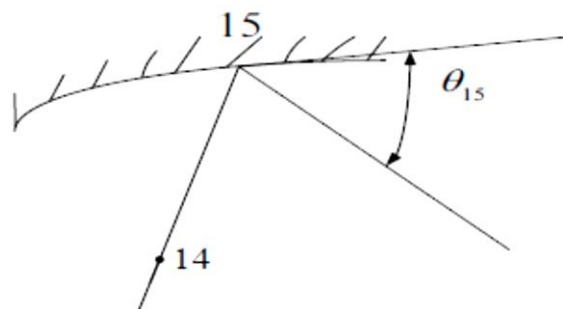


Figure 6: Wall Points

Along the centreline, the Mach number increases continuously. At point 34, it is assumed that the desired exit Mach number M_E is reached. The characteristic line from point 34 to 35 is the last line of the calculations. Again $\theta_{34}=\theta_{35}$, and the contour from point 33 to 35 is approximated with a straight-line segment with an average slope of $\frac{1}{2}(\theta_{33}+\theta_{35})$. The scheme of the calculation, concerning the internal points in the flow field, is illustrated in Figure (6). According to the right and left-running lines of Mach, the integrations of Equations 1 and 2 gives [1,2]:

$$y_{11} - y_9 = \lambda_{911}(x_{11} - x_9) \quad (28)$$

$$y_{11} - y_{10} = \lambda_{1011}(x_{11} - x_{10}) \quad (29)$$

$$\lambda_{911} = \tan\left(\frac{1}{2}(\theta_9 + \theta_{11}) - \frac{1}{2}(\mu_9 + \mu_{11})\right) \quad (30)$$

$$\lambda_{1011} = \tan\left(\frac{1}{2}(\theta_{10} + \theta_{11}) + \frac{1}{2}(\mu_{10} + \mu_{11})\right) \quad (31)$$

Equations 28 to 31 represent a system of nonlinear algebraic Equations with two unknowns (x_{11}, y_{11}) . The successive iteration algorithm is written as follows:

$$x_{11} = \frac{y_9 - y_{10} + \lambda_{1011}x_{10} - \lambda_{911}x_9}{\lambda_{1011} - \lambda_{911}} \quad (32)$$

$$y_{11} = y_9 + \lambda_{911}(x_{11} - x_9) \quad (33)$$

The relations 32 and 33 represent a system of Equations that can be employed in the calculation process through the use of iterations for the 2-D supersonic flow. The resolution of the system is done by the predictor-corrector algorithm [2].

The contour of the Nozzle Wall

The determination of the wall contour is done according to the process presented in Figure (7). The computing process has a recurrence continuation form. This calculation is an explicit procedure, that does not depend on the considered downstream results of the point. When the properties in a point are known, it is easy to determine such properties for the downstream adjacent point and go towards the next downstream points up to the exit section point. Here, the nozzle walls pass through point A, where we have $x_A=0$, $y_A=y^*=y_t$, and $M_A=M^*=1$ and $\theta_A=\theta^*$. The value of M^* is equal to the Mach number at point A right downstream of the expansion and corresponding to the Mach number at the first point on the last C- line.

The flow properties are employed in the calculation process for the determination of the position (x,y) of each wall point. Figure (7) presents a general view of the main considered points and parameters to determine the necessary Equations for obtaining the position of each wall point, P_i . The slope of the panel numbered $(i-1)$, between the points $i-1$ and i , is approximated by the angle α_{i-1} as illustrated in the above Figure. The position of point P_i can be determined by the following relations [27]:

$$x_{P_i} = \frac{y_i - y_{P_{i-1}} + x_{P_{i-1}} \tan(\alpha_{i-1}) - x_i \tan(\theta_i + \mu_i)}{\tan(\alpha_{i-1}) - \tan(\theta_i + \mu_i)} \quad (34)$$

$$y_{P_i} = y_{P_{i-1}} + (x_{P_i} - x_{P_{i-1}}) \tan(\alpha_{i-1}) \quad (35)$$

$$\alpha_{i-1} = \frac{1}{2}(\theta_{i-1} + \theta_i)$$

More details are given in Elshebani [27], where he illustrated the technique used in the calculation of the whole characteristics and properties.

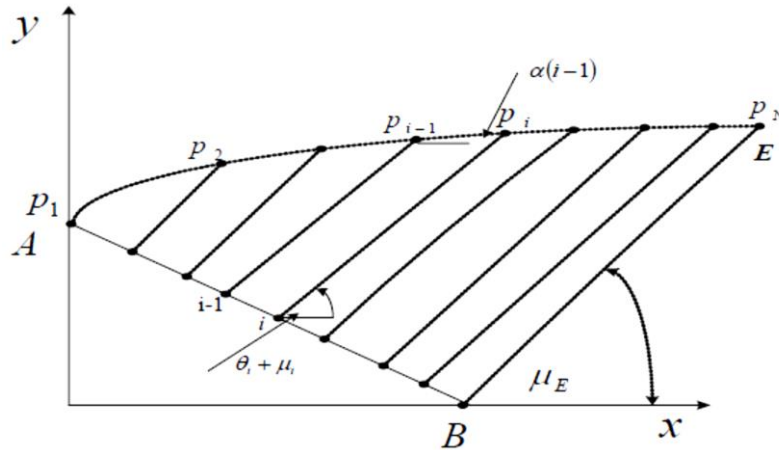


Figure 7: Presentation of points and parameters employed in the determination of wall points.

METHODOLOGY

Different stages of the calculations, concerning the geometry of the nozzle body and the behavior of the compressible flow through the desired C-D nozzle, are going to be described. Computer software subroutines are written and developed, where the logic used in the development of such software and its features are well presented elsewhere [27]. In the present design, it was decided to follow Foelsch, Beckwith and Moore and others in prescribing part of the supersonic flow field to be a region of radial flow. It was considered an advantage, however, to prescribe this region to lie within the nozzle to avoid straight segments in the walls. The present design may be considered to be a modification of Beckwith and Moore's method; accounts being taken for the more recent techniques. Using the MOC, the software was prepared to design a series of geometrically different C-D nozzles with specified exit Mach numbers and characteristics.

