# Final Project Report for Flow around a Diamond Obstacle

Computational Fluid Dynamics

Name: Fahad Jan

Degree: MS in Mechanical and Aerospace Engineering
Advisor: Professor Mohammed Hafez
Interests: Manufacturing, Design, and Robotics

# Table of Contents

# Notations:

$\gamma$ Ratio of specific heats

$\mu$ [Kg / m . s] Viscosity

$\rho$ [Kg / m^3] Density

$\tau$ [N/m^2] Viscous Tensor

c [m / s] Speed of sound

$c_p$ Coefficient of pressure.

p [Pascal] Pressure

T [K] Temperature

u [m/s] Velocity in x-direction

v [m / s] Velocity in y-direction

$\Delta t$ [s] Time step

$\Delta x$[m] Grid cell size in x-direction

$\Delta y$[m] Grid cell size in y direction

M Mach number

$\phi$ Velocity Potential

$\psi$ Streamfunction

# List of Figures:

# Summary:

In this study a computational fluid dynamics techniques are used to analyze a flow around a double wedge. The flow in interest is a steady inviscid supersonic flow. The flow around the diamond is conserved and obeys the three conservation laws: conservation of mass, conservation of energy, and conservation of momentum. The equation used is the hyperbolic stream function of irrotational flow. Implicit and explicit schemes are used to solve the equation numerically. A center difference and backward difference is carried out in the mesh domain to solve the partial differential equation. The simulation were carried using both software: Matlab and Ansys Fluent. The results obtained in both methods are in agreement with the analytical data available.

# Introduction:

The field of computational fluid dynamics has developed dramatically in the past few years. The progress of numerical methods with the aid of the technology computers boom has reached powerful results in understanding complex systems and designs. Simulations that used to be impossible to visualize in the mid-sixties are now one click away from programmers and engineers. Computational fluid dynamics is a very effective tool to solve complex problems. In many cases the cost associated with computational fluid dynamics is much cheaper than building a real prototype and doing the experiment multiple times. Computational fluid dynamics is important in many fields such as aerospace, vehicle design, and heat transfer.

In this report, I will be solving the stream function around a diamond obstacle using numerical method. The numerical solution for the stream function by finite differences requires construction of a mesh for the discretization of the PDE and of the initial and boundary conditions (Chattot). The solution will be calculated at the nodes of the mesh. A Cartesian grid is used to create the mesh and a constant time step ($\Delta x$ and $\Delta y$) is used to constrict the grid. The Cartesian grid is useful here because of the "regular" shape of the diamond. A centered difference scheme is implemented to solve the stream function. In this case, the scheme is second order accuracy. We are interested in finding the solution of the supersonic flow past the given geometry and no boundary condition is required downstream of the profile.

This report is prepared in support of the new integrated aerospace and mechanical engineering course. The course will be offered to the mechanical and aerospace engineering undergraduate students. The course will help the students to understand how airplanes fly by taking them through the various steps of designing, modeling, manufacturing and testing and airplane wing (Linke). This project is funded by the National Science Foundation's Improving Undergraduate STEM Education (IUSE) program under Award No. 1505080.

# Governing Equations:

Assuming 2-D, steady, inviscid and irrotational compressible flow conditions at infinity. The stream function used to solve the problem is an agreement with the following conservation laws:

1- Conservation of mass
2- Conservation of momentum
3- Conservation of energy

The stream function is defined using the conservation laws. The stream function equation for a linearized supersonic flow is of hyperbolic type and models wave propagation (Hafez and Chattot, Theoretical and Applied Aerodynamics And related Numerical Methods).

# Boundary Conditions:

I considered the following boundaries: (1) supersonic inflow (2) Lower and upper wall (3) Inlet velocity is known (4) No boundary conditions needed at the outlet.

# Numerical Methods:

The approach of solving the problem is based on the finite difference method (FD). First the domain is discretized on a Cartesian mesh system. The following equation is used to solve the problem.

$$-(M_\infty^2 - 1)\,\psi_{xx} + \psi_{yy} = 0$$

Let

$$\beta = \sqrt{M_\infty^2 - 1} \quad \text{For } M_\infty^2 \geq 1.0$$

The previous equation is the stream function for linearized supersonic flow. It is a second order PDE. The equation is hyperbolic. Both independent variables are represented in space. No boundary condition is required downstream of the profile (Chattot).

