JETIR.ORG | ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Effect of Variation in Geometrical Parameters on Ejector Performance Using CFD

Gajendra Singh Tomar[1], Adarsh Partap Singh Bais[2],
Lokesh Aurangabadkar[3], Dr. D V Singh[4],
Deepak Yadav[5], Ankit Patidar[6]

[1,2,5,6] UG student, Department of Mechanical Engineering, IIST, Indore, India

[5]Assistant Professor, Department of Mechanical Engineering, IIST, Indore, India

[6]Associate Professor, Department of Engineering, IIST, Indore, India

ABSTRACT

Supersonic ejectors are widely used in a variety of applications such as aerospace, propulsion and refrigerator. The main interest of this study is to establish a reliable hydrodynamics model for the supersonic ejector, which can be extended to refrigerator installation. Since the early 1900s, Supersonic Ejectors have been used in cooling systems / refrigerators. This project demonstrates the effects of computational fluid dynamics (CFD) on high ejector simulations for use in the refrigerator system. The proposed model was used in geometric variation as a change in throat radius and nozzle exit position using R134a. The effect on pressure, velocity, density and temperature was investigated by variation in geometry. The results show that CFD is a useful tool for building ejectors for refrigeration applications.

1. INTRODUCTION

Ejectors have long been used in cooling applications and as vacuum generators. They have not been favoured for use in refrigeration systems due to the relatively low COP (Coefficient of Performance) of ejector refrigeration systems in comparison with vapour-compression or absorption refrigeration systems. However, their simple mode of function, lack of moving parts and capability of driving a refrigeration device primarily through the use of waste heat or solar energy make them particularly attractive in this energy-conscious era. In addition, using waste heat or solar energy to power a refrigeration system will reduce the electrical energy consumption used to power vapour compression refrigeration systems, potentially reducing the emissions of greenhouse gasses that are associated with the production of electricity from fossil fuel burning power plants. The purpose of this paper is to present the results of CFD (Computational Fluid Dynamics) simulations of a supersonic ejector for use in a refrigeration system. This numerical model can be used as a design tool to improve the operation of ejectors, specifically by investigating the impact of various operating conditions and geometrical factors on the ejector entrainment and compression ratios. Many previous numerical models of ejectors have been limited to one-dimensional analyses. In order to better understand the function of ejector refrigeration systems, a commercially available CFD code has been used to create and solve a two-dimensional, axis-symmetric model of a supersonic ejector. The CFD code uses the (FVM) Finite Volume Method to solve the coupled partial differential equations of fluid flow. These studies include experimental investigations, one-dimensional and two-dimensional, axis-symmetric CFD models of ejectors. The goals of this article are: to compare the results of the present model with those from studies that used the R134a; to use the model to simulate a variation in ejector geometry that is substitute in throat radius and nozzle exit position; and to conduct a preliminary parametric study that will help identify key features that may impact ejector performance.

1.1 Ejector Theory

Ejector refrigeration system uses an ejector, a liquid pump and a vapor generator to replace the mechanical compressor. Fig.1 illustrates a simple ejector refrigeration system with its major components labelled. The generator receives heat from a low-cost, low-grade thermal energy source and heats up the refrigerant to produce high pressure and high temperature vapor known as the primary fluid that enters the ejector and accelerates through the ejector nozzle where the jet issuing from it entrains the low-pressure secondary flow coming from the evaporator. The resulting fluid mixture which is at an intermediate pressure passes through the condenser, where heat rejection process occurs, and leaves as liquid refrigerant. Most of the refrigerant leaving the condenser is pumped back to the generator and the rest enters the expansion valve to reduce its pressure down to that of the evaporator where another heat absorption process takes place. The system's performance is best described by its Coefficient of Performance (COP) and the ejector's entrainment ratio (ER).

1.2 Computational Fluid Dynamics (CFD) Theory

The governing equations for fluid flow such as equations for conservation of mass, momentum, and energy form a set of coupled, nonlinear partial differential equations. Analytical methods may not be possible to solve these equations for most engineering problems but, computational fluid dynamics is capable of dealing with these types of equations. CFD is a branch of fluid mechanics and is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena. The domain is discretized into finite sets of control volume where the equations in the form of Navier -Stokes partial differential equations are solved. These equations are converted into a set of algebraic equations at discrete points which are then solved numerically to render a solution with the appropriate boundary conditions.

Fluid flow through the ejector can be considered compressible, turbulent, steady-state and axis- symmetric. The Navier -Stokes continuity, momentum and energy equations provide the foundation in CFD simulation of fluid motion. This results in Reynolds-averaged Navier- Stokes equations. To find closure to this, a popular approach to turbulence modelling employs the Boussinesq hypothesis to relate the Reynolds stresses to the mean velocity gradients. The subsequent equations are written in Cartesian tensor form as:

THE MASS CONSERVATION EQUATION

$$\partial\rho/\partial t + \nabla.(\rho\vec{v}) = S_m$$

Sm = mass added to the continuous phase from dispersed second phase

For 2D axisymmetric geometries continuity equation is given by,

$$\partial\rho/\partial t + \partial/\partial x\,(\rho v_x) + \partial/\partial r\,(\rho v_r) + \rho v_r/r = S_m$$

vx = axial velocity

vr = radial velocity

x = axial coordinate

r = radial coordinate

THE MOMENTUM CONSERVATION EQUATION

$$\partial/\partial t\,(\rho\vec{v}) + \nabla.(\rho v\ v) = -\nabla P + \rho g + F$$

P = static pressure

ρg = gravitational body force

F = external body force

For 2D axisymmetric geometries the axial and radial momentum conservation equation is given by,

$$\partial/\partial t\ (\rho v_x) + 1/r\ \partial/\partial x\ (r\rho v_x v_x) + 1/r\ \partial/\partial r\ (r\rho v_r v_x) = -\partial P/\partial X + F_x$$

and

$$\partial/\partial t\ (\rho v_r) + 1/r\ \partial/\partial x\ (r\rho v_x v_r) + 1/r\ \partial/\partial r\ (r\rho v_r v_r) = -\partial P/\partial r + F_r$$

where

$$\nabla.\ v = \partial v_x/\partial x + \partial v_r/\partial r + v_r\ /r$$

ENERGY EQUATIONS

Conservation of energy is described by,

$$\partial/\partial t(\rho E) + \nabla.\ (v\ (\rho E + p)) = -\nabla.\ (\Sigma\ h_j\ J_j) + S_h$$

The stress tensor and energy equations are given in equations, respectively. The total energy equation takes into account the effects of viscous forces on fluid motion as this incorporates the viscous dissipation in it. The COP & Entrainment ratio are given as:

$$\mathrm{COP} = \mathrm{Q_e} / (\mathrm{Q_g} + \mathrm{W_p})$$

$$\mathrm{ER} = \mathrm{m_s} / \mathrm{m_p}$$

1.3 Ejector Geometry

Ejector, being the most critical component dictates the overall performance of the ejector refrigeration system. Thus, its configuration and geometry must be carefully determined and designed. Combinations of such dimensions and parameters are tried on in the CFD simulation one by one until near-optimum geometry is established which gives maximum entrainment ratio at the desired operating conditions of the refrigeration system for R134a refrigerant.

A general schematic drawing of the ejectors used in the numerical simulations. It is assumed that the ejectors are axis-symmetric, thus only the top half of the drawn ejector will be modeled in this work; there is a line of symmetry on the z-axis.

1.4 CFD Implementation

Two-dimensional (2-D) axis- symmetric model of the flow domain is used to minimize computational time. Quad mesh is employed using Ansys Meshing due to geometric simplicity; and is imported to Ansys Fluent v.19.2, for mesh checking and subsequently, for the simulation. This helps prevent diverging solution and makes the simulation process smooth. In addition, solver selected is density-based type with implicit formulation on the account that the flow is compressible. This type of solver computes the governing equations of continuity, momentum, and energy and species transport simultaneously; and afterwards, governing equations for additional scalars such as turbulence will be solved sequentially. For steady-state assumption with travelling shocks, implicit formulation may be more efficient.

2. Literature Review

(Zheng et al., 2018) Compared to absorption systems, ejector systems have advantages because of their simplicity, reliability, low installation and operational costs. Their typically low COP of around 0.2 can be improved substantially when combined with other systems such as absorption or vapour-compression.[1]

(Jeon et al., 2022)). An experiment was conducted with various ejector designs and working conditions. An ANN model of a household refrigeration cycle with an ejector was developed. The effects of the NXP on the performance were analyzed under various conditions. The correlation for the optimum NXP was proposed using the developed ANN model. The optimum NXP was proposed according to the operating and geometrical conditions.[2]

(Wen et al., 2021) In ejector-based refrigeration systems, the fluid phase state plays a key role in improving the ejector performance. In this study, the effects of liquid volume fraction on ejector performance were investigated by computational fluid dynamics simulations. The ejector was applied in a multi-evaporator refrigeration system used for refrigerated trucks with refrigerant R134a superheated gas and gas-liquid two-phase mixture. Under three operating modes (air-conditioning –freezing, refrigerating – freezing, and air-conditioning – refrigerating modes) of the ejector, by varying liquid volume fraction of the two inlets of the ejector in the range of 0–0.1, the variation trends of primary mass flow rate, secondary mass flow rate and entrainment ratio were obtained.[3]

(Zhang et al., 2020) An ejector containing phase changing gas-liquid flow process acts as a popular and decisive device in multiple industrial applications, including the hydrogen production, electricity production, fuel cells, refrigeration, petroleum industry and desalination systems. However, non-condensable gas is inevitable for the usual operation of phase-changing gas-liquid ejector in the trigeneration or electrolyze system for hydrogen production, and rarely research is concerned with this issue. In the present study, the effect of non-condensable gas contained in the condensable gas on the characteristics of gas-cantered water ejector is presented, with steam, water and air acting as the gas, liquid and non-condensable gas, respectively.[4]

(Miwa et al., 2021) The steam-water condensing injector (SI) is a device capable of producing high-pressure subcooled liquid streams by combining steam and subcooled liquid jet at a higher pressure than the inlet streams. The present study investigated the SI's pressure elevation mechanism using supersonic steam and a subcooled water jet.[5]

(Omidvar et al., 2016) In the present paper, a study of flow pattern in a variable geometry ejector used in solar refrigeration system was conducted in an effort to achieve the optimum geometry and improve the understanding of the important flow behaviors that lead to ejector performance drop. Since the entropy generation is equivalent to performance losses, this parameter was investigated in numerical simulations.[6]

(Smith et al.) A three-evaporator and dual-ejector refrigeration system was presented in this paper. By combining two single ejectors as a whole, the authors attempted to discuss the effects of key geometric dimensions of each stage on both stage performances with two-dimensional CFD simulation method.