The first stage deals with the preparation of the input database required. The input data represent the desired exit Mach number, the throat height, the number of the characteristic lines employed, the considered gas properties with the stagnation temperature and pressure. Many graphs have been drawn using the "Solid Works", with the aid of "Microsoft Excel" as a calculation tool. The second stage deals with writing the code of the design package that uses these input data. The expected output results represent the nozzle geometry and the characteristics of the flow inside the studied 2-D C-D nozzle under the given conditions for each scenario taken.

The MOC is employed, step-by-step, to construct graphically the nozzle flow model [5]. The x-y coordinates of these graphs have been stored in the unit matrices in the database and then they exported to a Visual Basic code. The Visual Basic code has been

developed to receive the operating conditions from the user and then scaling the unit model to match these conditions. From the available database according to the desired Mach number, the contour of the x-y profile of the nozzle can be plotted using the final array which represents all points along the wall of the desired nozzle.

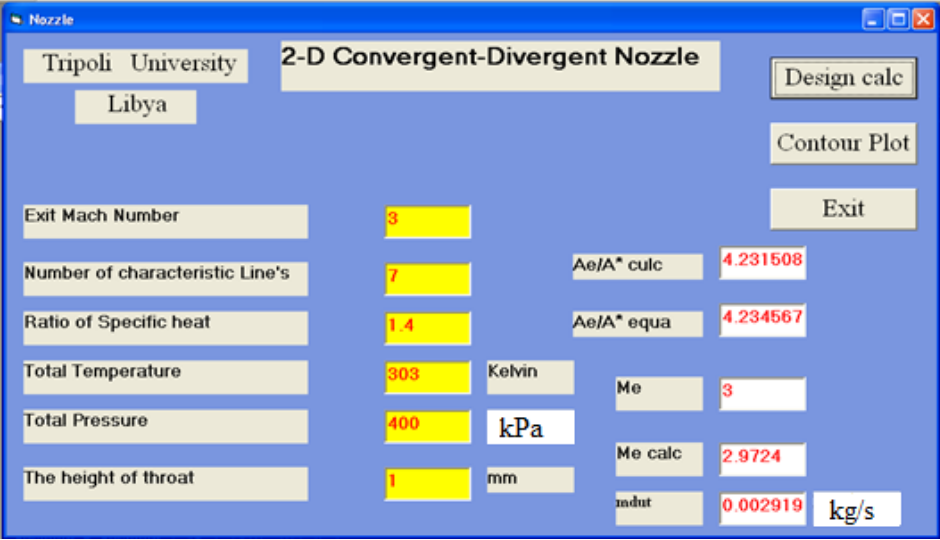


Figure 8: The main window of the software package.

SAMPLES OF SOFTWAREWINDOWS

Figure (8) represents the main window of the developed windows software including the input data and the output required results [27]. Figure (9) presents a window of nozzle wall contour plot.

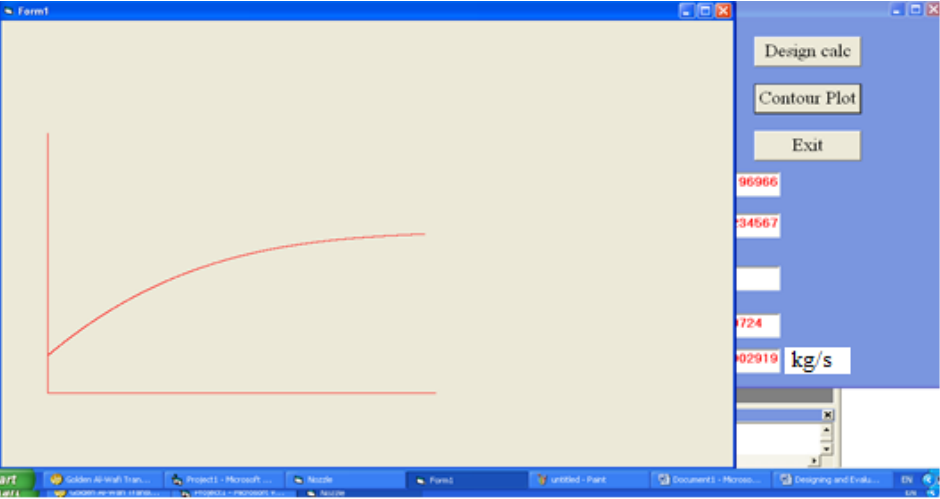


Figure 9: The window for the nozzle wall contour.

For different desired exit Mach numbers, contours were calculated and drawn for several specific Mach numbers for a unit throat height. These graphs represent the unit model of the nozzle for each value of such Mach numbers. Figure (10) shows the unit models for six different values of exit Mach numbers ranging from 1.5 to 4.

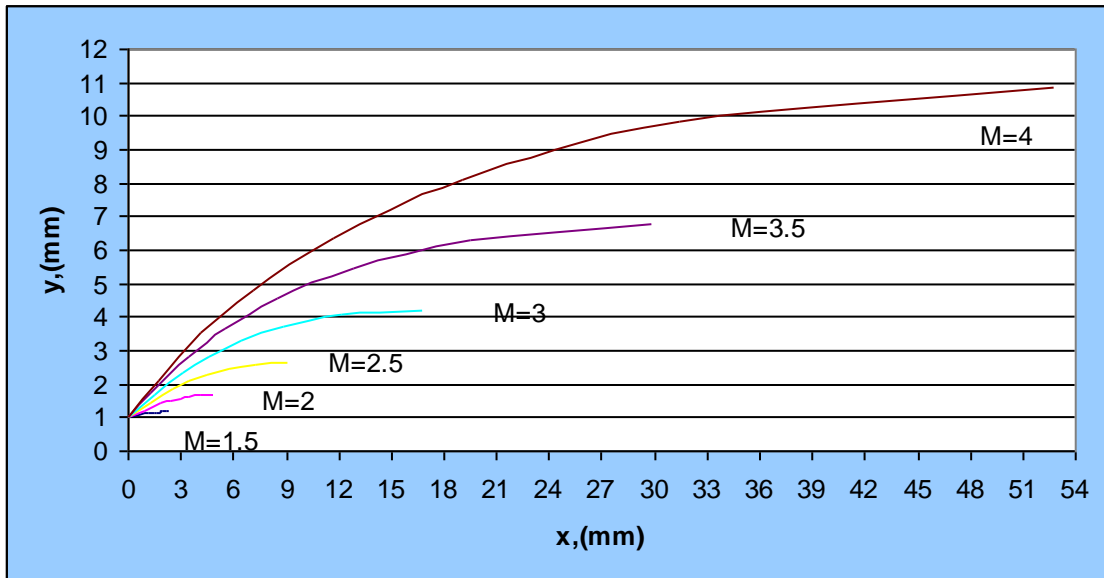


Figure 10: The unit models for six different values for the desired exit Mach numbers.

