11th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2021-153**
8-10 September 2021 - METU, Ankara TURKEY

SHOCK STRUCTURE PREDICTION METHOD FOR BLUNT-NOSED BODIES

Berkay DEMIR [1] and Engin LEBLEBICI [2]
Roketsan A.S.
Ankara, Turkey

Sıtkı USLU [3]
TOBB University of Economics and Technology
Ankara, Turkey

ABSTRACT

This paper presents a method to predict 2D and 3D shock shape for blunt-nosed bodies at supersonic and hypersonic speeds. In high-speed CFD problems, capturing the strong gradients accurately has always been an important and time-consuming process. However, this approach presents a more effective and fast method to capture gradients and simplifies mesh refinement process around the shock structures. The method includes both analytical and empirical formulations with CFD corrections. Validation of the 2D method have been made with SU2 v7.1.1. open-source code whose results agree well with experimental data for an axisymmetric spherically blunted-nose geometry. The shock structure predicted with the developed code for 2D and 3D problems with an angle of attack show a good agreement with the CFD results. Further studies will include improvement of methodology to make shock shape prediction process more effective and automatic.

INTRODUCTION

CFD, Computational Fluid Dynamics, is a powerful tool to design and analyse complex fluid flow problems. Nowadays, the main focus of CFD is to reduce the highly time consuming simulation times while conserving its fidelity. For supersonic and hypersonic flow problems an effective way to generate a suitable computational mesh is to refine and align the mesh along the strong gradient regions like the shock waves. In general CFD applications, a preliminary analysis is performed and contours of the result examined to capture the shock wave. Since design process constitutes thousands of runs, and each run has a different shock pattern, mesh alignment and refinement around the shock region becomes too time consuming. However, the mesh alignment and refinement around the shock structure can be done easily once the shock is predicted.In addition, the method can also reduce the number of analyses required for a design process. For instance, the shock borders can be obtained from the method, and according to that, the vehicle dimension limits can be determined without extra CFD computations.

In literature, some alternative approaches for defining the shock region are presented. One approach is using fully analytical methods which are not suitable for most geometries [Martel, J. D., & Jolly, B., 2015]. Another one is adaptive mesh refinement techniques, which change and define mesh refinement regions based on the previous solution results. However, the adaptive mesh techniques also increase the time of analysis [Herrera-Montojo, J., Fossati, M.,

[1] Engineer at Roketsan A.S., Email: berkay.demir@roketsan.com.tr
[2] Lead Engineer at Roketsan A.S., Email: engin.leblebici@roketsan.com.tr
[3] Assistant Professor in TOBB University of Economics and Technology, Email: suslu@etu.edu.tr

& Minisci., 2017]. However, the present methodology, that is suitable for all types of geometries, is distinguishable in terms of the run time CPU requirements.

Throughout the paper, firstly the methodology of shock shape prediction will be given. Afterwards, validation of SU2 v7.1.1 with experimental data and shock shape code's validation with CFD results will be presented. Finally, the results will be examined, and further studies will be explained.

METHOD

The shock structuring methodology can be applied to any case using only the free stream Mach number and the case geometry. The 2D shock shape prediction method consists of three main parts, which are detached shock, attached shock and CFD correction. First of all, 2D geometry need to be expressed as polylines. Then Billig's [Billig, 1967] empirical hyperbola technique is used to calculate the detached bow shock shape at the blunted-nose of the geometry. In this method the shock shape is defined as a hyperbola in which the LE (leading edge), radius ($\mathbf{R_c}$) and shock standoff distance (Δ) are calculated from (Equation 1) [Billig,1967]. The defined detached shock hyperbole curve is continuous to draw until the shock angle limit of the $\boldsymbol{\theta} - \boldsymbol{\beta} - \mathbf{M}$ equation is (Equation 2) reached. The limit of Equation 2 is determined by taking the derivative of $\boldsymbol{\theta}$ respect to $\boldsymbol{\beta}$ and equating the derivative to zero. The calculated $\boldsymbol{\beta}$ defines the stopping angle for detached shock and $\boldsymbol{\theta}$ defines the starting geometry angle for the attached shock calculations [Herrera-Montojo, J., Fossati, M., & Minisci, E., 2017].

