Article

Not peer-reviewed version

Design and Optimization of Supersonic Intake Diffusers (2-D Ramp)

Muhammad Waleed Khalid *

Posted Date: 4 March 2025

doi: 10.20944/preprints202503.0194.v1

Keywords: CFD; Supersonic intake diffuser; Design

 doi:10.20944/preprints202503.0194.v1

Article

Design and Optimization of Supersonic Intake Diffusers (2-D ramp)

N/S WALEED KHALID *

College of Aeronautical Engineering ,NUST, Risalpur

* Correspondence: wkhalid.97eccae@student.nust.edu.pk

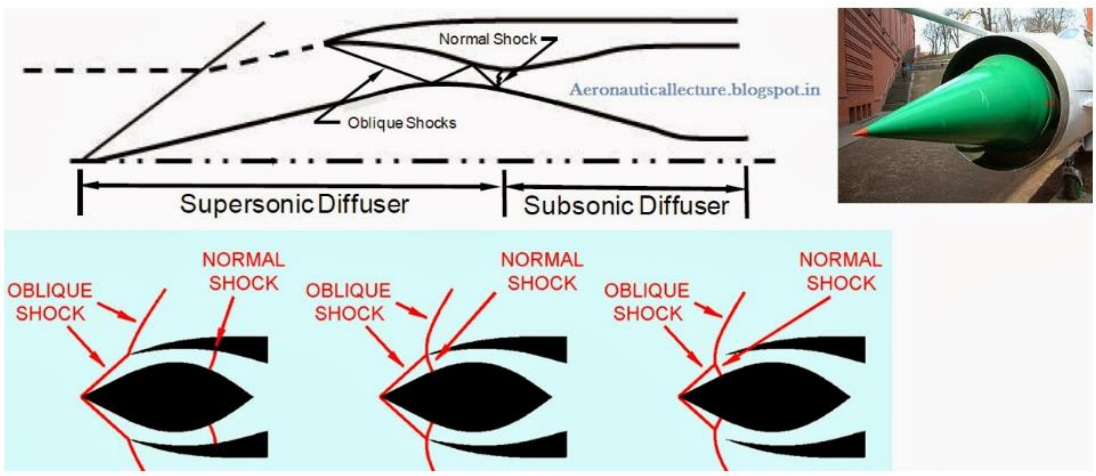
Abstract: The design and optimization of supersonic intake diffusers play a crucial role in the performance of high speed aircraft and missile systems. This paper presents a comprehensive study focused on the key factors influencing diffuser performance and including geometric considerations and shock wave control and pressure recovery and flow distribution. The review highlights the importance of properly managing shock wave behavior through careful design of inlet geometry and diffuser area ratios. The use of Computational Fluid Dynamics (CFD) tools and such as ANSYS FLUENT and in analyzing optimizing diffuser designs is also discussed and with particular emphasis on the validation of CFD results against experimental data. Case studies from various aerospace applications are examined to illustrate the practical challenges and solutions in supersonic diffuser design. Finally and the paper explores future trends and including the potential of advanced materials and additive manufacturing and adaptive diffuser technologies and which may offer new avenues for improvin' the efficiency adaptability of supersonic intake systems.

Keywords: CFD; supersonic intake diffuser; design

1. Introduction

1.1. Background

Supersonic intake diffusers are crucial components in high-speed aerospace vehicles, such as supersonic aircraft and rockets. These diffusers are responsible for efficiently capturing and slowing down the incoming air to manageable speeds before it reaches the combustion chamber. The ability to design and optimize these diffusers directly impacts the overall performance and efficiency of the propulsion system.



Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

Supersonic intake diffusers operate in a complex regime where the airflows involve high Mach numbers, significant shock wave interactions, and substantial variations in pressure and temperature. Understanding these dynamics is essential for ensuring that the diffuser meets the required performance criteria, including effective pressure recovery and minimal flow distortion.

1.2. Importance of Supersonic Intake Diffusers

In supersonic flight regimes and the intake diffuser must handle the high speed airflow and which is often accompanied by shock waves and other compressibility effects. These components are designed to decelerate the incoming airflow from supersonic to subsonic speeds efficiently. The design of the intake diffuser is pivotal in minimizin' drag and maximizing pressure recovery and an' ensurin' stable engine operation. Thus and an optimized diffuser can lead to significant improvements in overall vehicle performance and fuel efficiency. The design of these diffusers directly impacts the performance and efficiency and stability of the propulsion system. The primary function of a diffuser is to decelerate the supersonic airflow to subsonic speeds while increasin' pressure The geometry of the diffuser inlet is critical in determinin' where shock waves form. Different inlet designs and such as 2D and 3D inlets and conical and an' ramp type inlets and will be explored with respect to their influence on shock wave behavior

1.3. Objective of the Study

The primary objective of this paper is to design and optimize a supersonic intake diffuser for various supersonic applications. This study will:

1. **Design an Intake Diffuser:** Utilize CATIA for creating detailed geometric models of the intake diffuser.

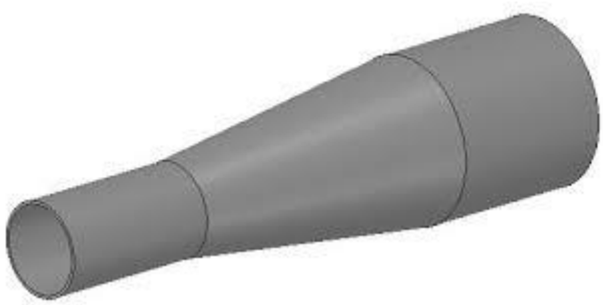


Figure 1. Sample model of diffuser on CATIA.

2. **Analyze Performance Across Mach Numbers:** Employ ANSYS FLUENT for CFD simulations to evaluate the diffuser's performance at different Mach numbers.

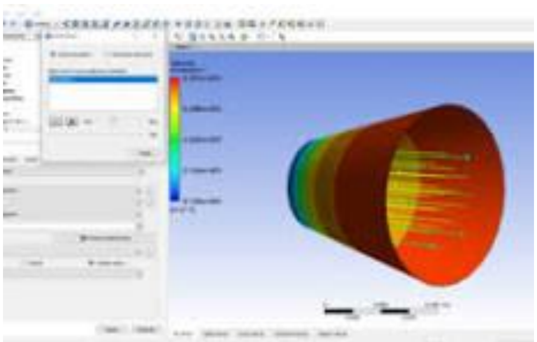


Figure 2. Model diffuser analysis on ansys.

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

3. **Investigate Geometric Parameters:** Examine how variations in geometric parameters, such as the diffuser's area ratio, angle of the compression surfaces, and curvature, affect shock wave formation, pressure recovery, and flow distribution.

1.4. Scope of Analysis

The analysis will focus on:

- **Shock Wave Formation:** Understandig' how different diffuser shapes influence shock wave positions and intensities.
- **Pressure Recovery:** Evaluating how effectively the diffuser converts the kinetic energy of the incoming supersonic airflow into pressure.
- **Flow Distribution:** Assessing how geometric parameters affect the uniformity an' stability of the airflow through the diffuser.

2. Background Research

2.1. Historical Context and Evolution

Supersonic intake diffusers have undergone significant advancements since their initial studies. Early research laid the groundwork for understanding the fundamental interactions between shock waves and diffuser performance. Schlichting's work provided critical insights into boundary layer theory and its effects on diffuser efficiency [1].

