

MECH 6111 Gas Dynamics

Numerical Study of a Converging Diverging Nozzle and its comparison with the respective 1-D Analytical Solution

Instructor:

Dr. Pierre Q. Gauthier

Submitted by:

Rahul Chug (Student ID: 40075138)

December, 2018

Abstract

Numerical simulation in a convergent divergent nozzle has been performed on ANSYS Fluent platform, and all the results are compared with their corresponding 1D analytical solution. A range of conditions were defined, which gave rise to normal shock in the divergent portion of the nozzle, and an oblique shock and expansion outside the nozzle. The results for Mach Number and pressure will be compared to the respective values of the analytical solution. Different plots and contours will also be plotted for different parameters with respect to the length of the nozzle.

Contents

Nomenclature

List of Figures

Error! Bookmark not defined.

List of Tables **5**

CHAPTER 1: INTRODUCTION **6**

1.1 Project Description 6

1.2 Introduction to Ansys Fluent 6

CHAPTER 2: PROBLEM SPECIFICATION **8**

2.1 1D Analytical solution 8

2.1.1 Flow of Air through the Convergent Divergent Nozzle causing Normal Shock
inside the Nozzle 8

2.1.2 Flow of Air through the Convergent Divergent Nozzle causing Oblique Shock and
expansion outside the nozzle 10

2.2 Numerical Methodology 12

2.3 Boundary Conditions 15

CHAPTER 3: DISCUSSION OF RESULTS **16**

3.1 Converging Diverging Nozzle causing Nozzle shock 16

3.2 Converging Diverging Nozzle causing oblique shock and expansion 18

CHAPTER 4: CONCLUSION **20**

REFERENCES

Nomenclature

Θ Oblique shock angle

δ wedge angle

ρ Density (kg/m^3)

Subscripts

i inlet

e exit

th throat

Superscript

* Critical position ($M=1$)

List of Figures

Fig.1.1 ANSYS Fluent process

Fig. 2.1 2D Geometry of the Convergent Divergent Nozzle

Fig. 2.2 2D Geometry of the Convergent Divergent Nozzle

Fig.2.3 Geometry of C-D Nozzle Causing Normal Shock

Fig.2.4 Mesh in C-D Nozzle Causing Normal Shock

Fig.2.5 Geometry of C-D Nozzle Causing an Oblique Shock and Expansion

Fig.2.6 Mesh in C-D Nozzle Causing an Oblique Shock and Expansion

Fig.3.1 Mach Number Contour of the C-D Nozzle with a Normal Shock

Fig.3.2 Mach Number Vs X Plot of the C-D Nozzle with a Normal Shock

Fig.3.3 Velocity Contour of the C-D Nozzle with a Normal Shock

Fig.3.4 Pressure Vs X Plot of the C-D Nozzle with a Normal Shock

Fig.3.5 Mach Number Contour of the C-D Nozzle with an oblique shock and expansion

Fig.3.6 Mach Number Vs X Plot of the C-D Nozzle with an oblique shock and expansion

Fig.3.7 Velocity Contour of the C-D Nozzle with an oblique shock and expansion

Fig.3.8 Pressure Vs X Plot of the C-D Nozzle with an oblique shock and expansion

List of Tables

Table 2.1 Simulation parameters in ANSYS Fluent

CHAPTER 1: INTRODUCTION

1.1 Project Description

The study consists of examining the performance of a Converging- Diverging Nozzle running under supersonic condition. The fluid used is ideal gas. The results were first determined through the analytical solution for different boundary conditions. Then, by keeping the same parameters, simulation was performed on the ANSYS platform. Boundary conditions were defined for the inlet, outlet and walls of the C-D Nozzle. All the results are plotted with respect to the nozzle length and the contours for Mach Number and Velocity have been shown. Further, they were compared with the results developed through the analytical method.

1.2 Introduction to Ansys Fluent

ANSYS Fluent is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries. It provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2-D triangular / quadrilateral, 3-D tetrahedral / hexahedral / pyramid / wedge / polyhedral, and mixed (hybrid) meshes. ANSYS Fluent also enables the user to refine or coarsen the mesh based on the flow solution. Mesh can either be read into ANSYS Fluent, or created using the “Mesh” module of Fluent. All remaining operations are performed within the “Solution” module, including setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, and post processing and viewing the results.

The ANSYS Fluent serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer. ANSYS Fluent also uses a utility called cortex that manages the user interface and basic graphical functions. Its parallel solver enables users to compute a solution using multiple processes that may be executing on the same computer, or on different computers in a network. Parallel processing in ANSYS Fluent involves an interaction between ANSYS Fluent, a host process, and a set of compute-node processes. ANSYS Fluent interacts with the host process and the collection of compute nodes using the cortex user interface utility.

The complete solution process of ANSYS Fluent is shown in Figure 3-1. In the first step, we need to conduct pre-analysis to obtain the boundary conditions and the required data for representing the real case. The “Geometry” module is used to generate the geometry model in ANSYS workbench or any other CAD software, such as AutoCAD, Solidworks, NX Nastran, CATIA, etc. Meshing quality determines the accuracy of the result. Different parameters that control the solution accuracy, boundary conditions, physics involved, material characteristics, etc. are entered in the “Physics Setup” module. In the “Solution” module, the solver will calculate the equations iteratively, updating the guess after each iteration, and stopping the

iterations when imbalances are below a preconfigured tolerance. And finally, the “Results” module is used for post-processing and analysis of the results.

