

دانشگاه صنعتی امیرکبیر  
( پلی تکنیک تهران )

**Amir Kabir University of Technology**  
**Faculty of Aerospace Engineering**

**Rocket Aero-Thermodynamics**

**Title**

# **The First Project of Rocket Aero- Thermodynamics Course**

**Supervisor**

**Dr. Ali Maddi**

**April 2023**



Ebrahim Safdarian 401129076

## Page Table of Contents

<b>1</b>	<b>Introduction .....</b>	<b>3</b>
<b>2</b>	<b>Nozzle design specifications.....</b>	<b>3</b>
2. 1	Vacuum conditions .....	4
2. 2	Shock conditions at the nozzle exit .....	5
2. 3	Sea level conditions .....	5
<b>3</b>	<b>Nozzle dimensions.....</b>	<b>5</b>
3. 1	Python code .....	6
3. 2	Creating geometry in CATIA .....	9
<b>4</b>	<b>Nozzle analysis in Ansys Fluent .....</b>	<b>9</b>
4. 1	Grid production.....	10
4. 2	Solver settings.....	14
4. 3	How to converge.....	15
4. 4	Design conditions .....	16
4.4.1	analysis .....	19
4. 5	Sea level conditions .....	19
4.5.1	analysis .....	23
4. 6	Shock conditions at the nozzle exit .....	23
4.6.1	analysis .....	26
4. 7	Vacuum conditions .....	27
4.7.1	analysis .....	30
4. 8	General analysis.....	30
<b>5</b>	<b>Requested charts.....</b>	<b>31</b>
5. 1	Thrust chart.....	31
5. 2	Specific impulse diagram .....	31

## 1 Introduction

The purpose of this project is to design and analyze a supersonic nozzle for an altitude of 8000 meters. In this project, the geometry of the nozzle was produced using the calculations performed in the relevant exercise and a Python code, and then the geometry of the nozzle was produced in the CATIA software and sent to the Ansys software for analysis and review of the calculations. The height at which the shock enters the nozzle is checked. The process of doing the project in the flowchart Figure1.1 can be seen.

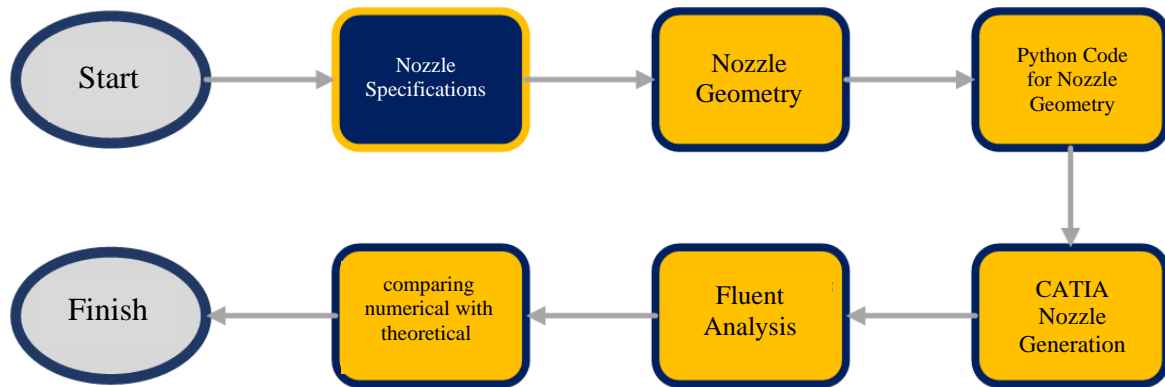


Figure1.1The process and steps of the project

## 2 Nozzle design specifications

The desired nozzle is designed for an altitude of 8000 meters and has the following specifications:

$$\begin{aligned} P_{01} &= 9.5 \text{ MPa} \\ T_{01} &= 2900 \text{ K} \\ T &= 2.5 \text{ MN} \\ h_{design} &= 8000 \text{ m} \end{aligned} \tag{1}$$

By performing one-dimensional calculations, we will have the following results in design conditions:

$$\begin{aligned}P_a &= 35.6 \text{ kPa} \\T_a &= 236.15 \text{ K} \\u_e &= 2197.5 \text{ m/sec} \\M_e &= 4.44\end{aligned}\tag{2}$$

For the ratio of outlet area to throat area and mass flow, we also have:

$$\begin{aligned}\dot{m} &= 1159.7 \text{ kg/sec} \\\frac{A_e}{A^*} &= 15.74 \\D_e &= 1.805 \text{ m} \\D^* &= 0.455 \text{ m}\end{aligned}\tag{3}$$

## 21. Vacuum conditions

In vacuum conditions, the following are true:

$$\begin{aligned}\dot{m} &= 1159.7 \text{ kg/sec} \\T &= 2.59 \text{ MN} \\u_e &= 2197.5 \text{ m} \\M_e &= 4.44\end{aligned}\tag{4}$$

## 22. Shock conditions at the nozzle exit

In the condition of shock in the exit of the nozzle, the following are established:

$$\begin{aligned}\dot{m} &= 1159.7 \text{ kg/sec} \\ P_a &= 812.74 \text{ kPa} \\ M_e &= 0.4247 \\ T &= 0.508 \text{ MN}\end{aligned}\tag{5}$$

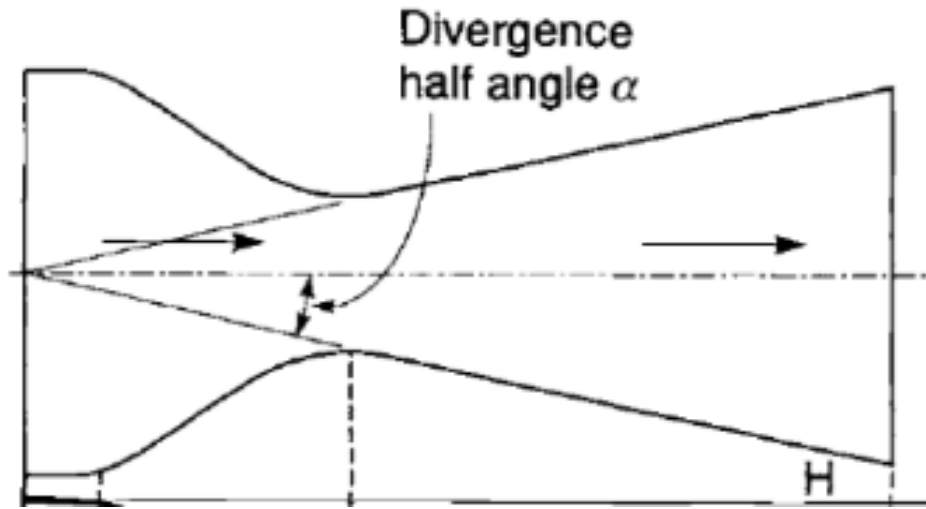
## 23. Sea level conditions

In sea level conditions, the following are established:

$$\begin{aligned}\dot{m} &= 1159.7 \text{ kg/sec} \\ T &= 2.33 \text{ MN} \\ u_e &= 2197.5 \text{ m} \\ M_e &= 4.44\end{aligned}\tag{6}$$

## 3 Nozzle dimensions

In relation 3, the characteristics of the area of the throat and the area of the outlet are specified. Now, assuming a conical nozzle with a half angle of 15 degrees, the length of the nozzle can be achieved. At Figure3.1 You can see the conical nozzle and the half angle.



**Figure3.1 Conical nozzle**

Now, assuming a conical nozzle, the geometric specifications of the nozzle will be as follows:

$$L_{convergent} = 0.85 \text{ m}$$

$$L = 3.37 \text{ m}$$

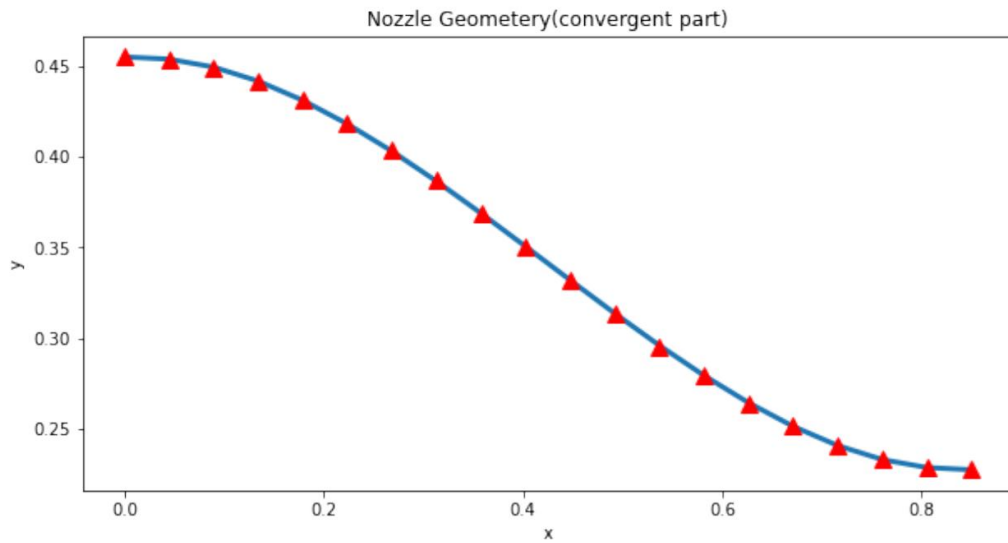
$$D_e = 1.805 \text{ m}$$

$$D^* = 0.455 \text{ m}$$

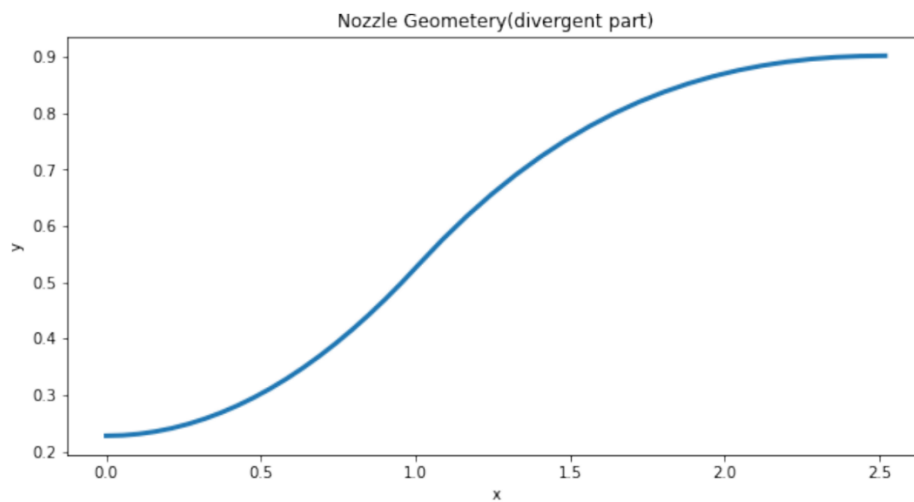
(7)

### 31. Python code

In the Python code, a cosine curve is used for the converging part, and a 2nd degree curve and a 3rd degree curve are used for the divergent part. For the proper entry and exit of the flow in the nozzle, the slope of the curves in the nozzle inlet, throat and nozzle outlet is considered to be zero. At Figure3.2 You can see the curve of the converging part, also in Figure3.3 The curve of the divergent part is visible.

