Rowan University

Modeling of J350W Rocket Motor

Group #1

Zach Pedrick, James Lee, Glen Vosgerichian
Spring 2025

Abstract

The purpose of this study was to model supersonic flow in the AeroTech J350W rocket nozzle using 2D axisymmetric CFD in ANSYS Fluent. Iterative refinement of mesh quality, solver settings, and boundary conditions allowed clear visualization of shock diamonds and flow expansion. Calculated thrust was 762.5 N—within 9.4% of expected peak thrust from numerical and manufacturer data. This validated our approach and demonstrated the effectiveness of CFD in predicting rocket nozzle performance.

Table of Contents

Abstract..... 1
Problem Statement and Significance..... 3
Geometry and domain..... 4
Grid Generation Strategy and Grid Study..... 6
Inlet Conditions..... 10
Solver Selection Strategy..... 11
Results..... 13
Discussion and Future Works..... 15
References..... 17
Appendix..... 18

Problem Statement and Significance

The goal of this project was to model supersonic flow through a solid rocket motor nozzle and calculate the resulting thrust using computational fluid dynamics (CFD). The client is interested in evaluating nozzle performance for a solid rocket motor under high-pressure, high-temperature conditions where compressible flow effects dominate.

The rocket motor selected was the AeroTech J350W, a commercially available motor with known thrust performance data. The problem required simulating the expansion of high-energy combustion gases through the nozzle and into ambient conditions, while accurately capturing features such as shock diamonds and supersonic acceleration.

Boundary and environmental conditions included fixed inlet pressure and temperature representative of peak combustion conditions, and a constant ambient pressure and temperature at the outlet. The simulation was carried out using a 2D axisymmetric domain to simplify the problem while maintaining geometric fidelity.

This work is significant because it demonstrates how CFD can be used to predict the behavior of supersonic nozzles, an essential component in propulsion system design. Understanding how flow properties change through a nozzle is critical for evaluating performance, optimizing design, and ensuring safe operation in both aerospace and rocketry applications. Additionally, CFD enables design validation without the cost or complexity of physical testing.

The modeling and simulation skills developed in this project are broadly applicable to launch vehicle design, jet propulsion systems, and atmospheric reentry studies—fields where high-speed compressible flow must be accurately resolved.

Geometry and domain

The nozzle geometry used in this simulation was based on technical drawings of the AeroTech J350W rocket motor. The initial geometry included the full internal combustion chamber and a large external domain, extending five times the nozzle exit diameter in the radial direction and ten times the combustion chamber length axially. This setup was intended to minimize boundary effects and allow for full plume expansion.

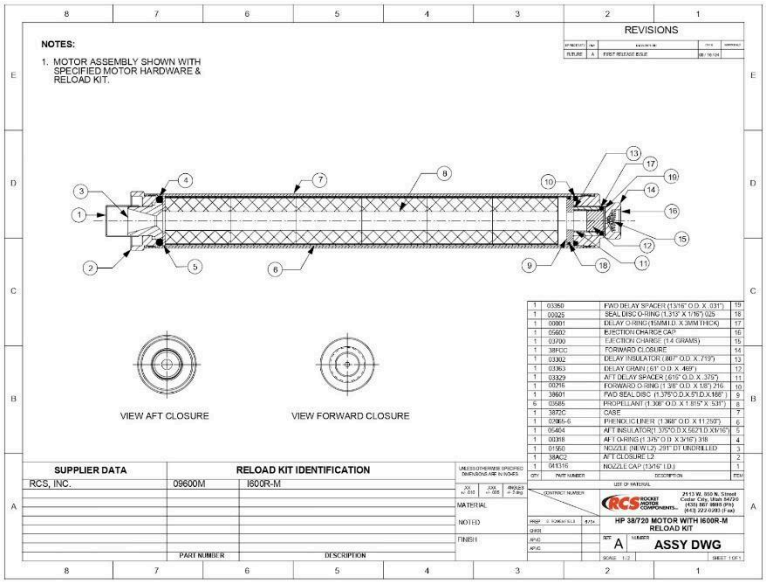


Figure 1 – Aerotech J450W Solid Rocket Motor Technical Drawing

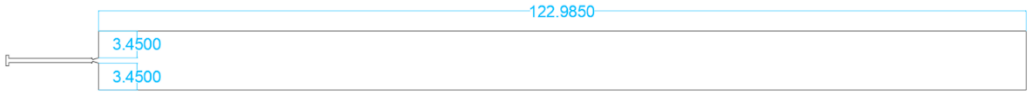


Figure 2 – Rocket Nozzle Preliminary Geometry

However, early simulations revealed that this oversized domain dramatically increased element count and computation time without improving accuracy near the nozzle exit. As a result, the external domain was significantly reduced while still capturing the interaction of the exhaust with the surrounding atmosphere.

Additionally, the internal combustion chamber was removed from the simulation domain. This simplification avoided convergence issues and allowed a pressure inlet to be applied directly at the nozzle inlet, representing chamber pressure conditions without modeling combustion.

The domain was further simplified by modeling only half of the nozzle as a 2D axisymmetric system. This reduced computational cost while preserving the rotational symmetry of the actual nozzle.

A small region was added above the nozzle in the final geometry to better visualize how the exhaust plume interacts with ambient air. This improved the accuracy of shock diamond formation and plume structure.