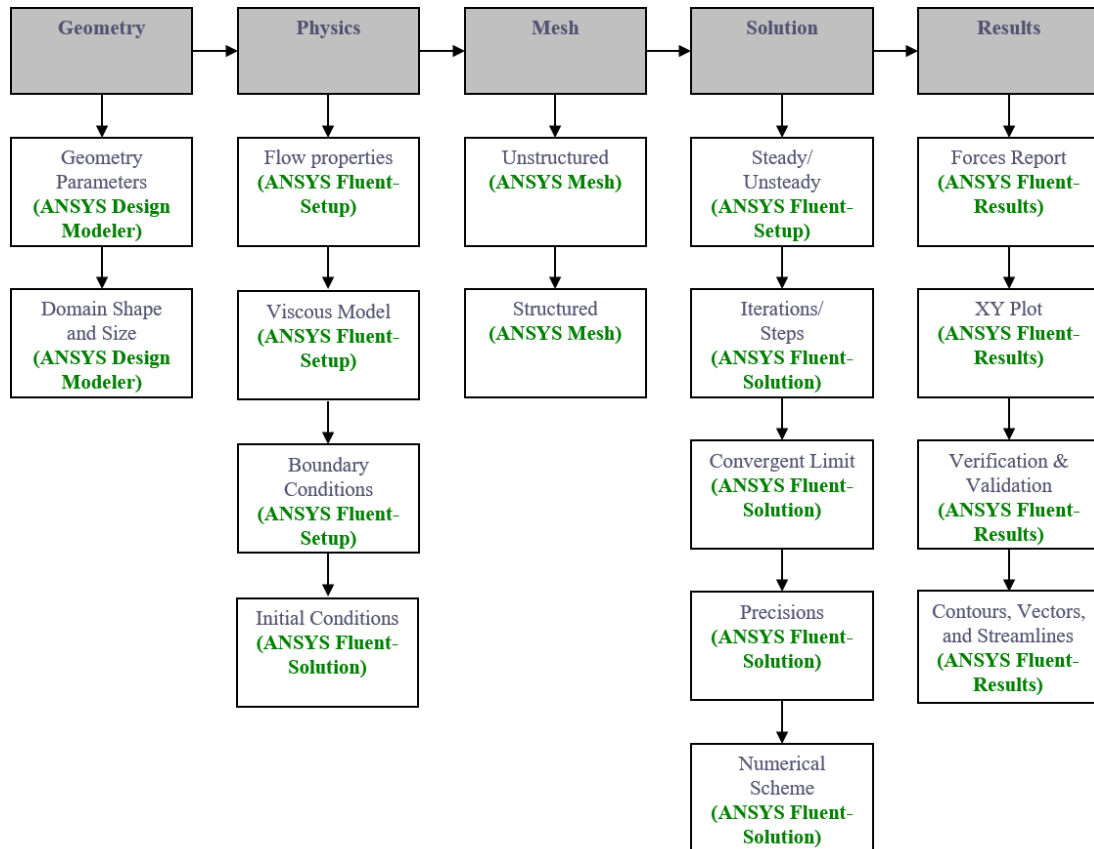


Fig.1.1 ANSYS Fluent process [1]

CHAPTER 2: PROBLEM SPECIFICATION

2.1 1D Analytical solution

2.1.1 Flow of Air through the Convergent Divergent Nozzle causing Normal Shock inside the Nozzle

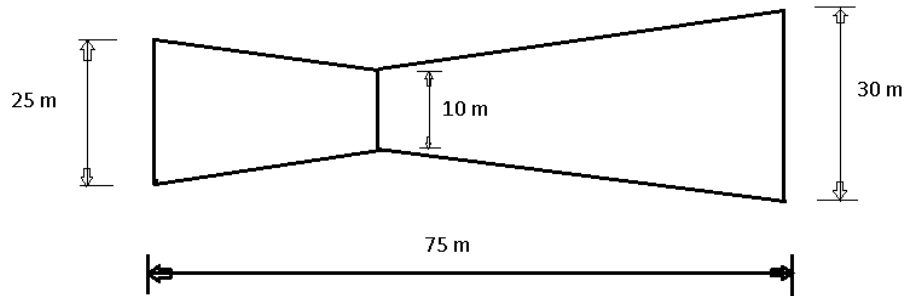


Fig. 2.1 2D Geometry of the Convergent Divergent Nozzle

At inlet,

$$\frac{A_i}{A_{th}} = 2.5 \rightarrow M_i = 0.2$$

$$\frac{P_i}{P_{0i}} = 0.9607 \text{ and } \frac{T_i}{T_{0i}} = 0.9886$$

Let $P_{0i} = 1 \text{ MPA}$ and $T_{0i} = 500 \text{ K}$

We get, $P_i = 960.7 \text{ kpa}$ and $T_i = 494.3 \text{ K}$

At the exit,

$$A_e / A_t = 3 \rightarrow \text{subsonic} \rightarrow M_e = 0.2$$

From IFT, $P_e / P_{0e} = 0.9725$ and $T_e / T_{0e} = 0.9921$

Since $P_{0e} = P_{0i} = 1 \text{ MPa}$ and $T_{0i} = 500 \text{ K} \rightarrow P_e = 972.5 \text{ Kpa}$, $T_e = 496.05$