The coordinate points are described with a curve-fit governor Equation defining the wall contour of the nozzle. The high quality and high accuracy of CNC-type machines lead to the effective production of such nozzle models. To obtain the desired Equation, the well-known Least Square Method for nonlinear relations is used to find the polynomial of the third degree. This polynomial has the following proposed form;

$$y = a + a_1x + a_2x^2 + a_3x^3$$

By using the same developed computer program, the coefficients of this Equation; a, a1, a2 and a3 are determined. Figure (11) shows the obtained Equation for 7 characteristic lines n, where the radius of the nozzle throat is $y_t=1$ mm, and the exit Mach number is $M= 3$.

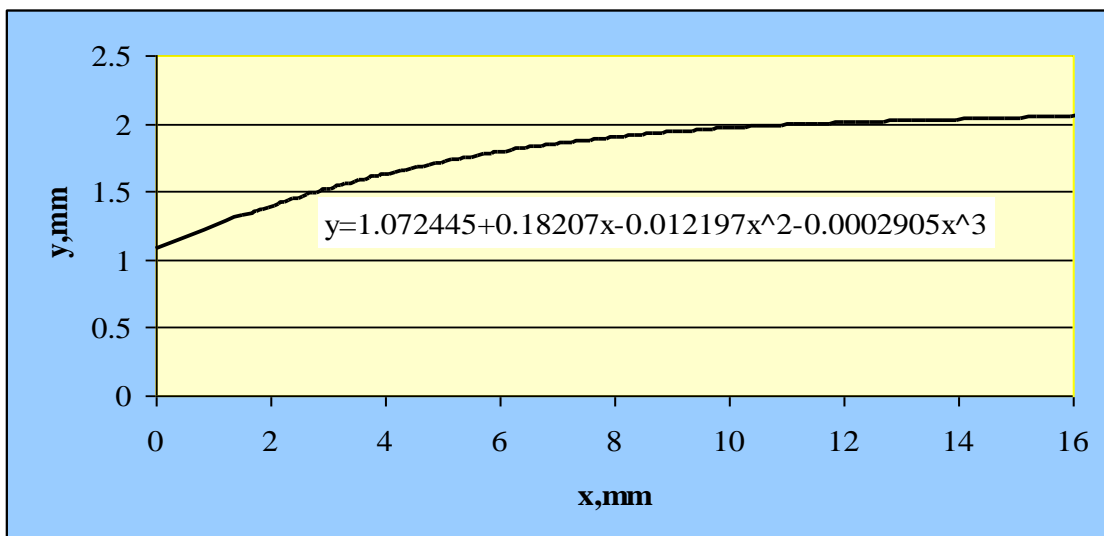


Figure 11: The Contour Curve-Fit Equation with the least square method.

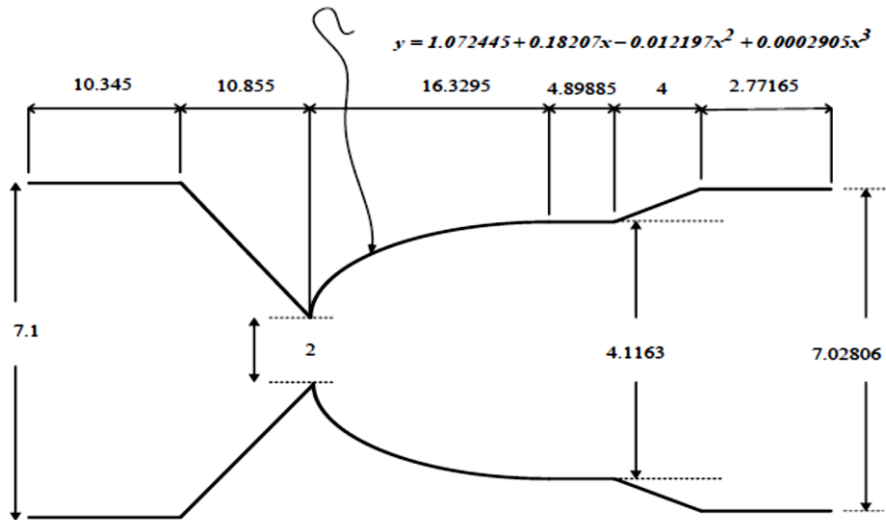


Figure 12: Dimensions of the considered C-D nozzle.

NOZZLE BEHAVIOUR WITH FLUENT SOFTWARE

Solver-FLUENT 5/6 is a computational fluid dynamics software. GAMBIT is used with "QUAD" mesh-type generation module. GAMBIT has a single interface for the geometry creation and meshing [27]. The air flow characteristics are determined using the CFD techniques. Figure (12) shows the geometry characteristics of the desired C-D nozzle. Different parameters are obtained, including Mach number, velocity, density, pressure, and temperature of the airflow along the axis of the considered nozzle.

CHARACTERISTICS OF THE C-D NOZZLE

The boundary conditions applied are represented in the inlet pressure, outlet pressure, wall contour and the condition of symmetry. The viscous model was defined for the airflow, where solver-FLUENT 5/6 and mesh-type "QUAD" were employed [27]. Table (1) summarises the considered given parameters. The iterations were set to be about 1000, where the software runs until the solution converges.

Table 1: Parameters considered for C-D Nozzle.

No.	Nozzle Parameters	Values
1	Inlet Diameter	7.1mm
2	Outlet Diameter	4.1163 mm
3	Throat Diameter	2 mm
4	Inlet Pressure	400kPa
5	The temperature of the gas at the inlet	303 K
6	Exit Mach	3
7	Boundary condition applied at the inlet	inlet pressure
8	Boundary condition applied at the outlet	outlet pressure
9	Boundary condition applied at the axis	symmetry
10	Boundary condition applied at the nozzle wall	wall
11	Atomizing gas used	air

The final form of the supersonic C-D nozzle is shown in Figure (13), where the constant-area of the duct extension is considered to be the test section in the proposed rig. This arrangement reduces the required pressure differential. Figures (10 to 13) identify

the solver feature, fluid properties, nozzle-operating conditions, flow mode, operating boundary conditions, and flow outlet pressure.