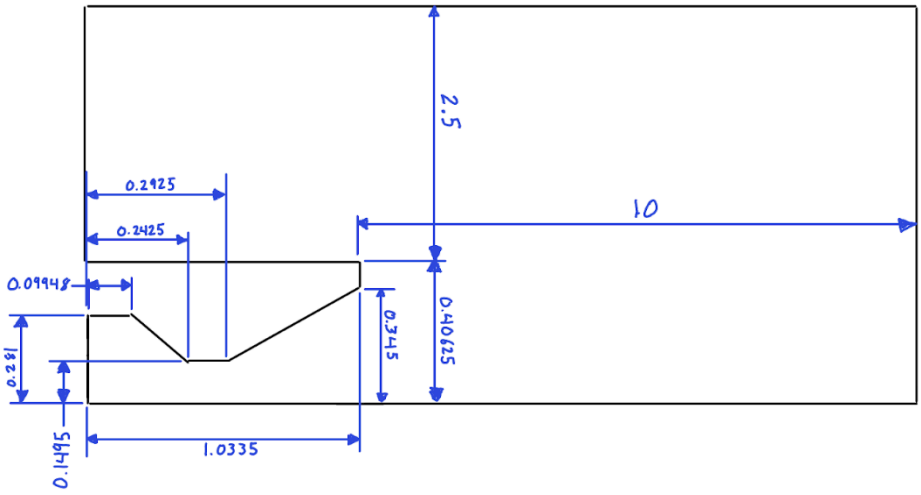


Figure 3 Rocket Nozzle Refined Geometry

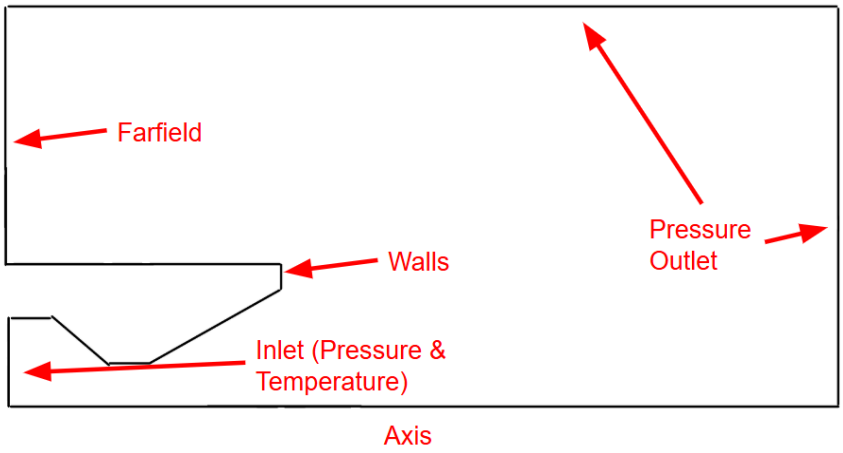


Figure 4 – Refined Boundary Conditions

Grid Generation Strategy and Grid Study

A structured multizone meshing approach was used to discretize the 2D axisymmetric domain of the nozzle. Initially, coarse meshes with a combination of triangular and quadrilateral elements were used to reduce computational time. However, these early meshes failed to resolve shock structures, exhibited poor convergence, and resulted in unrealistic flow behavior.

The team iteratively refined the mesh over 17 documented versions, adjusting edge sizing, biasing, and element types to improve resolution near critical flow regions—particularly the throat and nozzle exit. Quadrilateral elements were favored in the final mesh to improve skewness and orthogonal quality.

Mesh refinement parameters included:

- Smaller element sizes at the nozzle throat (down to 0.1 mm)
- Bias factors applied along edges to concentrate nodes where finer meshing was desired.
- Reduction of the external domain to limit element count
- Mesh metrics such as skewness, aspect ratio, and orthogonal quality were tracked at each stage

Table 1- Mesh Validation Summary

	Mesh 1	Mesh 2	Mesh 3	Mesh 4	Mesh 5
# of Elements	70,502	141,771	381,880	187,433	27,980
Element Quality	0.85791	0.8096	0.70738	0.79731	
Aspect Ratio	1.7797	1.9861	2.7808	2.0555	1.3557
Skewness	5.59E-02	6.26E-02	7.52E-02	6.52E-02	2.96E-02
Orthogonal Quality	0.99098	0.99003	0.98909	0.98963	0.99599
Run Time	~5-10 min.	42 min.	107 min.	~60 min.	~5-10 min.
Result	Converged	Did not converge	Did not converge	Did not converge	converged

The primary parameter monitored throughout the grid study was thrust at the nozzle exit, calculated using density, velocity, and pressure. Residual convergence and flow visualization (i.e., presence of shock diamonds) were also evaluated for each mesh.

During the mesh validation process, the following trends were observed:

- Meshes with more than ~100,000 elements failed to converge
- Meshes with lower aspect ratio and skewness performed best
- Over-refining the mesh led to longer runtimes but diminishing returns in solution quality