The initial conditions:

$$\psi(x, y) = y \qquad \text{For } x \leq 0$$

$$\psi_{top} = y_{top}$$

$$\frac{dy}{dx} = \frac{1}{\sqrt{M_\infty^2 - 1}} = tan\theta$$

$$\psi_y = \rho u$$

$$\psi_x = -\rho v$$

General Case (Implicit Scheme):

$$-\beta^2 \frac{\psi_{i+1,j} - 2\psi_{i,j} + \psi_{i-1,j}}{dx^2} + \frac{\psi_{i,j+1} - 2\psi_{i,j} + \psi_{i,j-1}}{dy^2} = 0$$

For i = 1, … and j = 1, …

The discrete solution will be defined at the nodes of the mesh. These equations were discretized using central difference in the y direction and backward difference in the x direction. The solution was marched forward by solving tridiagonal equations along the vertical lines.

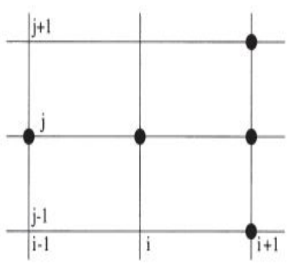


*Figure 1: The computational molecule for the implicit scheme (Chattot)*

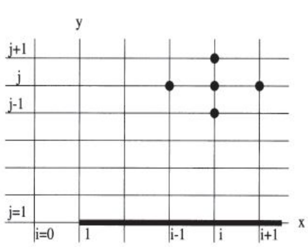


*Figure 2: The overall mesh system for the linearized supersonic flow (Chattot)*

The FDE for the boundary condition is

$$\frac{\psi_{i+1,2} - \psi_{i+1,1}}{dy^2} = 0$$

The first values for the first two columns were specified to solve the problem.

In supersonic flow the Mach angle is given by (Hafez and Chattot, Theoretical and Applied Aerodynamics And related Numerical Methods)

$$\sin\mu = \frac{a_\infty}{U} = \frac{1}{M_\infty}$$

The derivatives for equations:

$$\psi_{Btop(1-\frac{c}{2})} = \frac{\tau}{c}\,x \qquad \psi_{Btop(\frac{c}{2}-1)} = -\frac{\tau}{c}x + \frac{\tau}{2}$$

$$\psi_{Bbot(1-\frac{c}{2})} = -\frac{\tau}{c}\,x \qquad \psi_{Bbot\left(\frac{c}{2}-1\right)} = \frac{\tau}{c}x - \frac{\tau}{2}$$

$$\frac{\partial\psi_{Btop(1-\frac{c}{2})}}{\partial x} = \frac{\tau}{c} \qquad \frac{\partial\psi_{Btop(\frac{c}{2}-1)}}{\partial x} = -\frac{\tau}{c}$$

$$\frac{\partial\psi_{Bbot(1-\frac{c}{2})}}{\partial x} = -\frac{\tau}{c} \qquad \frac{\partial\psi_{Bbot(1-\frac{c}{2})}}{\partial x} = \frac{\tau}{c}$$

# Simulations:

# 1- MATLAB

The problem is defined in Matlab: A supersonic airflow around a diamond wedge. The code starts with specifying some parameters such as the Mach number, specific heat ratio, and angle attack. Then, a mesh is created and the embodied body is defined. The boundary conditions are defined in the code. Then, the stream function is solved point-by-point using the FOR loop column-by-column starting from the left boundary layer. Post processing and graphs are obtained at the end analysis. The matlab code is provided in Appendix 1.

# 2- Fluent

The simulations were done using the software FLUENT. A uniform supersonic stream encounter a wedge with a half angle of 15 degrees was simulated. The stream conditions were specified as the standard conditions. The table below specifies the values used in this simulation.

*Table 1: Values used for the fluent simulation*

| Quantity | Value |
|---|---|
| Mach Number | 3 |
| Static Pressure | 1 atm |
| Static Temperature | 300k |

The calculation has been performed by applying the conservation of energy, mass and momentum for 2D inviscid flow. The equations can be written as

$$\frac{\partial e}{\partial t} + u\,.\nabla\, e + \frac{p}{\rho}\,\nabla\,.\,u = 0$$
$$\frac{\partial \rho}{\partial t} + u\,.\nabla\,.\,u = 0$$
$$\frac{\partial u}{\partial t} + u\,.\nabla u = -\frac{\nabla p}{\rho}$$

Analytical solution:

$$\frac{p_2}{p_1} = \frac{2\gamma M_N^2 - (\gamma - 1)}{\gamma + 1}$$

Where

$M_N = M_\infty \sin\beta$ Normal Mach number

# Results and Discussion:

The double wedge obstacle showed a unique pressure distribution compared to any other shape of airfoils. Due to its geometry, the diamond airfoil produces shock waves at the trailing and leading edge, but it also produces an expansion fan in the middle of the body. This occurred due to the space augmentation in the flow at these specific regions.