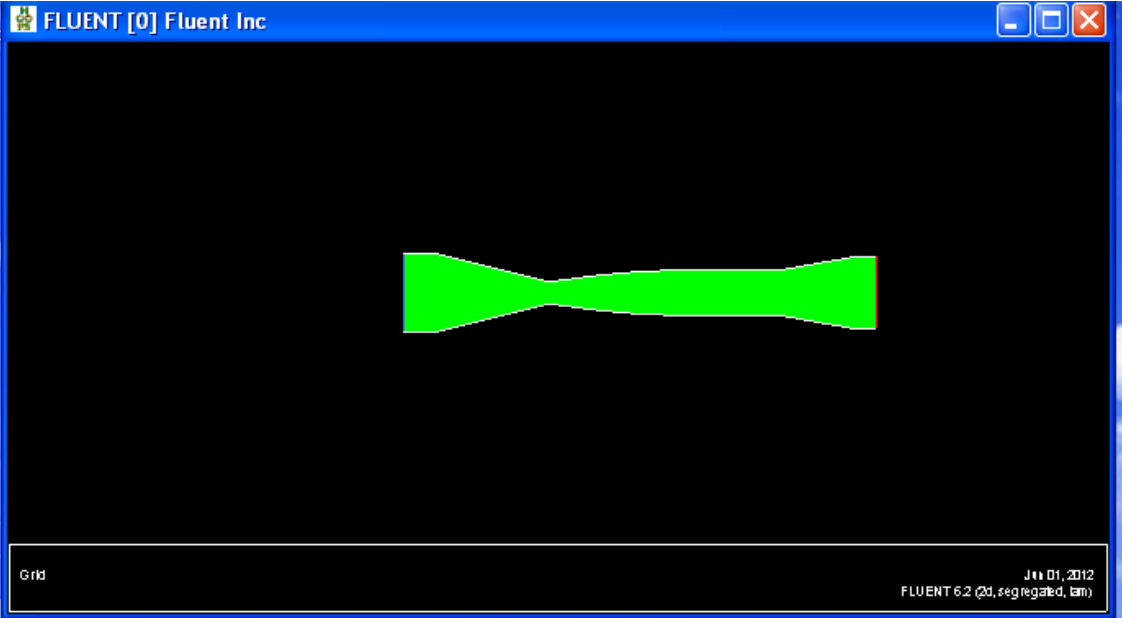


Figure 13: The final geometry of the designed C-D nozzle.

THE MAIN RESULTS

Referring to the isentropic process, the gas mass flow rate should be determined by the inlet stagnation conditions (P_o , T_o) and the nozzle throat area A^* [1]. The linear relationship between mass flow rate and throat area is shown in Figure (14). The thrust produced by the nozzle at any moment can be determined by the use of the well-known momentum integral Equation [1,27].

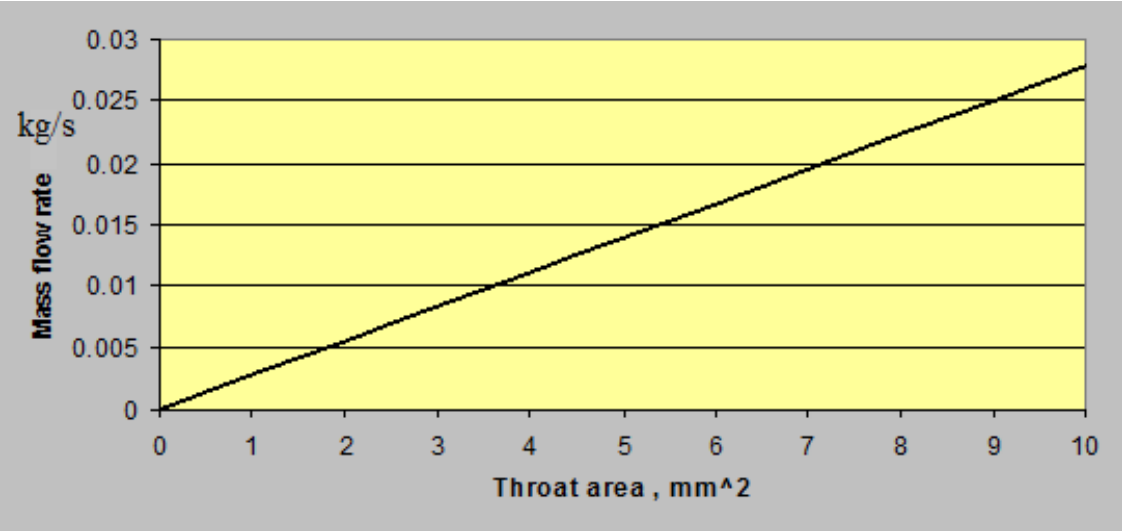


Figure 14: The mass flow rate relationship with the throat area.

The pressure ratio, P_o/P_b , may be defined as the ratio of the source stagnation pressure to the back static pressure that should be available to feed the nozzle with the

suitable flow to obtain the desired Mach number at the nozzle exit. This ratio could differ according to the nozzle geometrical and operating conditions [2].

The efficiency of nozzles is defined in different ways to give several indicators, depends primarily on the nozzle applications. They are used as pressure drop developers in the thermal engine ejectors or as thrust producers for the propulsion systems, or as kinetic energy generators in the firefighting systems. The most efficient process of converting high pressure and high temperature into kinetic energy is to expand the flow isentropic ally over a given pressure ratio. The real expansion processes in nozzles occur with little friction, and hence with a little increase in entropy [3,4].

Referring to the perfect gas Equation and the isentropic relations, the ratio of the cross-sectional area along the nozzle to the throat cross-sectional area can be obtained as a function of the local Mach number, as presented in Figure (15). For supersonic flows, it is clear that M increases as the ratio, A/A^* , increases, where for an exit Mach number of 3 the area ratio is 4.2345.