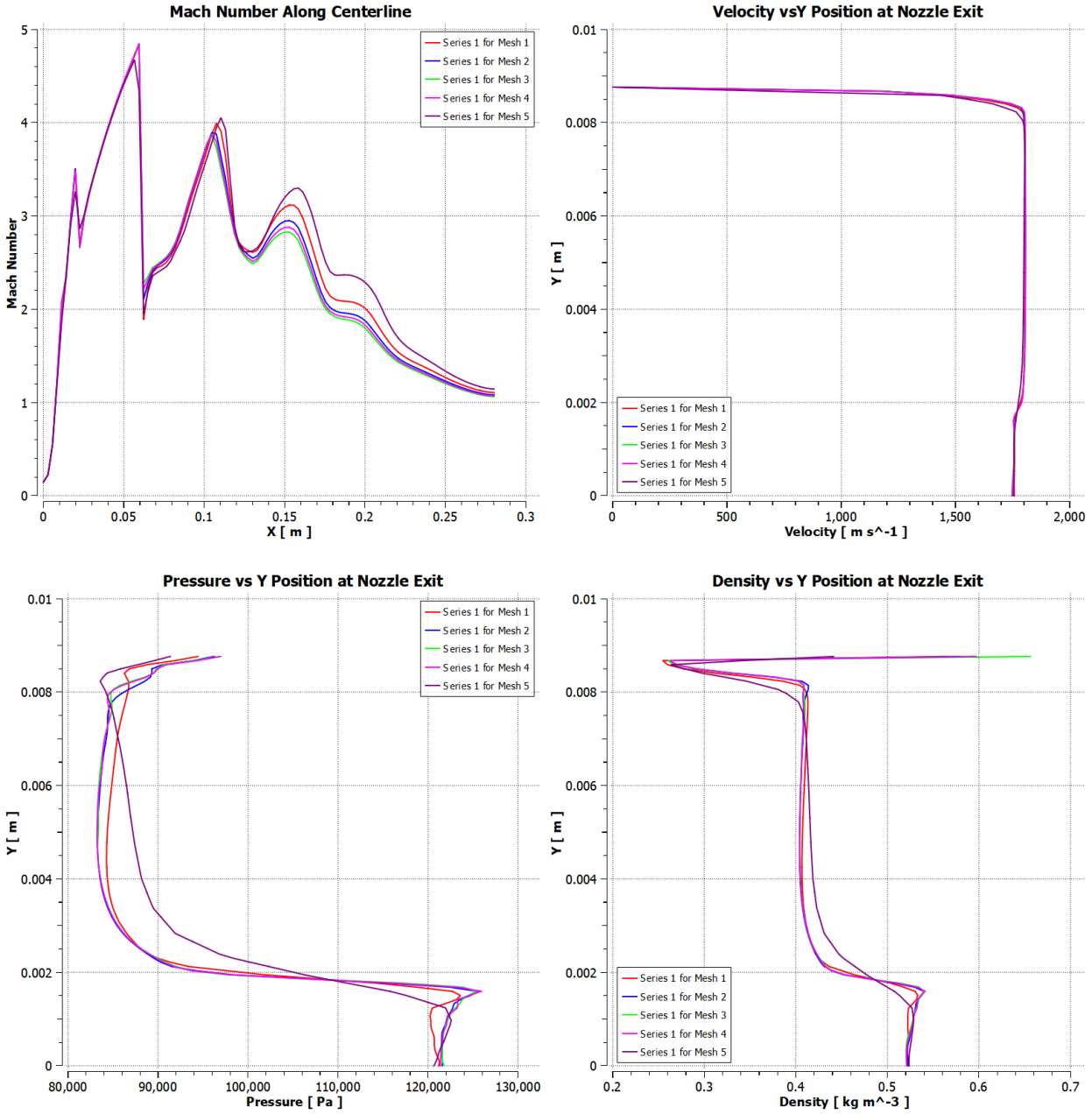


Figure 5 – Graphs of Validation Parameters for Various Mesh Densities

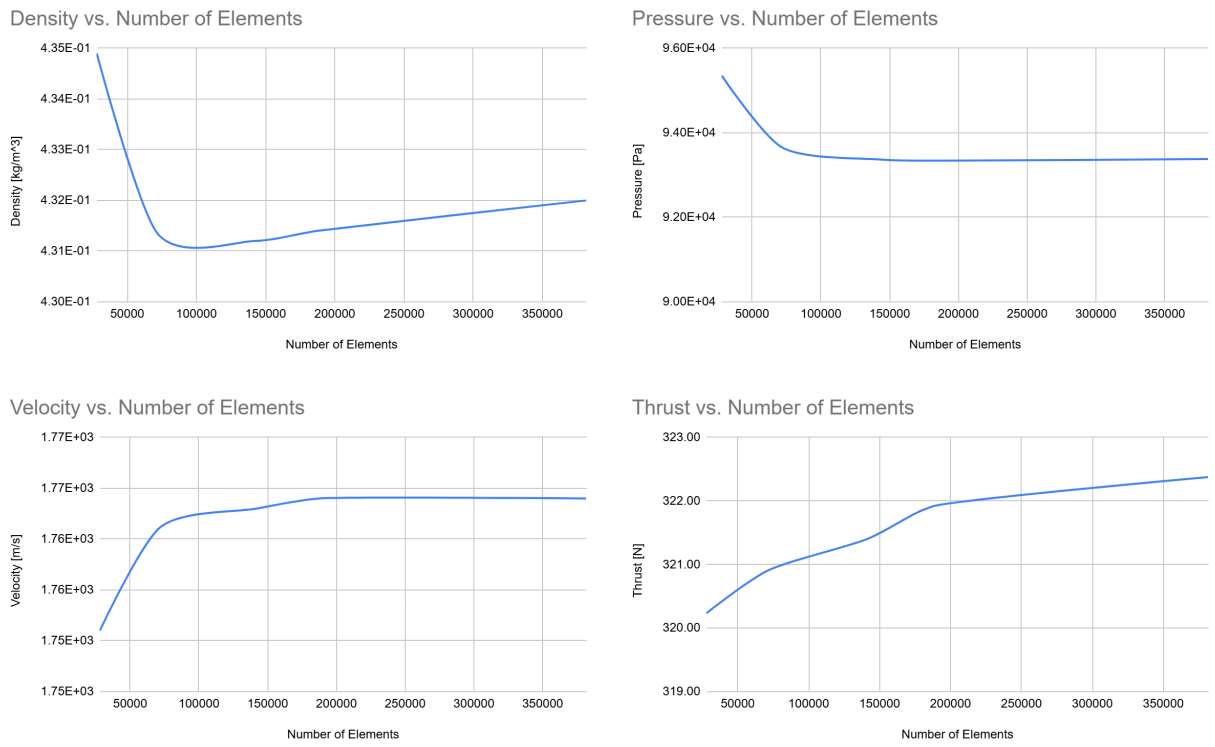


Figure 6 – Average Validation Parameter at Nozzle exit vs Number of Elements

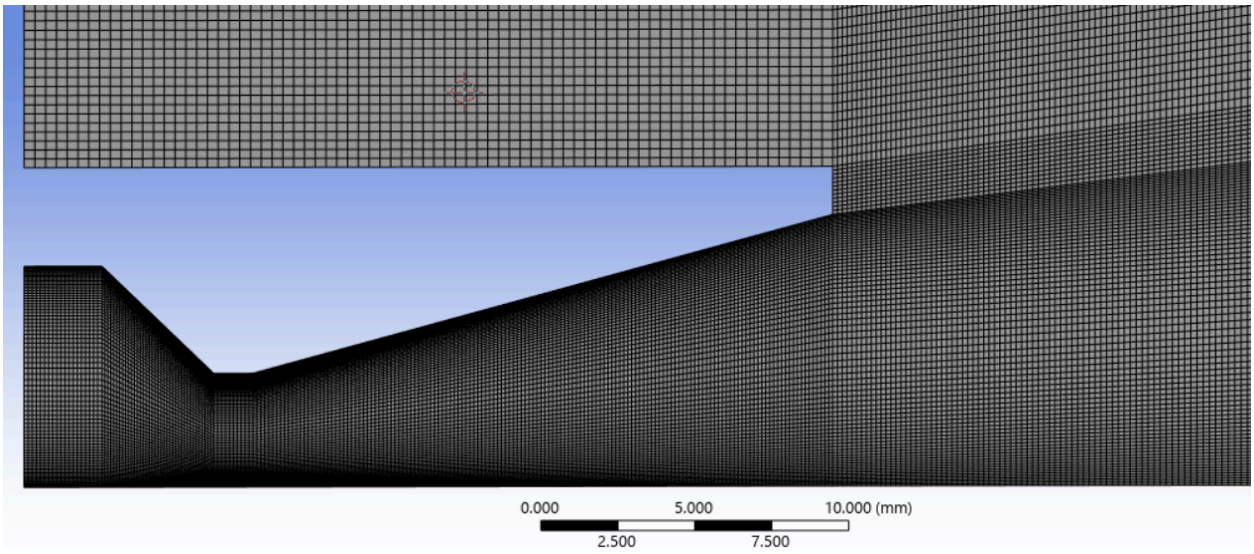


Figure 7 – Final Mesh Geometry at Rocket Nozzle

Based on the above graphs mesh 2 (iteration 13) was selected as the final mesh. As it balanced computational cost with physical accuracy as it was just past the point of being creating a mesh independent solution. This mesh had ~142,000 elements, a

skewness of 0.0626 and an aspect ratio of 1.9861. And while it did not converge we believed this could be fixed by making edits to the solver. As the only residual that did not converge was continuity.

Inlet Conditions

In order to set up the simulation the pressure and temperature of the combustion chamber was needed. After some research a rocket propellent burn simulation program called OpenMotor was found. This program had propellent data such as specific heat ratio, combustion temperature, characteristic velocity, exhaust gas molar mass, propellant density, burn rate coefficient, and pressure exponent for the Aerotech White Lightning fuel used in the J350W.

Throughout the course of this project multiple strategies were employed to determine the chamber pressure. The first was a method outlined in a paper on solid propellant formulations[1]. This use average mass flow through the nozzle (calculated using the propellent mass and burn time), nozzle throat area, and the characteristic velocity to estimate a chamber pressure use the following equation: $P_c = \frac{\dot{m}c^*}{A_t}$

Using this an estimated chamber pressure for the J350W of 742 psi was calculated. Because this method used an average mass flow rate a more accurate method was sought. After some research a new method was found on website called Richard Nakka's Experimental Rocketry Website and under the section on chamber pressure the following equation was found[2]: $P_c = \left[\frac{A_b}{A_t} * \frac{a^*\rho_P}{\sqrt{\frac{k}{R^*T_c}\left(\frac{2}{k+1}\right)^{\frac{k+1}{k-1}}}}\right]^{\frac{1}{1-n}}$

Where:

$A_b = grain\ burn\ area$

$k = specific\ heat\ ratio\ of\ the\ propellent$

$\rho_P = density\ of\ the\ propellent$

$R = specific\ gas\ constant\ of\ the\ exhaust$

$T_c = combustion\ temperature$

$a = burn\ rate\ coefficient$

$n = pressure\ exponent$