3. Methodology

3.1 Modeling of Ejector

A schematic diagram of a vapor-jet ejector showing the maximum values used as part of this study is shown. It is assumed that the ejector is axi-symmetric near the z- axis, so only a small cut of the ejector is shown. Basically, an ejector consists of two flexible pipes of the year. The refrigerator from the evaporator enters axially into the space outside the first mouth through the outer tube. The second flow is accelerated by the length mixed with the main flow. The mixed flow is extended with a diffuser in length until it comes out of the ejector.

3.2 Meshing

Ansys offers standard, high-performance, automated, intelligent integration software that produces the most suitable mesh of precise, efficient multi-physics solutions - from simple, automatic connectivity to a highly designed net. The smart automation is built into software to make meshing a painless and accurate task, delivering the necessary gradient scanning solution accordingly to get reliable results.

3.3 CFD Analysis

The ejector modeling using computational fluid dynamics (CFD) equipped to create points of interest for the flow field and fluid structures is considered for numerical flow domain solutions. Certain types of fluids including heat exchange, radiation, turbulence, etc. in any operating conditions or geometric model can be analyzed using CFD. Information that is difficult to obtain in test settings can be successfully tested using CFD.

3.4 Result

In the design cases of the R134a refrigerator, the analysis predicts pressure, congestion, speed, temperature. This indicates that the exit mode is very hot. A computer domain contains elements where very few of them fall into the hybrid region, as planned. The same pattern occurs with good gas reflection in another refrigerator. Thus, from a mathematical point of view, the flow is basically one phase.

3.5 Change in Geometry & Parameters

These studies include exploratory research, one-dimensional and two-dimensional, axis-symmetric CFD ejector models. The objectives of this article are: to compare the results of the current model with those from studies using refrigerators R134a; using a model to mimic the geometric variation of the ejector replacing the throat radius and the outlet area; as well as conducting initial parametric research that will help identify key factors that may influence ejector performance.

3.6 Processing of Meshing and CFD Analysis

Ejector, being the most important component defines the full functionality of the ejector refrigerator system. Therefore, its configuration and geometry must be carefully determined and designed. A combination of such sizes and parameters is attempted in each CFD simulation until a near-top geometry is established which provides the maximum penetration of the required operating conditions of the refrigerator system R134a.

3.7 Result

The Supersonic Ejector design used in the Vapor Refrigeration System has a major impact on its overall performance. Although there has been a lot of research on the effectiveness of Supersonic Ejectors, this study explains the flow parameters of the flow and changes in different parameters such as pressure, density, velocity and temperature by doing variation in nozzle exit position and throat radius.

3.8 Result Comparison

At refrigeration operating conditions, there is a corresponding fixed configuration ejector that will operate under optimum conditions. A fixed geometry ejector is determined for given design conditions such as pressure, density, velocity, and temperature that operates on near-optimal ejector operation. Some important conclusions are drawn as follows.

The maximum and minimum density in ejector whose throat radius is 4.49 are $1.401e^{+01}$ and $7.134e^{-01}$ and when radius is 2.49 the results are $1.400e^{+01}$ and $5.471e^{-01}$ and when radius is 3.49 the results are $1.400e^{+01}$ and $6.242e^{-01}$ and now when length of nozzle exit position is 21.32 the maximum and minimum density the results are $1.400e^{+01}$ and $6.231e^{-01}$ and when length of nozzle exit position is 15.32 the results are $1.400e^{+01}$ and $6.294e^{-01}$ and when length of nozzle exit position is 18.32 the results are $1.400e^{+01}$ and $6.242e^{-01}$.

The maximum and minimum pressure in ejector whose throat radius is 4.49 the results are $4.005e^{+05}$ and $1.806e^{+04}$ and when radius is 2.49 the results are $4.005e^{+05}$ and $1.399e^{+04}$ and when radius is 3.49 the results are $4.005e^{+05}$ and $1.566e^{+04}$ now when length of nozzle exit position is 21.32 the maximum and minimum pressure are $4.005e^{+05}$ and $1.563e^{+04}$ and when length of nozzle exit position is 15.32 the results are $4.005e^{+05}$ and $1.584e^{+04}$ and when length of nozzle exit position is 18.32 the results are $4.005e^{+05}$ and $1.566e^{+04}$.

The maximum and minimum temperature in ejector whose throat radius is 4.49 the results are $3.510e^{+02}$ and $2.784e^{+02}$ and when radius is 2.49 the results are $3.511e^{+02}$ and $3.047e^{+02}$ and when radius is 3.49 the results are $3.510e^{+02}$ and $2.810e^{+02}$ now when length of nozzle exit position is 21.32 the maximum and minimum temperature are $3.510e^{+02}$ and $2.810e^{+02}$ and when length of nozzle exit position is 15.32 the results are $3.510e^{+02}$ and $2.810e^{+02}$ and when length of nozzle exit position is 18.32 the results are $3.510e^{+02}$ and $2.810e^{+02}$.

The maximum and minimum velocity in ejector whose throat radius is 4.49 the results are $3.795e^{+02}$ and $0.000e^{+00}$ and when radius is 2.49 the results are $3.633e^{+02}$ and $0.000e^{+00}$ and when radius is 3.49 the results are $3.688e^{+02}$ and $0.000e^{+00}$ now when length of nozzle exit position is 21.32.

The maximum and minimum velocity are $3.680e^{+02}$ and $0.000e^{+00}$ and when length of nozzle exit position is 15.32 the results are $3.675e^{+02}$ and $0.000e^{+00}$ and when length of nozzle exit position is 18.32 the results are $3.688e^{+02}$ and $0.000e^{+00}$.

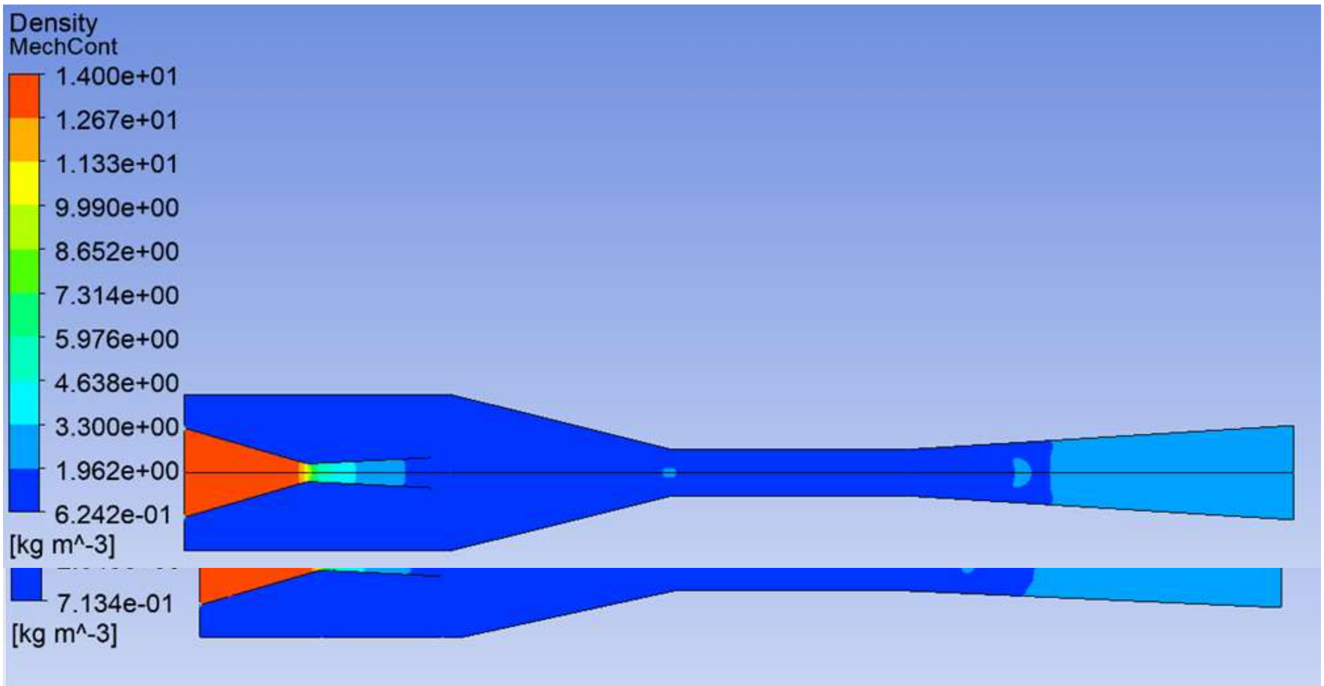
4. Results And Discussion

For the given design conditions for refrigerant R134a, the analysis over pressure, density, velocity, temperature when substitute in throat radius and nozzle exit position. Now the maximum and minimum values of analysis of density among all changes are in which ejector whose throat radius is 4.49 are $1.401e^{+01}$ and whose radius is 2.49 are $5.471e^{-01}$.

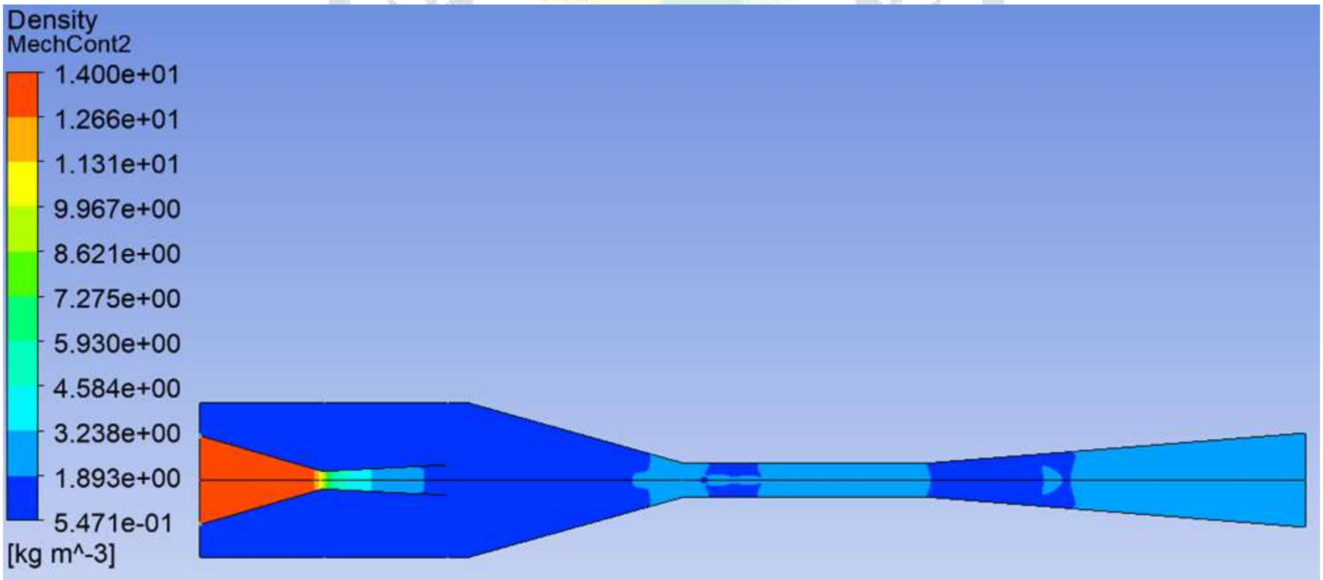
Minimum and the maximum and minimum values of analysis of pressure among all changes are same in all ejector that is $4.005e^{+05}$ and whose radius is 2.49 are $1.399e^{+04}$ are minimum and the maximum and minimum values of analysis of density among all changes are in which ejector whose throat radius is 2.49 are $3.511e^{+02}$ and whose radius is 4.49 are $2.784e^{+02}$ are minimum and the maximum and minimum values of analysis of density among all changes are in which ejector whose throat radius is 4.49 are $3.795e^{+02}$ and same in all ejector that is $0.000e^{+00}$.