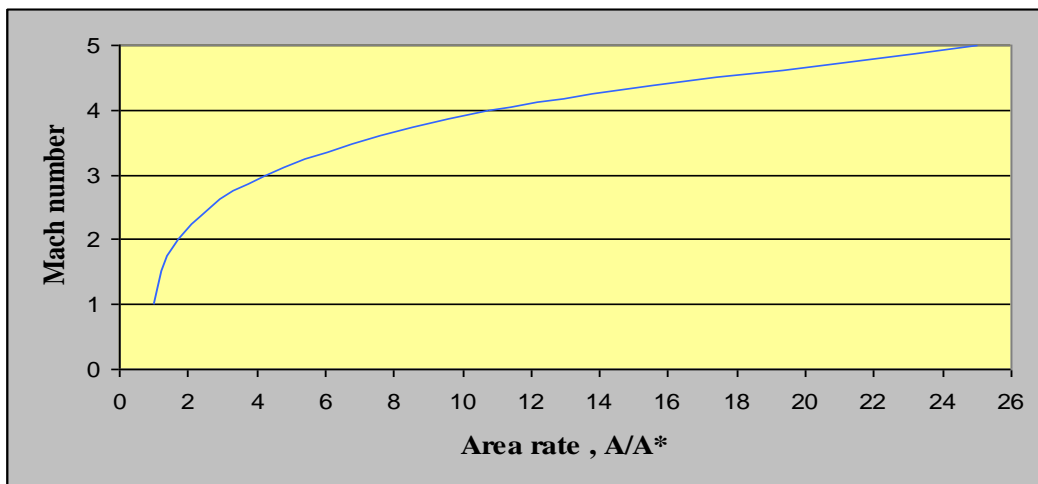


Figure 15: Area-Mach relationship for the supersonic nozzle diverging section.

The present program leads to detailed geometry and operation-related results. These are collected in Tables that show geometry characteristics and flow properties in each point of the considered grid [27]. These include characteristic line constants, local Mach numbers, flow angles, the locations of the points, ratios of the pressures, temperatures, densities, and others. The results include the determination of the values of x and y coordinates of the nozzle wall contour points.

Figure (16) shows the final wall contour of the nozzle for the diverging-section. There are two curves; one curve represents the present wall contour obtained from the developed software, and the other is plotted by the use of the numerical curve-fit equation, that was described above, which can be read as;

$$y = 1.072445 + 0.18207x - 0.012197x^2 - 0.0002905x^3 \quad (36)$$

The resulted two contours are so matched to a large extent. To construct such a nozzle, this contour may be employed in an advanced industrial process with CNC machines.

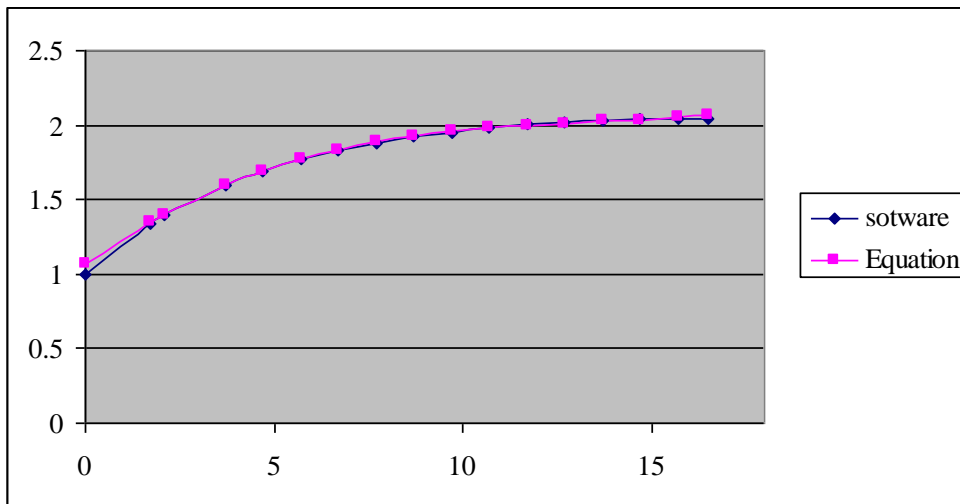


Figure 16: Contour of the diverging section of the desired nozzle.

Figure (17) presents the geometry of different wall contours for the same desired exit Mach number of 3, but with a set of six different values for the throat-height. These are obtained by the use of the developed software, where the Figure shows the geometrical similarity of the six different nozzles. The sizes of such nozzles are related to each other by applying a specific length scale factor.

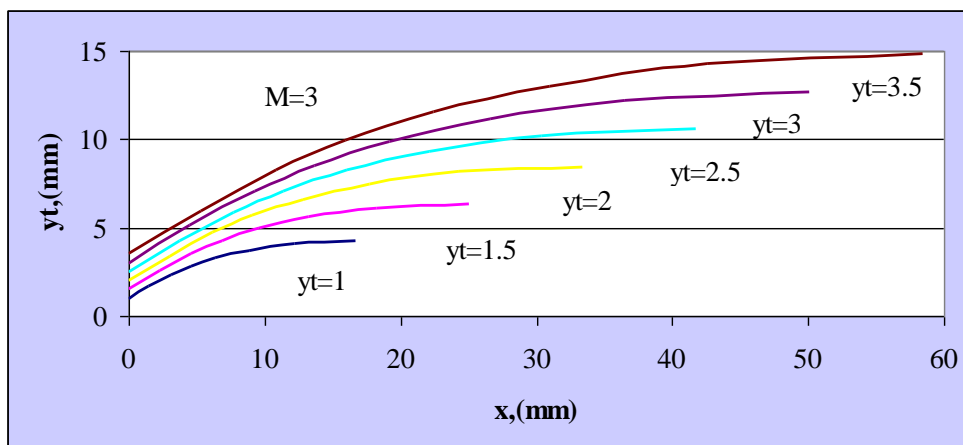


Figure 17: Contours of the diverging-section of the desired nozzle for different values of the throat area.

The computed area ratio was obtained as; $A_E/A^*=2.391$, however, from reference [1], the corresponding computed area ratio is $A_E/A^*=2.33$. By comparison, these are within 0.5 and 3 percent, respectively, relative to the isentropic value presented elsewhere in [1]. The FLUENT software is used here to validate the developed software, to compare the obtained results, and to clarify the behaviour of the supersonic flow field. FLUENT results are highly identical to those obtained through the use of the mathematical relationships related to the dynamics of the supersonic flow of an ideal gas through nozzles, more details could be traced in [27]. Here, different coloured FLUENT-related windows are presented. Figure (18) shows coloured contours of the axial velocity field inside the C-D nozzle. The red zone represents the high-velocity field at the exit region, with a maximum value of 624 m/s, while the blue area represents the low-velocity field

at the entrance region, with a static starting region. The expansion wave lines are well demonstrated.