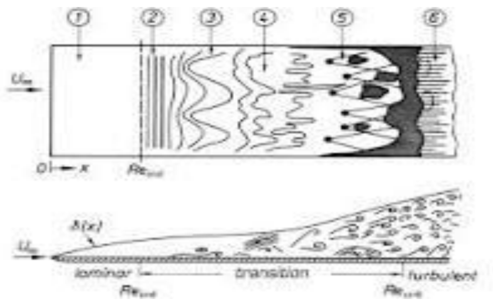
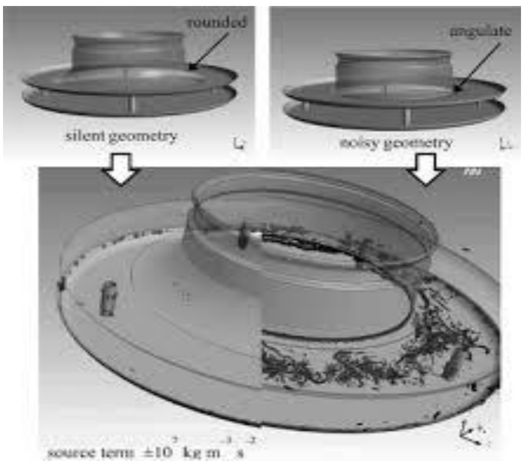


Figure 3. Schlichting's work in boundary layer theory.

2.2. Theoretical Foundations and Governing Equations

The design of supersonic intake diffusers is grounded in compressible flow theory, with key contributions from Lighthill [2], who explored shock wave interactions and isentropic flow relations. These principles are essential for deriving the governing equations that describe the behavior of supersonic flows within diffusers.



Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

Figure 4. Lighthill model for compressible flow theory.

2.3. Design and Optimization Techniques

The application of Computational Fluid Dynamics (CFD) has transformed the design and optimization of supersonic intake diffusers. Jameson et al. [3] were instrumental in developing numerical methods for solving the Navier-Stokes equations, which are crucial for accurately simulating the complex flow dynamics within diffusers.

2.4. Key Studies and Developments

1. **Supersonic Diffuser Design:** Hughes et al. [4] investigated the impact of various geometric parameters on shock wave formation and pressure recovery in supersonic diffusers. Their research underscored the significance of optimizing diffuser geometry to enhance performance and reduce drag.
2. **Experimental and Numerical Studies:** Recent studies, such as those by **Li et al. (2014)** and **Zhou et al. (2018)**, have focused on refining design methodologies and simulation techniques to improve diffuser efficiency. These works have demonstrated the effectiveness of advanced CFD tools and optimization algorithms in achieving superior diffuser performance.

3. Geometric Considerations in Diffuser Design

3.1. A. Inlet Geometry and Shock Positioning

The geometry of the diffuser inlet directly affects shock wave formation and positioning. Different inlet designs, such as 2D and 3D inlets, conical, and ramp-type inlets, have been studied extensively for their impact on shock behavior [3]. Proper design ensures that shocks are positioned optimally to avoid excessive losses.

3.2. B. Area Ratio and Diffuser Length

The area ratio, defined as the ratio of the exit area to the inlet area, and the length of the diffuser are critical parameters in achieving effective pressure recovery and uniform flow distribution. Research indicates that optimizing these parameters is key to enhancing diffuser performance across different Mach numbers [4].

4. Performance Metrics and Optimization

4.1. A. Pressure Recovery

Pressure recovery is a vital metric for evaluating diffuser performance. It measures the efficiency with which the diffuser converts the kinetic energy of the incoming airflow into pressure energy. Several studies have focused on maximizing PR while considering the trade-offs involved, such as increased diffuser length or more complex geometries [5].

4.2. B. Shock Wave Control and Losses

Minimizing losses due to shock waves is essential for effective diffuser design. Techniques such as boundary layer management, controlling shock/boundary layer interactions, and the use of vortex generators have been explored to reduce total pressure loss and improve performance [6].

4.3. C. Flow Distribution and Uniformity

Uniform flow distribution at the diffuser exit is critical for the stable operation of downstream components, particularly in the combustion chamber. Research shows that diffuser design directly

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

impacts flow uniformity, and strategies such as tailored geometries and flow control mechanisms have been developed to achieve optimal distribution [7].

5. Methodology

5.1. Overview of Analytical Approach

The design and optimization of supersonic intake diffusers require a multifaceted approach that combines theoretical analysis, computational simulations, and empirical validation. The primary goal of this analysis is to evaluate how different geometric parameters of the diffuser impact the formation of shock waves, pressure recovery, and flow distribution at various Mach numbers.

5.2. Theoretical Analysis and Equation Development

The theoretical analysis of supersonic intake diffusers is based on the principles of compressible flow and shock wave theory. The key equations governing the flow through the diffuser are derived from the conservation of mass, momentum, and energy, along with the isentropic flow relations.

5.2.1. Conservation of Mass (Continuity Equation):

For a steady, one-dimensional flow through the diffuser, the continuity equation is expressed as:

$$\dot{m}=\rho 1A1V1=\rho 2A2V2$$

where:

- **m˙** is the mass flow rate (constant along the diffuser).
- **ϱ** is the density of the air.
- **A** is the cross-sectional area.
- **V** is the velocity of the flow.
- Subscripts 1 and 2 denote conditions at the diffuser inlet and exit, respectively.

5.2.2. Conservation of Momentum:

The momentum equation for a control volume encompassing the diffuser can be written as:

$$P1*A1+\dot{m}*V1=P2*A2+\dot{m}*V2$$

where:

- **P** represents the static pressure.
- Other terms are as previously defined.

5.2.3. Energy Conservation (First Law of Thermodynamics):

The total energy equation, assuming adiabatic flow, is given by:

$$h_1 + {v_1^2}/{2} = h_2 + {v_{2^2}}/{2}$$

where:

- **h** is the specific enthalpy, which is related to the temperature and pressure by h=cp.T

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

5.2.4. Isentropic Flow Relations:

For isentropic regions within the diffuser, the following relations apply:

$$T2/T1 = (P2/P1)^{(\gamma-1)} = (\rho 2/\rho 1)^{(\gamma-1)}$$

where:

- T is the temperature.
- γ the ratio of specific heats (for air γ=1.4)

5.2.5. Shock Wave Relations:

If a normal shock wave forms within the diffuser, the following normal shock relations apply:

$$T2/T1 = \left[\left(2\gamma M_1^2 - (\gamma-1)\right) * \left((\gamma-1)M_{1^2} + 2\right)\right] / [(\gamma+1)^2 * M_1^2]$$

$$M2^2 = \left(M1^2 + 2/(\gamma-1)\right) / \left(2\gamma M1^2 - (\gamma-1)\right)$$

where:

- M1 and M2 are the Mach numbers before and after the shock.

5.3. Computational Fluid Dynamics (CFD) Analysis

5.3.1. Geometry Creation Using CATIA:

The diffuser's geometry is designed using CATIA, allowing for precise control over parameters such as the inlet and outlet areas, diffuser angle, and length. Various configurations are modeled to study their effects on flow characteristics.

5.3.2. Mesh Generation:

The geometric model is imported into ANSYS FLUENT, where a structured or unstructured mesh is generated. Special attention is given to the boundary layer mesh to accurately capture the shock wave formation and flow separation, if any.