Using this equation a chamber pressure of 530 psi was calculated. But after completing the mesh validation and a low thrust was found it was thought that the inlet conditions could be incorrect. So more research was done in order to find a way to calculate the chamber pressure using the actual thrust produced by the motor. This was found on Richard Nakka's Experimental Rocketry Website and under the section on motor thrust.

$$F = A_t * P_c * \sqrt{\frac{2k^2}{k-1}\left(\frac{2}{k+1}\right)^{\frac{k+1}{k-1}}\left[1 - \left(\frac{P_e}{P_c}\right)^{\frac{k-1}{k}}\right]} + \left(P_e - P_a\right) * A_e$$

Where:

$P_e = \ nozzle\ exit\ pressure\ (assumed\ to\ be\ 1\ atm)$

$P_a = \ ambient\ air\ pressure$

Plugging in the thrust data from thrustcurve.org the chamber pressure vs time can be calculated. This method resulted in an initial chamber pressure of 1738 psi and this is what was used as the inlet pressure for the final simulation.

Solver Selection Strategy

Multiple solvers and turbulence models were tested during the course of the project to achieve physically accurate and numerically stable results for the supersonic nozzle flow. The evolution of solver strategy played a critical role in overcoming convergence issues and capturing key flow features like shock diamonds and velocity gradients.

Solver Evolution and Rationale

The initial approach used a pressure-based solver, which is standard for steady-state simulations. However, due to the highly compressible, high-speed flow regime of rocket exhaust, this setup failed to accurately resolve shock structures or converge reliably in early meshes.

The team transitioned to a density-based solver, better suited for supersonic flows where large changes in density and pressure occur. While this solver improved physical realism, it made convergence challenging and computationally expensive, particularly on under-refined meshes.

Ultimately, the team returned to the pressure-based solver, which—when paired with appropriate turbulence models, discretization schemes, and relaxation factors—produced the most stable and accurate results.

Viscosity Model Selection

Three viscosity models were tested:

- Spalart–Allmaras: Provided quick results but did not resolve shocks well.
- Reynolds Stress Model (RSM): Physically detailed but very slow and unstable.
- Realizable k-epsilon: Offered the best trade-off between realism and solver performance. It effectively captured boundary layer behavior and shock structures, and was used in the final validated simulation.