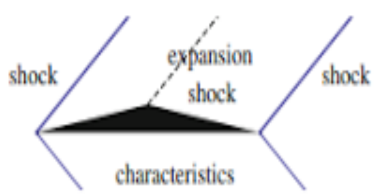


*Figure 3: Supersonic flow will result in two shock waves and an expansion fan as shown in the picture (Chattot).*

Figure 3, illustrates the location of the shock waves and expansion fan. Shocks can be differentiated from expansion waves by noticing that the u velocity decreases (pressure increases) across them. On the other hand, the u velocity increases (pressure decrease) across the expansion fan (Chattot). In both Matlab and Fluent results, the pressure increased where the shock wave occurred; whereas, pressure decreased where the expansion fan occurred.

The theta-beta-M chart available in many textbooks can be used to compare calculated numerical data. With wedge angle theta at 15 degrees the shock wave beta is about 32 degrees (White).

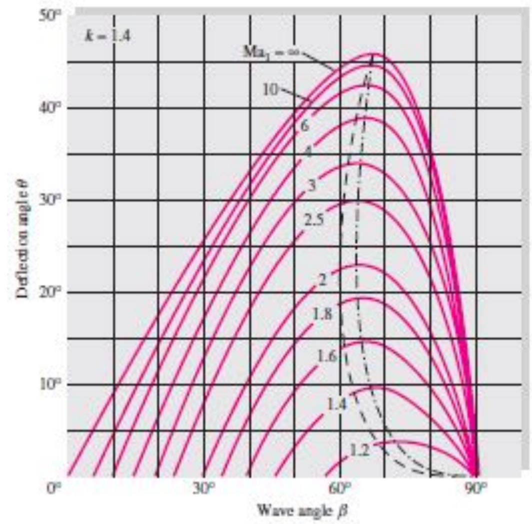


*Figure 4: The theta-beta-M chart available in many textbook can be used to compare calculated numerical data. With wedge angle theta at 15 degrees the shock wave beta is about 32 degrees (White).*

The solution implemented the Cartesian grid to solve for the unknowns. The weakness of the method is losing accuracy near the embodied boundary. However, this problem wasn't encountered in diamond body because the spacing of the grid was chosen based on the dimension of the double edge. The following equation was used to determine the grid size.

$$\frac{dx}{dy} = \tan\theta$$

Another limitation of this method is that the above linear theories will not hold when the Mach number approach 1 (Chattot).

# Results from Matlab:

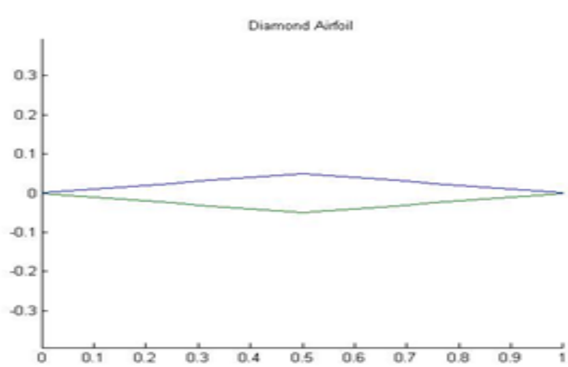


*Figure 5: The picture shows the location and dimension of the embodied body in the mesh.*

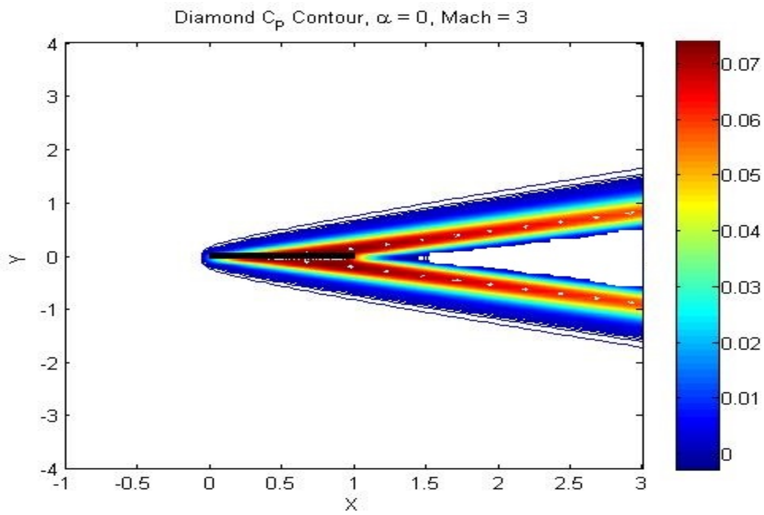


*Figure 6: The picture shows the coefficient of pressure contour obtained from Matlab for the linearized body.*