Also,

$$A_e / A_t = 3 \rightarrow \text{supersonic} \rightarrow M_e = 2.64$$

$$\text{From IFT, } P_e / P_{0e} = 0.04711 \text{ and } T_e / T_{0e} = 0.4177$$

$$\text{Since } P_{0e} = P_{0i} = 1 \text{ MPa and } T_{0i} = 500 \text{ K} \rightarrow P_e = 47.11 \text{ KPa, } T_e = 208.85 \text{ K}$$

Assume if there is a shock at the exit of the nozzle,

$$M_{ex} = 2.64 \rightarrow \text{From NST} \rightarrow M_{ey} = 0.5$$

$$\text{From NST, } P_{ey} / P_{ex} = 7.964 \text{ and } T_{ey} / T_{ex} = 2.2797$$

$$\text{As } P_e = P_{ex} = 47.11 \text{ KPa and } T_e = T_{ex} = 208.85, \text{ we get}$$

$$P_{ey} = 375.18 \text{ KPa and } T_{ey} = 476.11$$

$$\text{Let, } P_b = 500 \text{ KPa}$$

Therefore, $P_{ey} < P_b \rightarrow$ shock is inside the nozzle

Now to calculate Mach Number at the exit, we will equate the mass flow rate of throat to that of the exit,

$$m_{th} = m_e$$

$$\rho^* A^* V^* = \rho_e A_e V_e$$

$$\frac{P^*}{RT^*} A^* M^* \sqrt{(\gamma RT^*)} = \frac{P_e}{RT_e} A_e M_e \sqrt{(\gamma RT_e)}$$

On rearranging,

$$\left(\frac{P^*}{P_{01}}\right) \left(\frac{P_{01}}{P_e}\right) \sqrt{\frac{T_{01}}{T^*}} \left(\frac{A^*}{A_e}\right) = M_e \sqrt{\frac{T_{01}}{T_e}}$$

$$\left(\frac{P^*}{P_{01}}\right) \left(\frac{P_{01}}{P_e}\right) \sqrt{\frac{T_{01}}{T^*}} \left(\frac{A^*}{A_e}\right) = M_e \sqrt{\left(1 + \frac{\gamma - 1}{2} M_e^2\right)}$$

$$\text{At the throat, } M=1 \rightarrow \text{IFT} \rightarrow P^* / P_{01} = 0.5283 \text{ and } T^* / T_{01} = 0.8333$$

$$(0.5283) \left(\frac{1000}{500} \right) \sqrt{\frac{1}{0.8333} \left(\frac{1}{3} \right)} = M_e \sqrt{(1 + 0.2M_e^2)}$$

On solving, we get, $M_e = 0.1445$

2.1.2 Flow of Air through the Convergent Divergent Nozzle causing Oblique Shock and expansion outside the nozzle

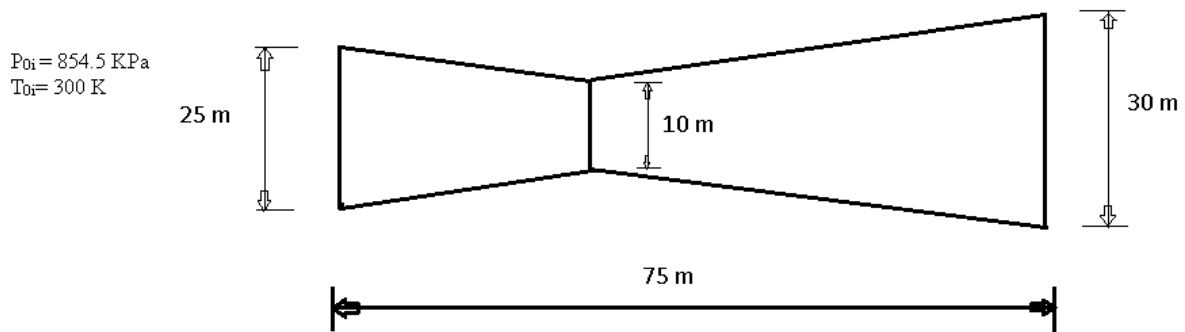


Fig. 2.2 2D Geometry of the Convergent Divergent Nozzle

At inlet,

$$A_i / A_{th} = 2.5 \rightarrow \text{IFT} \rightarrow M_i = 0.2$$

$$\frac{P_i}{P_{0i}} = 0.9607 \text{ and } \frac{T_i}{T_{0i}} = 0.9886$$

Let $P_{0i} = 854.5 \text{ KPa}$ and we get, $P_i = 820.92 \text{ kPa}$ and $T_i = 296.58 \text{ K}$

At the exit,

$$A_e / A_t = 3 \rightarrow \text{subsonic} \rightarrow M_e = 0.2$$

From IFT, $P_e / P_{0e} = 0.9725$ and $T_e / T_{0e} = 0.9921$

Since $P_{0e} = P_{0i} = 854.5 \text{ KPa}$ and $T_{0i} = 300 \text{ K} \rightarrow P_e = 831 \text{ KPa}$, $T_e = 297.63$