$$x = R + \Delta - R_c cot^2\theta \left[\left(1 + \frac{y^2\, tan^2\theta}{R_c^2} \right)^{1/2} \right]$$

$$\frac{R_C}{R} = 1.386 \exp[1.8/(M-1)^{0.75}]$$

$$\Delta/R = 0.386 \exp(4.67/M^2) \tag{1}$$

$$tan\theta = 2\, cot\beta \frac{M_1^2 \sin^2\beta - 1}{M_1^2(\gamma + \cos 2\beta) + 2} \tag{2}$$

After defining the starting geometry angle ($\boldsymbol{\theta}$), the attached shock calculation procedure is examined at the intersection point of the perpendicular line from geometry starting point (which has the angle of $\boldsymbol{\theta}$) and Billig's hyperbola curve. The attached shock calculation is based on the tangent wedge method. Firstly, the shock angle for the first polyline part is calculated with using the tangent wedge method and Mach wave of the expansion fan is produced at the end point of the line. Mach wave inclinations is calculated with $\boldsymbol{\mu} = \mathbf{arcsin}(\mathbf{1}/\mathbf{M_2})$ equation where $\mathbf{M_2}$ refers to after shock Mach number. The oblique shock of the first section is drawn until the intersection point of the shock and Mach waves reached and continue for the second polyline section with using the intersection point of the first section as starting point. This process continues until the end of the polylines and was presented basically with the Figure 1 [Herrera-Montojo, J., Fossati, M., & Minisci, E., 2017].

The shock shape obtained with the methodology, which has been presented so far, for a blunted-nose geometry is shown with white points in the Figure 2. As can be deduced from the figure, the shock shape can't be captured correctly. Therefore, the CFD correction factor is defined based on Mach number. This factor used to coincide the calculated sonic line (shown with yellow points) with the sonic line of the CFD results (shown with red points). Therefore the calculated Mach wave angles ($\boldsymbol{\mu}$) is multiplied by these factors. Result of the shock shape prediction with CFD correction is presented with black points which show good agreement with the shock shape determined from CFD calculation.