Final Solver Configuration

The final validated setup used:

- Solver Type: Pressure-based (coupled scheme)
- Viscosity Model: Realizable k-epsilon
- Energy Equation: Enabled
- Methods:
    - Second-order upwind for all variables
    - High order term relaxation for all variable: 0.1
- Controls:
    - Pressure: 0.5
    - Momentum: 0.5
    - Density: 0.5
    - Body Forces: 1
    - Turbulent Kinetic Energy: 0.85
    - Turbulent Dissipation Rate: 0.85
    - Turbulent Viscosity: 1.0
    - Energy: 0.65

These settings produced well-behaved residuals, resolved shock diamonds, and yielded exit flow values that aligned with physical expectations.

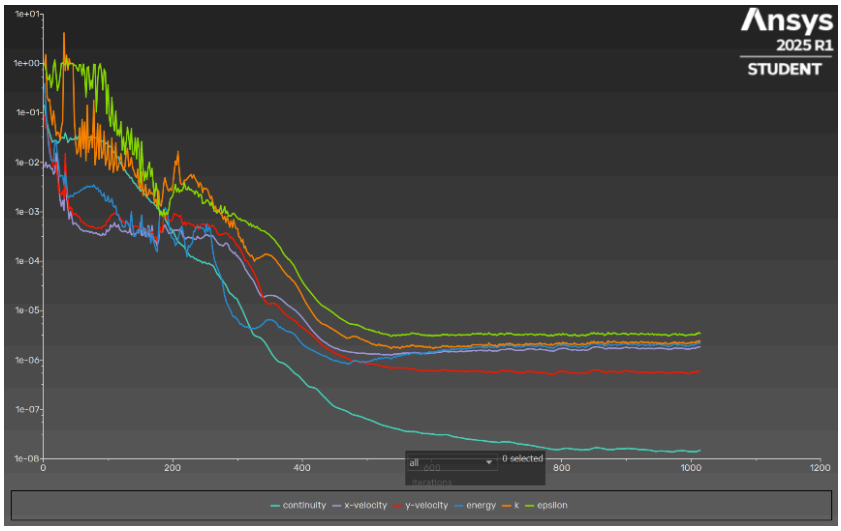


Figure 8 – Residuals for Final Solver Selection

Results

The final simulation, conducted using the refined mesh and the realizable k-epsilon turbulence model with the pressure-based coupled solver, produced high-quality outputs representative of supersonic nozzle flow behavior. This section summarizes the primary flow characteristics and quantitative results obtained from the simulation.

Flow Field Contours

- Mach Number Contours:
  The flow accelerated through the nozzle, reaching supersonic speeds at the throat and forming visible shock diamonds in the external plume. These features confirmed proper expansion behavior and compressibility effects.
- Pressure Contours:
  A sharp drop in pressure was observed from the inlet to the nozzle exit, consistent with expansion through a converging-diverging nozzle.
- Velocity Contours:
  The maximum velocity at the nozzle exit was approximately 1770 m/s, in line with theoretical expectations for supersonic exhaust under high chamber pressure.

- Density Contours:
  As expected, density decreased as the flow expanded and accelerated, reaching a minimum at the exit.

Table 2 – Nozzle Exit Values (Averaged)

Quantity	Value
Pressure	212 kPa
Velocity	1770 m/s
Density	0.977 kg/m³
Exit Area	2.41 × 10^{-4} m²

These values were used in the thrust equation

$$F = (\rho_{exit} * V_{exit}^2 + (P_{exit} - P_{ambient})) * A_{exit}$$

which yielded a thrust of 762.5 N.