Also,

$$A_e / A_t = 3 \rightarrow \text{supersonic} \rightarrow M_e = 2.64$$

$$\text{From IFT, } P_e / P_{0e} = 0.04711 \text{ and } T_e / T_{0e} = 0.4177$$

$$\text{Since } P_{0e} = P_{0i} = 854.5 \text{ KPa and } T_{0i} = 300 \text{ K} \rightarrow P_e = 40.24 \text{ KPa, } T_e = 125.31 \text{ K}$$

Assume if there is a shock at the exit of the nozzle,

$$M_{ex} = 2.64 \rightarrow \text{From NST} \rightarrow M_{ey} = 0.5$$

$$\text{From NST, } P_{ey} / P_{ex} = 7.964 \text{ and } T_{ey} / T_{ex} = 2.2797$$

$$\text{As } P_e = P_{ex} = 40.24 \text{ KPa and } T_e = T_{ex} = 125.31, \text{ we get}$$

$$P_{ey} = 320.47 \text{ KPa and } T_{ey} = 285.67 \text{ K}$$

$$\text{Consider } P_b = 100 \text{ KPa}$$

Therefore, $P_{ey} > P_b \rightarrow$ oblique shock outside the nozzle

$$P_b / P_e = 2.48 \rightarrow \text{NST} \rightarrow M_{en} = 1.5 \rightarrow \text{NST} \rightarrow M_{2n} = 0.701, T_2 / T_e = 1.333$$

$$\text{Which gives, } T_2 = 167.03 \text{ K}$$

$$\text{Also, } T_2 / T_{02} = 167.03 / 300 = 0.556 \rightarrow \text{IFT} \rightarrow M_2 = 2$$

$$\text{Now, } M_{en} = M_e \sin \theta \rightarrow \theta = 34.62^\circ$$

and

$$M_2 = \frac{M_{2n}}{\sin(\theta - \delta)}$$

$$2 = \frac{0.701}{\sin(34.62 - \delta)}$$

$$\delta = 14.10^\circ$$

Now $M_2 = 2$, $\delta_2 = \delta = 14.10^\circ \rightarrow \theta_2 = 44.2^\circ$

$M_{2n} = M_2 \sin \theta_2 = 1.394 \rightarrow \text{NST} \rightarrow M_{3n} = 0.742$

$P_3 / P_2 = 2.1 \rightarrow P_3 = 210 \text{ KPa}$ ($P_2 = 100 \text{ KPa}$)

Since $P_3 > P_b$ (100 KPa), expansion will occur

At $M_2 = 2 \rightarrow \text{IFT} \rightarrow P_2 / P_{02} = 0.1278 \rightarrow P_{02} = 782.47 \text{ KPa}$

At $M_{2n} = 1.394 \rightarrow \text{NST} \rightarrow P_{03} / P_{02} = 0.964 \rightarrow P_{03} = 754.3 \text{ KPa}$

3-4- Expansion

$P_{03} = P_{04} = 754.3 \text{ KPa}$

$$\frac{P_4}{P_{04}} = \frac{100}{754.3} = 0.132$$

$\rightarrow \text{IFT} \rightarrow M_4 = 1.98$

2.2 Numerical Methodology

First the “Geometry” module in ANSYS Fluent has been used to create the 2-D geometry of the convergent divergent nozzle. Fig.2.3 shows the 2-D nozzle geometry, which cause a normal shock in the divergent part of the nozzle. Fig.2.5 shows the 2-D nozzle geometry, which cause an oblique shock and expansion outside the nozzle. Next, using the “Mesh” module, different meshes have been defined to obtain the desired results for the shocks. Figure 2.4 and Fig.2.6 shows the mesh for the two different geometries. The simulation parameters have been set in the “Setup” module, and shown in Table 2.1. These values have been chosen in such a way as to match the problem characteristics and the analytical solution parameters. Finally, the simulation has been run until convergence was achieved.