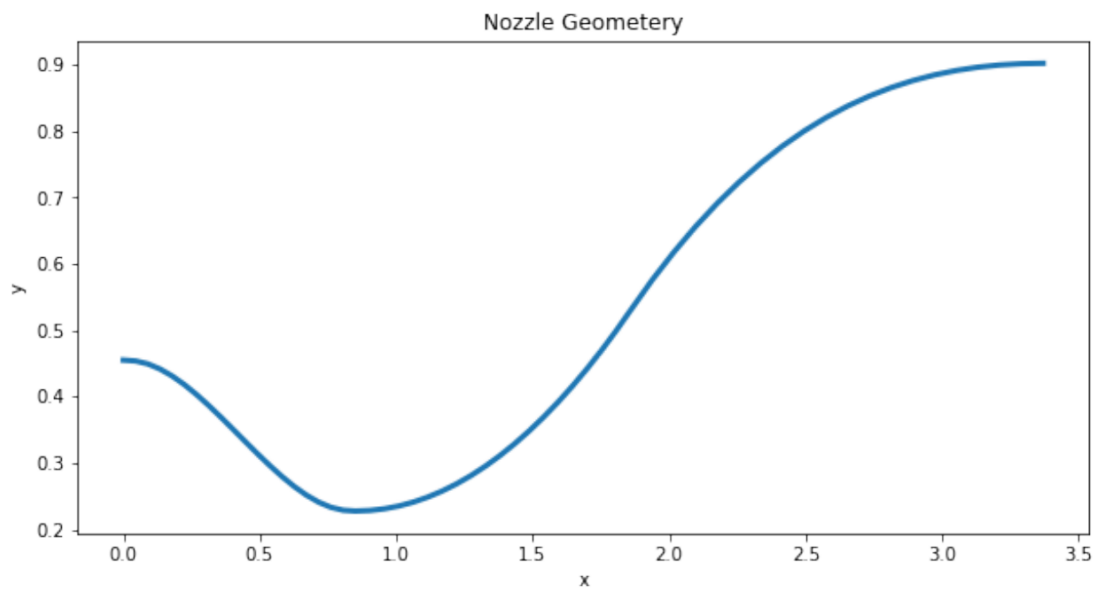


**Figure3.2 Convergent part curve**

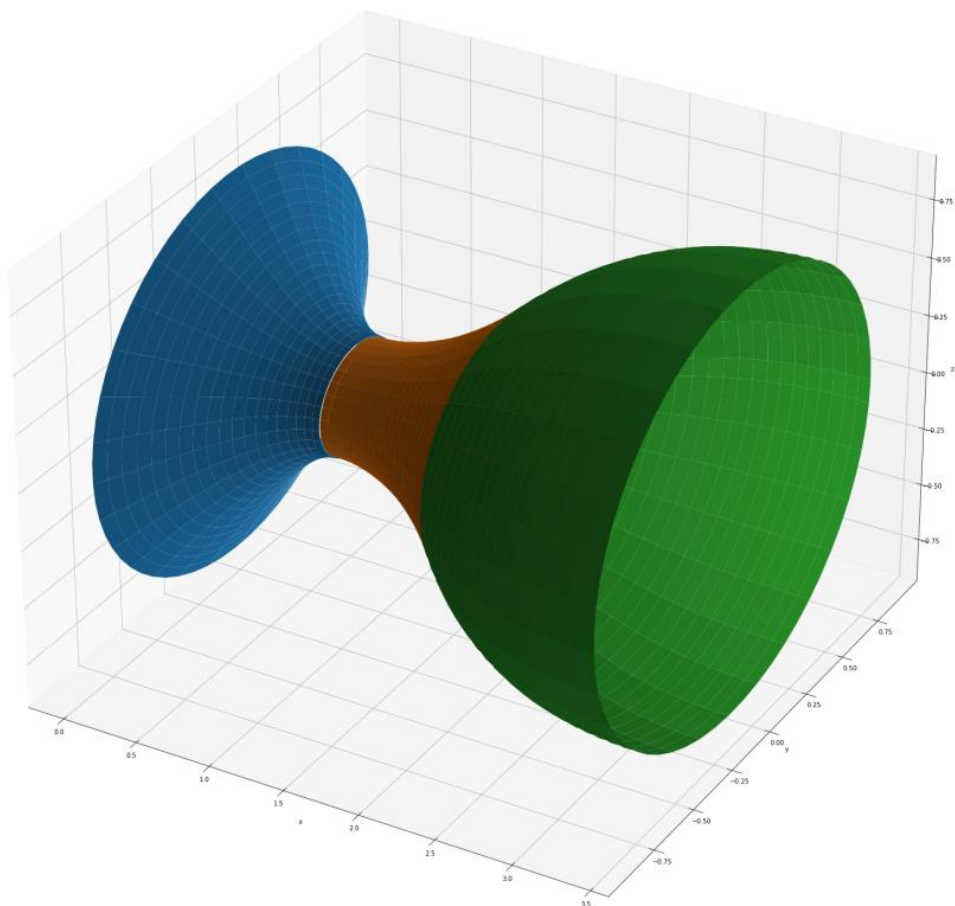


**Figure3.3 Divergent part curve**

At Figure3.4 You can see the general curve of the nozzle, also in Figure3.5, the three-dimensional Figure of the nozzle can be seen.



**Figure3.4 The overall curve of the nozzle**



**Figure3.5 Three-dimensional Figure of the nozzle**



### 32. Creating geometry in CATIA

Having the geometry created by the Python code, it can be created in the CATIA software. Geometry created in CATIA in Figure3.6 can be seen.