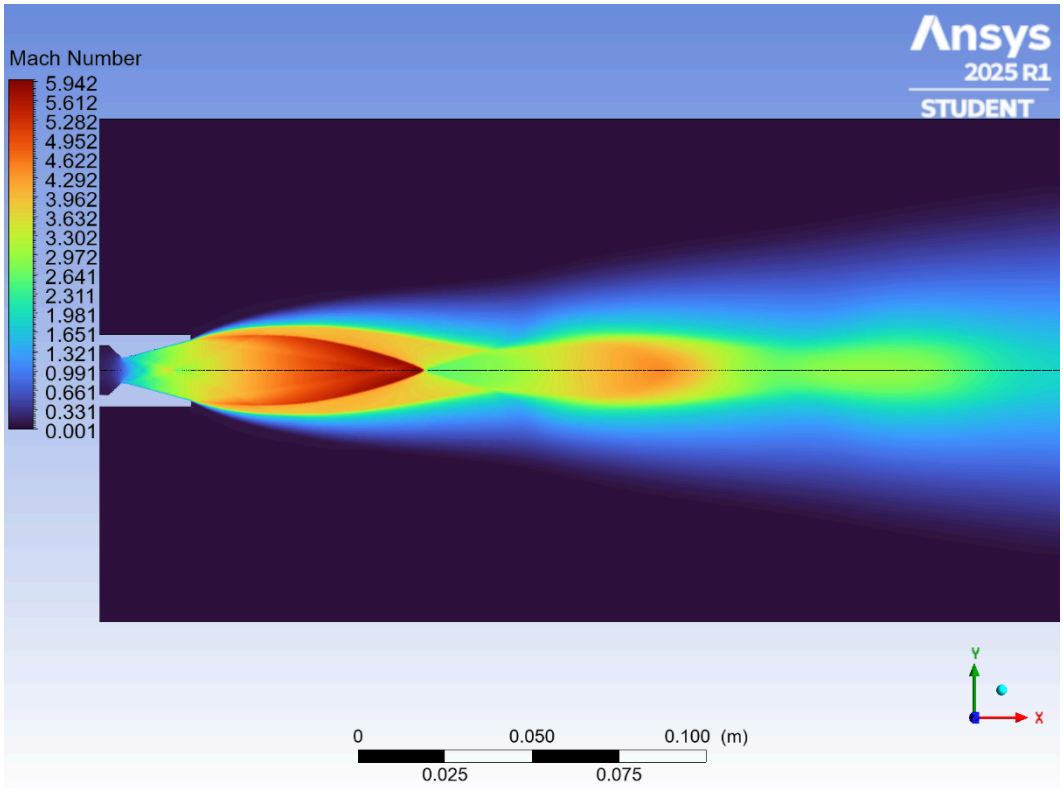


Figure 9 – Mach Number Contour Plot

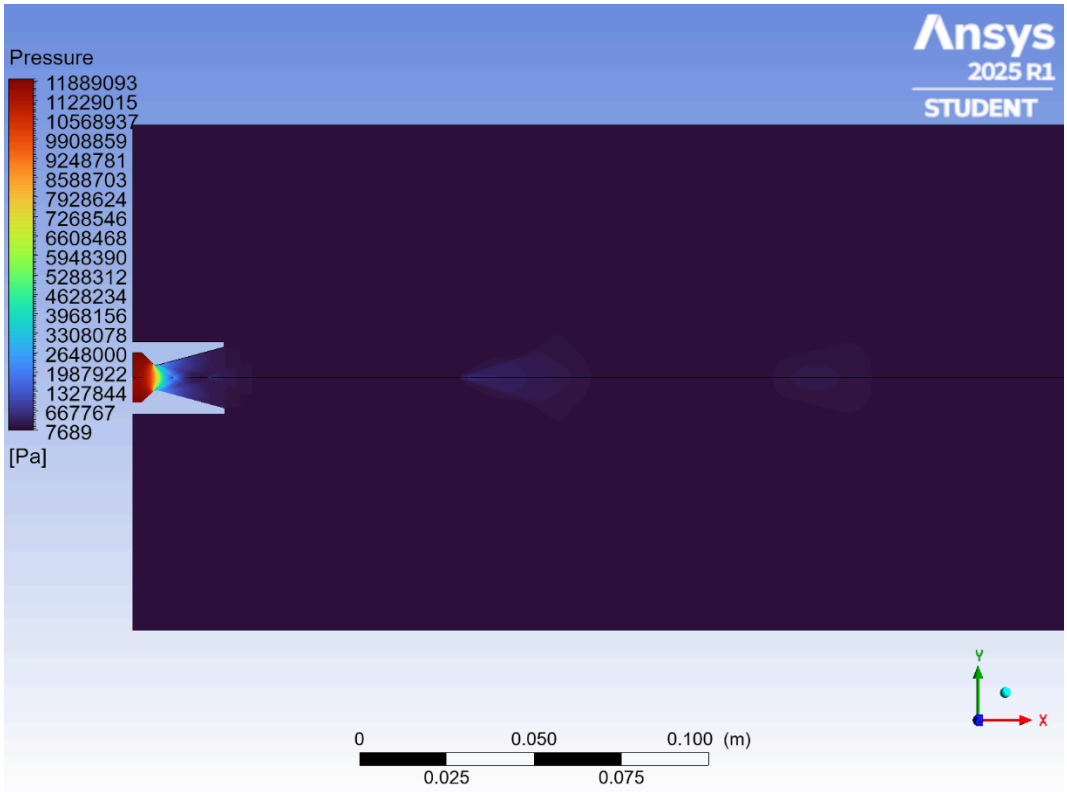


Figure 10 – Pressure Contour Plot

Discussion and Future Works

The results of this simulation showed strong agreement with expected physical behavior of a supersonic rocket nozzle. The visible formation of shock diamonds in the Mach number contours, a clean supersonic expansion, and a calculated thrust within 9.4% of the peak manufacturer thrust confirm the reliability of the final configuration.

The final thrust output of 762.5 N compares well to the expected 841.4 N, especially considering that combustion effects were not directly modeled. Flow properties at the nozzle exit—pressure, velocity, and density—aligned with theoretical expectations for high-pressure rocket exhaust expansion at sea-level ambient conditions.

Key Observations

- Use of the realizable k-epsilon model and the pressure-based coupled solver proved most effective for capturing compressible flow in this geometry.
- Mesh refinement significantly impacted solution stability and flow realism. The final mesh (Iteration 13) produced well-behaved residuals and visible shocks without excessive runtime.

- The inclusion of an external region above the nozzle exit enhanced the accuracy of external plume structure.

Limitations

Despite these successes, several limitations remain:

- The simulation assumes steady-state flow, whereas real rocket combustion is transient.
- A fixed inlet pressure and temperature were used rather than a time-dependent combustion profile.
- The use of air as the working fluid does not exactly match the combustion products of APCP (Ammonium Perchlorate Composite Propellant).
- The 2D axisymmetric model cannot capture any 3D or asymmetric flow behavior.

Future Work

To build on this project, future efforts could include:

- Implementing transient simulations to study how thrust evolves over time.
- Modeling combustion physics directly to capture chamber pressure dynamics and fuel regression.
- Using more accurate fluid properties, such as real combustion gas mixtures instead of air.
- Expanding to a 3D simulation to analyze off-axis effects or clustered nozzle arrangements.

Additional validation against thrust data over time (rather than just peak thrust) could provide even stronger correlation between simulation and experimental performance.