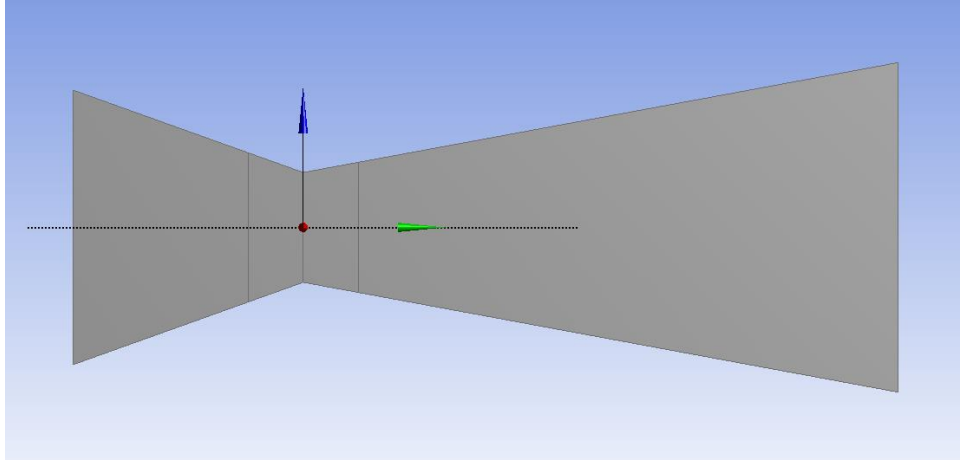


Fig.2.3 Geometry of C-D Nozzle causing Normal Shock

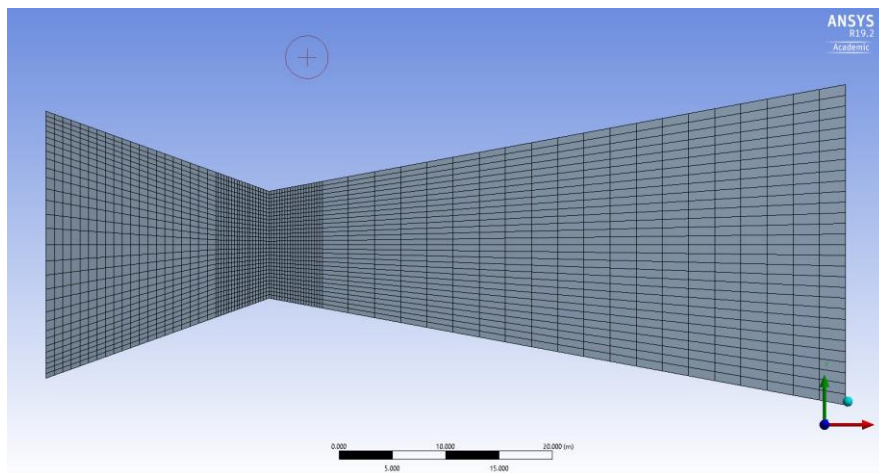


Fig.2.4 Mesh in C-D Nozzle causing Normal Shock

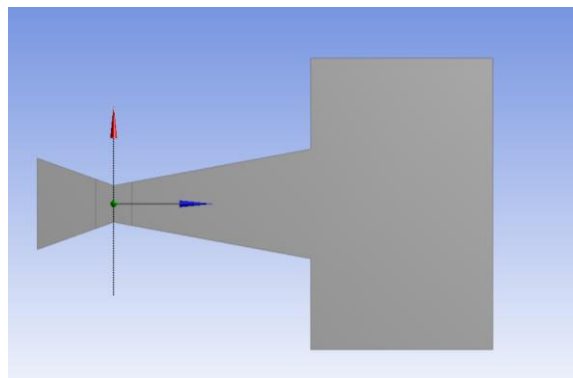


Fig.2.5 Geometry of the C-D Nozzle causing Oblique shock and Expansion

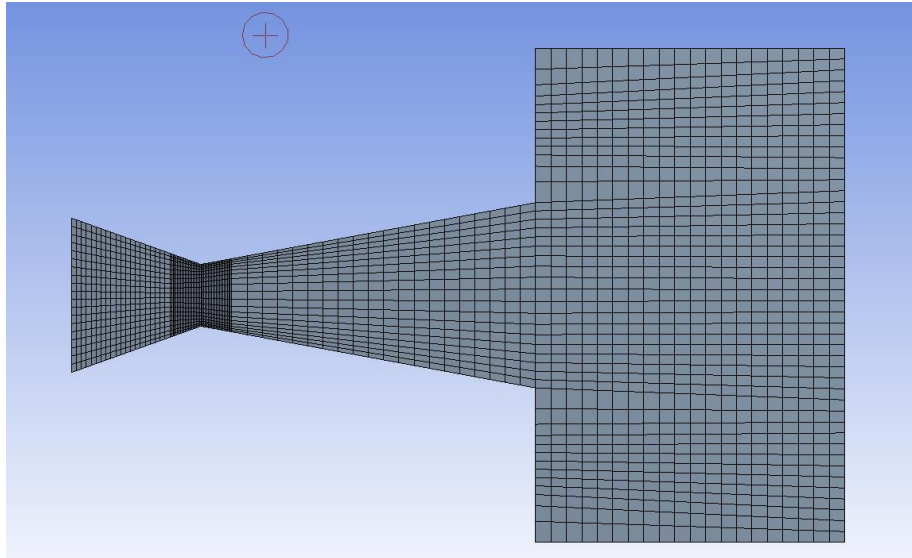


Fig.2.6 Mesh in C-D Nozzle causing Oblique shock and Expansion

Table 2.1 Simulation parameters in ANSYS Fluent

| Option settings | Selection |
|----------------------|-----------------------|
| Time | Steady |
| Models | Inviscid |
| Material | Ideal gas |
| Density | 1.2 kg/m ³ |
| Solver | Density based |
| Residuals Criteria | 0.001 |
| Time step size | 0.1 |
| Number of time steps | 500 |

2.3 Boundary Conditions

a) C-D Nozzle [Fig. 2.3]

Pressure Based: Inlet- $P_0 = 1 \text{ MPa}$ and $T_0 = 500 \text{ K}$

Pressure Based: Outlet- $P_b = 500 \text{ KPa}$

Walls: Zero Momentum

b) C-D Nozzle with Flow Domain [Fig.2.5]