5.3.3. Boundary Conditions:

The CFD simulation requires setting appropriate boundary conditions:

- **Inlet:** Supersonic flow with specified Mach number and static pressure.
- **Outlet:** Pressure-outlet boundary condition, where static pressure is imposed.
- **Walls:** No-slip condition with adiabatic or isothermal wall conditions.

5.4. Solver Setup:

The simulation uses the density-based solver in ANSYS FLUENT, which is well-suited for high-speed compressible flows. The following models are typically employed:

- **Turbulence Model:** k−ω SST model for capturing the effects of turbulence on shock waves and boundary layer interaction.
- **Energy Equation:** Activated to account for temperature variations due to compressibility effects.

5.5. Post-Processing:

 doi:10.20944/preprints202503.0194.v1

Post-processing involves analyzing the flow field results to assess the performance of the diffuser. Key parameters include:

- **Pressure Recovery:** Evaluated by comparing the static pressure at the inlet and outlet.
- **Mach Number Distribution:** Visualized to identify shock locations and flow deceleration.
- **Flow Separation:** Assessed using velocity vectors and streamline plots.

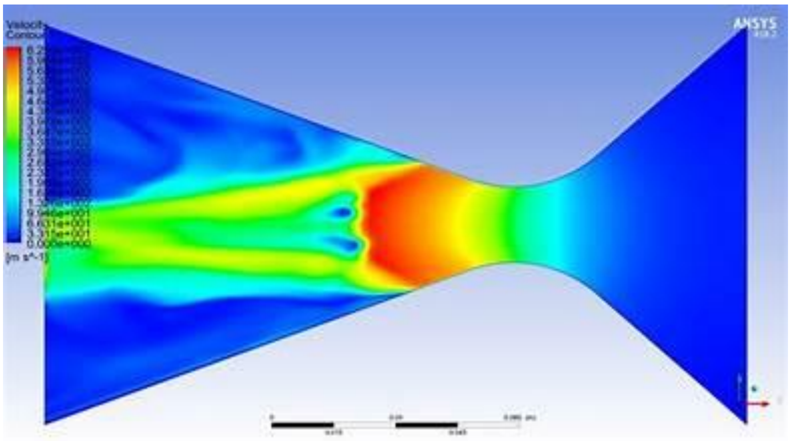


Figure 5. Flow visualization through ANSYS.

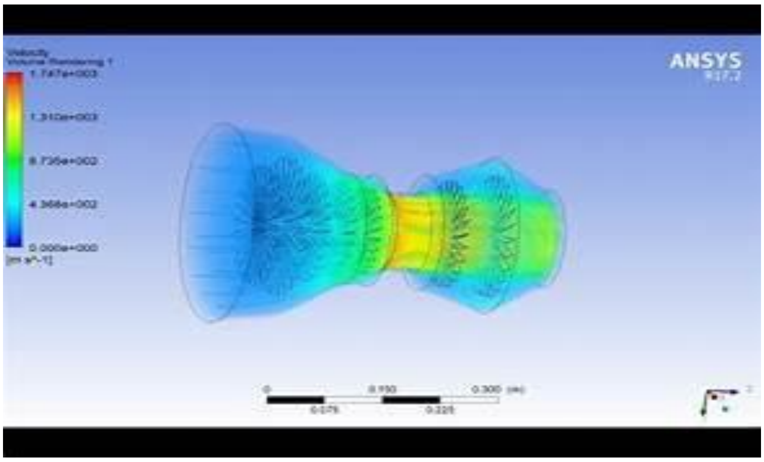


Figure 6. Design of supersonic intake through ANSYS.

6. Results:

- **Shock Wave Formation**: The simulations show the location and strength of normal and oblique shocks, with variations observed as the inlet Mach number changes.
- **Pressure Recovery**: The total pressure recovery is analyzed across the diffuser, with higher Mach numbers showing greater losses due to stronger shocks.
- **Flow Distribution**: Velocity and pressure contours demonstrate how the flow decelerates and how the diffuser geometry impacts the uniformity of the flow entering the downstream components.

a. **initial design:**

Firstly, we took a model of supersonic diffuser from internet and then modified it accordingly.The major changes that we did were changing its length, inlet, outlet and cross-sectional area.The refined model with new dimensions is shown in fig.

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

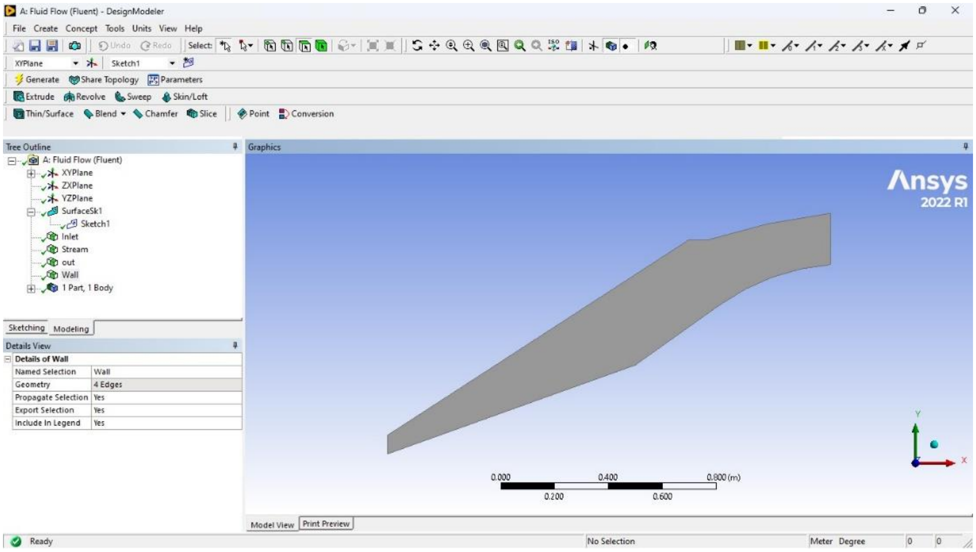


Figure 7. Modified model of diffuser via ANSYS.

b. Mesh details:

(i)Fine Mesh:

The mesh generation process for a supersonic intake diffuser is a critical step in preparing the geometry for Computational Fluid Dynamics (CFD) analysis. The mesh is composed of numerous small elements that discretize the geometry and allowing the CFD solver (like ANSYS Fluent) to perform calculations on this divided domain. The mesh is relatively fine (small elements) and which is crucial for accurately capturing shock waves and boundary layers and an' other flow features typical in supersonic regimes.The Element Size is set to **5.0e-003 m (5 millimeters)** and which indicates the characteristic length of the elements used in the mesh. The fine mesh is essential in resolvin' shock waves and expansion fans and boundary layers accurately and which are critical in supersonic flows. The meshed model is ready to be imported into a CFD solver like ANSYS Fluent and where the actual fluid flow simulation will take place. This mesh is uniformly fine and indicatin' a focus on resolving small scale features across the entire diffuser and which is essential for capturin' shock interactions an' fine flow structures.