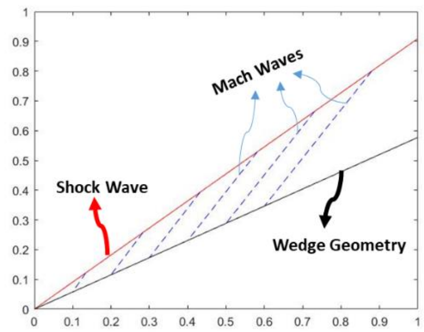

Figure 1: 2D shock shape for 30° wedge geometry at M=5

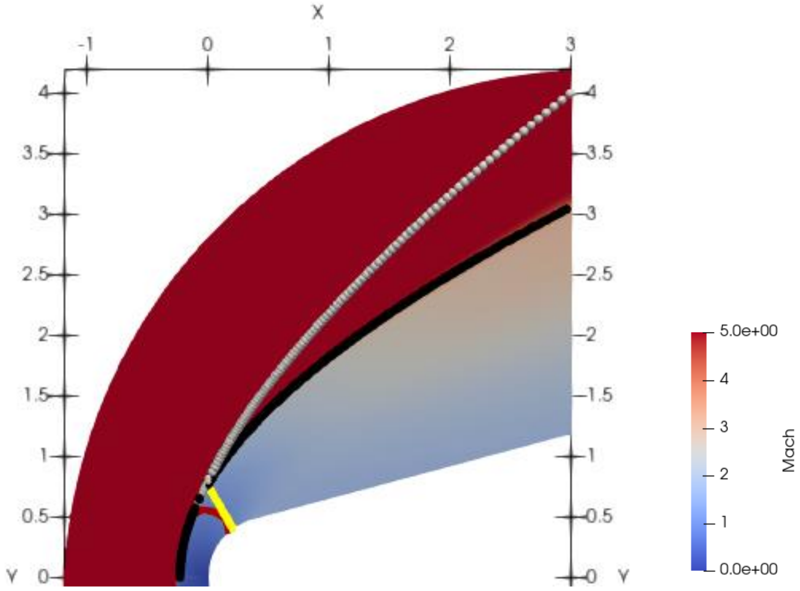

Figure 2: 2D shock shape for blunted-nose body at zero angle of attack

Three-dimensional shock shape prediction method is similar to the 2D version in many ways. However, geometrical manipulations for the 3D implementation of the method are much more complicated. Therefore, CGAL (The Computational Geometry Algorithms Library), which is geometrical manipulation library of C++, is used. Briefly, 3D geometry is divided into cells with surface meshes for 3D shock shape prediction. Then, the surface streamlines are calculated with using CGAL, and 2D methodology is implemented along the streamlines with some modification [Herrera-Montojo, J., Fossati, M., & Minisci, E., 2017].

In addition, unlike the 2D methodology, Billig's hyperbola formula for spherically conic geometries is used to calculate shock standoff distance and hyperbola radius (Equation 3).

$$\frac{R_C}{R} = 1.143\ \exp[0.54/(M-1)^{1.2}]$$

$$\Delta/R = 0.143\exp(3.24/M^2) \tag{3}$$

Surface streamlines calculation process starts with the triangular surface mesh. Firstly, the freestream velocity is projected to the each vertex of the surface mesh with equation $\mathbf{V_v} = \mathbf{V_\infty} - (\mathbf{n}.\mathbf{V_\infty}).\mathbf{n}$ where $\mathbf{V_\infty}$ is freestream velocity and n is normal unit vector for each vertex. Then, Surface velocity is calculated using 2D barycentric triangle coordinates function of the CGAL with the formula $\mathbf{V_s} = \mathbf{W_1}.\mathbf{V_{v1}} + \mathbf{W_2}.\mathbf{V_{v2}} + \mathbf{W_3}.\mathbf{V_{v3}}$ where W defines barycentric averaging weight. Calculated surface velocities are used in the fluid kinematic equation $\left[\frac{\mathbf{dx}}{\mathbf{dt}} = \mathbf{V_s(x)}\ \textbf{for x direction}\right]$for x, y and z directions. To calculate the surface streamlines the kinematic equation are iterated with a numerical integration method until the distance between two calculated flow points are below the defined limit [CGAL 5.2.1 Documentation,2021].

In addition to tangent wedge method, tangent cone method is also examined for 3D shock shape predictions. Tangent cone method is determined by simplifying the continuity equation in cylindrical coordinates for axisymmetric conical flow. The simplified equation, which is named as Taylor-Maccoll Equation, is presented in Equation 4. The equation is an ordinary differential equation with only one dependent variable V_r. The numerical solution procedure is designed as ordinary differential equation solver starts the iterations with geometry θ angle and increases the angle until it reaches the shock angle for the calculated V_r [Squire, 1991]

$$\frac{\gamma-1}{2}\left[V_{max}^2 - V_r^2 - \left(\frac{dV_r}{d\theta}\right)^2\right]\left[2V_r + \frac{dV_r}{d\theta}\cot\theta + \frac{d^2V_r}{d\theta^2}\right] - \frac{dV_r}{d\theta}\left[V_r\frac{dV_r}{d\theta} + \frac{dV_r}{d\theta}\left(\frac{d^2V_r}{d\theta^2}\right)\right] = 0 \tag{4}$$

RESULTS

The SU2 v7.1.1. is used for CFD calculations. Validation work is performed for 2D blunt-body at a freestream Mach number of 2. 2D steady, RANS (Reynolds-averaged Navier–Stokes) equations are solved with Spalart–Allmaras turbulence model. The domain is meshed with about 26,000 structured quadrilateral cells. The CFD result for the distribution of pressure coefficient shows very good agreement compared to the experimental measurements of Sahoo et al. [2016]. The validated solution procedure and settings are used in other CFD studies.