Pressure Based: Inlet- $P_0 = 854.5 \text{ KPa}$ and $T_0 = 300 \text{ K}$

Pressure Based: Outlet- $P_b = 100 \text{ KPa}$

Walls: Zero Momentum

CHAPTER 3: DISCUSSION OF RESULTS

3.1 Converging Diverging Nozzle causing Nozzle shock

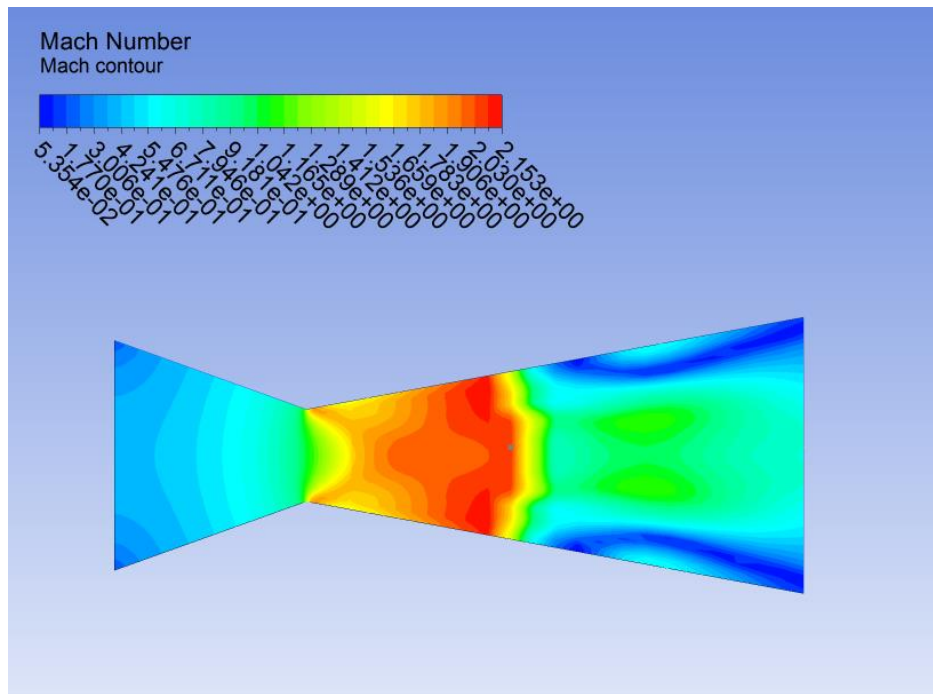


Fig.3.1 Mach Number Contour of the C-D Nozzle with a Normal Shock

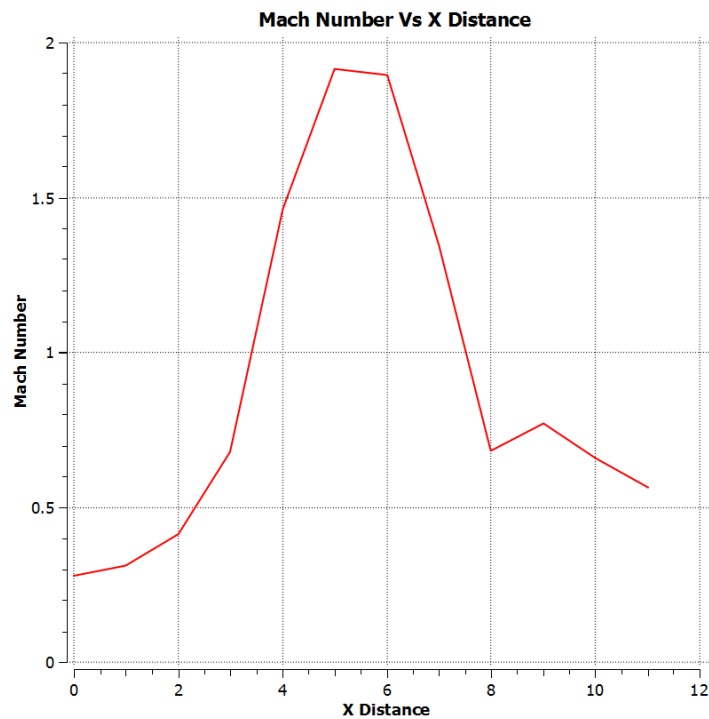


Fig.3.2 Mach Number Vs X Plot of the C-D Nozzle with a Normal Shock

From Fig. 3.2, we can observe that the Mach Number increases in the beginning of the nozzle, and in the divergent portion, the flow encounters a sudden shock due to which the Mach Number decrease drastically and reaches the subsonic limit. The average Mach number values at the exit and just after the shock are almost similar to the results developed through the analytical solution.