Figure (19) shows coloured contours of the density field inside the C-D nozzle. The red zone represents the high-density field at the entrance region, with a maximum value of 4.55 kg/m^3 , while the blue area represents the low-density field at the exit region. The maximum rate of change of density occurs near the nozzle throat, while it is almost uniform and decreases smoothly along the converging-section of the nozzle. The density variations along the diverging-section are non-uniform along the cross-section, where it decreases along the nozzle radius with minimum values at the wall.

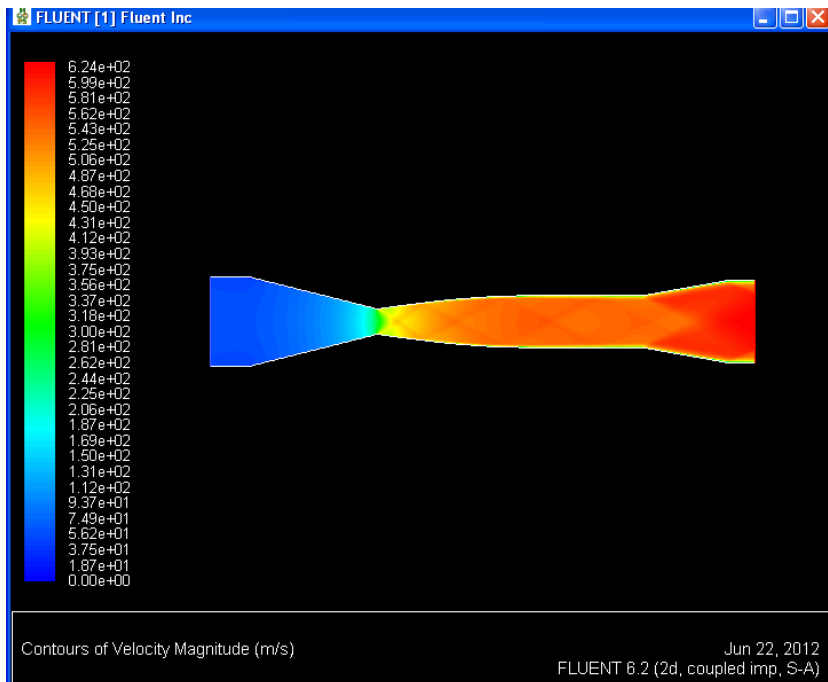


Figure 18: Contours of the axial velocity field inside the C-D nozzle.

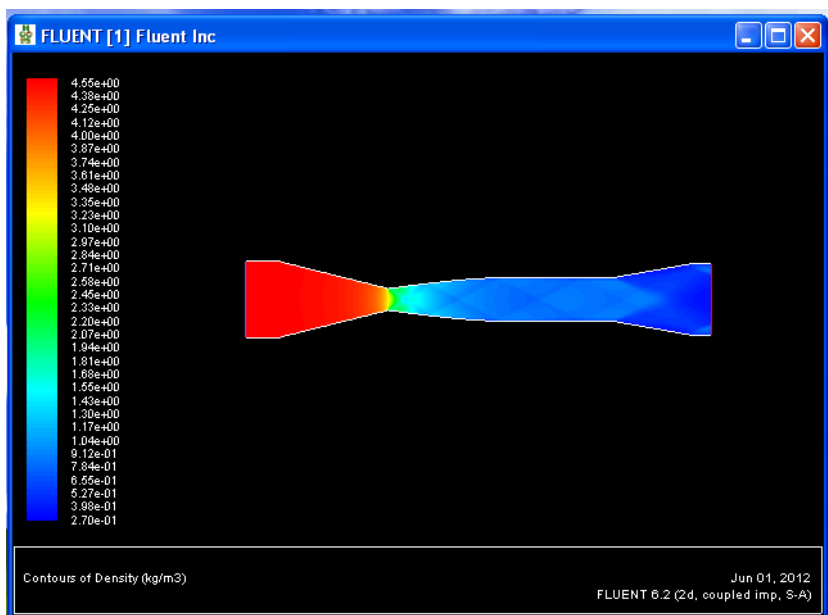


Figure 19: Axial density distribution inside the C-D nozzle.

Figure (20) shows colored contours of the static pressure field inside the C-D nozzle. The red zone represents the high-pressure field at the entrance region, with a maximum value of 400 kPa, while the blue area represents the low-pressure field at the exit region, which is equal to the atmospheric pressure. The maximum rate of change of static pressure occurs near the nozzle throat, while it is almost uniform and decreases smoothly along the converging-section of the nozzle. The static pressure variations along the diverging-section are near to be uniform across the cross-section, where it decreases along the nozzle.