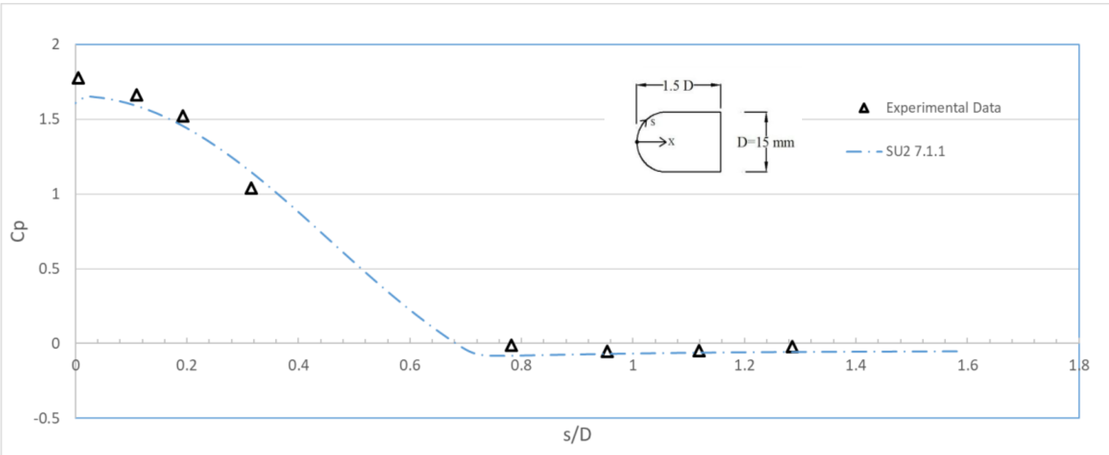

Figure 3: 2D blunt-body pressure distribution along the non-dimensional length (s/d) [Sahoo, 2016]

The results of the 2D and 3D shock prediction calculations are compared with CFD results. The spherically blunted-nose geometry with 0.5 m radius, 3m length and 15-degree cone angle is used for the validations of the methodology. Prediction code and CFD results for 2D approach are presented for freestream Mach number 5 in Figure 2 for zero angle of attack and in Figure 4 for 4 degree angle of attack. The calculated shock shapes are presented with black points in the Mach contours and it shows good agreement with the CFD results.