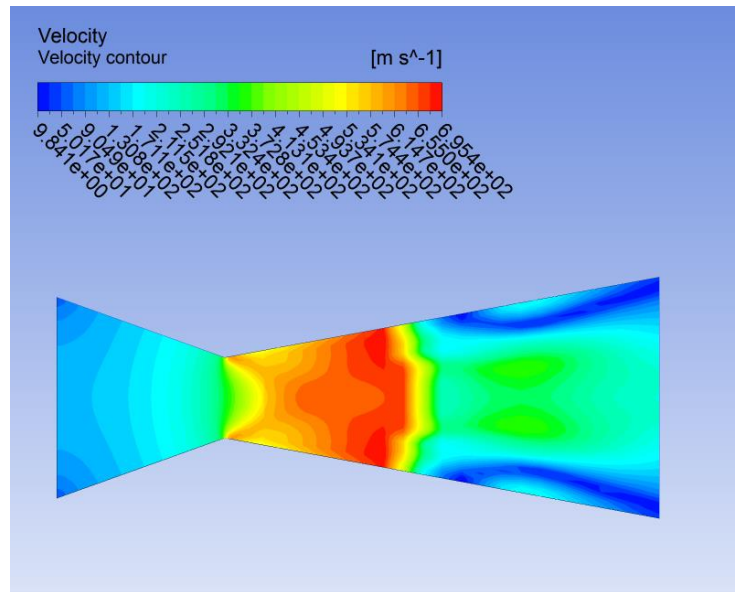


Fig.3.3 Velocity Contour of the C-D Nozzle with a Normal Shock

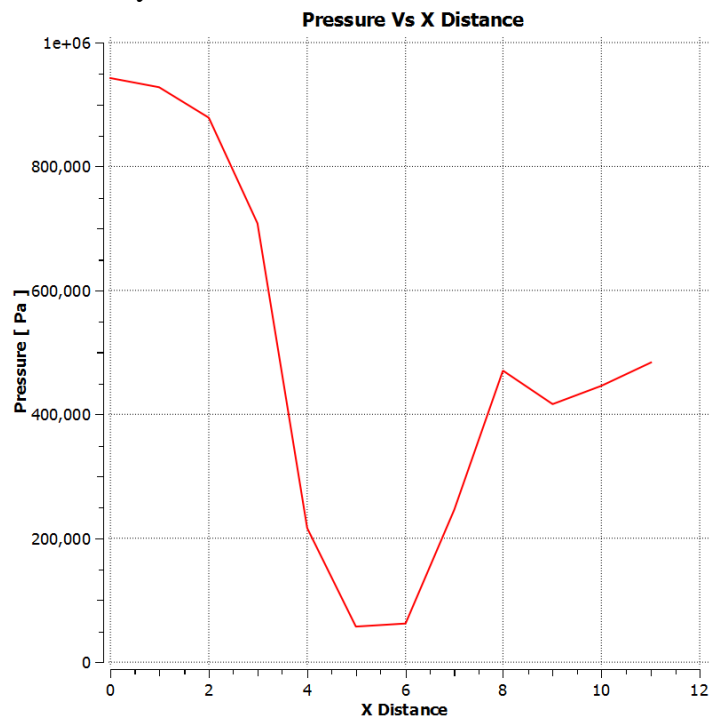


Fig.3.4 Pressure Vs X Plot of the C-D Nozzle with a Normal Shock

3.2 Converging Diverging Nozzle causing oblique shock and expansion

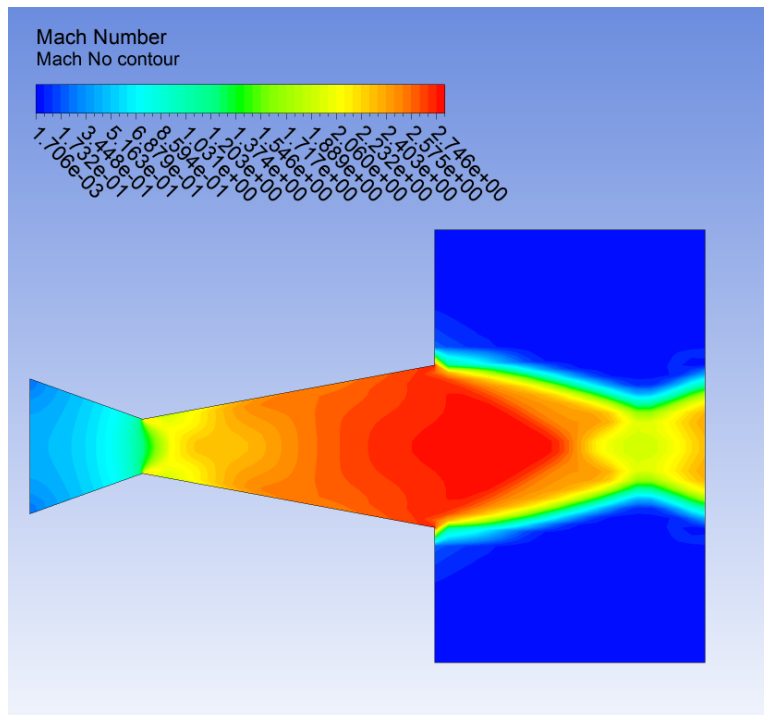


Fig.3.5 Mach Number Contour of the C-D Nozzle with an oblique shock and expansion