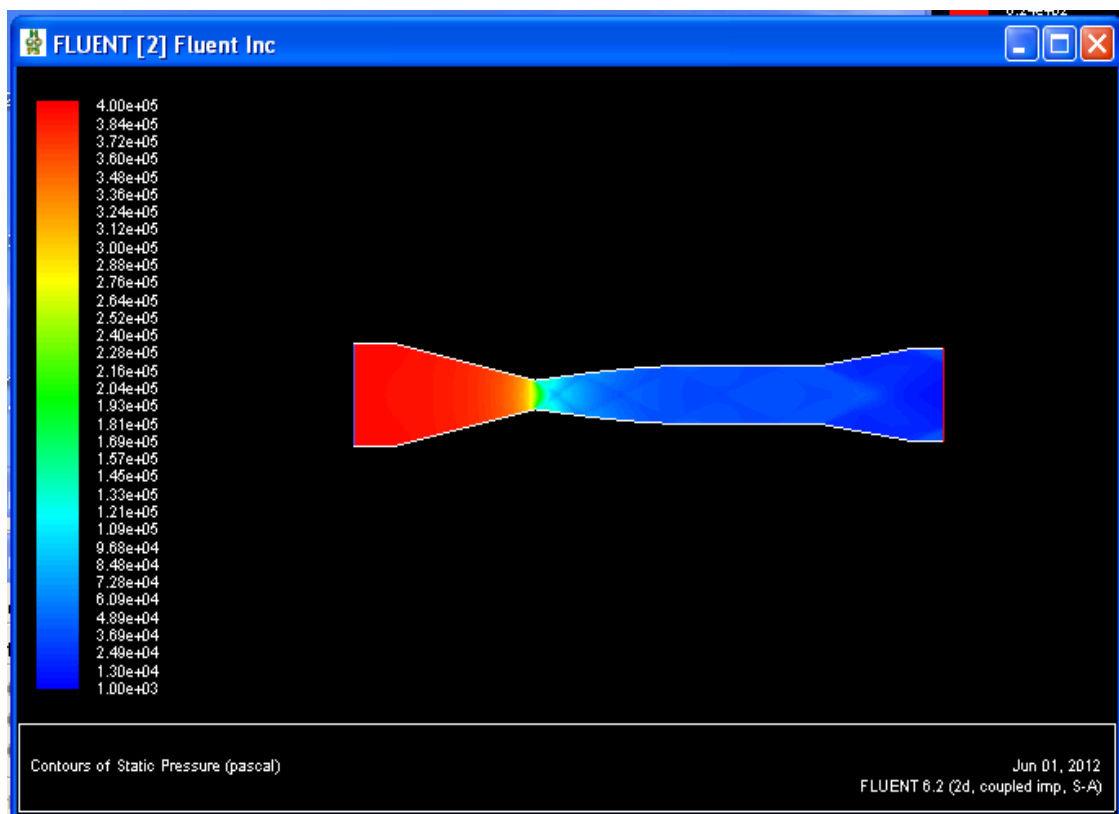


Figure 20: Axial static pressure distribution inside the C-D nozzle.

Figure (21) shows colored contours of the static temperature field inside the C-D nozzle. The red zone represents the high static temperature field at the entrance region, with a maximum value of the assumed value of 303 K, while the blue area represents the low static temperature field at the exit region. The maximum rate of change of static temperature occurs near the nozzle throat, while it is almost constant along the converging-section of the nozzle. The static temperature variations along the diverging-section are widely non-uniform across the cross-section, where it decreases along the nozzle radius with maximum values at the wall.

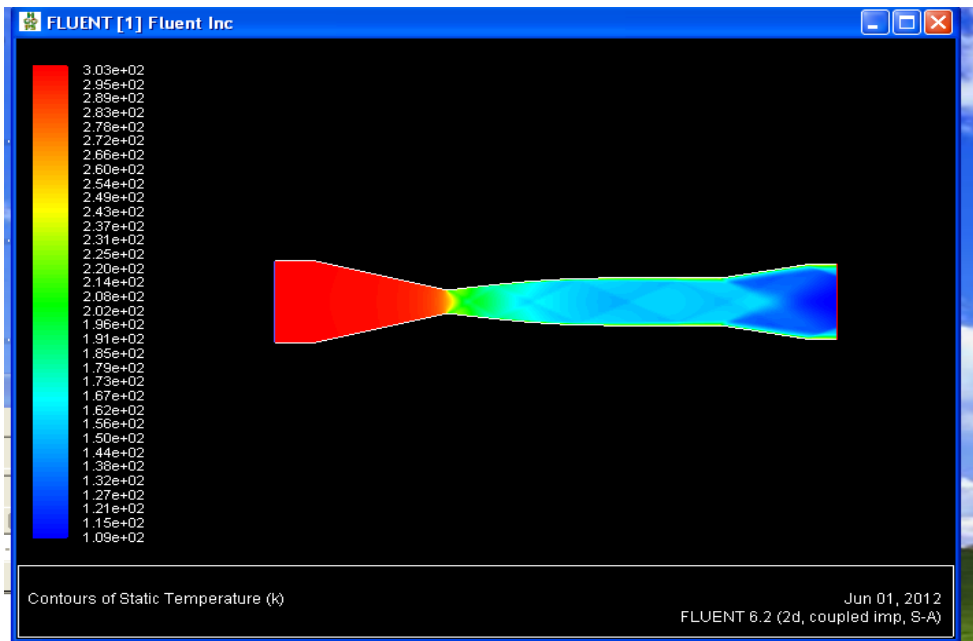


Figure 21: Axial static temperature distribution inside the C-D nozzle.

Figure (22) shows the Mach number variations along the contour of the wall of the diverging-section of the C-D nozzle, where it is 3 at the exit. It seems to have a high Mach number gradient near the throat, while it increases smoothly along the rest of the nozzle.

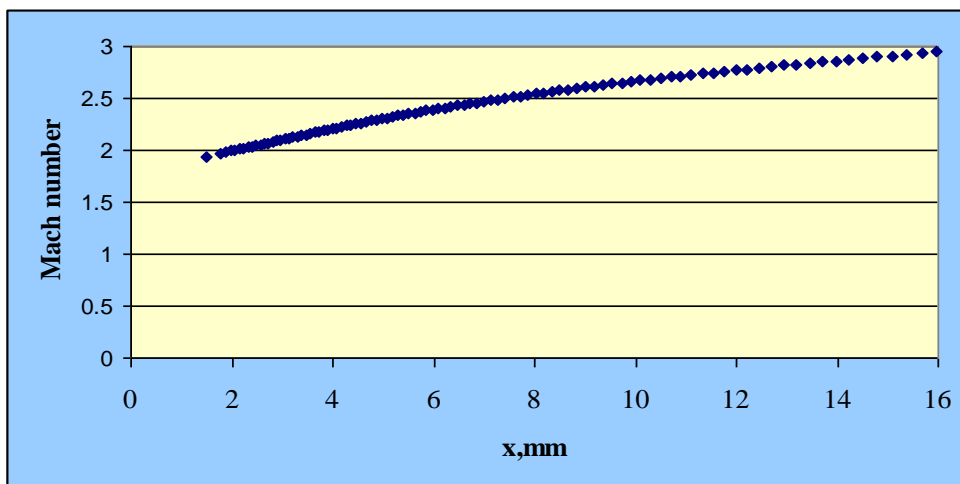


Figure 22: Mach number variations along the contour of the wall of the diverging-section of the nozzle.

CONCLUSIONS AND RECOMMENDATIONS

The MOC proved to be an adequate and proper method. This method with the Prandtl-Meyer relations should be used to build the appropriate mathematical model in order to calculate the flow properties of the supersonic flow field under different assumptions and conditions. To generate the required mesh, the graphical representation should be strongly linked to the mathematical modelling equations. The developed software, written in Visual-Basic language, is a very useful tool for the designer in producing an accurate and immediate solution for supersonic flow in C-D nozzles. This

software could be done with a flexible link with other CAD packages. The functionality of the software has been validated by solving many problems with known solutions. The developed software includes also an error tracking system.