References

[1] Lysien, K., Stolarczyk, A., & Jarosz, T. (2021). Solid Propellant Formulations: A Review of Recent Progress and Utilized Components. Materials (Basel, Switzerland), 14(21), 6657. https://doi.org/10.3390/ma14216657

[2] Nakka, R. (n.d.). Richard Nakka's Experimental Rocketry Site. www.nakka-Rocketry.net. https://www.nakka-rocketry.net/

[3] AeroTech J350W • ThrustCurve. (2016). Thrustcurve.org. https://www.thrustcurve.org/motors/AeroTech/J350W/

Appendix

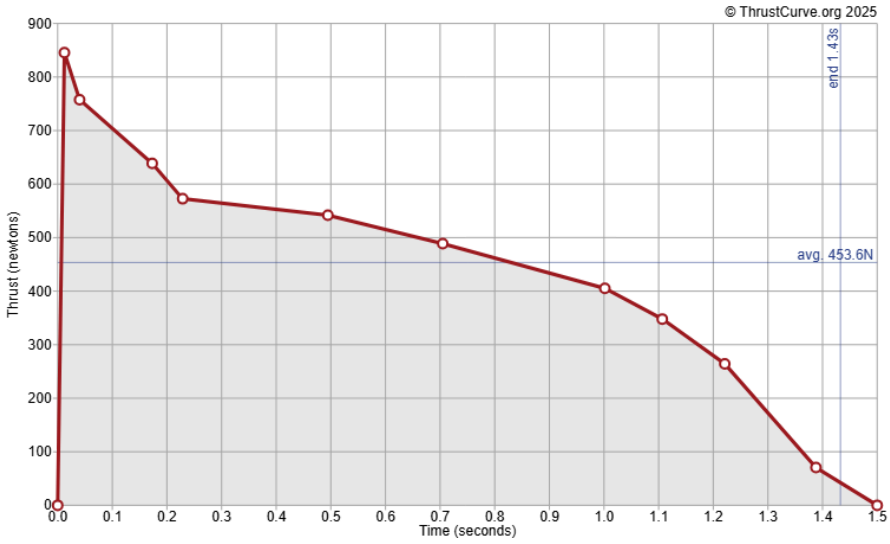


Figure 11 – Thrust Curve of the Aerotech J350W from [Thrustcurve.org]

Table 2 – Edge Sizing for Meshes Used in the Mesh Validation Study

Mesh	*1*	*2*	*3*	*4*	*5*
Method	*Mulitzone*	*Mulitzone*	*Mulitzone*	*Mulitzone*	*Mulitzone*
Face Mesh	*Quadrilaterals*	*Quadrilaterals*	*Quadrilaterals*	*Quadrilaterals*	*Quadrilaterals*
Edge 1	*50*	*75*	*100*	*88*	*25*
Edge 1 Behavior	*Hard*	*Hard*	*Hard*	*Hard*	*Hard*
Edge 1 Bias Factor	*5*	*5*	*5*	*5*	*5*
Edge 2	*50*	*75*	*100*	*88*	*25*
Edge 2 Behavior	*Soft*	*Soft*	*Soft*	*Soft*	*Soft*
Edge 2 Bias Factor	*5*	*5*	*5*	*5*	*5*
Edge 3	*0.2 mm*	*0.133 mm*	*0.1 mm*	*0.114*	*0.4*

Edge 3 Behavior	*Hard*	*Hard*	*Hard*	*Hard*	*Hard*
Edge 3 Bias Factor	*No Bias*	*No Bias*	*No Bias*	*No Bias*	*No Bias*
Edge 4	*100*	*150*	*200*	*175*	*50*
Edge 4 Behavior	*Hard*	*Hard*	*Hard*	*Hard*	*Hard*
Edge 4 Bias Factor	*2*	*2*	*2*	*2*	*2*
Edge 5	*0.5 mm*	*0.333 mm*	*0.25 mm*	*0.286*	*1 mm*
Edge 5 Behavior	*Hard*	*Hard*	*Hard*	*Hard*	*Hard*
Edge 5 Bias Factor	*4*	*4*	*4*	*4*	*4*
Edge 6	*50*	*75*	*100*	*88*	*25*
Edge 6 Behavior	*Hard*	*Hard*	*Hard*	*Hard*	*Hard*
Edge 6 Bias Factor	*No Bias*	*No Bias*	*No Bias*	*No Bias*	*No Bias*