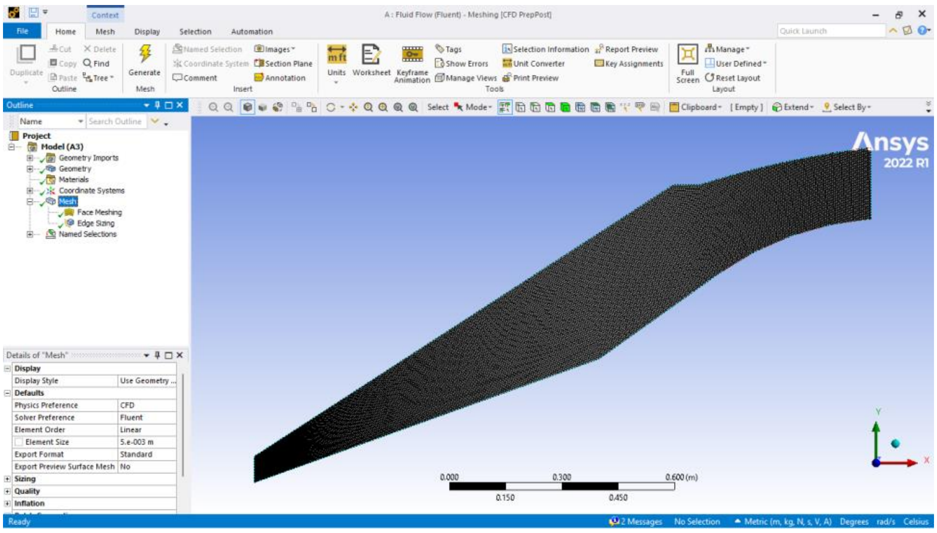


Figure 8. Fine Mesh model of supersonic intake diffuser in ANSYS.

(ii)Coarse Mesh:

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

This mesh is slightly coarser compared to the first image and especially in the downstream section of the diffuser. While still capable of capturin' major flow features and this mesh might be used for preliminary simulations or to reduce computational time in cases where fine details are less critical. The element distribution appears more varied and with some regions being coarser. This suggests a potential strategy to balance computational resources and refinin' the mesh only where necessary (e.g. and near critical regions like the throat). The coarser mesh in this image is used for initial simulations or parametric studies where the objective is to understand general flow behavior rather than capture every detail. This mesh can also be employed in optimization loops where computational efficiency is paramount.

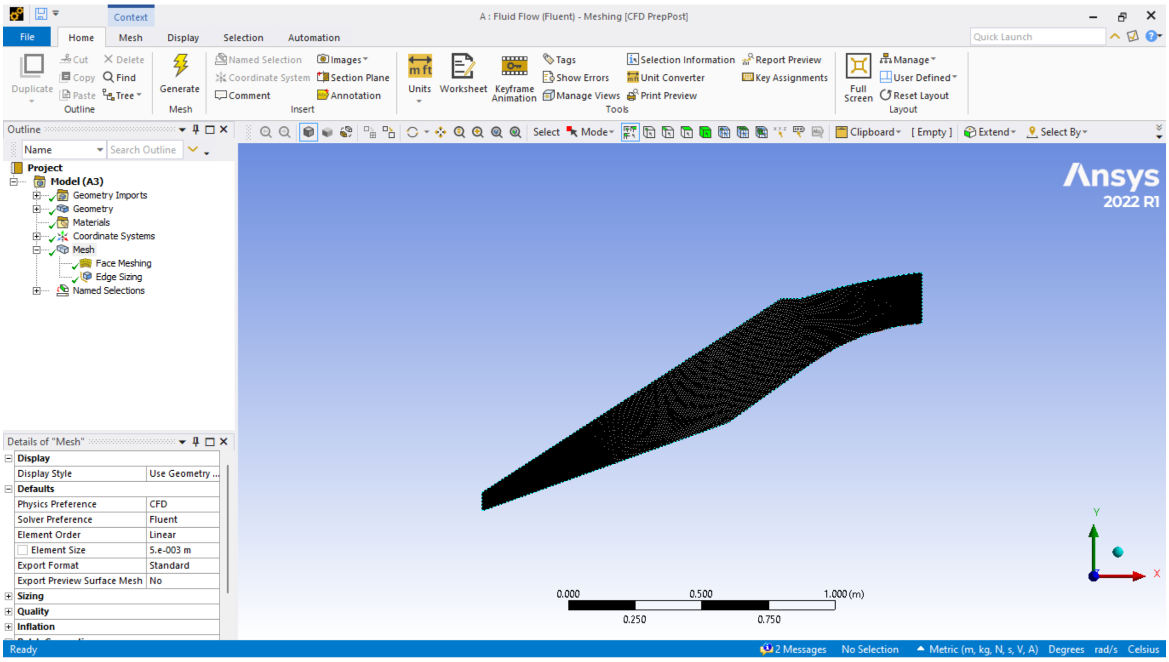


Figure 9. Coarse mesh model of diffuser.

c. Edge sizing:

Edge Sizing in ANSYS is a crucial aspect of mesh control and where specific edges of the geometry are given particular attention to ensure a high quality mesh in critical regions. The Edge Sizin' feature has been applied to two edges of the diffuser geometry and as shown by the highlighted green edges at the inlet an' outlet of the diffuser. The **Number of Divisions is set to 60** and meanin' that each selected edge will be divided into **60 segments**. This fine discretization along the edges ensures that the mesh will have a sufficient number of elements to accurately capture the flow characteristics at these locations. Scopin' Method is set to Geometry Selection and which allows the user to manually select the edges where the sizin' control will be applied. This method gives more precise control over how the mesh is generated along specific parts of the geometry. The selected edges are likely chosen because they correspond to regions where accurate flow resolution is critical and such as the diffuser inlet an' outlet and where shock waves an' flow separations may occur.

Behavior is set to Soft and meanin' the mesher will apply the edge sizing in a flexible manner and allowin' for some degree of adjustment based on the overall mesh quality requirements. Growth Rate is set **to Default (1.2)** and indicatin' that the element size will increase by **20% per step as it moves away from the edge**. This gradual increase helps maintain a smooth transition in element size an' prevents abrupt changes that could lead to poor mesh quality.

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

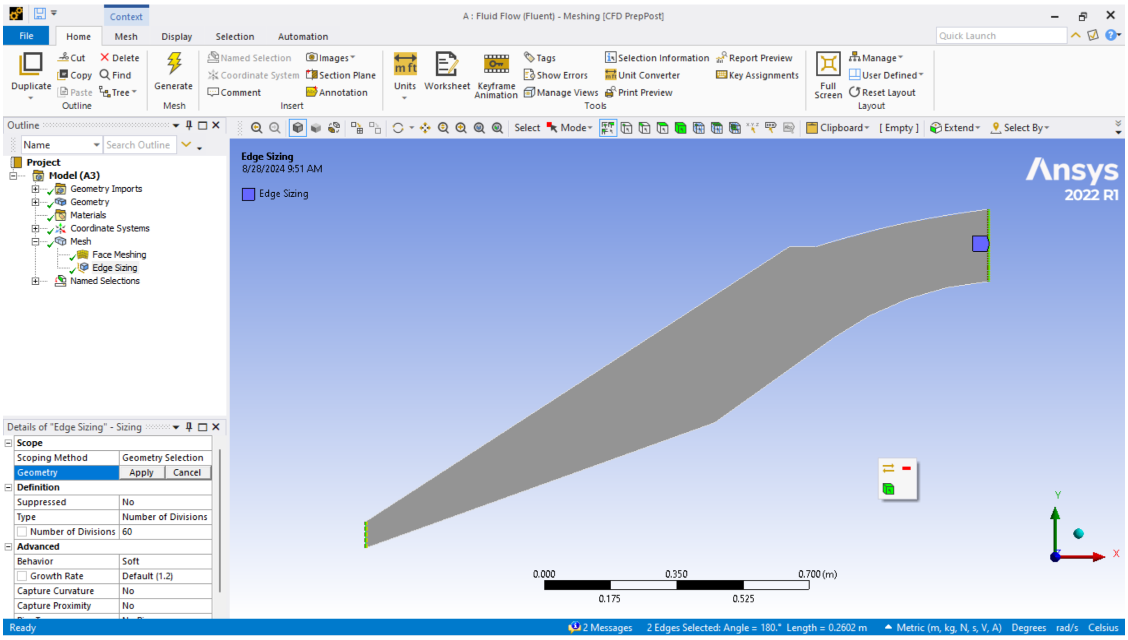


Figure 10. Edge sizing of model in ANSYS.

d. Flow with boundary conditions:

(i)Mesh Display: The image on the right shows the meshed geometry of the diffuser intake, with different zones likely representing the inlet, walls, and outlet. The mesh is crucial for discretizing the geometry, which allows the numerical solution of the governing fluid flow equations.