The achieved curve-fit wall surface equation is a valuable input parabolic formula, for the high-accuracy CNC-type of production machines. FLUENT gave a great help. Practice should not be ignored. This study could be extended to cover the effects of velocity boundary layer and/or thermal boundary layer on the behaviour of such supersonic flow. The consequences of enhancing flow behaviour by imposing sound propagation could be considered.

LIST OF SYMBOLS

θ	Flow angle for steady flow.	A	Cross-sectional area (mm ²)
θ^*	The maximum angle at the throat of the nozzle.	L	Length of the nozzle (mm)
α	The inclination angle of the characteristic line.	N	Number of characteristics line
β	The inclination angle of the velocity vector	y_t	Required throat height (mm)
INT[θ^*]	The integer value of the maximum angle of the nozzle	c	Speed of sound (m/s)
v	Prandtl-Meyer function.	C+	Left-running characteristic lines
μ	Mach cone angle	C-	Right-running characteristic lines
γ	Specific heat ratio	u	Velocity component along x-axis (m/s)
ρ	Gas density (kg/m ³)	L	Length of the plug nozzle (mm)

REFERENCES

- [1] Anderson, John D., "Modern Compressible Flow with Historical Perspectives", 3rd ed., McGraw Hill, New York, 2003.
- [2] Anderson, J. Jr., "Computational Fluid Dynamics; The Basics with Applications", McGraw-Hill Book Company, New York, 1995.
- [3] Zucro, M. J., and Hoffman, J. D., "Gas Dynamics", New York, Wiley, Vol. 1 and Vol. 2, 1976.
- [4] Ascher, H. S., "The Dynamics and Thermodynamics of Compressible Fluid Flow, The Ronald Press Company", Volume I and II, New York, 1954.
- [5] Meyer R. E., Goldstein S., "The Method of Characteristics for Problems of Compressible Flow Involving Two Independent Variables", Journal of Mechanics and Applied Mathematics, 1948.
- [6] Yu, Y. N., "A Summary of Design Techniques for Axisymmetric Hypersonic Wind Tunnels", AGAR Dograph 35, Nov 1958.
- [7] Jones, T. Robert, "Effects of sweepback on Boundary Layer and separation, Separation NACA" Technical Report 884, P487-489, January 01, 1947.
- [8] Puckett A. E., "Supersonic Nozzle Design, Journal of Applied mechanics", Transaction of ASME, Vol. 13, P265-270, No. 4, 1946.
- [9] Friedrichs, K. O., "Theoretical Studies on the Flow-Through Nozzles and Related Problems", App. Maths Panel 82-IR, AMG-NYU No. 43, New York University, April 1944.

- [10] Nilson, E., "Analytic Correction of a Two-Dimensional Nozzle for Uniform Exhaust Flow", U.A.C. Meter Report No. 15, 1948.
- [12] Baron, J. R., "Analytic Design of a Family of Supersonic Nozzles by The Friedrichs Method", WADC Technical Report, 54-279, June 1954.
- [13] "A New Method of Designing Two Dimensional Laval Nozzles for a Parallel and Uniform Jet", Report NA-46-235-1, North American Aviation, Inc., Downey, California, March 1946.
- [14] Beckwith, Ivan E., and Moore, John A., "An Accurate and Rapid Method for The Design of Supersonic Nozzles", NACA Technical NMTE 3322, Langley Aeronautical Laboratory, Langley Field, Va., February 1955.
- [15] Riise, H. N., "Flexible-plate nozzle design for two-dimensional supersonic wind tunnels", JPL report No. 20-74, CIT, June 1954.
- [16] Rosen, J. C., "The design and calibration of a variable Mach number nozzle", J. Aeronaut. Sci., 22, No. 7, pp. 484-490, 1955.
- [17] Sivells, J.C., "Design of two-dimensional continuous-curvature supersonic nozzles", J. Aeronaut. Sci., 22, No. 10, pp. 685-692, 1955.
- [18] Rao, G. V. R., "Exhaust Nozzle Contour for Optimum Thrust, Jet Propulsion", Vol. 28, No. 6, June 1958, pp. 377-382.
- [19] Allman, J. G., and Hoffman, J. D., "Design of Maximum Thrust Nozzle Contours by Direct Optimization Methods," AIAA/SAE Joint Propulsion Conference, Vol. 14, Aug. 1978.
- [20] Zebbiche, T. And Youbi, Z., Supersonic "Two-Dimensional Minimum Length Nozzle Design at High Temperature", Emirates Journal for Engineering Research, 11 (1), 91-102, 2006.
- [21] Khan, M. A., Sardiwal, S. K., Sharath, M. V. Sai, Chowdary, D. Harika, "Design of a Supersonic Nozzle using Method of Characteristics:", International Journal of Engineering Research & Technology (IJERT), Vol. 2 Issue 11, November 2013.
- [22] Salhi, M., Zebbiche, T. and Mehalem, A., "Stagnation pressure effect on the supersonic flow parameters with application for air in nozzles", Aeronaut J, 2016, 120, (1224), pp 313–354.
- [23] Sarah, Elizabeth Adams, "The Design and Computational Validation of a Mach 3 Wind Tunnel Nozzle Contour", Master of Science Thesis, Aerospace Engineering, University of Tennessee, Knoxville, TN, USA, 12-2016.
- [24] Chi Zhang, Peng Wen, Yueming Yuan, Xuejun Fan, "Evaluation and optimal design of supersonic nozzle for laser-assisted oxygen cutting of thick steel sections", Springer-Verlag London 2016.
- [25] M White, A I Sayma, and C N Markides, "Supersonic flow of non-ideal fluids in nozzles: An application of similitude theory and lessons for ORC turbine design and flexible use considering system performance", Journal of Physics: Conf. Series 821, 2017.
- [26] Zebbiche, T., "Stagnation pressure effect on the supersonic minimum length nozzle design", The Aeronautical Journal, Volume 123, Issue 1265, pp. 1013-1031, July 2019.

- [27] Walid Hamaidia, Toufik Zebbiche, Mohamed Sellam, Aberrazak Allali, "Performance improvement of supersonic nozzles design using a high-temperature model", *Proceeding of the Institution of Mechanical Engineers, J. of Aerospace Engineering*, February 27, 2019. Volume: 233 issue: 13, page(s): 4895-4910
- [28] Mohamed K. Elshebani, "Designing, and Evaluation of Supersonic Nozzle", M.Sc. Thesis, Mechanical and Industrial Engineering Department, University of Tripoli, January 2012.