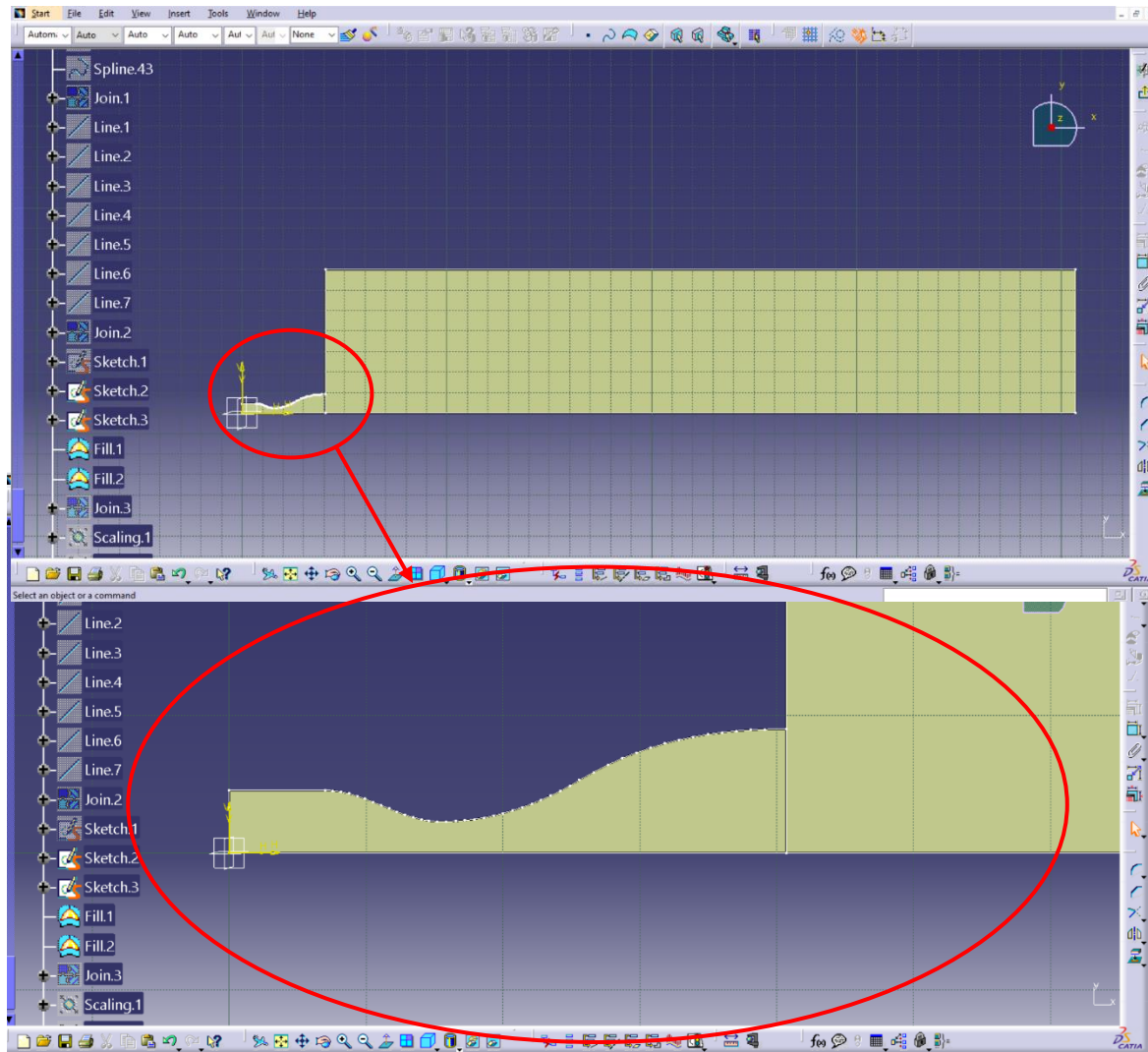


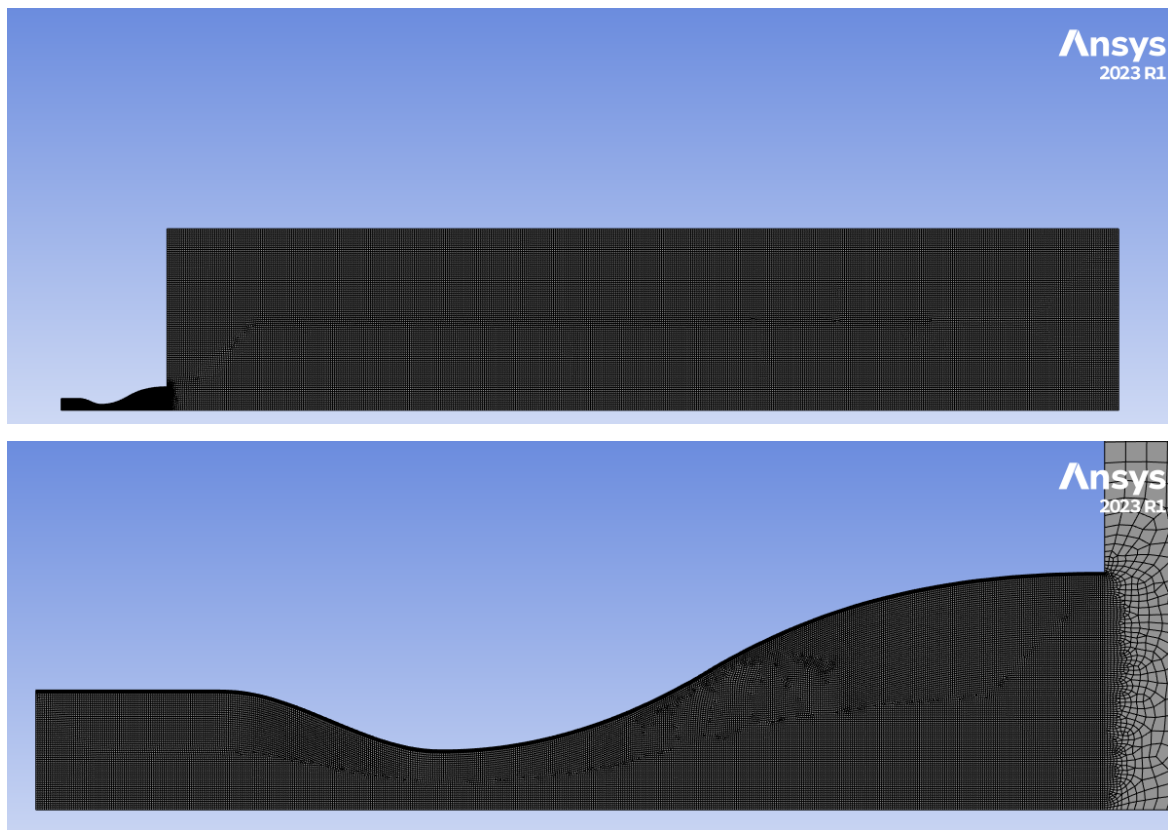
Figure3.6 Nozzle geometry in CATIA

## 4 Nozzle analysis in Ansys Fluent

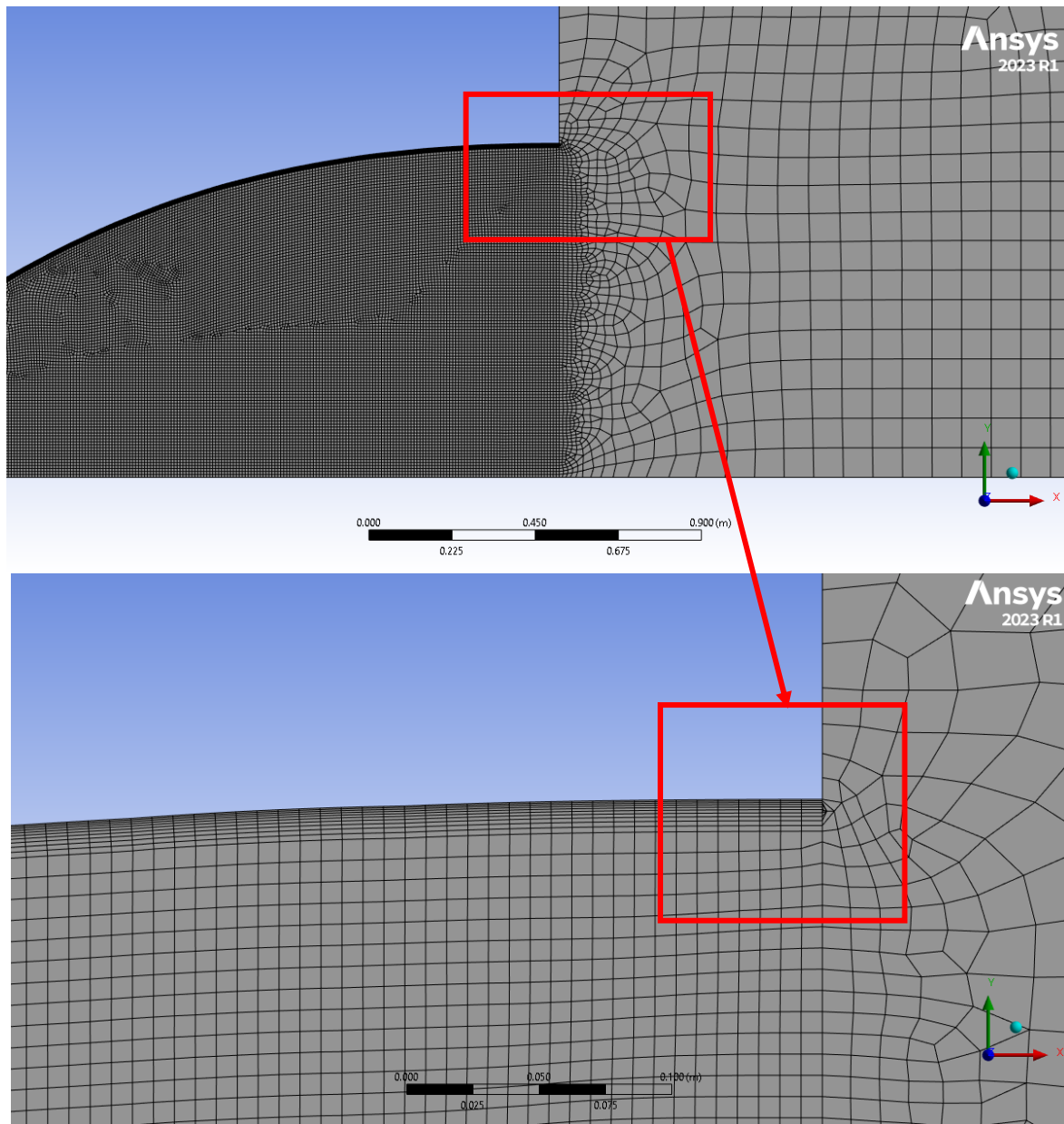
To analyze the nozzle and validate the theoretical results, the designed nozzle will be checked in design conditions, vacuum conditions, sea level conditions and shock conditions at the nozzle exit.

#### 41. Grid production

Grid generated for nozzle geometry can be seen in Figure4.1 And Figure4.2. **Dimensions of the area behind the nozzle** It is equal to nine times the length of the nozzle  $\times$  ten times the exit radius of the nozzle.

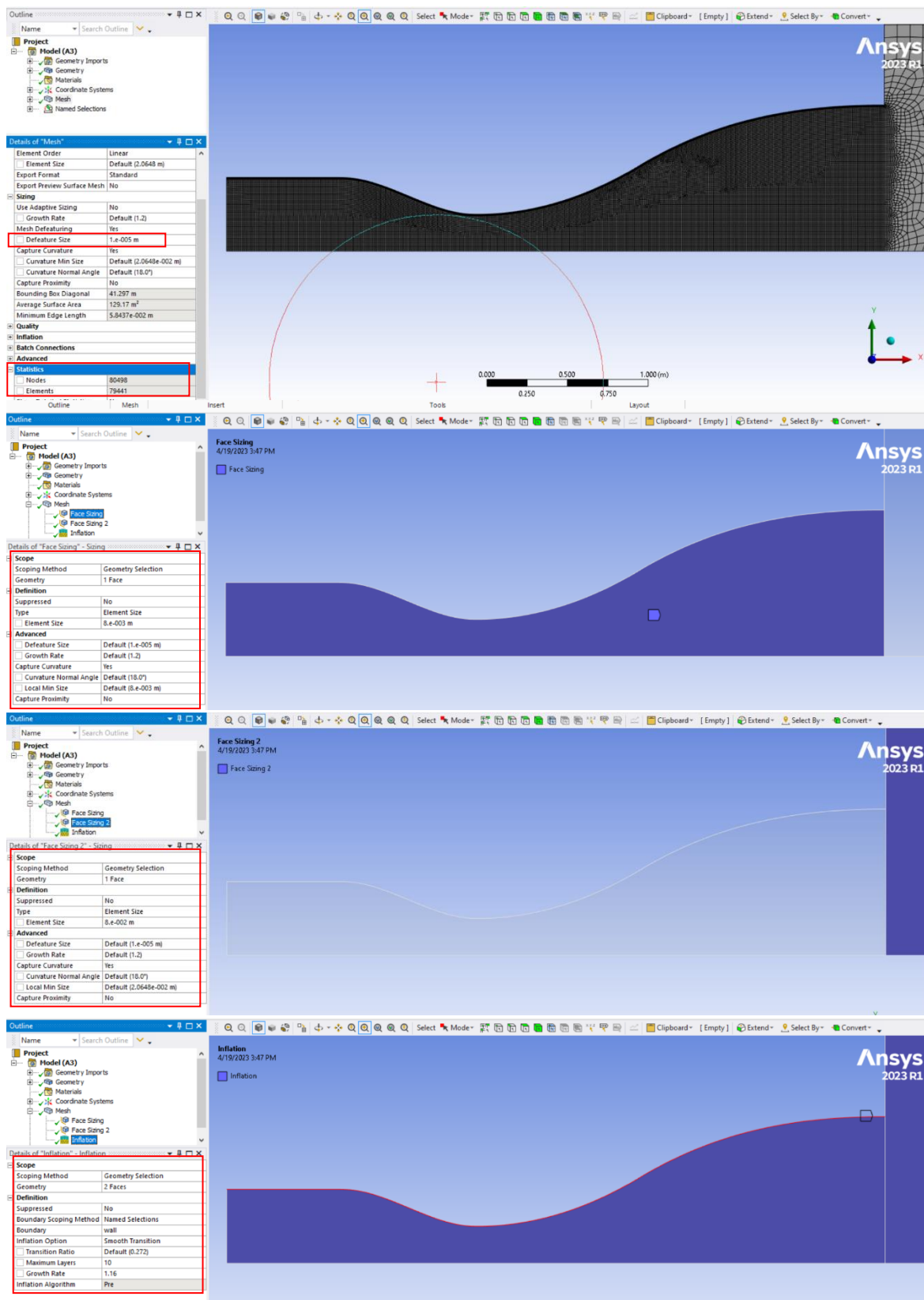


**Figure4.1 Generated Grid for the nozzle**



**Figure4.2 Generated Grid for the nozzle**

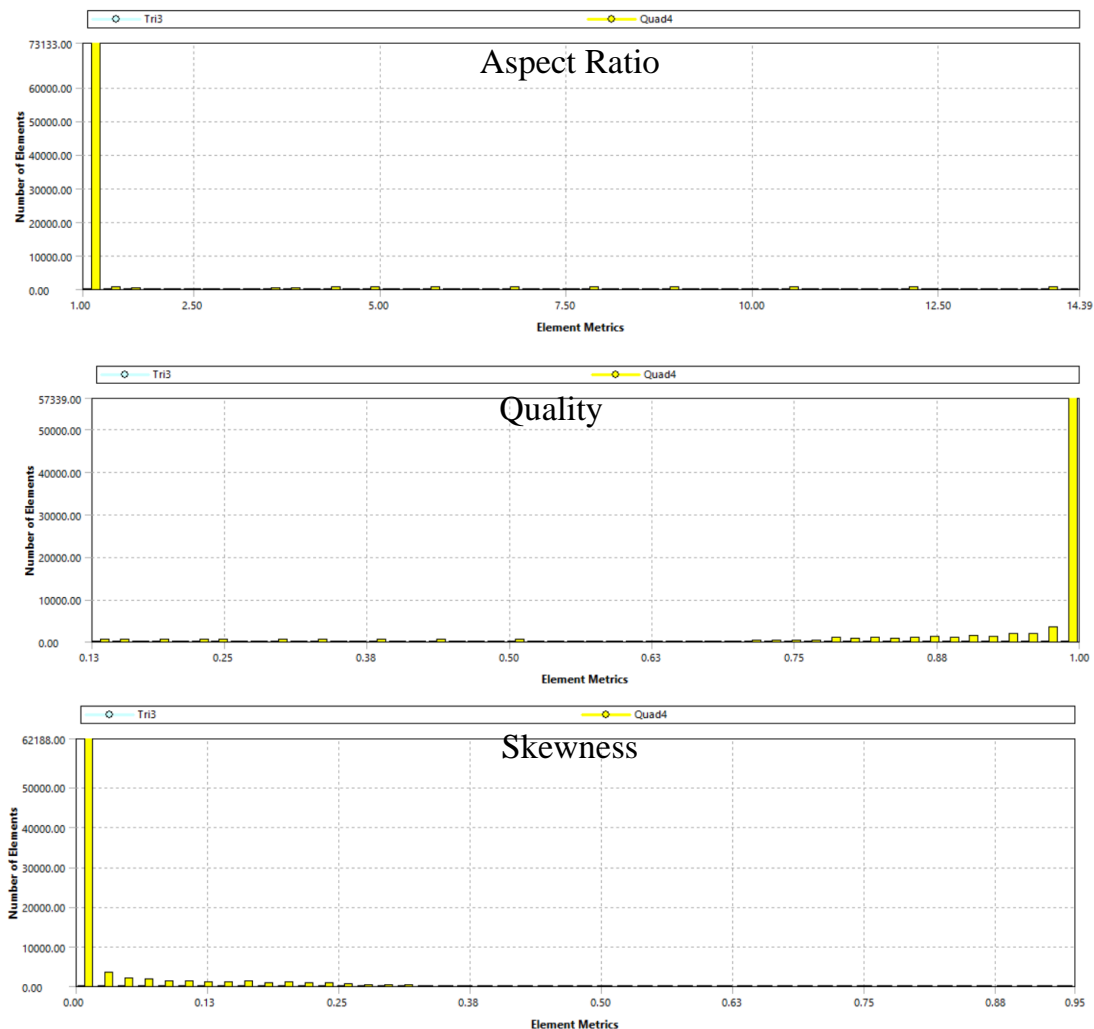
Mesh settings in Figure4.3 can be seen



**Figure4.3 Generated mesh settings for the nozzle**

The quality of the mesh Figure4.4 It is visible, which is in accordance with the figure within the permitted and appropriate limits. Also, the mesh check output is as follows:

Domain Extents: x-coordinate: min (m) = 2.286000e-12, max (m) = 4.070000e+01  
 y-coordinate: min (m) = 0.000000e+00, max (m) = 6.994375e+00 Volume statistics:  
 minimum volume (m3): 8.157623e-07 maximum volume (m3): 2.806245e-01  
 total volume (m3): 5.633876e+03 minimum 2d volume (m3): 1.445639e-07  
 maximum 2d volume (m3): 8.017979e-03 Face area statistics:  
 minimum face area (m2): 2.908336e-04 maximum face area (m2): 1.123706e-01  
 Checking mesh.....Done.