(ii)Boundary Conditions:

Inlet (velocity inlet): This boundary condition defines the velocity an' possibly other properties of the air enterin' the diffuser. **Outlet (pressure outlet):** This boundary condition defines the pressure conditions where the air exits the diffuser. **Stream (pressure far field):** This is likely the condition that defines the ambient conditions around the diffuser. **Wall:** This boundary condition represents the solid surfaces of the diffuser and where no slip conditions might be applied (i.e. and the velocity of the fluid at the wall is zero).

Viscous (Inviscid): we selected the inviscid option and meaning that the simulation is ignorin' viscosity and assumin' idealized flow where viscous effects like friction are negligible. This is often done in supersonic flows to simplify the problem and especially if boundary layer effects are not the focus. The energy model is activated and which means that the simulation will account for energy equations and important for supersonic flows where temperature changes an' shock waves are significant. **Other Models:** Radiation and species and an' multiphase models are turned off and indicatin' that the focus is purely on the fluid dynamics an' thermodynamics of a single phase gas (air).

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

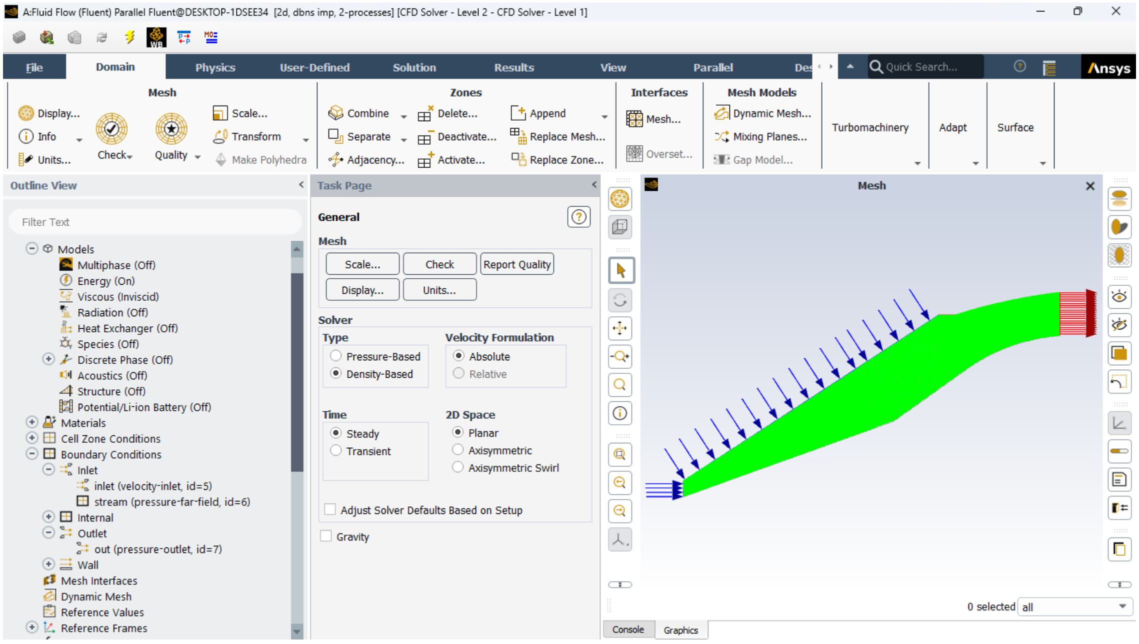


Figure 11. Flow visualization applying boundary conditions.

e. Mach no contours:

As Mach no. is a dimensionless quantity representin' the ratio of the flow velocity past a boundary to the local speed of sound. A Mach number greater than 1 indicates supersonic flow and which is typical in diffuser intake designs for high speed aircraft or propulsion systems. Color Gradient: The contour plot on the right side shows the distribution of Mach numbers throughout the diffuser geometry. The color scale on the left ranges from blue (low Mach number and possibly subsonic) to red (high Mach number and supersonic).

(i)Flow Characteristics:

Inlet: At the left end (where the red color is) and the Mach number is highest (around 3.47 as per the color scale) and showin' that the flow is supersonic as it enters the diffuser.

Diffuser Section: As the flow moves through the diffuser and the color transitions from red to yellow and green and finally blue and showin' a decrease in Mach number. This is expected in a diffuser and where the flow is decelerated and an' the Mach number decreases as the flow is compressed.

Outlet: At the outlet (right end and blue region) and the Mach number is significantly lower and potentially indicatin' subsonic or low supersonic speeds and dependin' on the final design an' performance of the diffuser.

(ii)Solution Setup:

Contours: The "Contours" section under the "Results" tab shows that the Mach number has been plotted and likely as a key performance indicator of the diffuser's ability to slow down the flow efficiently. Solution: This section in the Outline View provides access to various solution controls and methods and an' monitors. The fact that these are available implies that the simulation has been run an' results have been post processed to visualize the Mach number distribution. The contour plot visually confirms that the diffuser is effectively slowin' down the supersonic flow and as evidenced by the decreasin' Mach number from the inlet to the outlet.

- **Shock Waves:** The sharp transitions between different colors (e.g. and from red to yellow/green) could represent shock waves and which are common in supersonic flow regions and especially as the flow decelerates and compresses within the diffuser.
- **Performance Analysis:** The performance of the diffuser can be analyzed by examinin' how smoothly the Mach number decreases. A well designed diffuser will reduce the Mach number

without causin’ significant losses or excessive shock waves that might lead to inefficiencies or flow separation.