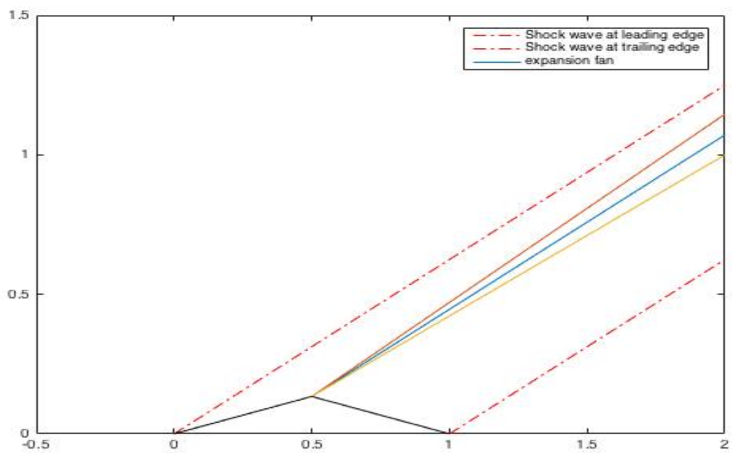


*Figure 7: Shock waves are located at the leading and trailing edge. Expansion fan is located in the middle of the diamond as shown the above picture.*

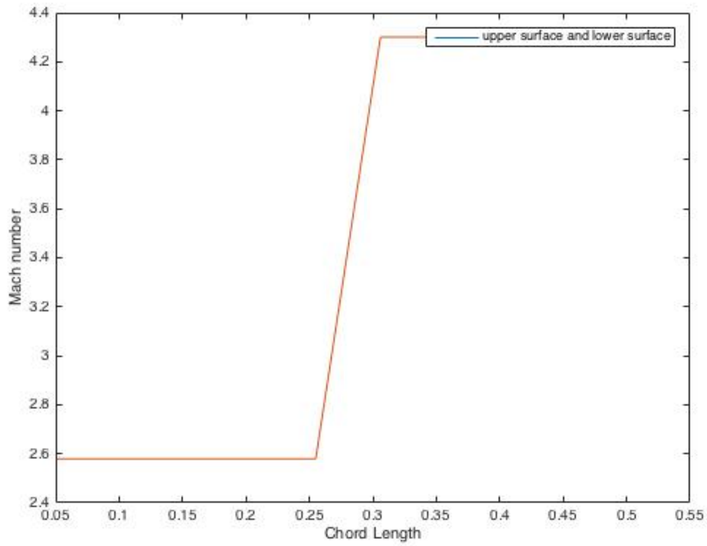


*Figure 8: The plot shows the Mach number for the first half of the diamond (0 - 0.5).*

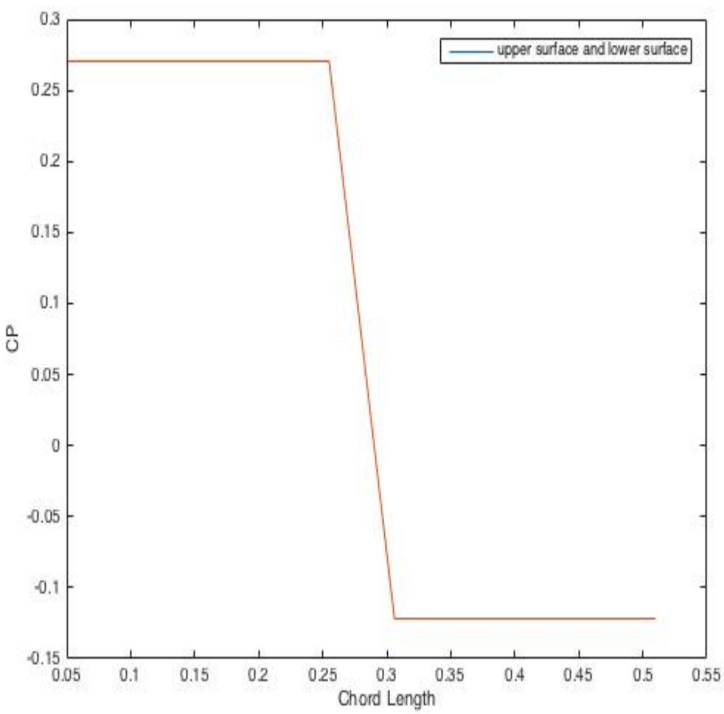


*Figure 9: The shape of the diamond is symmetric and the attack angle is zero. Therefore, both upper and lower surface face the same pressure.*