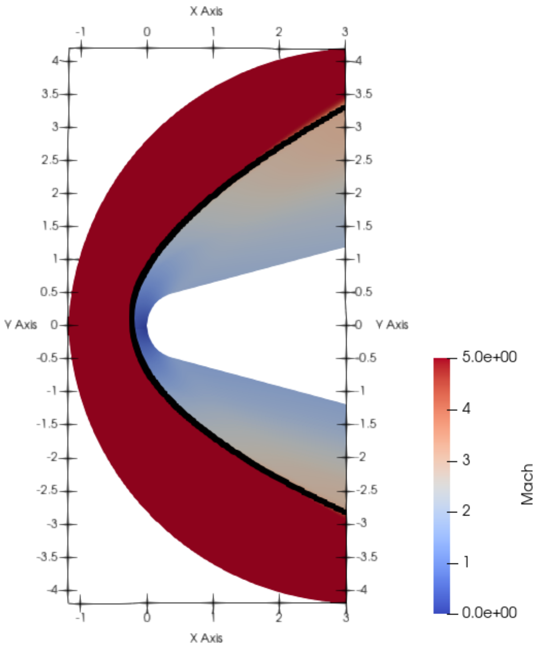

Figure 4: 2D shock shape for blunted-nose body at 4° angle of attack

The same spherically blunted-nose geometry and free stream conditions are also used in 3D methodology validation studies. Since the studies with CGAL are not fully completed, surface streamline tool of Paraview is used as an alternative. The calculated streamlines are presented in Figure 5. In 3D shock structure methodology, streamlines are used as 2D geometry input and calculations are performed for each of the selected streamlines to predict an accurate shock structure. 3D prediction code calculations and CFD results are presented in Figure 6. The calculated shock shapes are presented with black points in the density gradient contours and it shows good agreement with the CFD results.