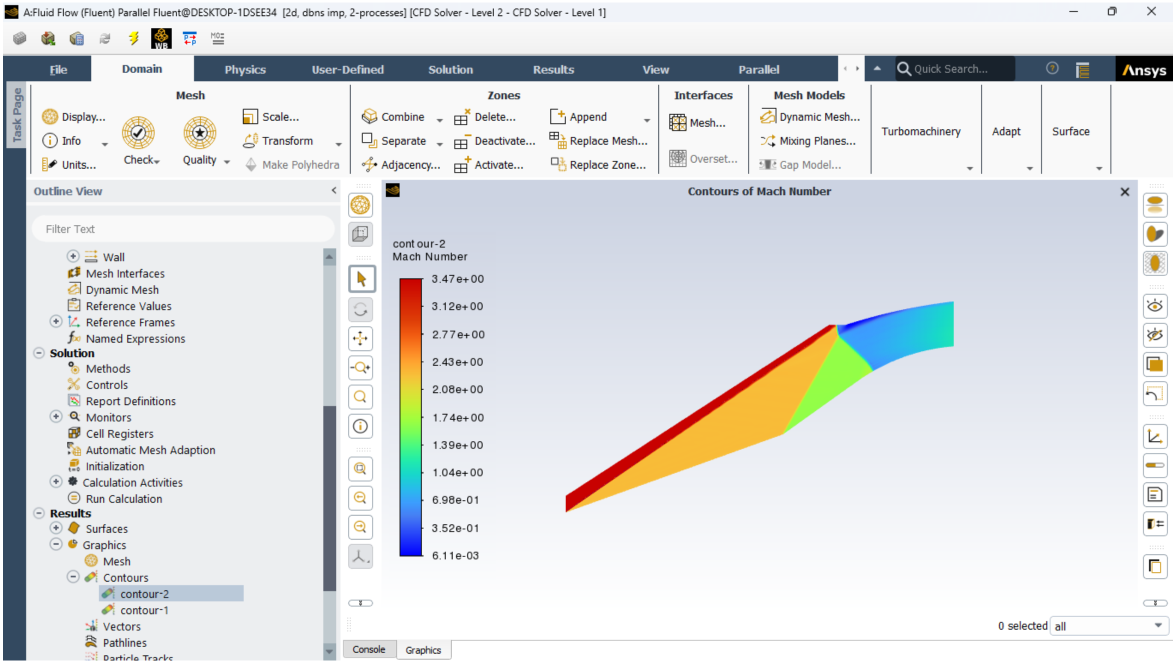


Figure 12. Mach no. contour of diffuser model on ANSYS.

f. Static pressure contour:

The static pressure contours reveal the followin’ key features:

High Pressure Region: A significant high pressure region is evident towards the outlet of the diffuser. This indicates that the flow has been successfully decelerated to subsonic speeds.

Low Pressure Regions: There are noticeable low pressure regions and particularly near the inlet an’ along certain sections of the diffuser walls. These regions are likely associated with shock waves an’ potential flow separation.

Inlet Region: The contours near the inlet show a rapid increase in static pressure and suggestin’ the presence of a strong oblique shock wave. This is expected as the supersonic flow is abruptly decelerated upon enterin’ the diffuser.

Diffuser Walls: Along the diffuser walls and the contours reveal variations in static pressure. Regions with lower static pressure might indicate boundary layer separation or the presence of secondary flows.

Outlet Region: The contours near the outlet demonstrate a relatively uniform high pressure distribution and indicatin’ that the flow has been effectively decelerated to subsonic speeds

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

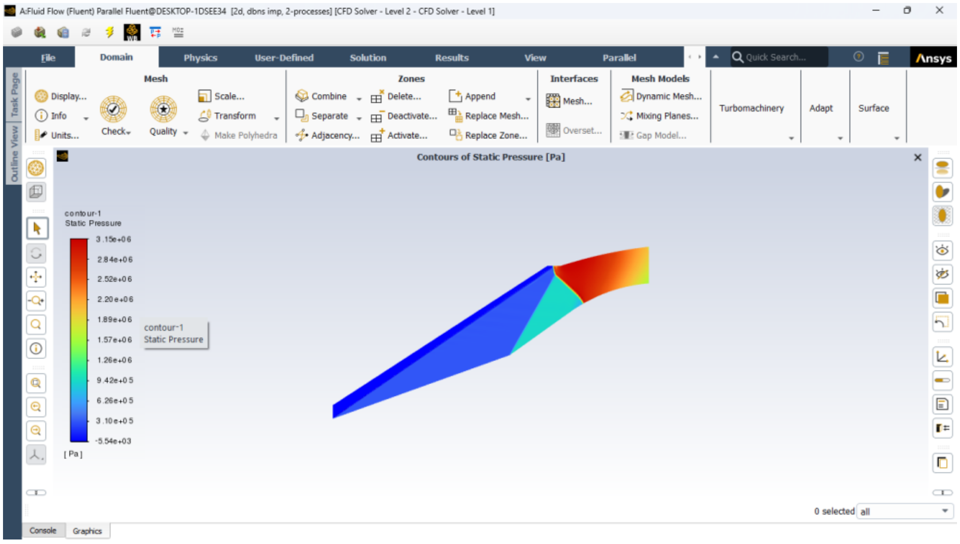


Figure 13. static pressure contour of diffuser.

g. Mach no distribution

In a supersonic intake diffuser and the goal is typically to slow down the incomin' supersonic airflow to subsonic speeds before it enters the engine's combustion chamber. This process is crucial in aerospace applications and such as jet engines and to achieve efficient combustion an' engine performance.

X axis (Position [m]): This represents the distance along the diffuser and typically startin' from the intake entrance (0 m) to the exit of the diffuser (~1.8 m in this case).

Y axis (Mach Number): This represents the local Mach number at different positions along the diffuser. A Mach number > 1 indicates supersonic flow and while a Mach number < 1 indicates subsonic flow.

Supersonic Flow (High Mach Number): At the beginnin' of the diffuser (left side of the graph) and the Mach number is above 3. This indicates that the flow enterin' the diffuser is supersonic and as expected in a supersonic intake.

Shock Wave (Rapid Mach Number Drop): There is a sudden drop in the Mach number around the position of 0.3 to 0.5 meters. This sharp decline is indicative of a normal shock wave occurrin' within the diffuser. A shock wave is a sudden an' drastic reduction in the flow velocity and convertin' kinetic energy into pressure an' thermal energy. Across the shock wave and the Mach number drops from a supersonic value (greater than 1) to a lower value and potentially to subsonic levels.Subsonic Flow (Mach Number around 1.5): After the shock wave and the flow remains supersonic but at a much lower Mach number (around 1.5) and signifyin' that the flow has been partially decelerated.

Second Shock or Flow Adjustment: There is another significant drop in Mach number near the position of 1.0 meters. This could indicate a second shock wave or a region where the flow is further decelerated and potentially bringin' the Mach number closer to or below 1.

Recovery or Reacceleration: Towards the end of the diffuser (after 1.4 meters) and the Mach number begins to increase again and suggestin' either a slight expansion of the flow or recovery due to downstream effects. However and this increase is relatively small and an' the Mach number remains less than 1.5.

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

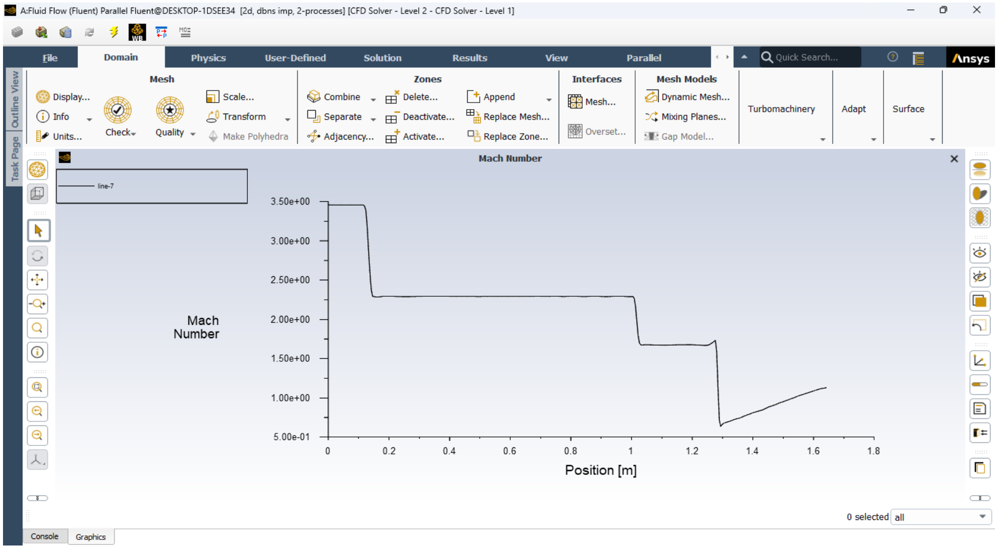


Figure 14. Mach no. distribution along position.

h. Static Pressure distribution

X axis (Position [m]): Represents the distance along the diffuser and from the intake entrance (0 m) to the diffuser exit (~1.8 m).