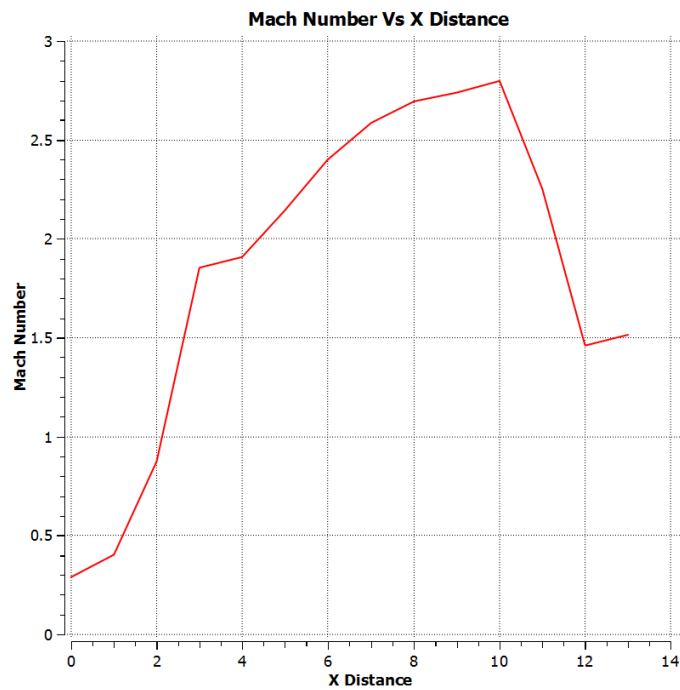


Fig.3.6 Mach Number Vs X Plot of the C-D Nozzle with an oblique shock and expansion

Similar to the previous case, here also the Mach decreases after the oblique shock , but unlike the earlier case, it again rise because of the expansion after the shock.

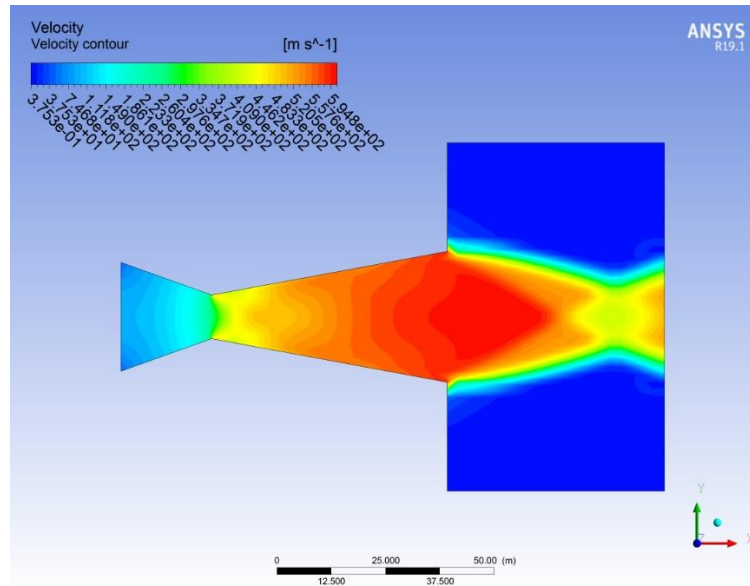


Fig.3.7 Velocity Contour of the C-D Nozzle with an oblique shock and expansion

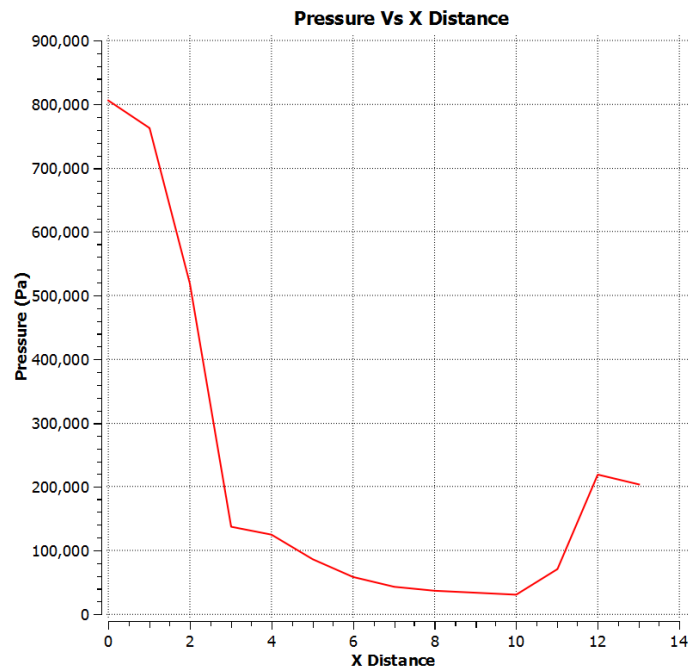


Fig.3.8 Pressure Vs X Plot of the C-D Nozzle with an oblique shock and expansion

CHAPTER 4: CONCLUSION

Ansys Fluent has been used to simulate two different steady incompressible inviscid flow cases; C-D Nozzle without and with the flow domain at the outlet to encounter several losses inside and outside the nozzle. The geometry and mesh characteristics of both cases have been chosen in such a way as to closely mimic that of the analytical solution and it was found that all variables presented in the results were in good agreement with the calculated analytical parameters.

References

1. Source: SlidePlayer. Published by Ashley Baker. <https://slideplayer.com/slide/6663703/>