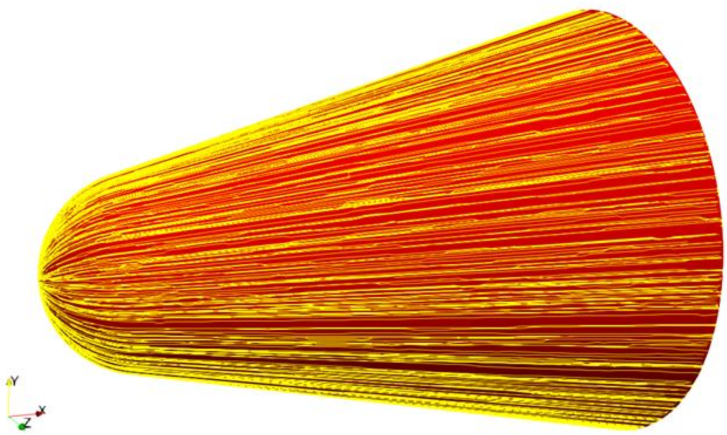

Figure 5: 2D surface streamlines for blunted-nose body

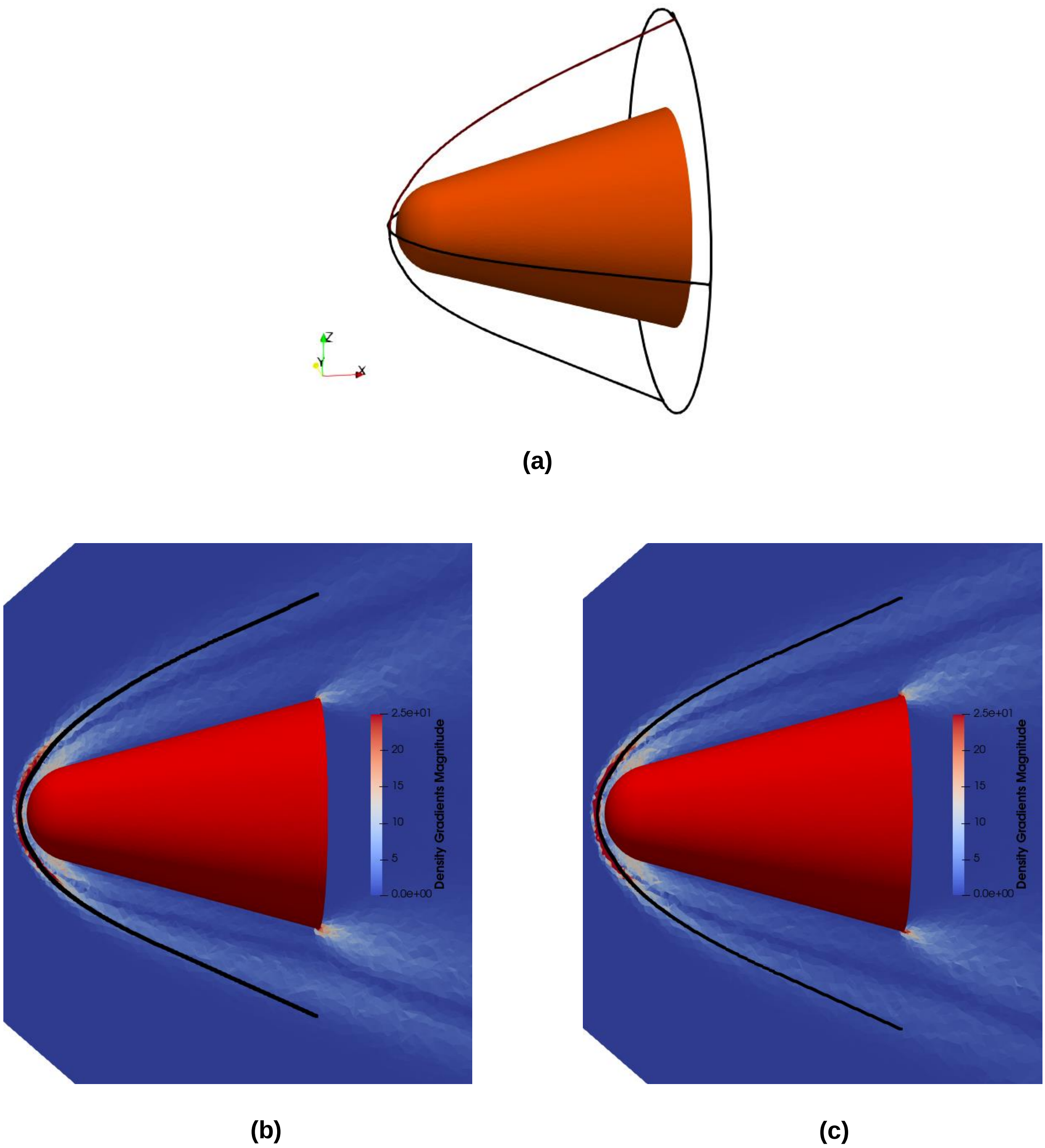

Figure 6: 3D shock shape for blunted-nose body at 4° angle of attack
a) 3D shock structure, b) Shock-structure and density gradient contour on the x-z plane, c) Shock-structure and density gradient contour on the x-y plane

In conclusion, the 2D-3D shock shape prediction methodology for supersonic and hypersonic flow is developed in the present work. The methodology is validated against the CFD results and it shows that a very good agreement between the results obtained from both approaches. The 3D shock shape methodology will be examined in more detail with using CGAL libraries for calculation of streamlines as future work.

References

Billig, F. S. (1967). Shock-wave shapes around spherical-and cylindrical-nosed bodies. Journal of Spacecraft and Rockets, 4(6), 822-823.

CGAL 5.2.1 Documentation. Retrieved March 20, 2021, from https://www.cgal.org/documentation.html

Herrera-Montojo, J., Fossati, M., & Minisci, E. (2017). Shock-conforming mesh generation for aerodynamic analyses at supersonic regimes. *Computers & Fluids*, *157*, 276-293.

Mohandas, S., Siddharth, R. K., & John, B. (2019). Reduction of wave drag on parameterized blunt bodies using spikes with varied tip geometries. *Acta Astronautica*, *160*, 25-35.

Martel, J. D., & Jolly, B. (2015). Analytical shock standoff and shape prediction with validation for blunt face cylinder. In *AIAA Atmospheric Flight Mechanics Conference* (p. 0523).

Sahoo, D., Das, S., Kumar, P., & Prasad, J. K. (2016). Effect of spike on steady and unsteady flow over a blunt body at supersonic speed. *Acta Astronautica*, *128*, 521-533.

Squire, L. C. (1991). Modern Compressible Flow: with historical perspective—Second edition. JD Anderson. McGraw-Hill Book Co (UK), McGraw Hill House, Shoppenhangers Road, Maidenhead, Berks, SL6 2QL. 1990. 650 pp. Illustrated.£ 18.95. *The Aeronautical Journal*, *95*(947), 248-248.