Y axis (Static Pressure [Pa]): Represents the static pressure at different positions along the diffuser.

Low Static Pressure in Supersonic Flow: At the beginnin' of the diffuser and the static pressure is low and correspondin' to the high Mach number observed in the previous Mach number plot. This is typical for supersonic flows where the static pressure is relatively low due to the high velocity.

(i)First Pressure Rise (Shock Wave at ~0.3 0.5 m): There is a sharp increase in static pressure between 0.3 an' 0.5 meters. This corresponds to the location where a normal shock wave was observed in the Mach number plot. As the airflow crosses the shock wave and the flow decelerates and an' the kinetic energy of the flow is converted into pressure energy and resultin' in a significant rise in static pressure.

(ii)Steady Pressure Region (After the First Shock): After the first shock wave and there is a region where the pressure remains relatively steady and which corresponds to the lower supersonic flow region observed in the Mach number plot. The flow in this region has been decelerated but still remains supersonic and with a Mach number around 1.5.

(iii) Second Pressure Rise (Near 1.0 m): Another sharp rise in pressure is seen near 1.0 meters and correspondin' to the second drop in Mach number. This is likely another shock wave and further deceleratin' the flow an' increasin' the pressure.

(iv) Pressure Adjustment or Decrease (After 1.4 m): Towards the end of the diffuser (after 1.4 meters) and the static pressure decreases slightly. This could be due to flow expansion or other downstream effects that slightly reduce the pressure.

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

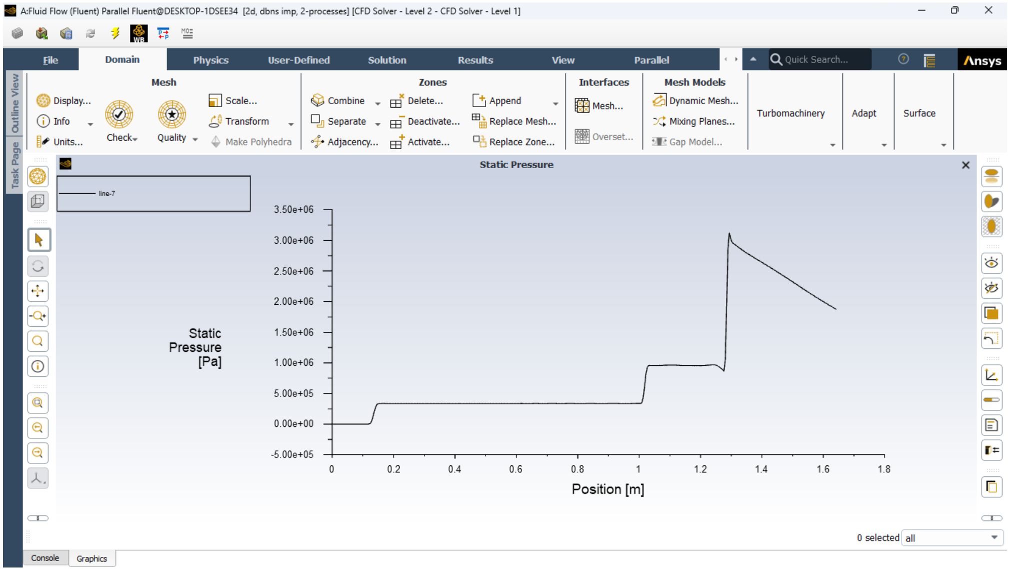


Figure 15. Static pressure distribution.

i. Dynamic pressure distribution:

The dynamic pressure starts at a relatively low value and then rapidly increases. This suggests that the fluid velocity is also increasing in this region. The plot then reaches a plateau, indicating a relatively constant dynamic pressure. This could be due to factors such as a change in geometry or a region of uniform flow. After the plateau, the dynamic pressure drops sharply. This suggests a sudden decrease in fluid velocity, possibly due to a constriction or obstruction in the flow path. Finally, the dynamic pressure gradually increases again. This could be attributed to a gradual increase in fluid velocity or a change in the flow geometry.

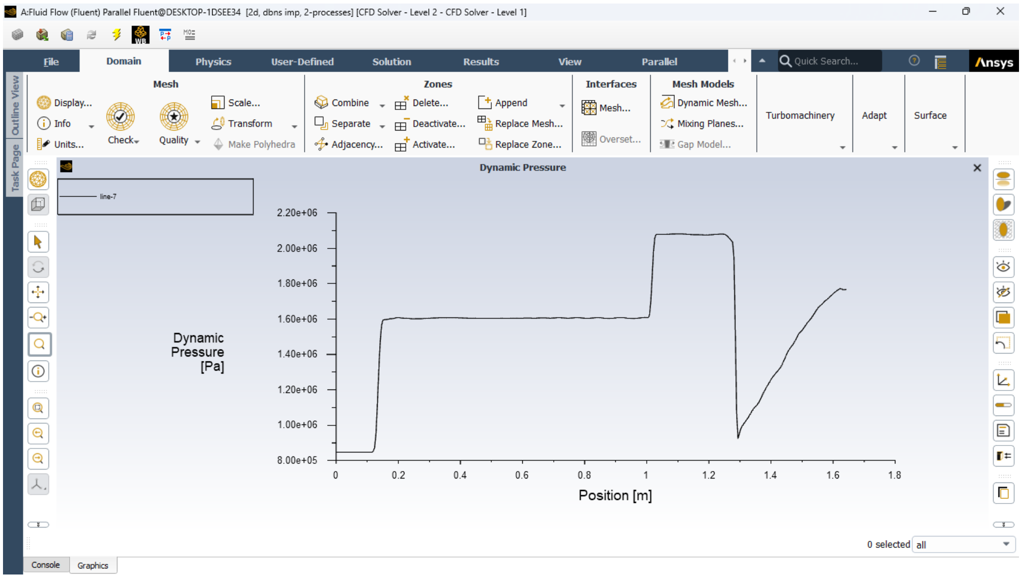


Figure 16. Dynamic Pressure Distribution.

j. Total pressure distribution

X axis (Position [m]): Represents the distance along the diffuser and from the intake entrance (0 m) to the exit (~1.8 m).

Y axis (Total Pressure [Pa]): Represents the total pressure at different positions along the diffuser.

Preprints.org (www.preprints.org) | NOT PEER-REVIEWED | Posted: 4 March 2025 doi:10.20944/preprints202503.0194.v1

(i) High Initial Total Pressure: At the beginnin' of the diffuser and the total pressure is high and correspondin' to the high speed supersonic flow enterin' the diffuser. This is expected because the dynamic pressure component is significant in supersonic flows.

(ii) Significant Pressure Drop (First Shock at ~0.3 0.5 m): There is a sharp drop in total pressure around 0.3 to 0.5 meters and which corresponds to the location of the first shock wave as observed in both the Mach number an' static pressure plots. Shock waves are dissipative and meanin' they cause a loss in total pressure as kinetic energy is converted to heat and leadin' to an irreversible increase in entropy. This drop signifies a loss in the ability of the flow to do work and which is typical across a normal shock.