**Figure4.4 The quality of the produced mesh**

## 42. Solver settings

The solver settings are shown below.

**Create/Edit Materials**

Models: Multiphase - Off, Energy - On, Viscous - Realizable k- $\epsilon$ , Scalable Wall Fn, Radiation - Off, Heat Exchanger - Off, Species - Off, Discrete Phase - Off, Acoustics - Off, Structure - Off, Potential/Electrochemistry - Off

Name: air

Material Type: fluid

Fluent Fluid Materials: air

Mixture: none

Order Materials by: Name (selected), Chemical Formula

Properties:

- Density [kg/m<sup>3</sup>]: ideal-gas
- Cp (Specific Heat) [J/(kg K)]: constant, 1006.43
- Thermal Conductivity [W/(m K)]: constant, 0.0242
- Viscosity [kg/(m s)]: sutherland

**Pressure Inlet**

Zone Name: inlet

Zone Name: field

Momentum: Reference Frame: Absolute, Gauge Total Pressure [Pa]: 9500000, Supersonic/Initial Gauge Pressure [Pa]: 9000000, Direction Specification Method: Normal to Boundary, Prevent Reverse Flow: ☐

Turbulence: Specification Method: Intensity and Length Scale, Turbulent Intensity [%]: 1, Turbulent Length Scale [m]: 0.03

Momentum: Gauge Pressure [Pa]: 812740, Mach Number: 0.01, Axial-Component of Flow Direction: 1, Radial-Component of Flow Direction: 0

Turbulence: Specification Method: Intensity and Length Scale, Turbulent Intensity [%]: 1, Turbulent Length Scale [m]: 0.03

**Solution Methods**

Implicit: ☐ Explicit: ☒ Flux Type: Roe-FDS

Spatial Discretization: Gradient: Green-Gauss Node Based, Flow: Second Order Upwind, Turbulent Kinetic Energy: Second Order Upwind, Turbulent Dissipation Rate: Second Order Upwind

Pseudo Time Method: Off

Transient Formulation: ☐ Non-Iterative Time Advancement, ☐ Frozen Flux Formulation, ☐ Warped-Face Gradient Correction, ☐ High Order Term Relaxation, ☐ Convergence Acceleration For Stretched Meshes, ☐ High Speed Numerics

**Solution Controls**

Courant Number: 0.5

Under-Relaxation Factors: Turbulent Kinetic Energy: 0.8, Turbulent Dissipation Rate: 0.8, Turbulent Viscosity: 1

Solid: 1

Buttons: Default, Equations..., Limits..., Advanced...



### 43. How to converge

to converge each case initially with the initialization values specified in Figure4.5 It has been run for a coarse grid and then the obtained results have been interpolated to the fine grid.

The screenshot displays the 'Task Page' for 'Solution Initialization' in ANSYS Fluent. The 'Initialization Methods' section has 'Standard Initialization' selected. The 'Reference Frame' is set to 'Relative to Cell Zone'. The 'Initial Values' section, highlighted with a red box, contains the following values:

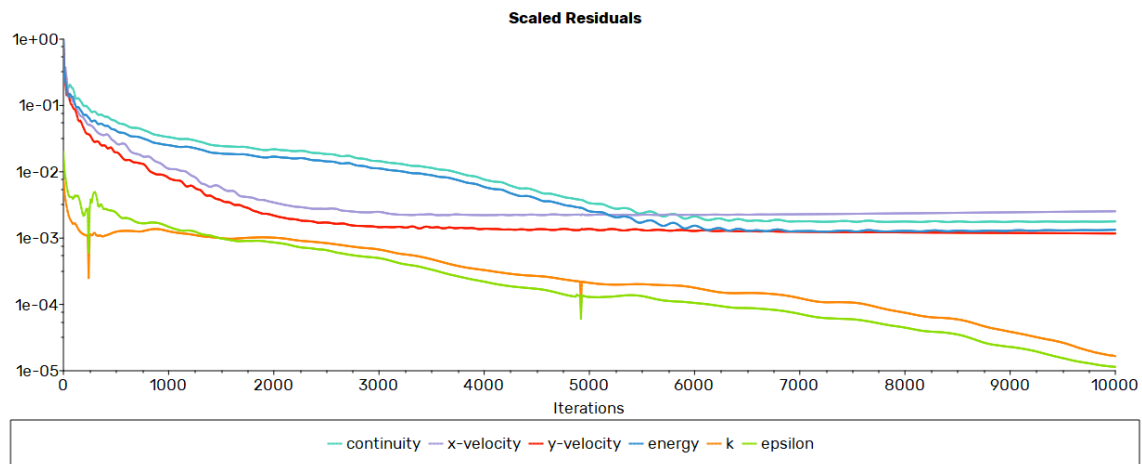
Parameter	Value
Gauge Pressure [Pa]	500000
Axial Velocity [m/s]	300
Radial Velocity [m/s]	0
Turbulent Kinetic Energy [m <sup>2</sup> /s <sup>2</sup> ]	1
Turbulent Dissipation Rate [m <sup>2</sup> /s <sup>3</sup> ]	1
Temperature [K]	500

Below the 'Initial Values' section, there are buttons for 'Initialize', 'Reset', and 'Patch...'. At the bottom, there are buttons for 'Reset DPM Sources', 'Reset LWF', 'Reset Statistics', and 'VOF Check'.

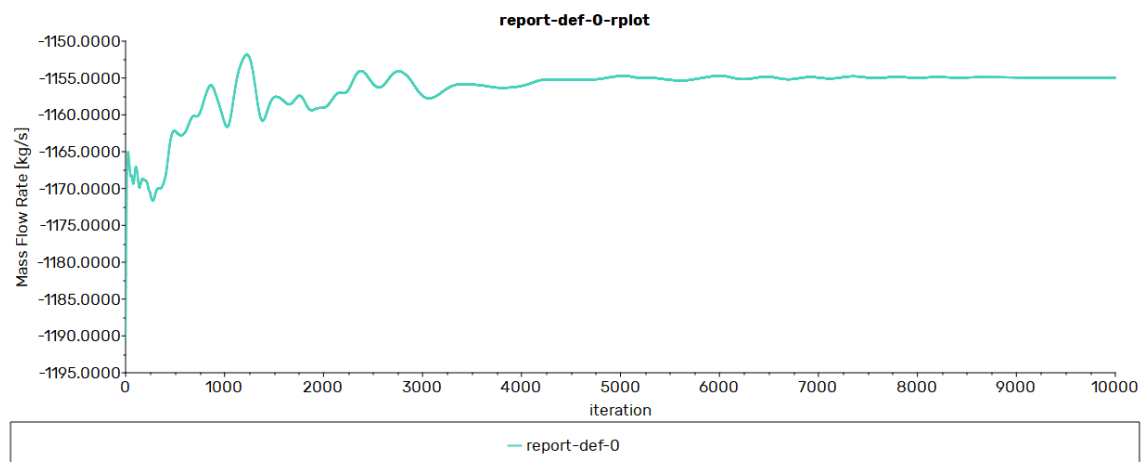
**Figure4.5 Initialization values**

#### 44. Design conditions

In design conditions, the following results are obtained in Fluent. The behavior of residuals during the corresponding solution process can be seen in Figure4.6, as well as the mass flow behavior during the corresponding solution process can be seen in Figure4.7. Corresponding static pressure contours can be seen in Figure4.8. Also static temperature contours in Figure4.9 and speed contours in Figure4.10 are visible.