# Results from Fluent:

To begin the analysis a half diamond was used to set up the simulations. If a higher accuracy is needed, the size of the mesh should be smaller. However, the smaller the mesh the longer the time it takes to run the simulation. In this report only a second order scheme was used. A third order scheme will result in higher accuracy (Hafez, Kwak and Oshima, Computional Fluid Dynamics Review). The advantage of using Fluent to run the simulation is the user friendly interference.

The result from fluent turned out be extremely helpful. Both the implicit and explicit schemes are used. The implicit scheme with second order upwind approach converged after 960 iterations. The table provided in the Appendix shows the calculated velocities in both x and y direction after each iteration.  On the other hand, the explicit method with the MUSCL approach did not converge.

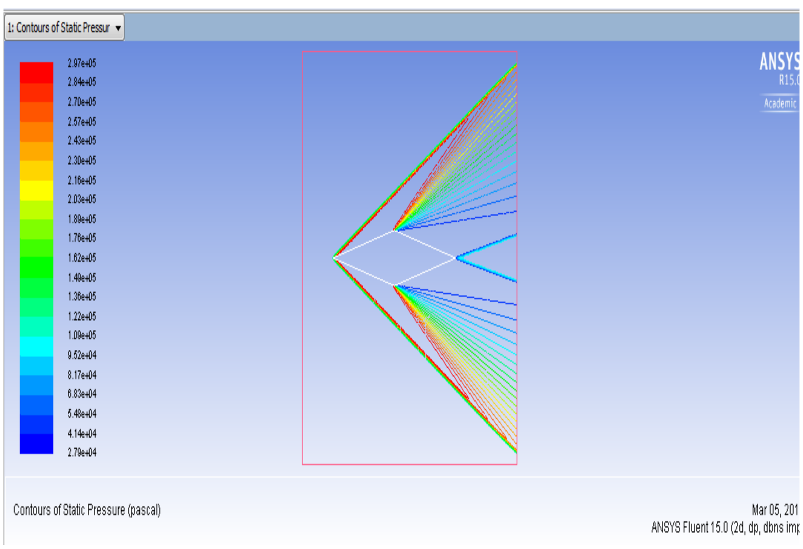


*Figure 10: This picture shows the static contour for pressure using the second order upwind implicit method*

The results make sense as the left side of the diamond faces more pressure due to the upcoming fluid (inlet). On the other hand, the pressure on the right side is lower due to the shock wave. This is an agreement with analytical data. However the back shock wave was not correct.

# Conclusion:

In this work, I used finite difference to solve the stream function for a steady inviscid supersonic flow around a diamond. The domain is discretized into a Cartesian grid. Central difference in the y direction and backward difference in the x direction are used in solving the hyperbolic PDE stream function. The solution was marched forward by solving tridiagonal equations along the vertical lines. Both Matlab and fluent are used to simulate the flow around the diamond. The results from both Matlab and fluent are compared. The solution obtained from the numerical simulation is in a good agreement with shock expansion theory.  By simulating the diamond wedge, one can simply use the same technique on any other airfoil of interest in supersonic flow.

# Acknowledgment:

This project is funded by the National Science Foundation's Improving Undergraduate STEM Education (IUSE) program under Award No. 1505080. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation.

# Works Cited:


Hafez, Mohammed and J. J. Chattot. *Theoretical and Applied Aerodynamics: And related Numerical Methods*. n.d.

Hafez, Mohammed, Dochan Kwak and K Oshima. *Computional Fluid Dynamics Review*. Hackensack, NJ: World Scientific, 2010.

J., Chattitm J. *Computional Aerodynamics ans Fluid Dynamics: An Introduction*. Berlin: Springer, 2002.

Linke, Babara. *An integrated STEM approach for aerospace studies*. 17 03 2016. <http://research.engineering.ucdavis.edu/stemaerospace/>.

White, Frank M. *Fluid Mechanics*. New York: McGraw-Hill, 2008.