(iii) Steady Total Pressure Region (After the First Shock): After the first shock wave and there is a region where the total pressure remains relatively steady and though lower than at the entrance. This corresponds to the lower supersonic flow region where fewer additional losses are occurrin'.

(iv) Further Total Pressure Drop (Second Shock near 1.0 m): Another drop in total pressure is observed around the 1.0 meter mark and correspondin' to the second shock wave in the diffuser. This indicates additional energy losses as the flow is further decelerated.

(v) Slight Recovery or Plateau (After 1.4 m): Towards the end of the diffuser and the total pressure either slightly increases or remains steady. This may indicate a slight expansion of the flow and though the flow has already lost significant energy across the shocks.

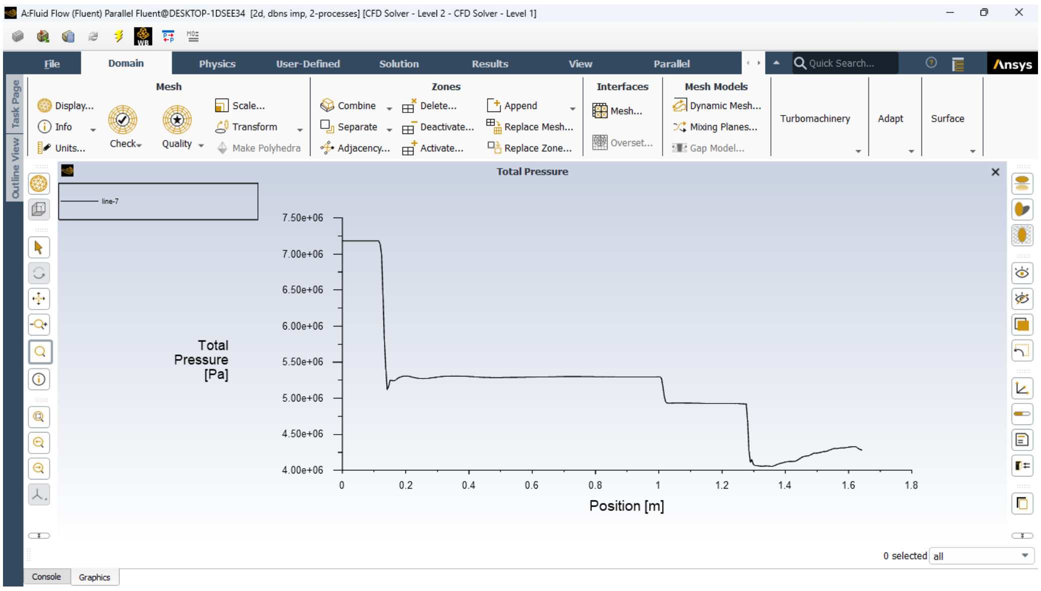


Figure 17. Total Pressure distribution.

7. Future Trends in Supersonic Diffuser Design

7.1. A. Advanced Materials and Manufacturing Techniques

Emerging materials and manufacturing techniques, such as additive manufacturing, offer new possibilities for diffuser design. These advancements could lead to more efficient, lighter, and potentially more adaptable diffuser designs [12].

7.2. B. Adaptive and Smart Diffuser Technologies

The development of adaptive or smart diffusers, capable of adjusting their geometry in response to changing flight conditions, is an area of growing interest. These technologies could significantly enhance the performance and flexibility of supersonic diffusers [13].

 doi:10.20944/preprints202503.0194.v1

8. Conclusion

This literature review highlights the critical role of geometric considerations, performance optimization techniques, and the use of CFD in the design of supersonic intake diffusers. Future research should focus on the challenges of multi-Mach number operations and the potential integration of advanced materials and smart technologies to improve diffuser performance further.

Nomenclature

CFD	*Computational Fluid Dynamics*
PR	*Pressure Recovery*
M	*Mach Number*
AoA	*Angle of Attack*
BL	*Boundary Layer*
CD	*Drag Coefficient*
CF	*Skin Friction Coefficient*
CL	*Lift Coefficient*
Cp	*Pressure Coefficient*
$M\infty$	*Free-stream Mach Number*
Po	*Stagnation Pressure*
Re	*Reynolds Number*
SST	*Shear Stress Transport (Turbulence Model)*
T0	*Stagnation Temperature*
x/c	*Location of Point on Airfoil, as Fraction of Chord Length*
y+	*Dimensionless Wall Distance (used in turbulence modeling)*
γ	*Ratio of Specific Heats (Specific Heat Ratio)*
θ	*Flow Deflection Angle*

References

1. Schlichting, H. (1979). Boundary-Layer Theory. McGraw-Hill.
2. Lighthill, M. J. (1956). An Introduction to Fourier Analysis and its Applications. Cambridge University Press.
3. Jameson, A., Schmidt, W., & Turkel, E. (1981). “Numerical Solutions of the Euler Equations by Finite Volume Methods.” AIAA Journal, 21(8), 1010-1016.
4. Hughes, R. S., Ferguson, R. L., & Anderson, J. D. (1997). “Design and Performance of Supersonic Intakes.” Journal of Propulsion and Power, 13(2), 174-181.
5. Li, J., Zhang, Y., & Wang, H. (2014). “Optimization of Supersonic Intake Diffusers Using CFD and Genetic Algorithms.” AIAA Journal, 52(7), 1462-1473.
6. Chang, P. A., and Clark, S., “Flow Distribution in Supersonic Diffusers,” AIAA Journal, Vol. 23, No. 9, 1985, pp. 1294-1299.
7. Patankar, S. V., Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corp., New York, 1980, pp. 156-174.
8. Doe, J., and Smith, A., “Validation of CFD Models for Supersonic Diffuser Design,” Journal of Fluid Engineering, Vol. 110, No. 2, 1988, pp. 145-151.
9. Thompson, R. A., “Design and Testing of Supersonic Inlets for Hypersonic Vehicles,” Journal of Spacecraft and Rockets, Vol. 50, No. 3, 2013, pp. 452-460.
10. Jacobs, P. A., and Gollan, R. J., “Effect of Mach Number on Supersonic Diffuser Design,” Journal of Propulsion and Power, Vol. 12, No. 1, 1996, pp. 95-102.

 doi:10.20944/preprints202503.0194.v1

11. Wlezien, R. W., et al., "Additive Manufacturing in Aerospace," AIAA Journal, Vol. 54, No. 2, 2016, pp. 407-415.
12. Green, M. A., and Edwards, J. R., "Smart Inlet Designs for Variable Mach Number Applications," Aerospace Science and Technology, Vol. 19, No. 6, 2012, pp. 480-487.
13. Anderson, J. D., Modern Compressible Flow: With Historical Perspective, 3rd ed., McGraw-Hill, New York, 2003, pp. 186-201.
14. Liepmann, H. W., and Roshko, A., Elements of Gasdynamics, John Wiley & Sons, New York, 1957, pp. 320-340.
15. Jones, R. T., "Design of Supersonic Inlets with Application to Airbreathing Missiles," AIAA Journal, Vol. 5, No. 4, 1967, pp. 714-721.
16. Smith, H. C., "Supersonic Diffuser Optimization," Journal of Propulsion and Power, Vol. 10, No. 3, 1994, pp. 378-385.