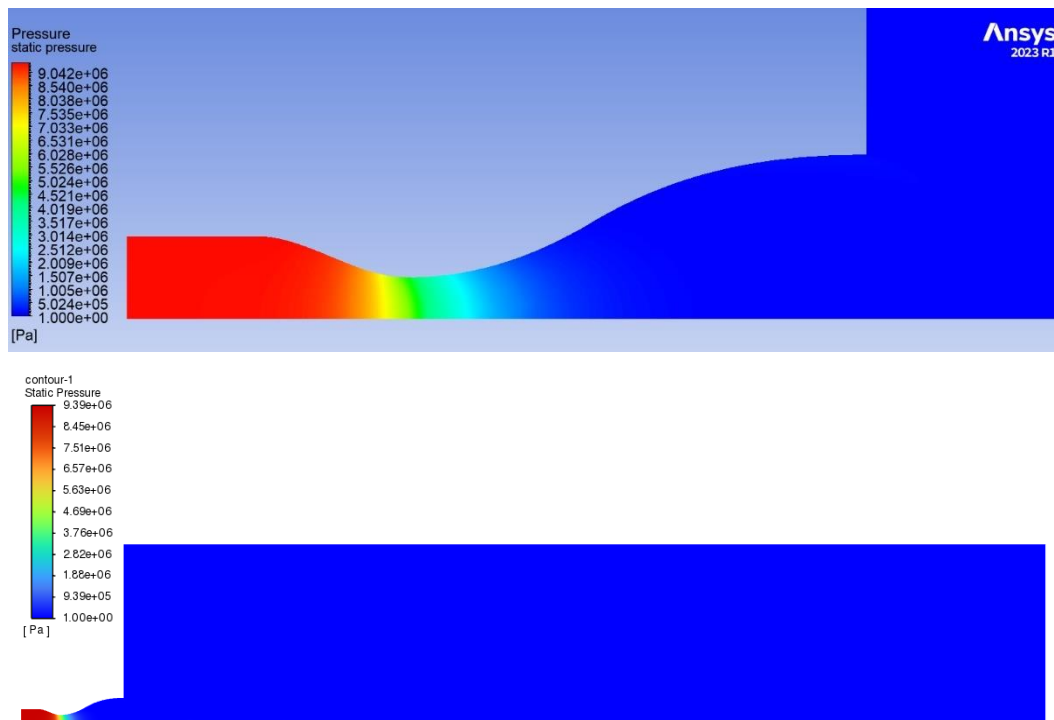


**Figure4.6 The behavior of resins during the solution process in design conditions**

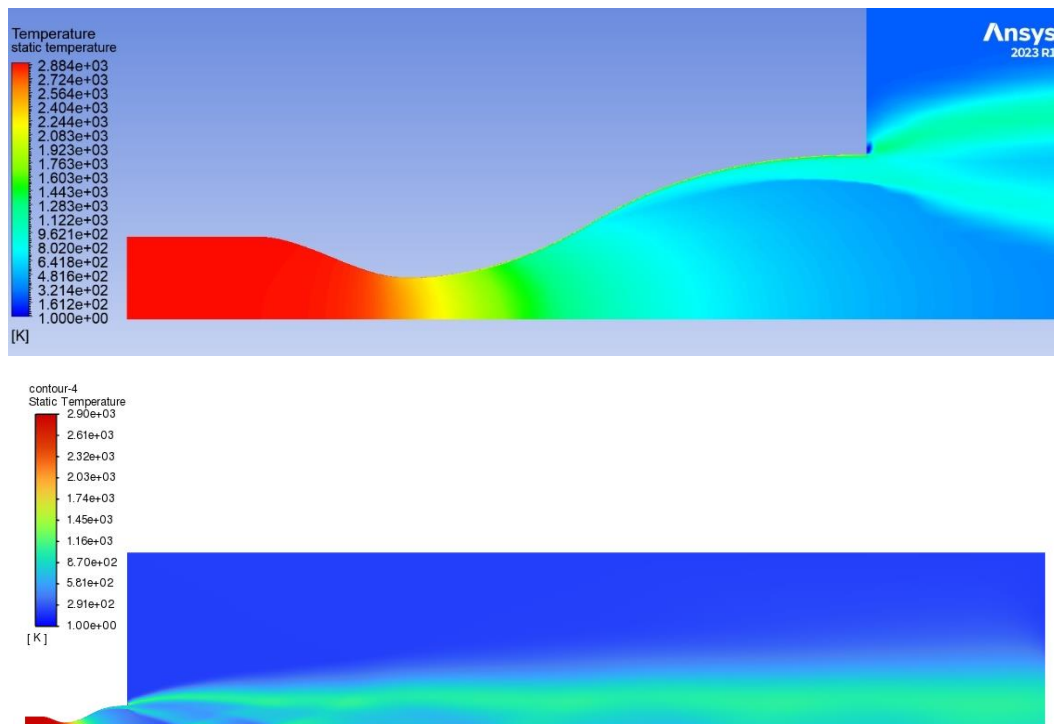


**Figure4.7 The mass flow behavior of the nozzle during the solution process in the design conditions**

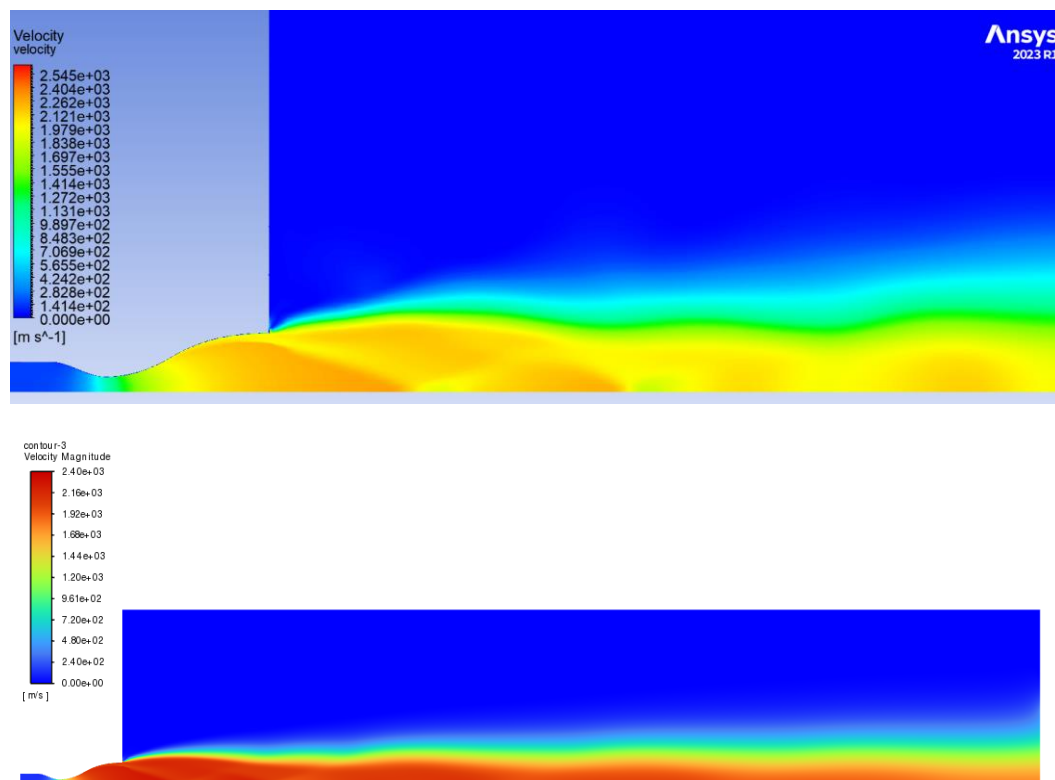




**Figure4.8 Static pressure contours in design conditions**



**Figure4.9 Static temperature contours at design conditions**



**Figure4.10 Speed contours in design conditions**

The output of mass flow, Mach and speed in the software is as follows:

Area-Weighted Average

Mach number

Interior 4.5823589

Mass Flow Rate [kg/s]

inlet 1154.6982

Net 1154.6982

Area-Weighted Average Velocity Magnitude [m/s]

Interior 2147.9188

By comparing the results obtained from Fluent with the theoretical results, we will have:

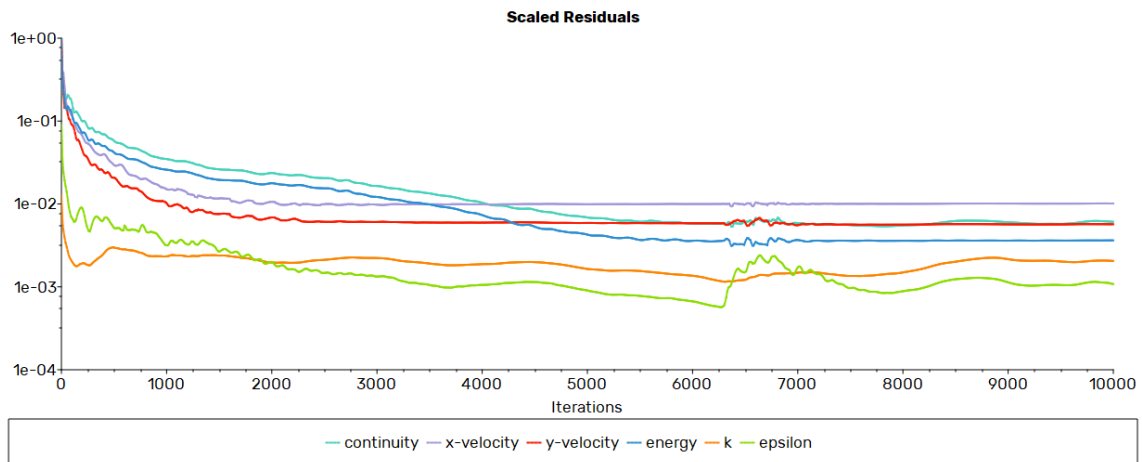
$$\begin{aligned} T_{fluent} &= \dot{m}u_e = 1159.7 \times 2147.92 = 2.48 \text{ MN} \\ error_{u_e} &= \frac{|2197.5 - 2147.92|}{2147.92} \times 100 = 2.31\% \\ error_{\dot{m}} &= \frac{|1159.7 - 1154.7|}{1154.7} \times 100 = 0.43\% \\ error_T &= \frac{|2.5 - 2.48|}{2.48} \times 100 = 0.81\% \end{aligned} \quad (8)$$

#### 41.4. analysis

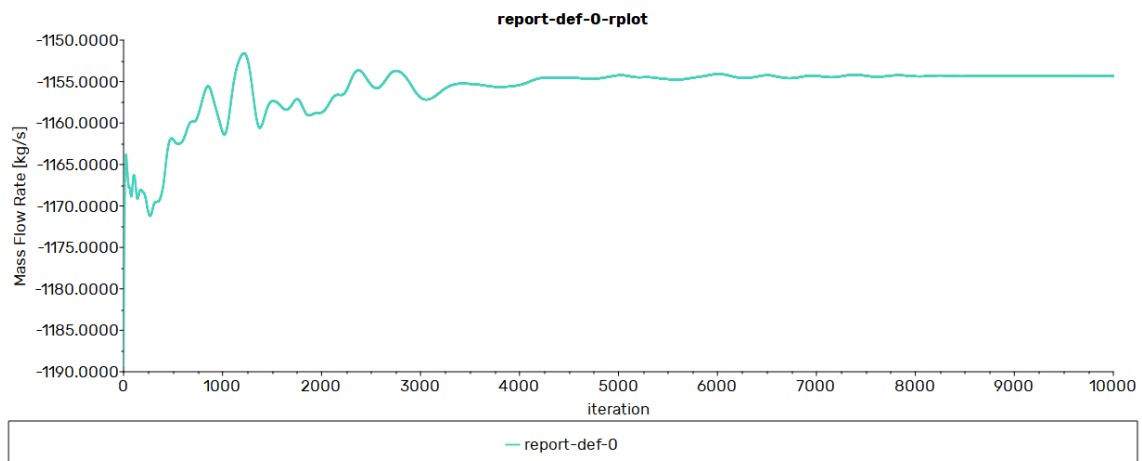
As can be seen, the theoretical results and numerical solution results are in good agreement and the resulting error percentage is low and favorable. ChartsFigure4.6AndFigure4.7They show good convergence in the solution. As in static pressure contoursFigure4.8It can be seen, the static pressure before the throat is almost constant and equal to the chamber pressure. Static temperature contoursFigure4.9and speed contoursFigure4.10They show well the formation of oblique shocks in the nozzle exit and the effect of the boundary layer.

#### 45. Sea level conditions

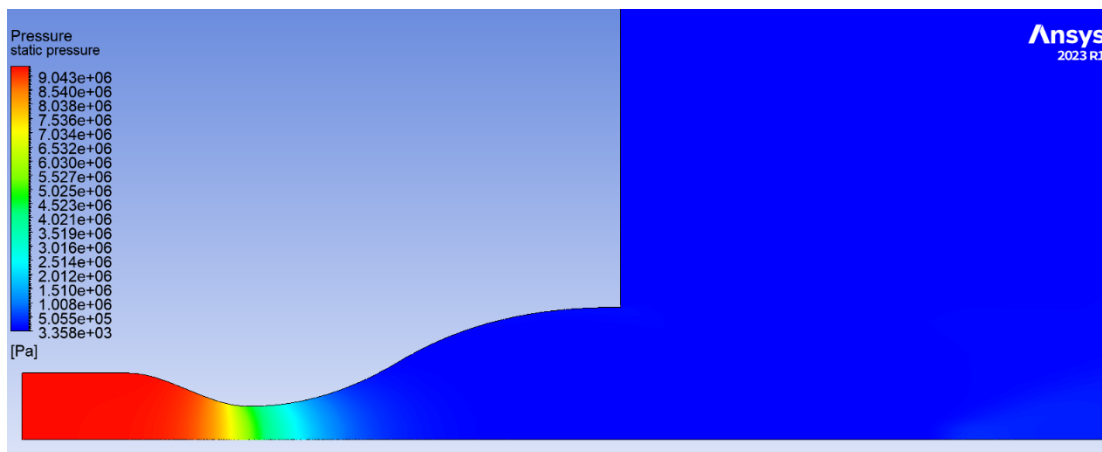
In sea level conditions, the following results are obtained in Fluent. The behavior of residuals during the corresponding solution process Figure4.11 It is, as mass dlow rate behavior during solution Figure4.17 is. Static pressure contours according to Figure4.13 is. Also Static temperature contours in Figure4.14 and velocity contours speed in Figure4.15 are visible