1. Density

Throat radius is 4.49:

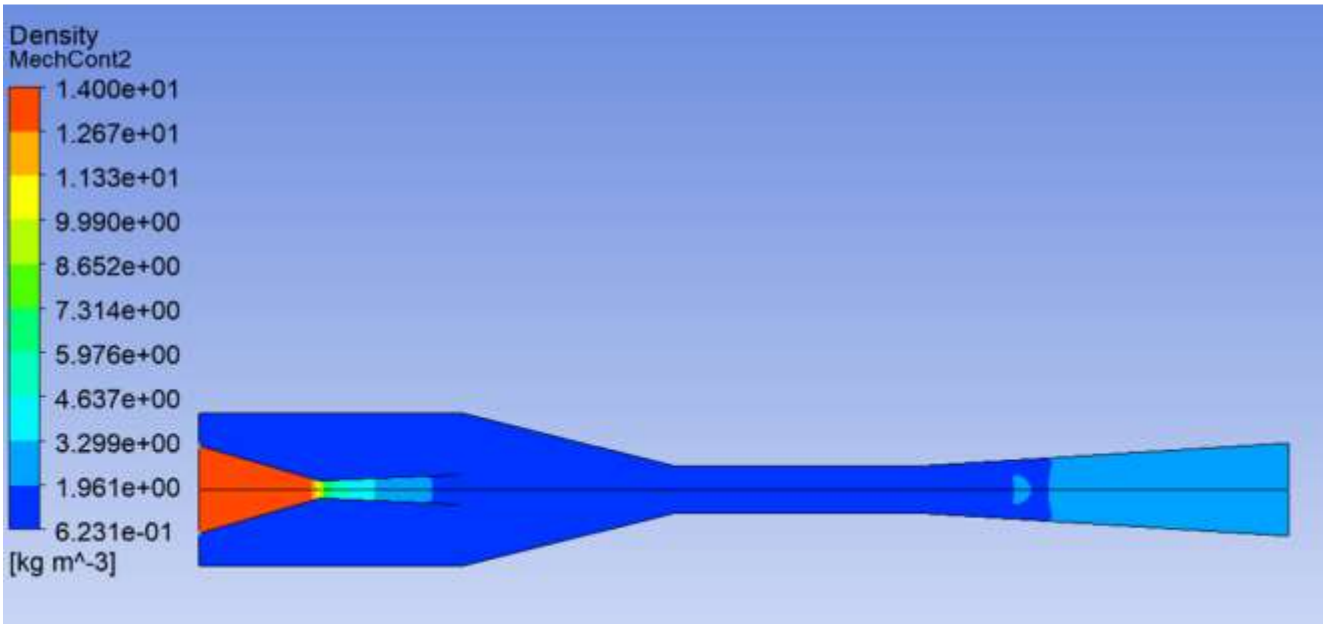


Throat radius is 2.49:

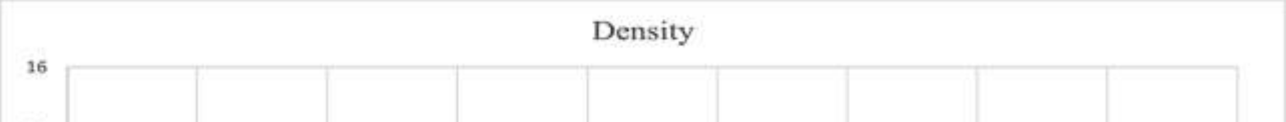


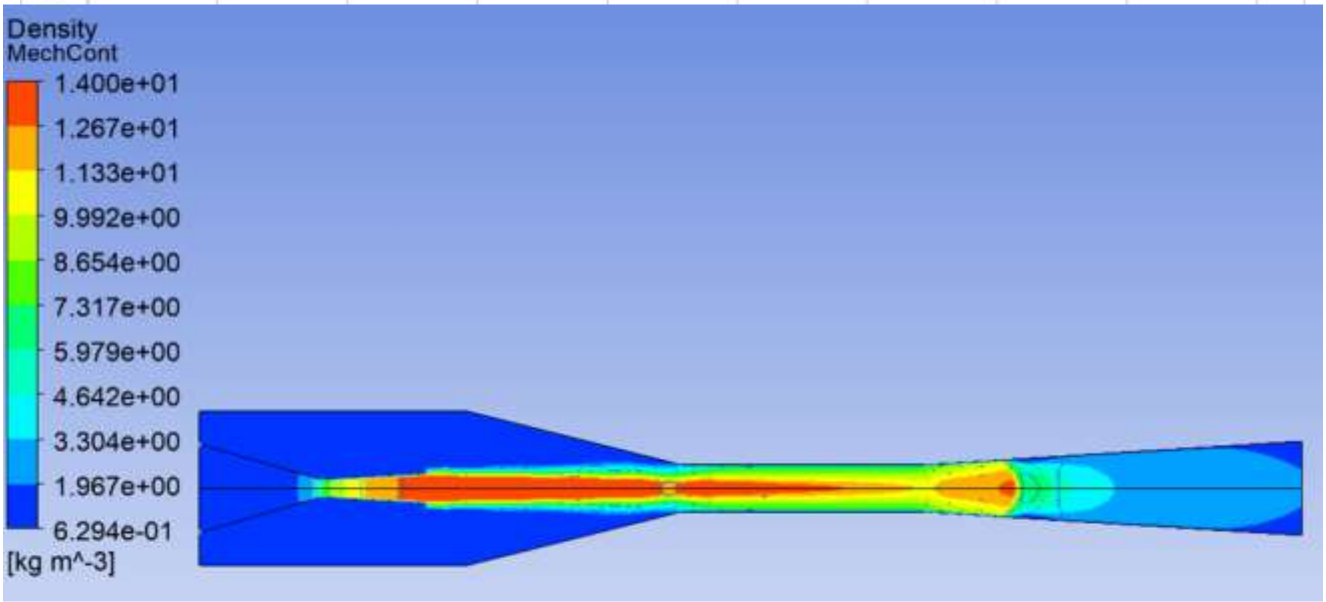
Throat radius is 3.49:

Graph of Density to compare change in Throat radius:



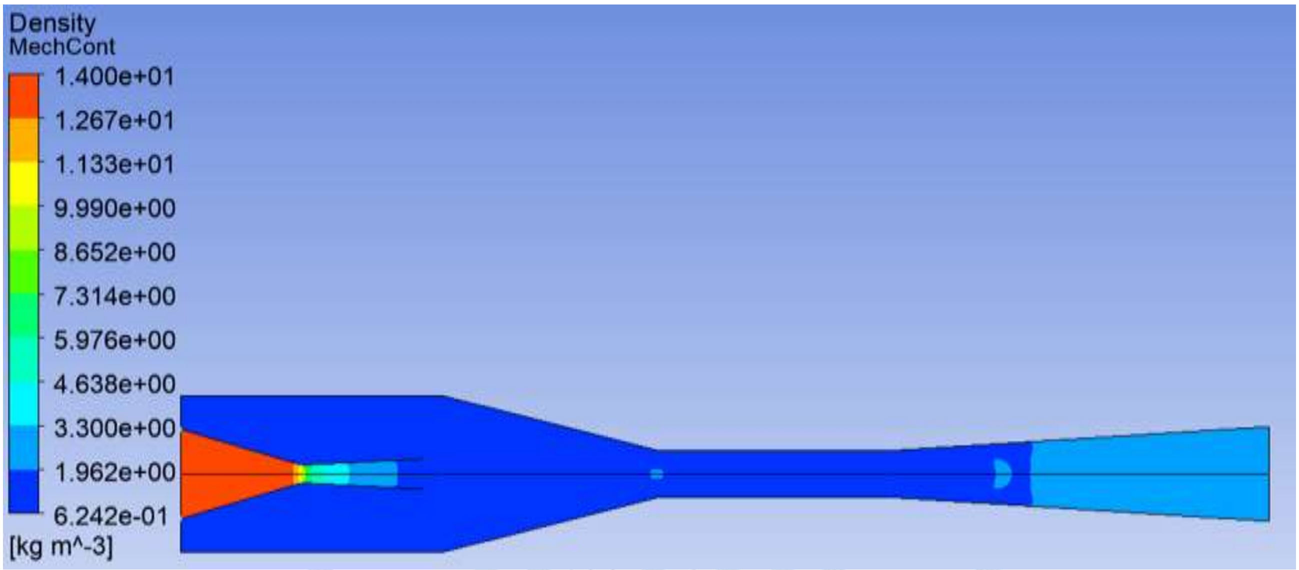
The Figure shows variation of density with increase in axial length of the Ejector. Now as seen from the graph, the density decreases with increases in Ejector Throat Radius however in the radius of 2.49 variation is more, so maximum value at 0.08 or minimum value at 0.12. The least variation in the radius of 4.49 in which minimum value at 0.06 and maximum value at 0.16 in diffuser.