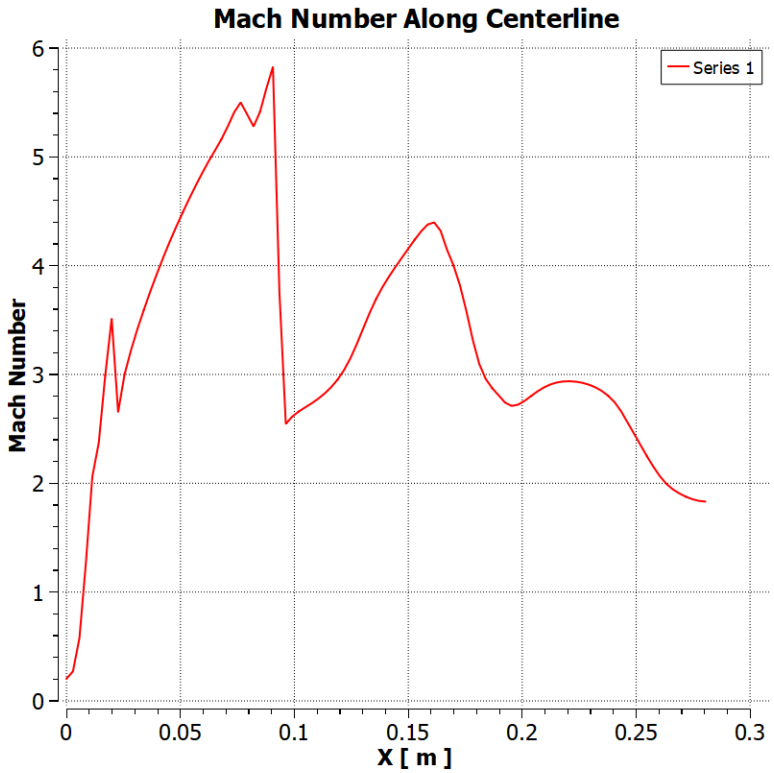


Figure 12 – Mach Number Along Centerline of Final Simulation

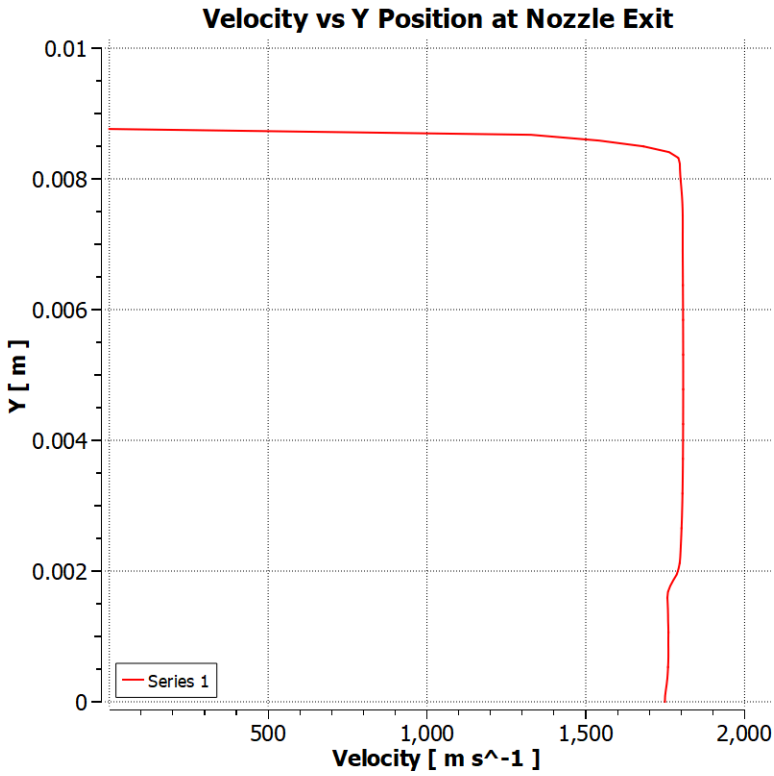


Figure 13 – Velocity vs Y Position at Nozzle Exit of Final Simulation

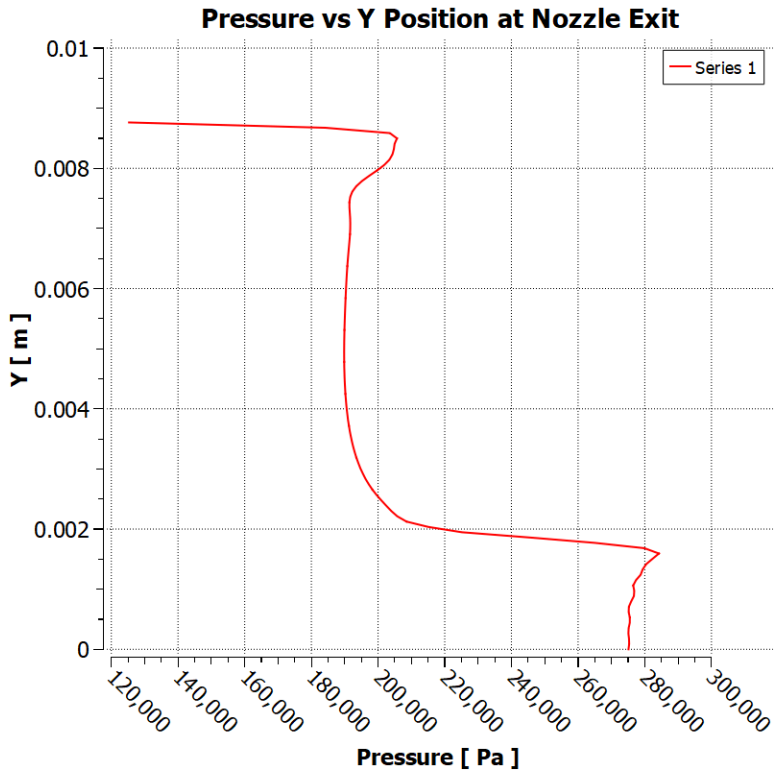


Figure 14 – Pressure vs Y Position at Nozzle Exit of Final Simulation

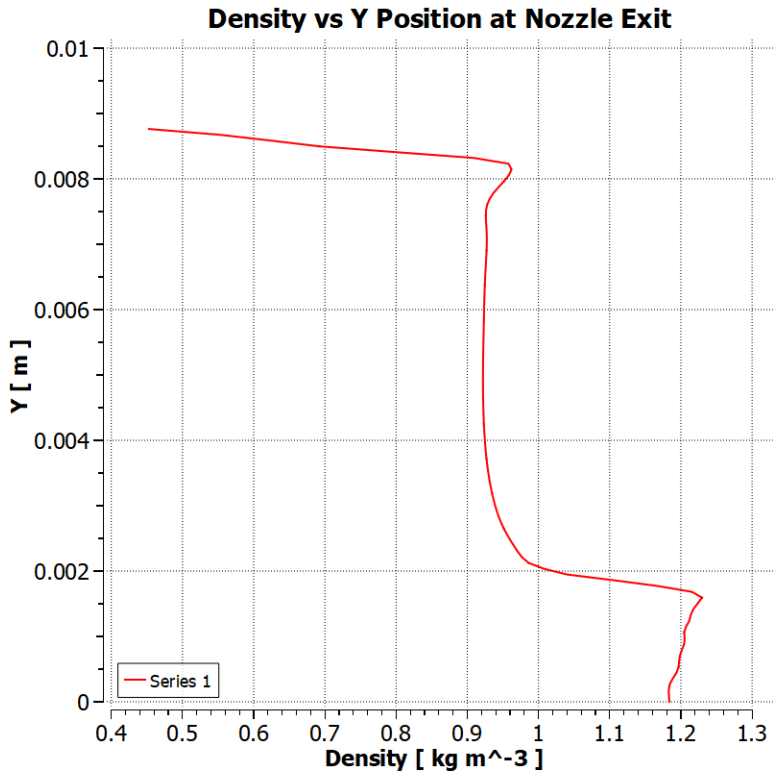


Figure 15 – Density vs Y Position at Nozzle Exit of Final Simulation