**Figure 4.11** The behavior of residuals during the solution process in sea level conditions

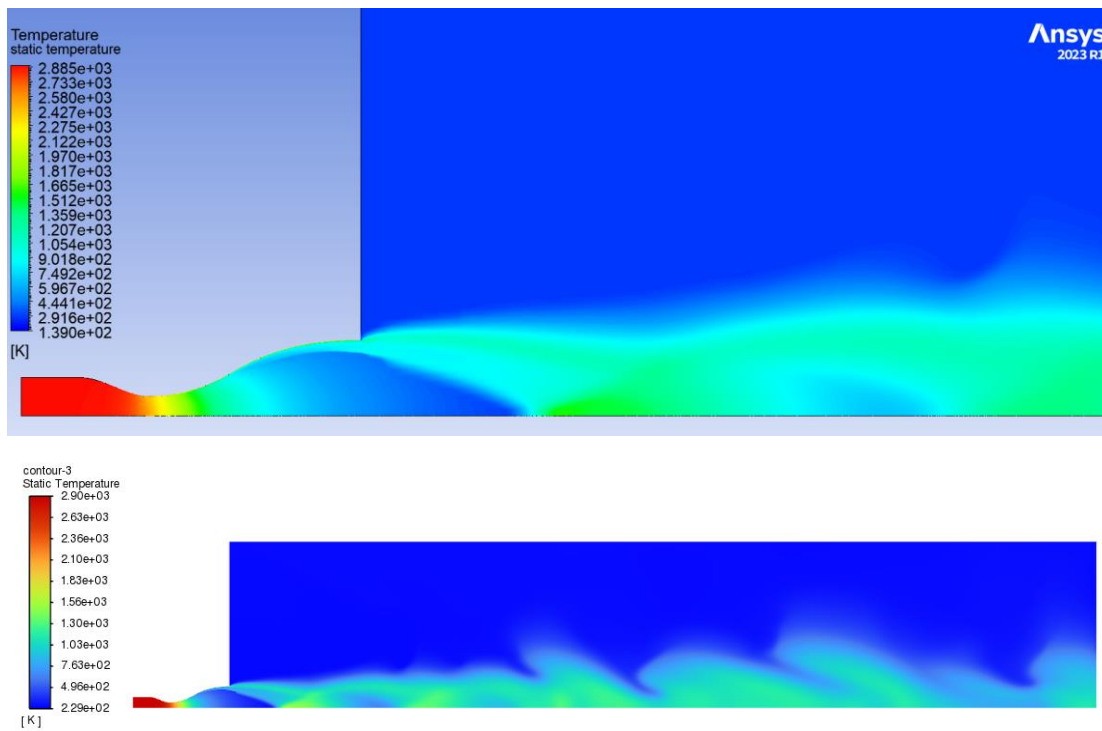


**Figure 4.12** The mass flow behavior of the nozzle during the solution process in sea level conditions

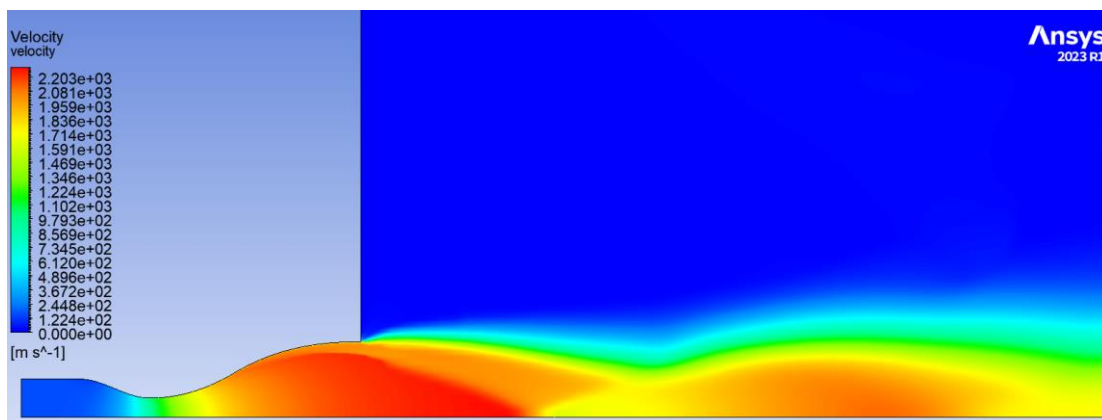


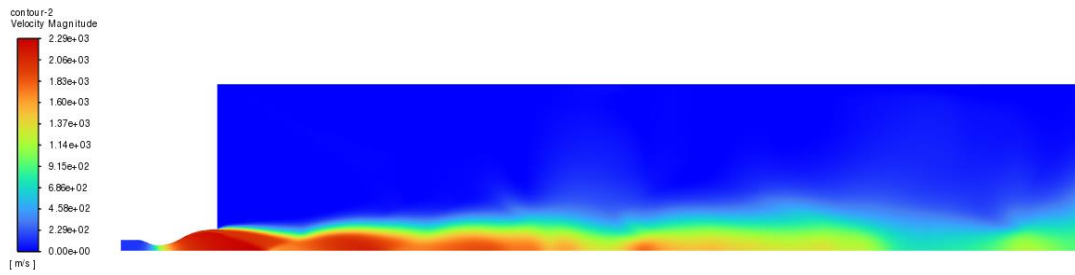


**Figure 4.13 Static pressure contours at sea level conditions**



**Figure 4.14 Static temperature contours at sea level conditions**





**Figure4.15 Velocity contours at sea level conditions**

The output of mass flow, Mach and speed in the software is as follows:

Mass Flow Rate [kg/s]

-----  
inlet 1154.1031

-----  
Net 1154.1031

Area-Weighted Average

Velocity Magnitude [m/s]

-----  
Interior 2146.1613

Area-Weighted Average Mach Number

----- interior 4.5754657

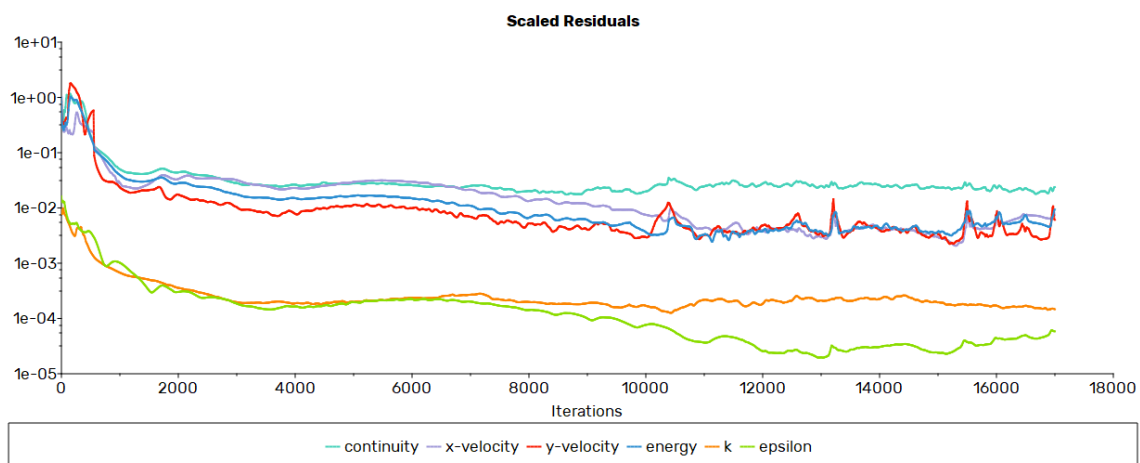
(9)

#### 41.5. analysis

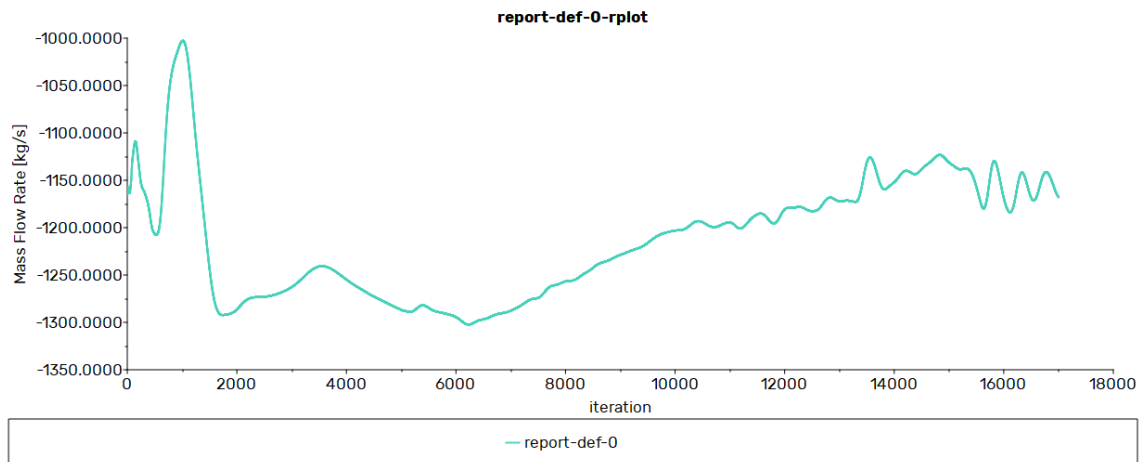
As can be seen, the theoretical results and numerical solution results are in good agreement and the resulting error percentage is low and favorable. Charts Figure 4.11 and Figure 4.12 show good convergence in the solution. As in static pressure contours Figure 4.13 it can be seen, the static pressure before the throat is almost constant and equal to the chamber pressure. Static temperature contours Figure 4.14 and speed contours Figure 4.15 show well the formation of oblique shocks in the nozzle exit and the effect of the boundary layer.

#### 46. Shock conditions at the nozzle exit

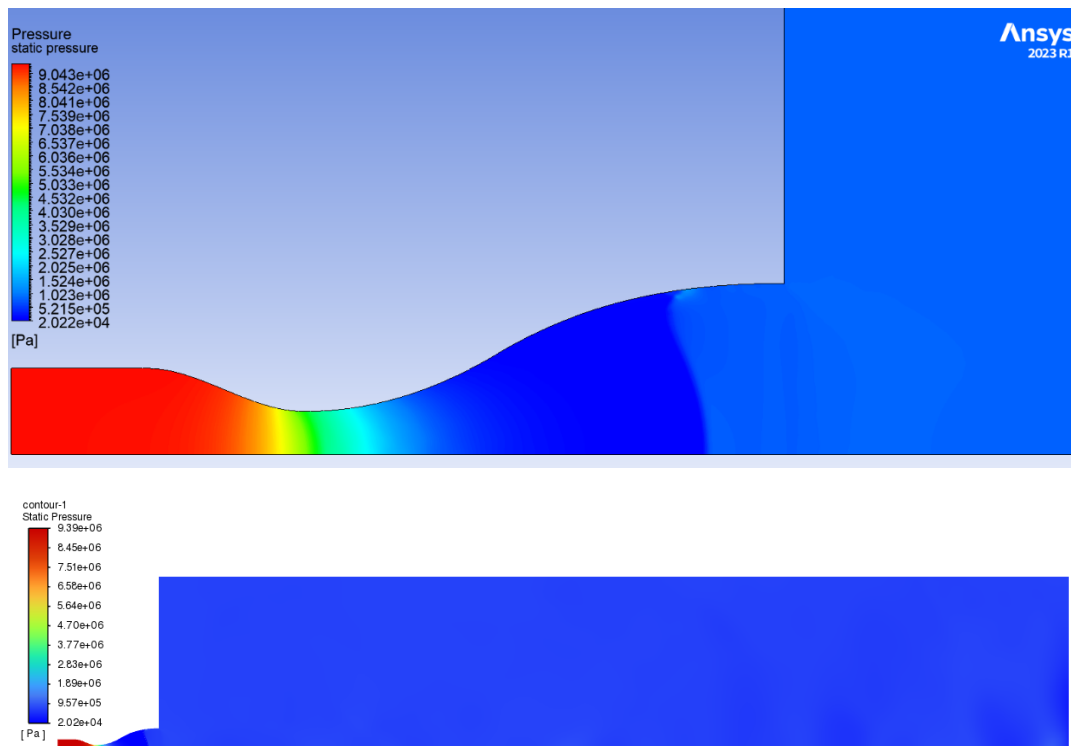
In the condition of shock in the nozzle exit, the following results are obtained in Fluent. The behavior of residuals during the corresponding solution process Figure 4.16. It is, as mass flow rate behavior during solution Figure 4.17 is. Static pressure contours according to Figure 4.18 is. as well contours of Static temperature in Figure 4.19 and velocity contours in Figure 4.20 are visible



**Figure 4.16 The behavior of residuals during the solution process under shock conditions at the nozzle exit**

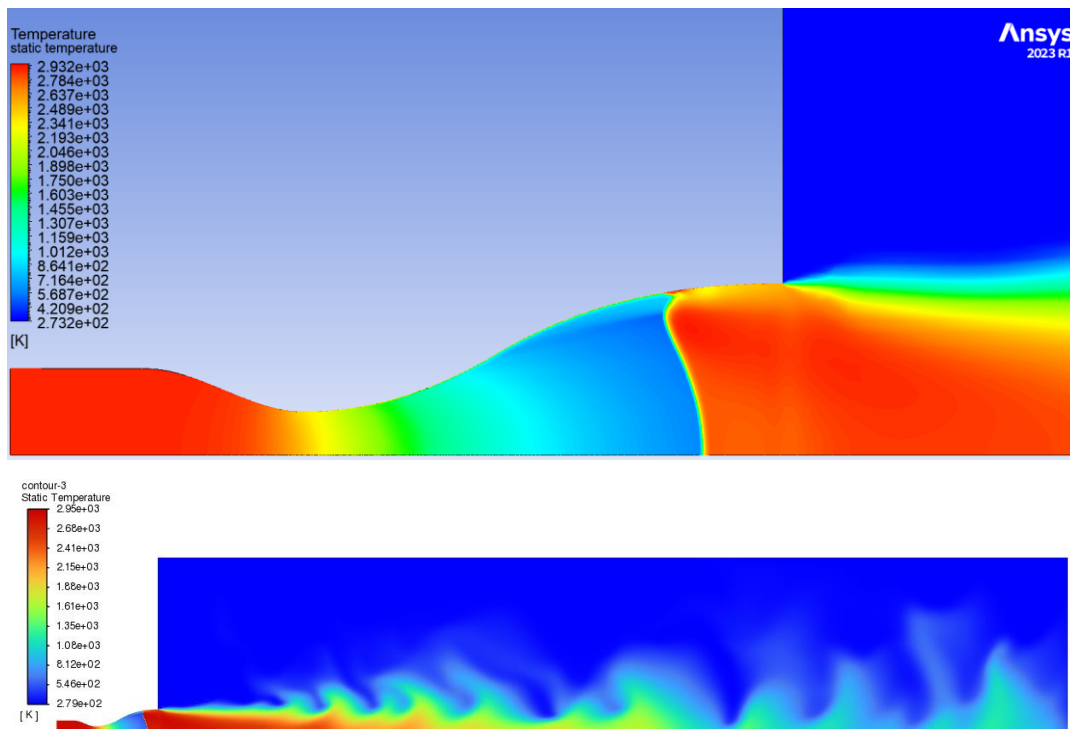


**Figure4.17** Nozzle mass flow rate behavior during the solution process in shock conditions at the nozzle exit

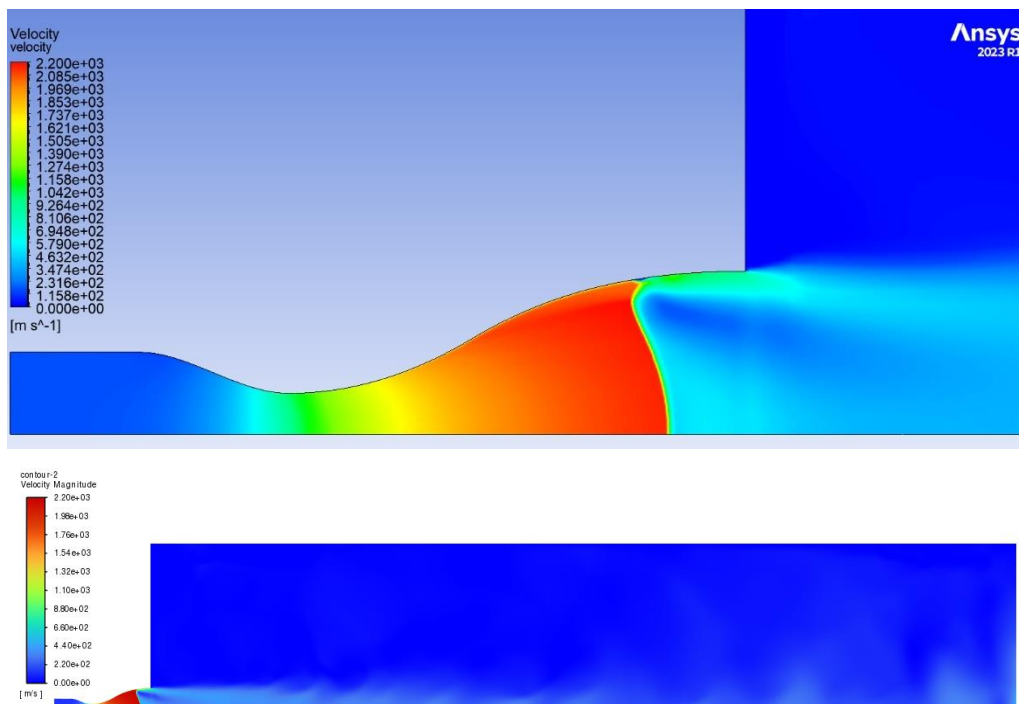


**Figure4.18** Static pressure contours in shock conditions at the nozzle exit





**Figure 4.19 Static temperature contours in shock conditions at the nozzle exit**



**Figure 4.20 Velocity contours in shock conditions at the nozzle exit**

The output of mass flow, Mach and speed in the software is as follows:

Area-Weighted Average

Velocity Magnitude [m/s]

Interior 457.21263

Area-Weighted Average

Mach number

Interior 0.43835978

Mass Flow Rate [kg/s]

inlet 1154.3774

Net 1154.3774

$$T_{fluent} = \dot{m}u_e = 1154.37 \times 457.21 = 0.53 \text{ MN}$$

$$error_{u_e} = \frac{|457.08 - 457.21|}{457.21} \times 100 = 0.02\%$$

$$error_{\dot{m}} = \frac{|1159.7 - 1154.37|}{1154.37} \times 100 = 0.46\%$$

$$error_T = \frac{|0.53 - 0.53|}{0.53} \times 100 = 0.0 \dots \%$$

(10)

#### 41.6. analysis

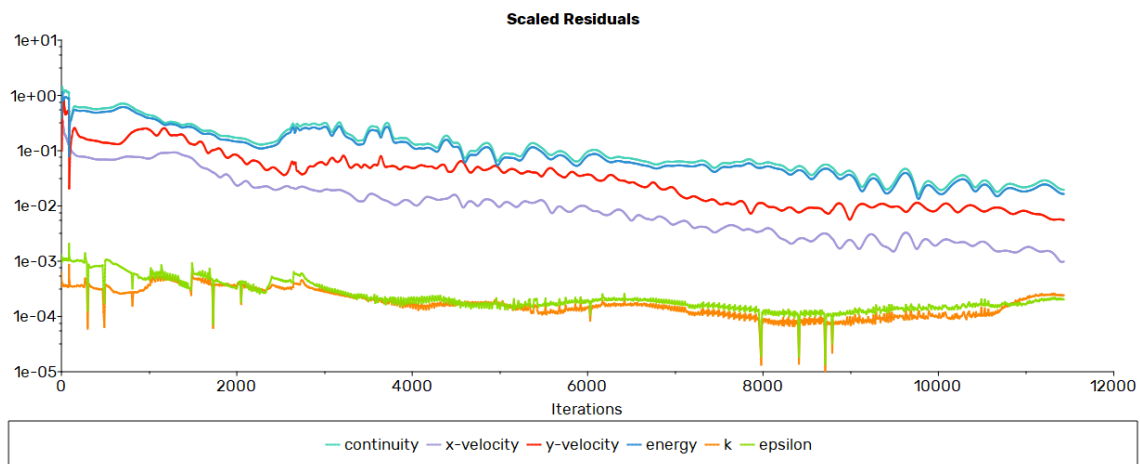
As can be seen, the theoretical results and numerical solution results are in good agreement and the resulting error percentage is low and favorable. Charts Figure4.16 And Figure4.17 They show a good convergence in the solution, but with the continuation of the solution and more iterations, more convergence was not achieved and a fluctuating behavior was observed. As in static pressure contours Figure4.18 It can be seen that the static pressure before the throat is almost constant and equal to the pressure of the chamber, and the formation of a normal shock in the exit can also

be seen. Static temperature contours Figure4.19 and speed contours Figure4.20 They show the formation of normal shock in the nozzle exit, but the difference that can be seen in the theoretical and numerical results is that the normal shock is formed in the pressure of another environment exactly in the nozzle exit.

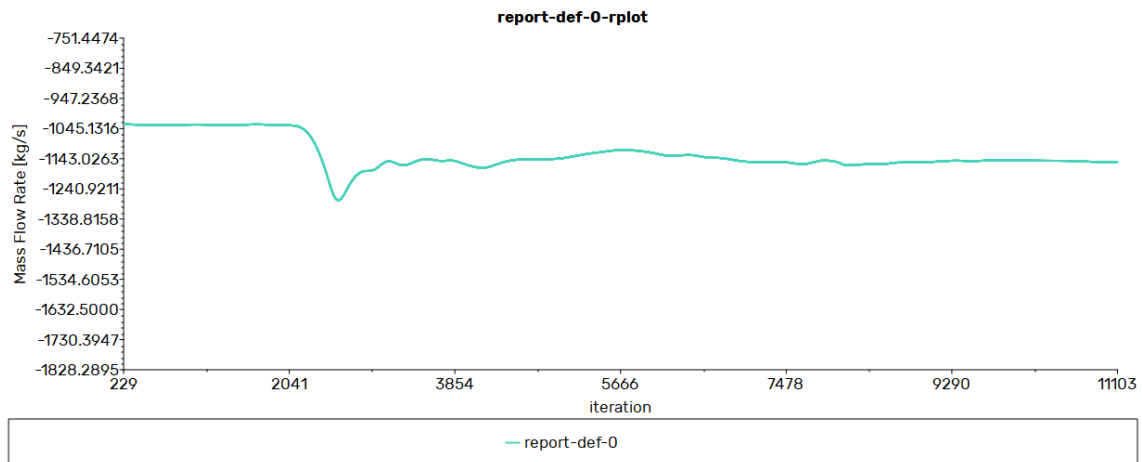
#### 47. Vacuum conditions

In vacuum conditions, the following results are obtained in Fluent. The behavior of residuals during the corresponding solution process Figure4.21 is, as well as the flow behavior during the corresponding solution process Figure4.22 is.

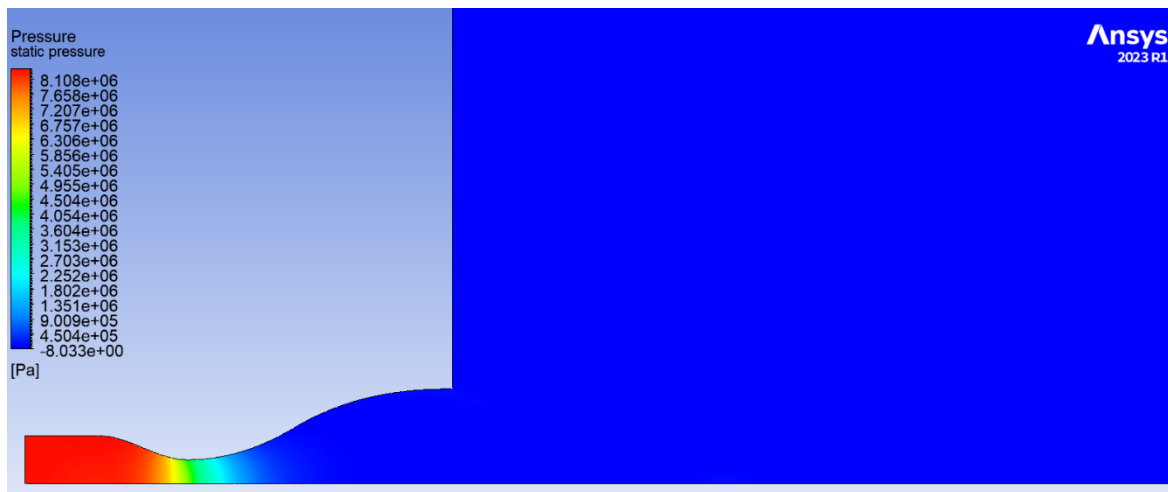
Corresponding static pressure contours Figure4.23 is. Also static temperature contours in Figure4.24 and speed contours in Figure4.25 are visible



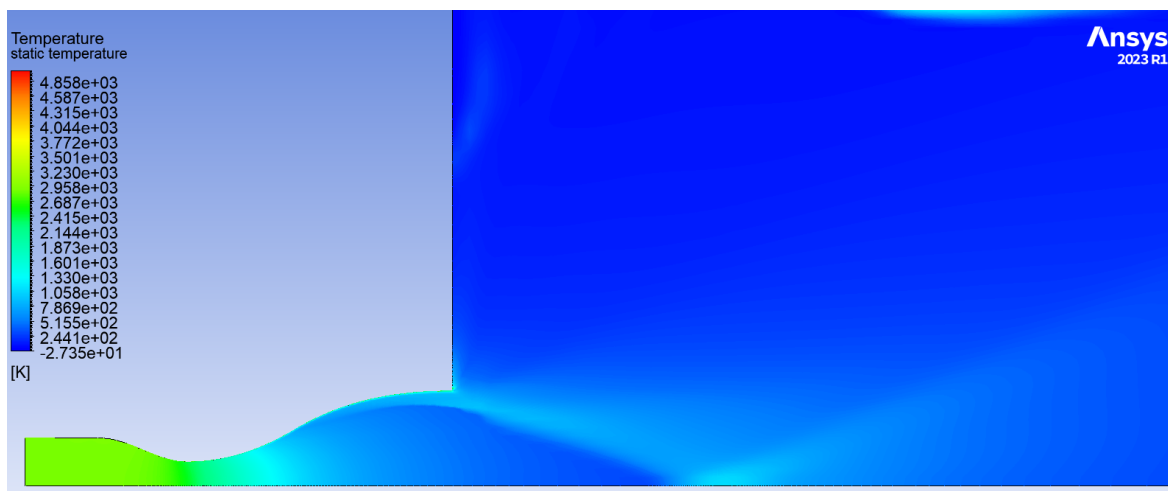
**Figure4.21 The behavior of resins during the solution process in vacuum conditions**



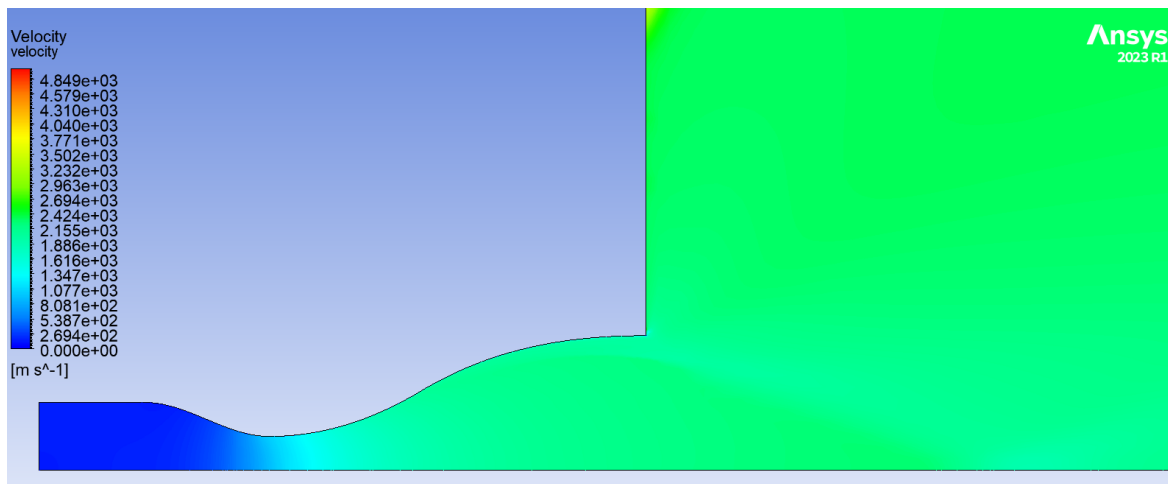
**Figure4.22** The mass flow behavior of the nozzle during the solution process in vacuum conditions



**Figure4.23** Static pressure contours in vacuum conditions



**Figure4.24** Static temperature contours in vacuum conditions



**Figure4.25 Velocity contours in vacuum conditions**

The output of mass flow, Mach and speed in the software is as follows:

Area-Weighted Average  
Mach number

-----  
Interior **4.5779364**

Area-Weighted Average  
Velocity Magnitude [m/s]

-----  
Interior **2147.6625**

Mass Flow Rate [kg/s]

-----  
inlet **1140.2629**

-----  
Net 1140.2629

By comparing the results obtained from Fluent with the theoretical results, we will have:

$$\begin{aligned}
T_{fluent} &= \dot{m}u_e + (P_e - P_a)A_e \\
&= 1140.26 \times 2147.66 \\
&\quad + 35.6 \times 2.562 \times 1000 = 2.54 \text{ MN} \\
error_{u_e} &= \frac{|2197.5 - 2147.66|}{2147.66} \times 100 = 2.32\% \\
error_{\dot{m}} &= \frac{|1159.7 - 1140.26|}{1140.26} \times 100 = 1.7\% \\
error_T &= \frac{|2.59 - 2.54|}{2.54} \times 100 = 1.96\%
\end{aligned} \tag{11}$$

#### 41.7. analysis

As can be seen, the theoretical results and numerical solution results are in good agreement and the resulting error percentage is low and favorable. Charts Figure4.21 And Figure4.22 They show good convergence in the solution. As in static pressure contours Figure4.23 It can be seen, the static pressure before the throat is almost constant and equal to the chamber pressure. Static temperature contours Figure4.24 and speed contours Figure4.25 They show the formation of oblique shocks in the exit of the nozzle and the effect of the boundary layer.

#### 48. General analysis

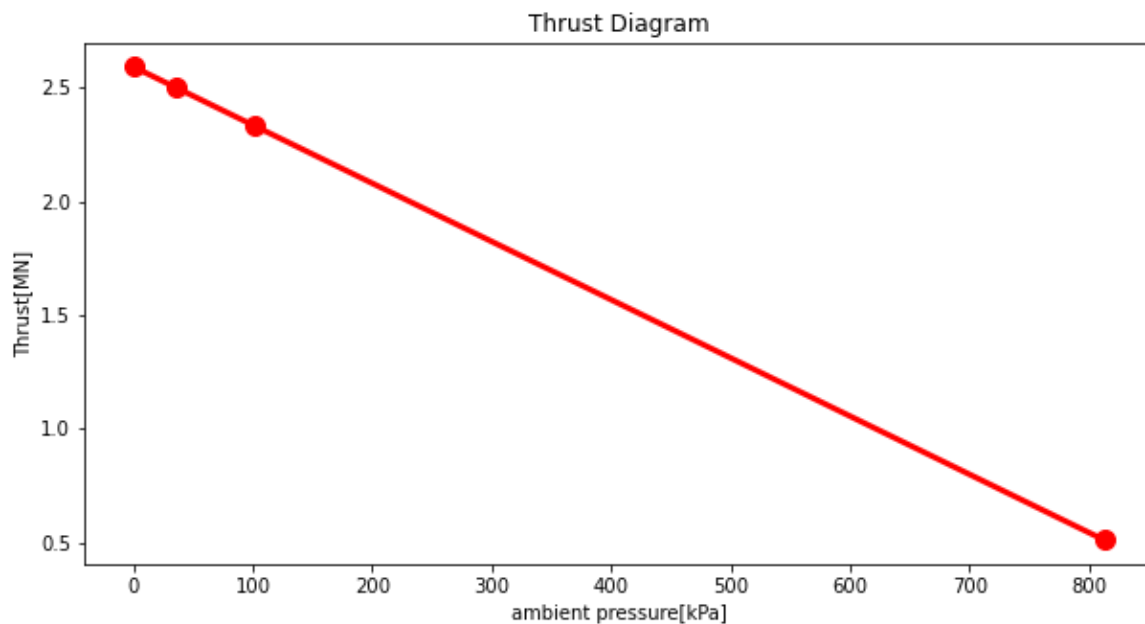
As in static pressure contours Figure4.8, Figure4.13, Figure4.18 And Figure4.23 It can be seen that the static pressure before the throat is the same in all cases and has a similar behavior, but in the case of the shock pressure at the nozzle exit, this is true only at the first moment, but after that, the pressure behind the nozzle changes the behavior of the static pressure before the throat.

## 5 Requested charts

In this part, the desired graphs are drawn from the vacuum pressure to the shock pressure at the nozzle exit.

### 51. Thrust chart

Chart requested for thrust Figure5.1 is.

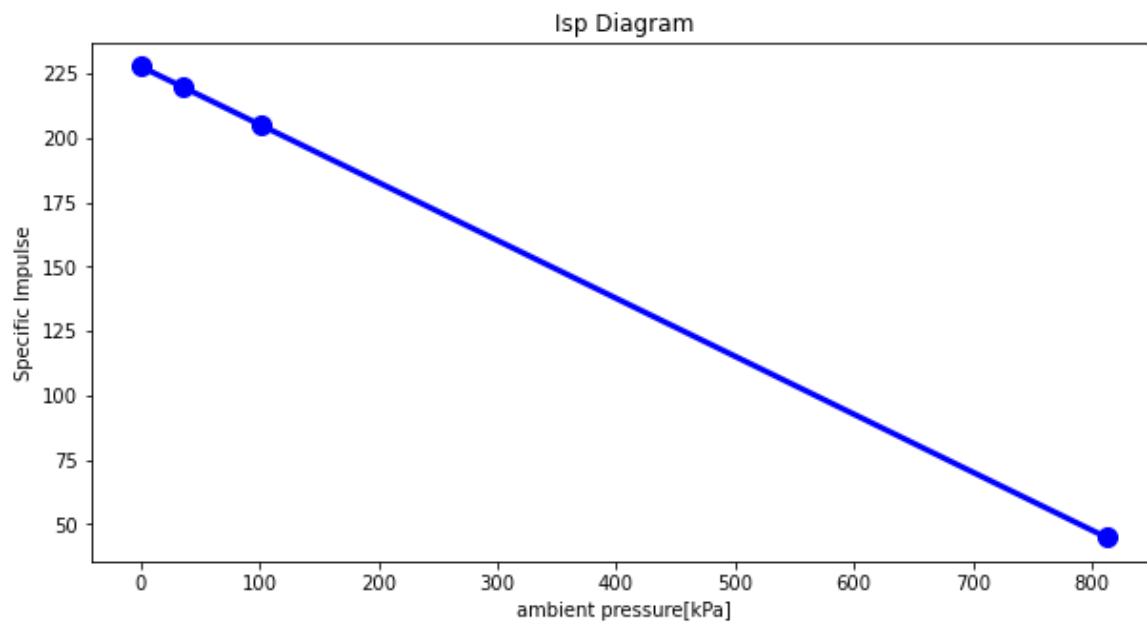


**Figure5.1 Thrust diagram from vacuum pressure to shock pressure at nozzle exit**

As can be seen, this diagram changes linearly and the reason for this is that in different pressures, mass flow rate, outlet velocity and outlet pressure are constant and only the ambient pressure changes in relation to the thrust.

### 52. Special impact diagram

Requested graph for matching special hitFigure5.2is.



**Figure 5.2 Specific impact diagram from vacuum pressure to shock pressure at nozzle exit**

As can be seen, this diagram has a behavior similar to thrust, because the mass flow rate is constant and the thrust decreases, so the specific impact also decreases.



# End

**Ebrahim Safdarian**  
**401129076**  
**April 2023**