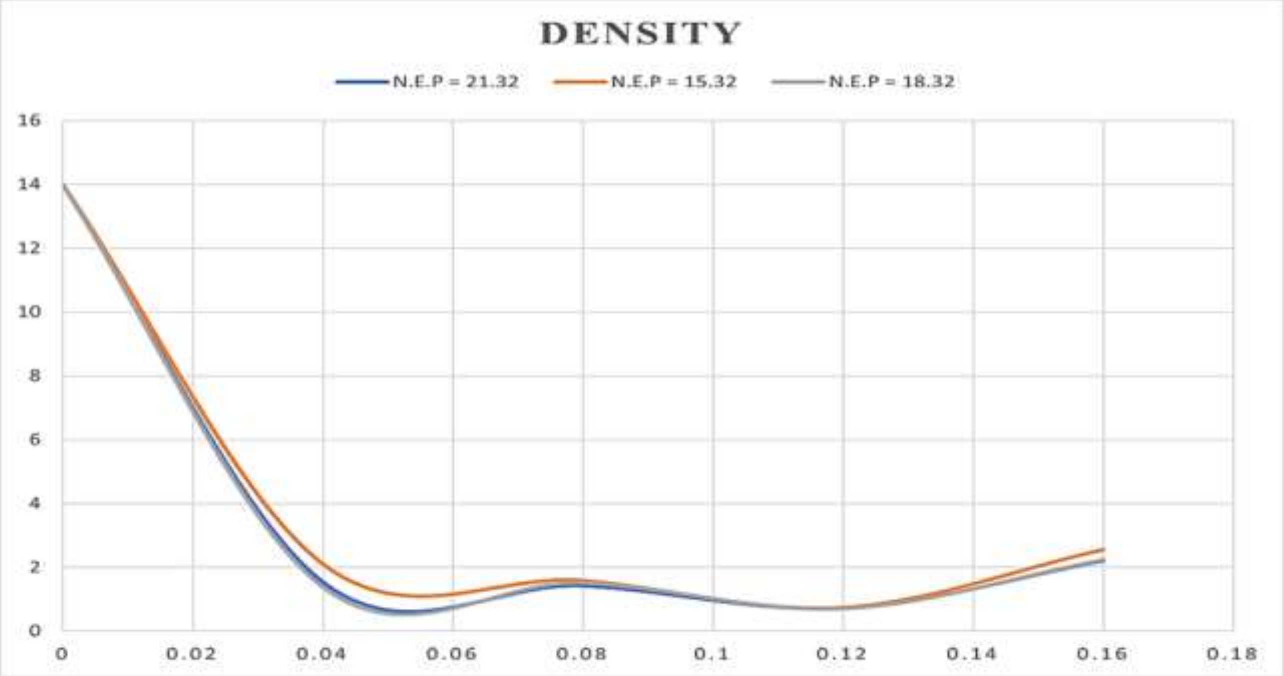


Nozzle exit position is 21.32:

Nozzle exit position is 15.32:



Nozzle exit position is 18.32:

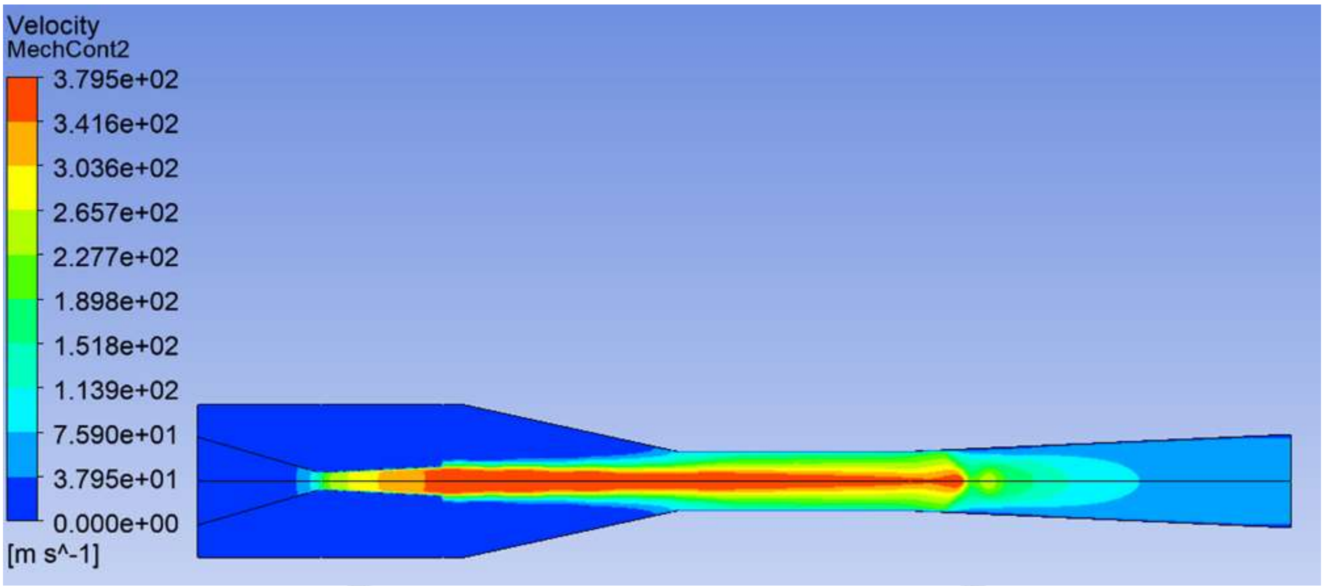


Graph of Density to compare change in length of exit position:

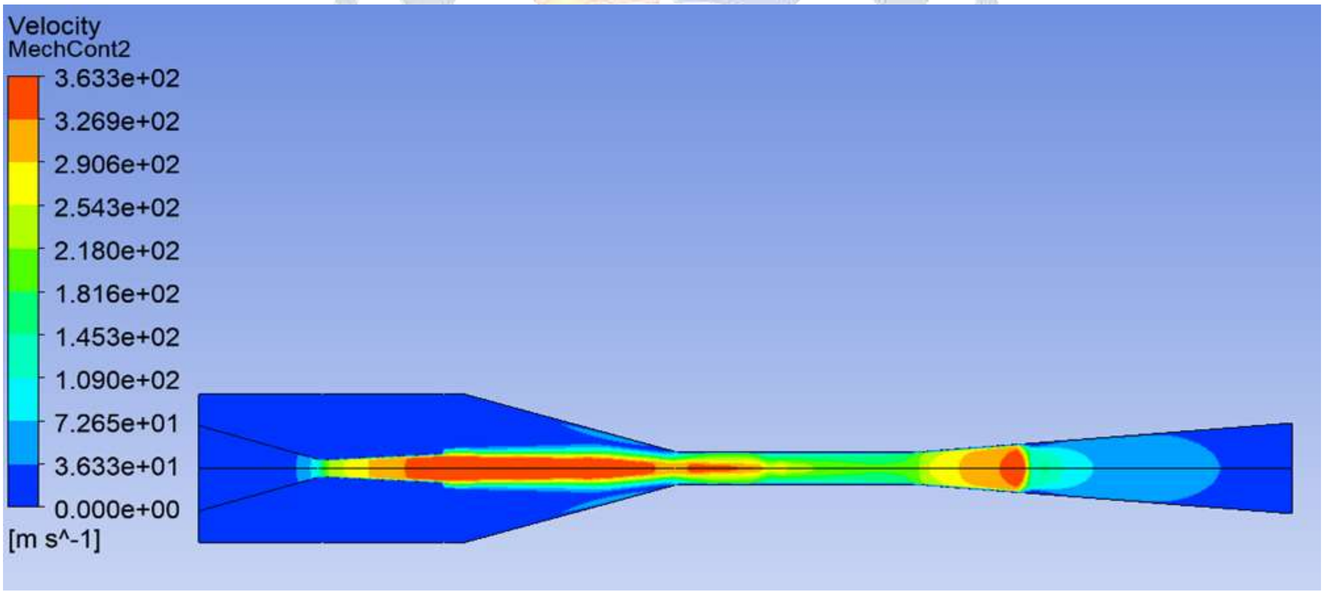
As seen from Figure, with increase in axial length of the Ejector, density of refrigerant R134a is decreases in primary nozzle. When the length of exit position is 15.32, then the mean of density is maximum and at length of exit position 21.32 or 18.32 the variation is minimum in primary nozzle and throat.

2. Velocity

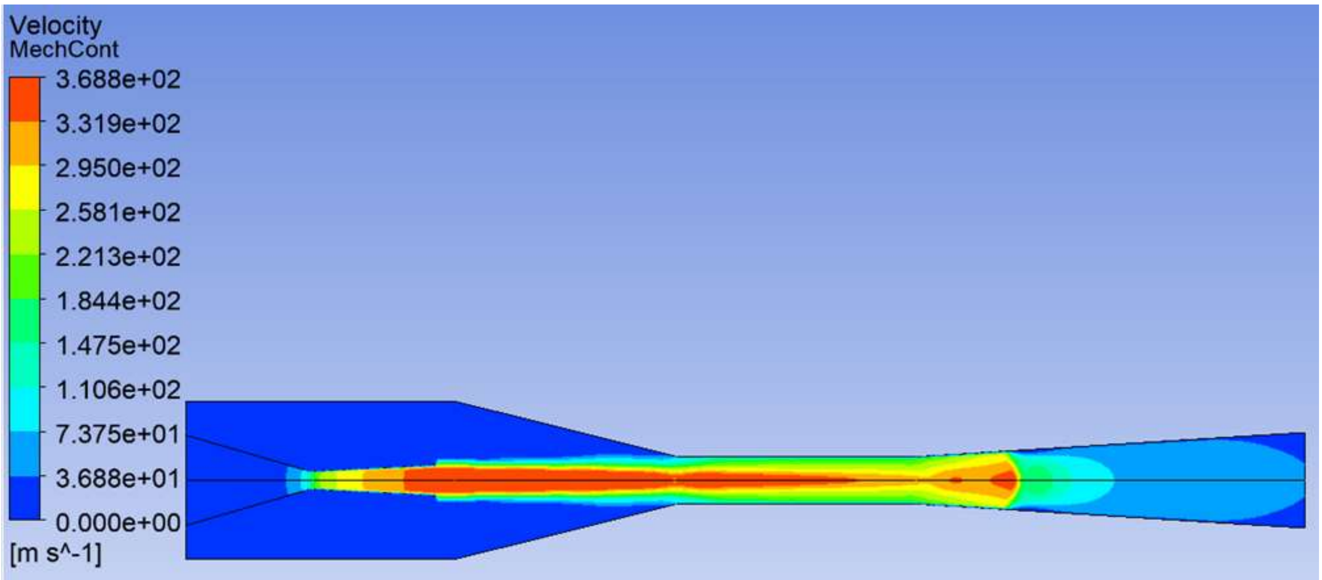
Throat radius is 4.49:



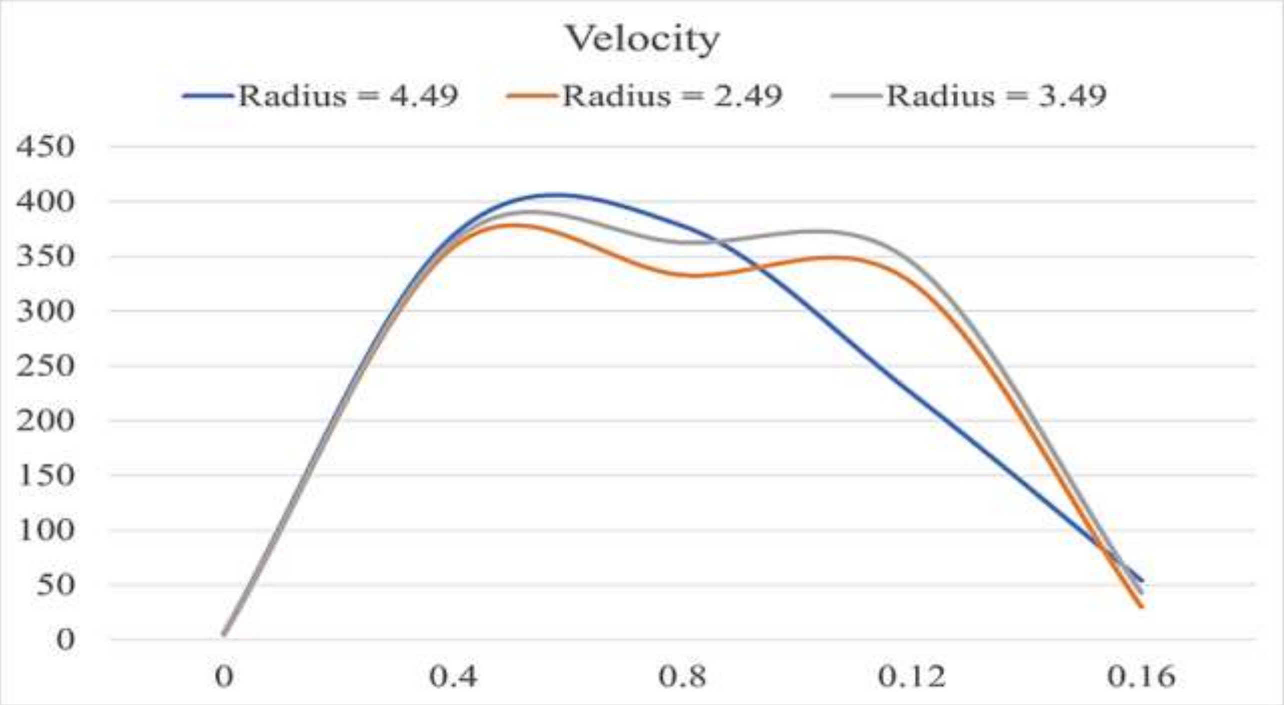
Throat radius is 2.49:



Throat radius is 3.49:

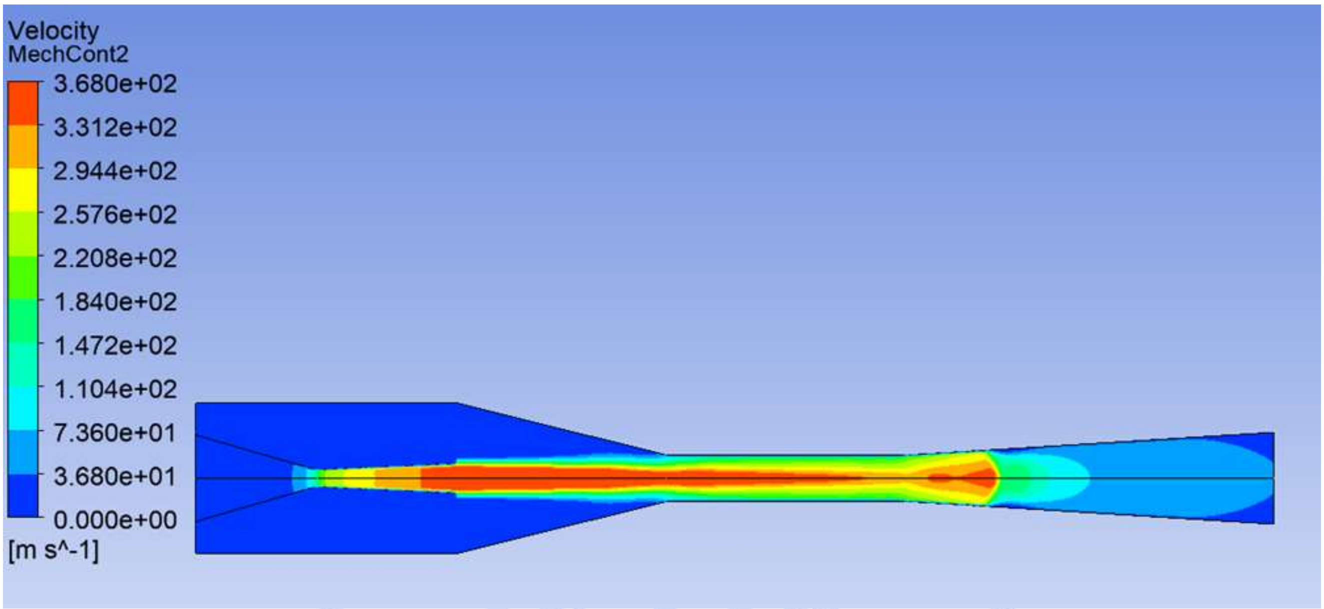


Graph of Velocity to compare change in Throat radius:

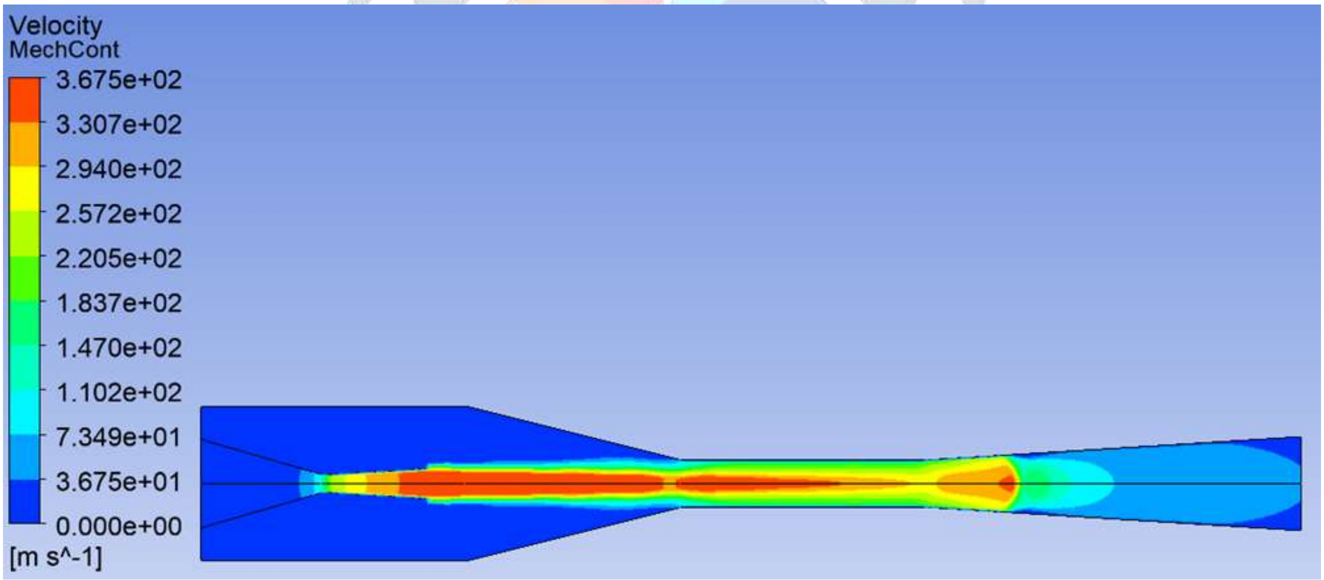


The Figure shows variation of Velocity with increase in axial length of the Ejector. Now as seen from the graph, the velocity decreases with increases in Ejector Throat Radius however in the radius of 2.49 or 3.49 variation is more, so maximum value at between 0.5 or minimum value at 0.8. The least variation in the radius of 4.49 in which minimum value at 0.16 and maximum value at 0.6 in exit of primary nozzle.

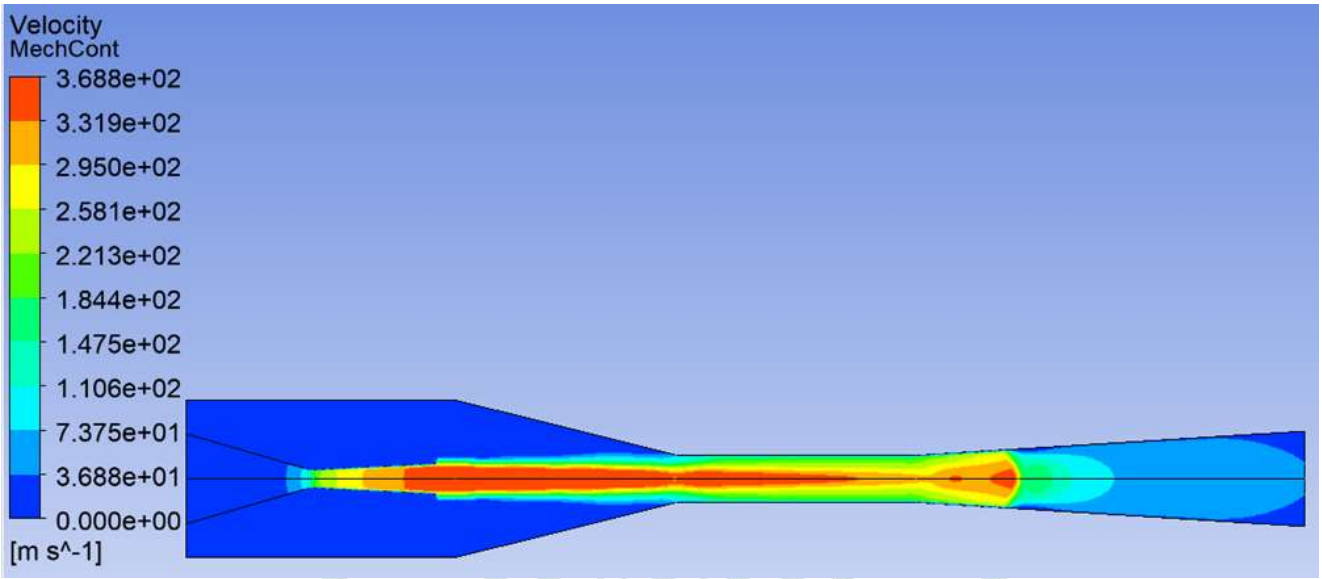
Nozzle exit position is 21.32:



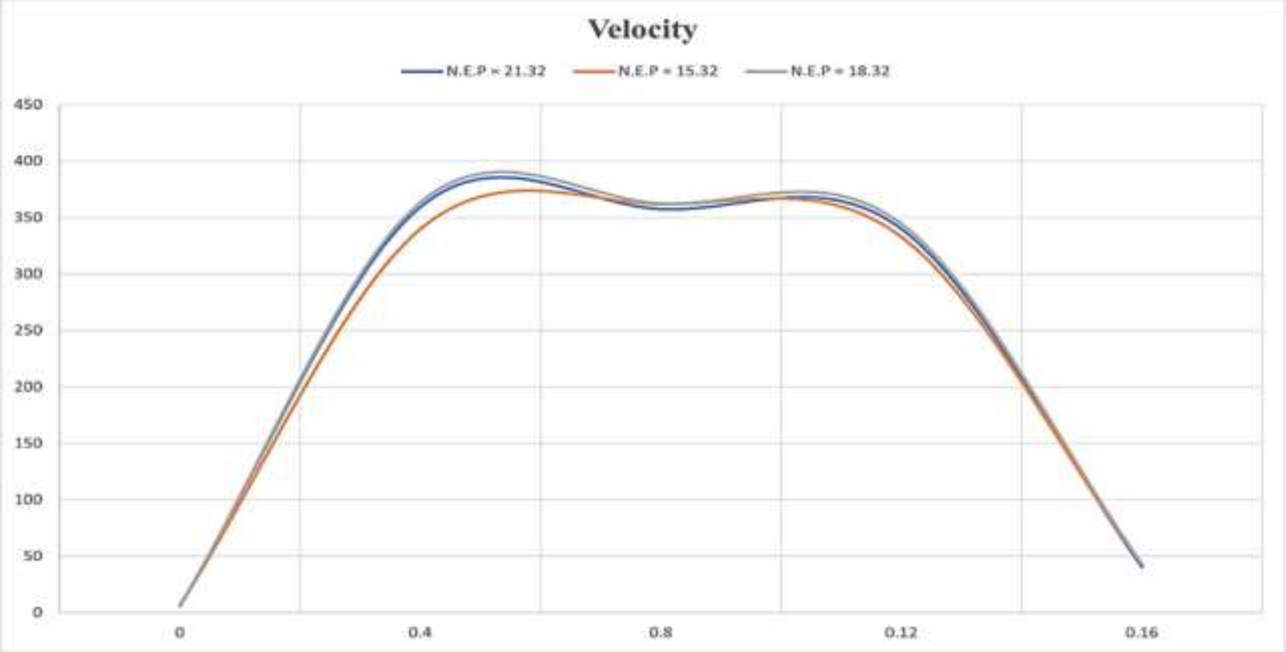
Nozzle exit position is 15.32:



Nozzle exit position is 18.32:



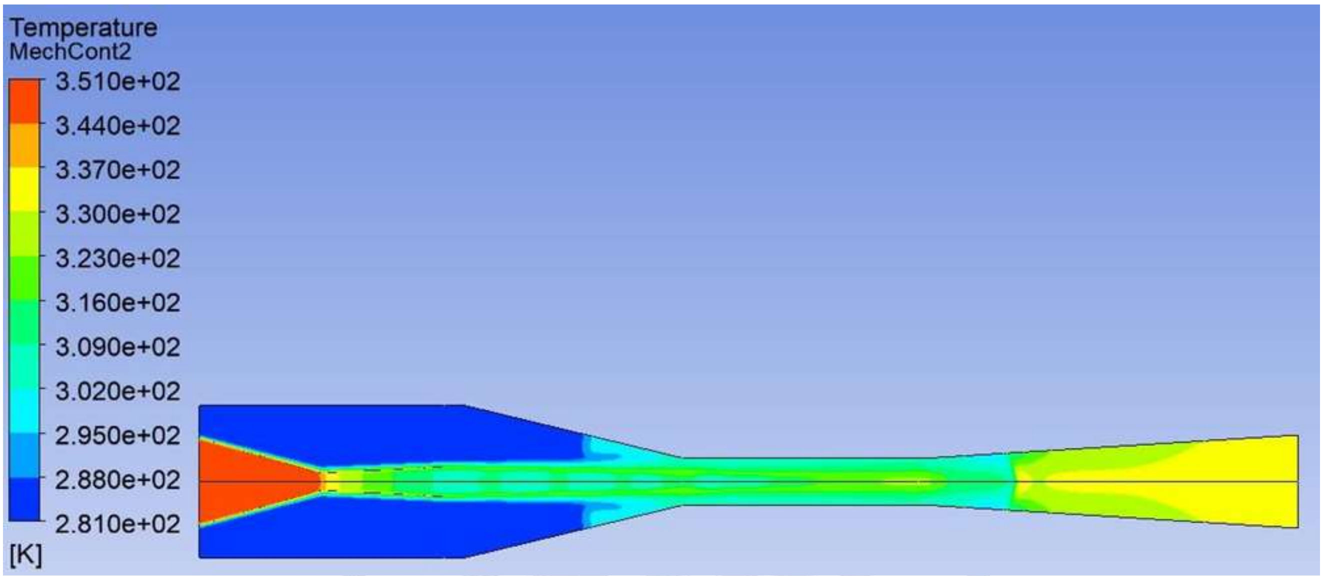
Graph of Velocity to compare change in length of exit position:



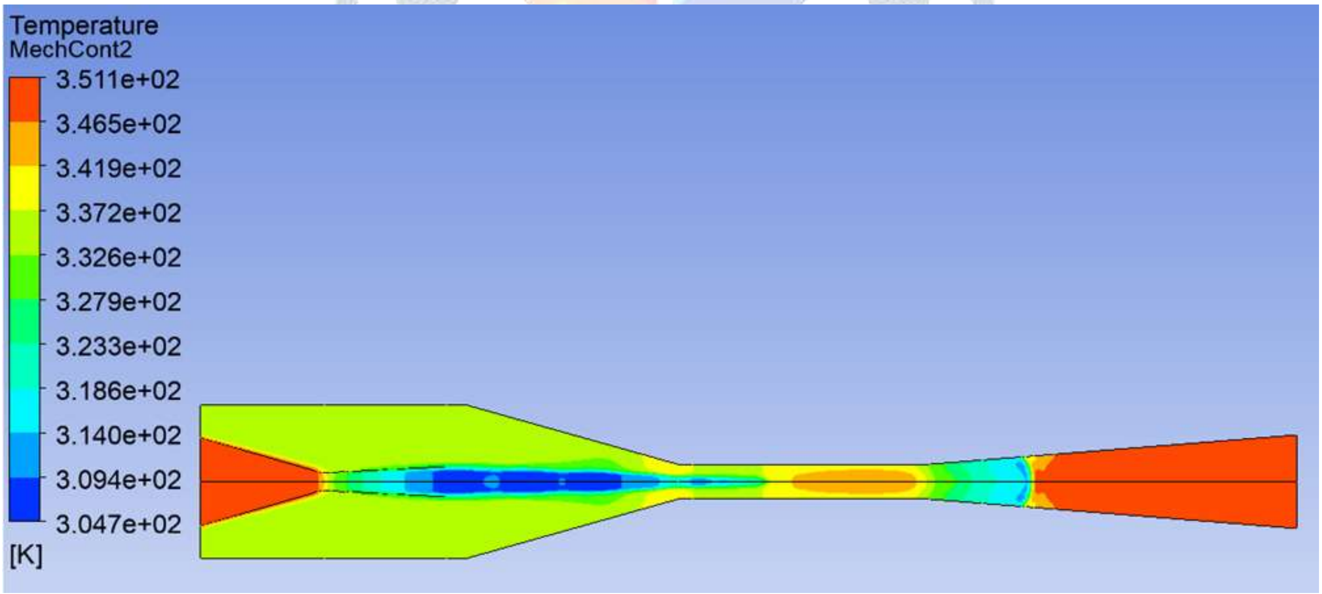
As seen from Figure, with increase in axial length of the Ejector, velocity of refrigerant R134a is increases in primary nozzle. When the length of exit position is 15.32, then the mean of velocity is minimum and at length of exit position 21.32 or 18.32 the variation is maximum in primary nozzle and throat.

3. Temperature

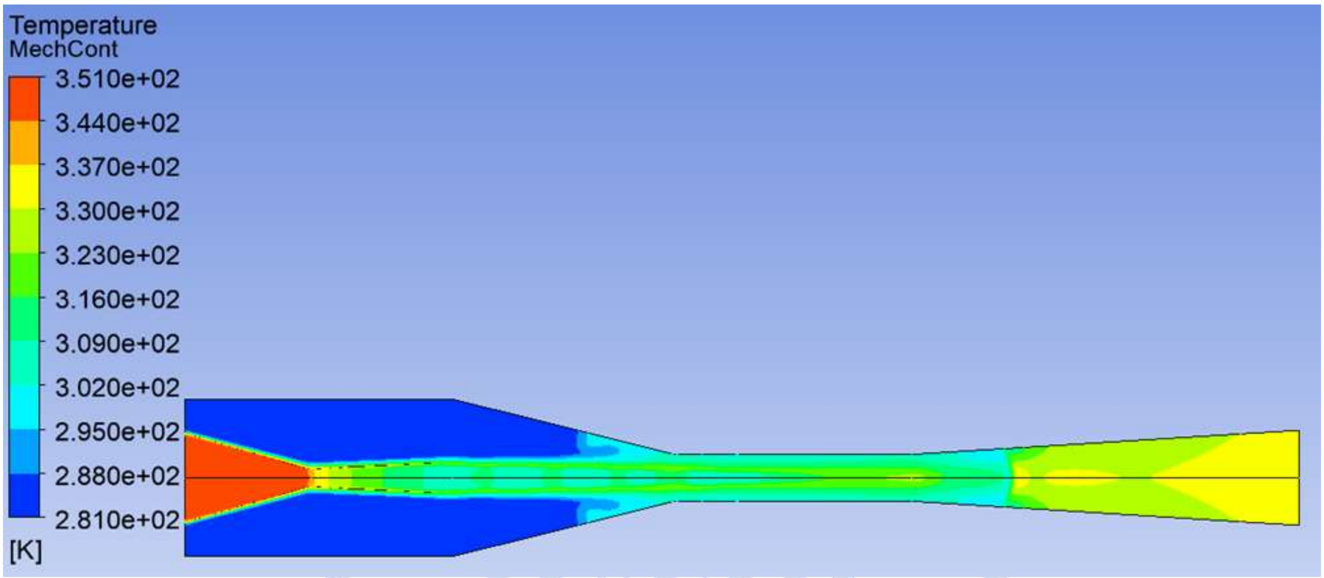
Throat radius is 4.49:



Throat radius is 2.49:

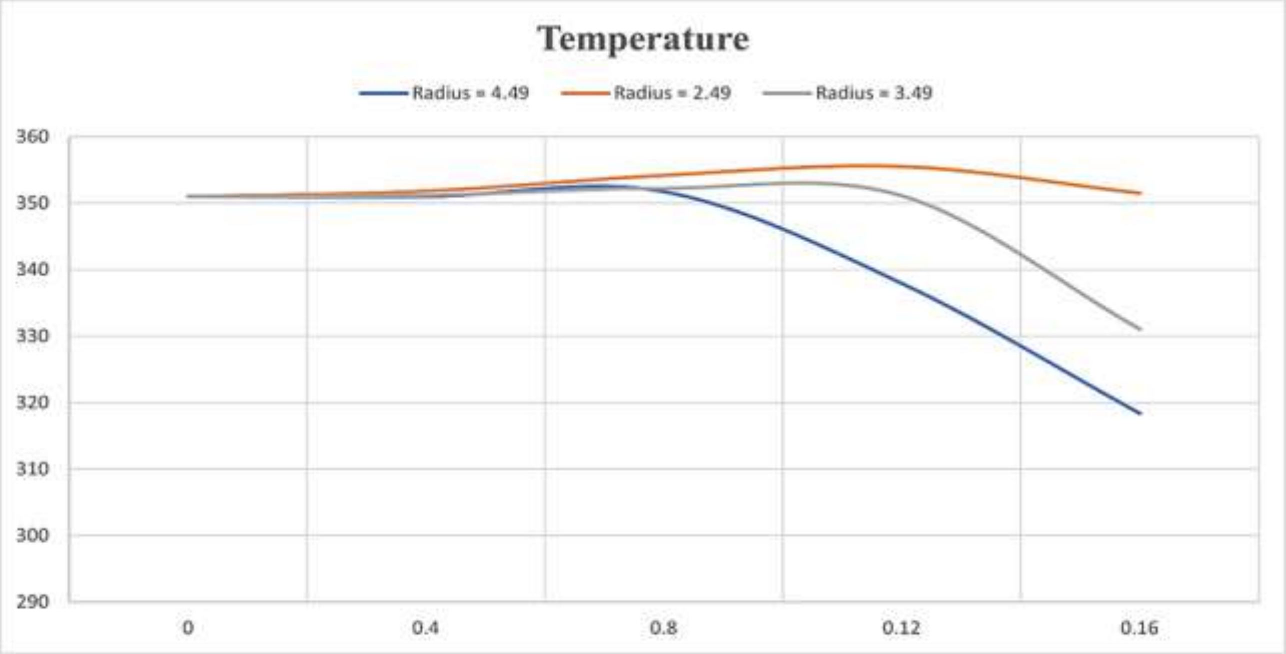


Throat radius is 3.49:

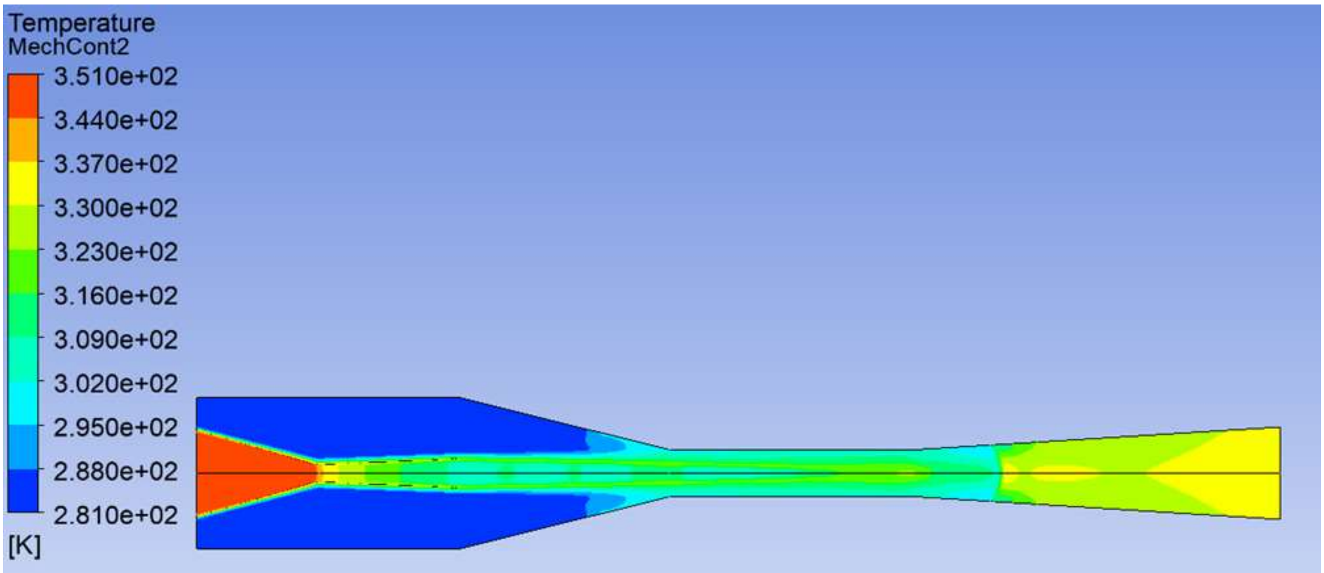


Graph of Temperature to compare change in throat radius:

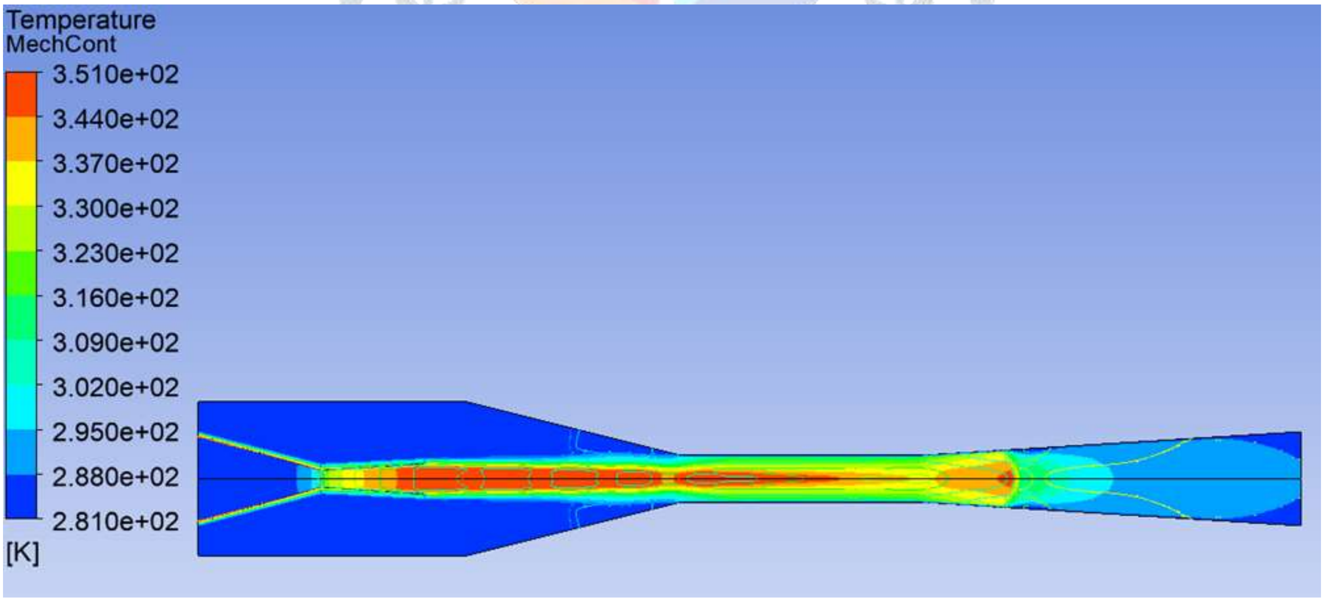
The Figure shows variation of temperature with increase in axial length of the Ejector. Now as seen from the graph, the temperature decreases with increases in Ejector Throat Radius however in the radius of 4.49 variation is more, so maximum value at between 0.8 or minimum value at 0.16. The least variation in the radius of 2.49 in which minimum value at 0.16 and maximum value at 0.12 in secondary nozzle.



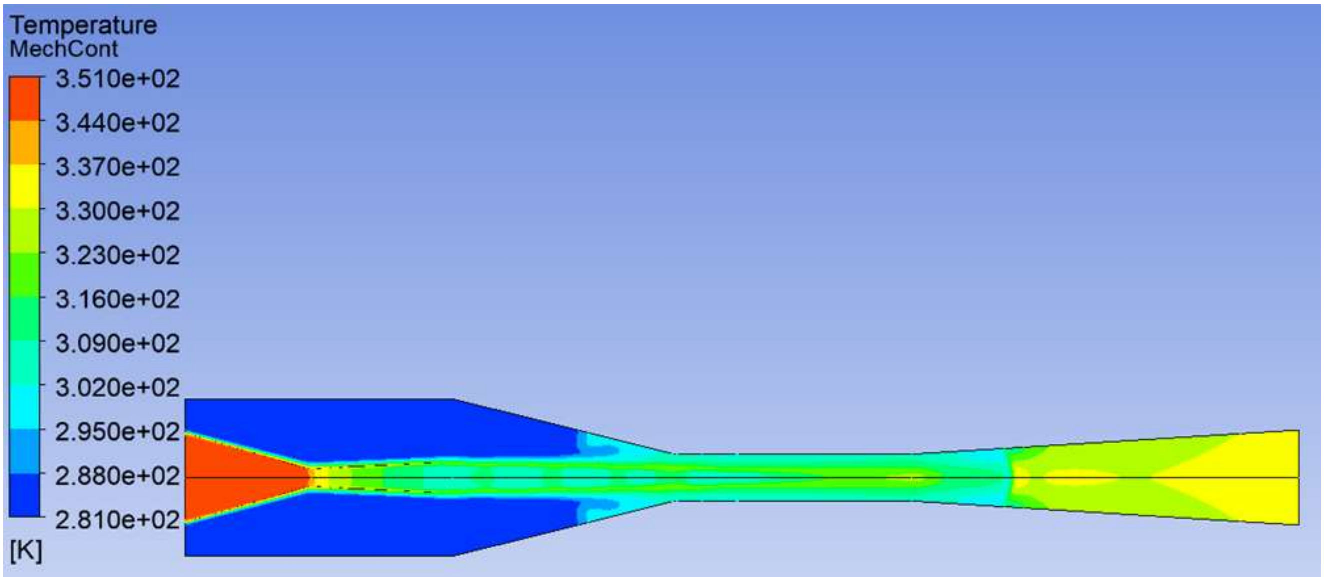
Nozzle exit position is 21.32:



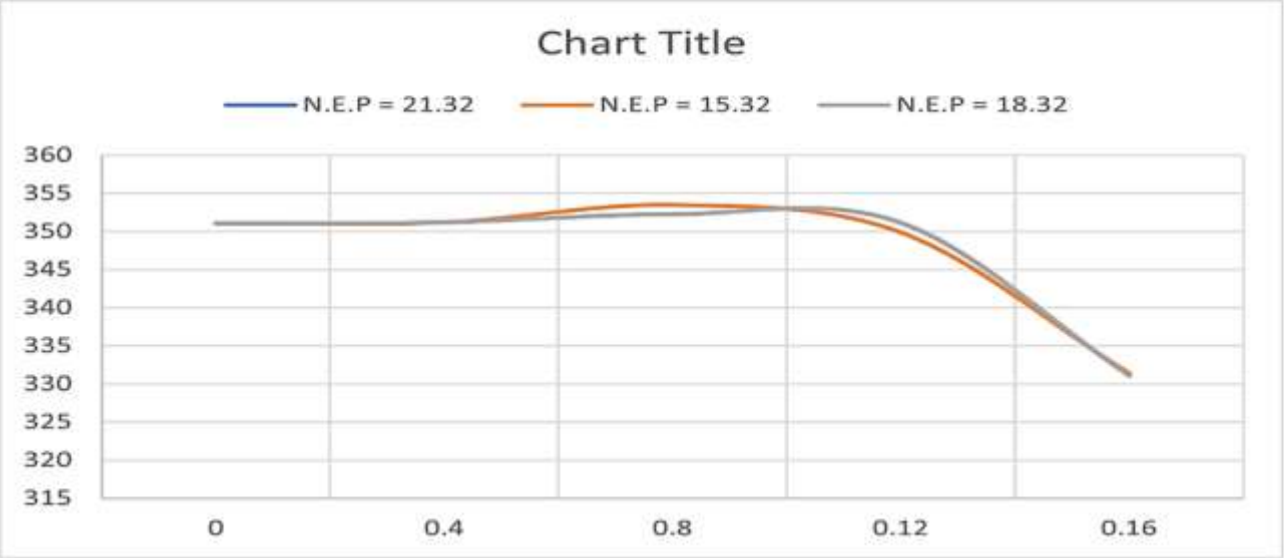
Nozzle exit position is 15.32:



Nozzle exit position is 18.32:



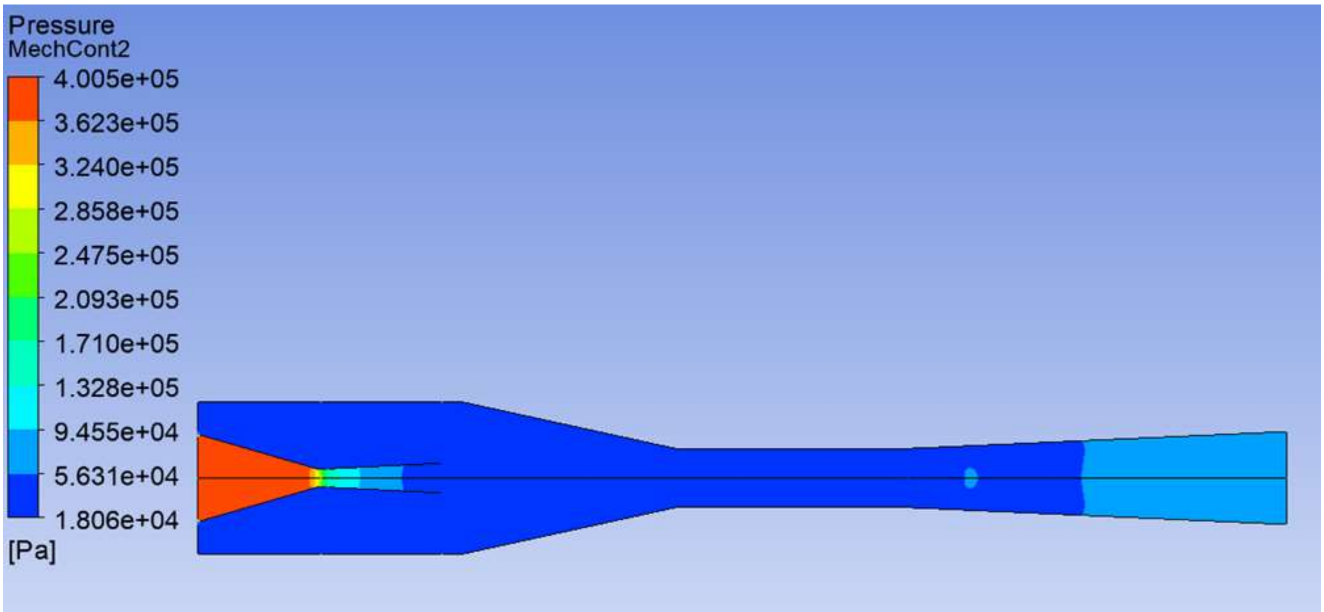
Graph of Temperature to compare change in length of nozzle exit position:



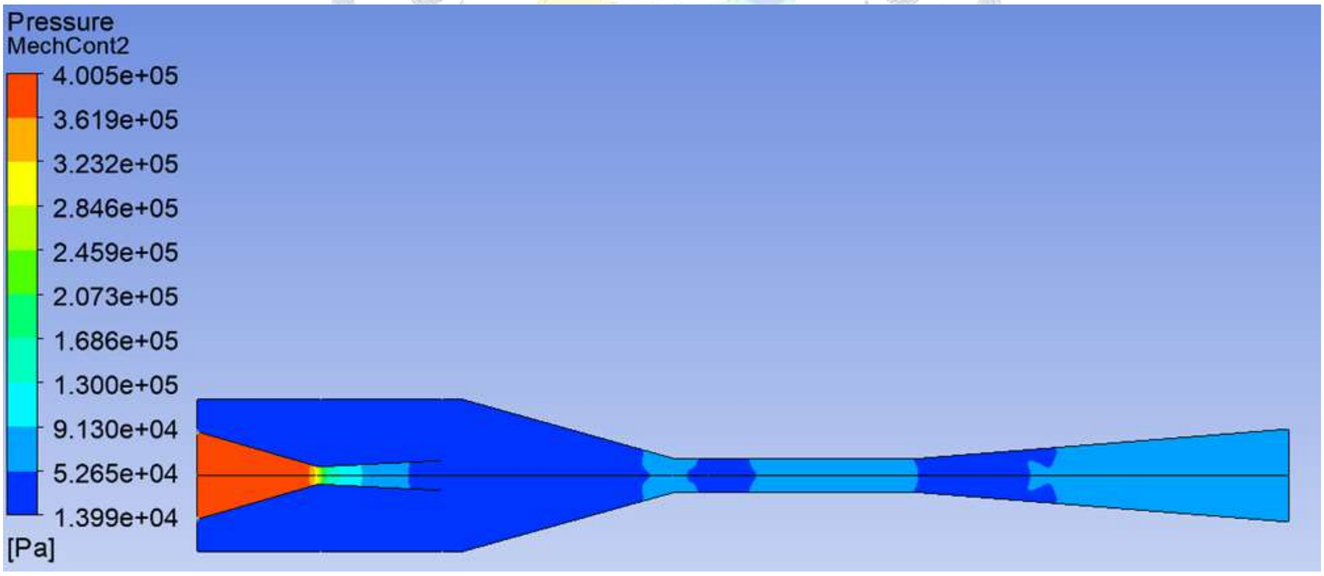
As seen from Figure, with increase in axial length of the Ejector, temperature of refrigerant R134a is increases in primary nozzle. When the length of exit position is 15.32, then the mean of velocity is maximum and at length of exit position 21.32 or 18.32 the variation is minimum in primary nozzle and throat.

4. Pressure

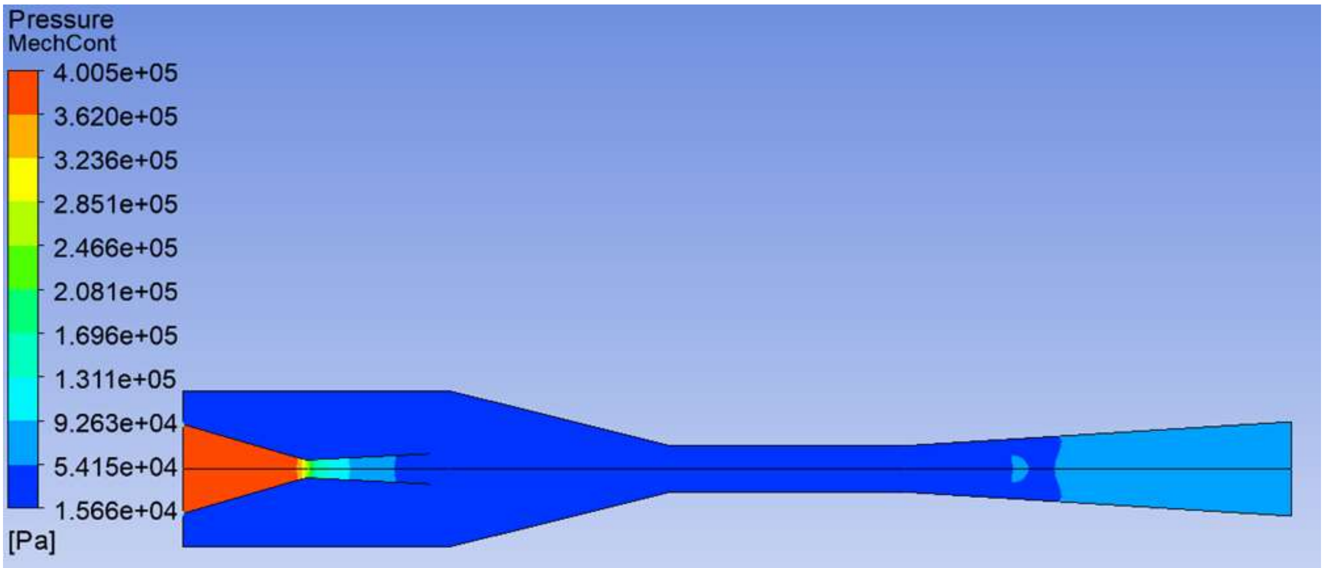
Throat radius is 4.49:



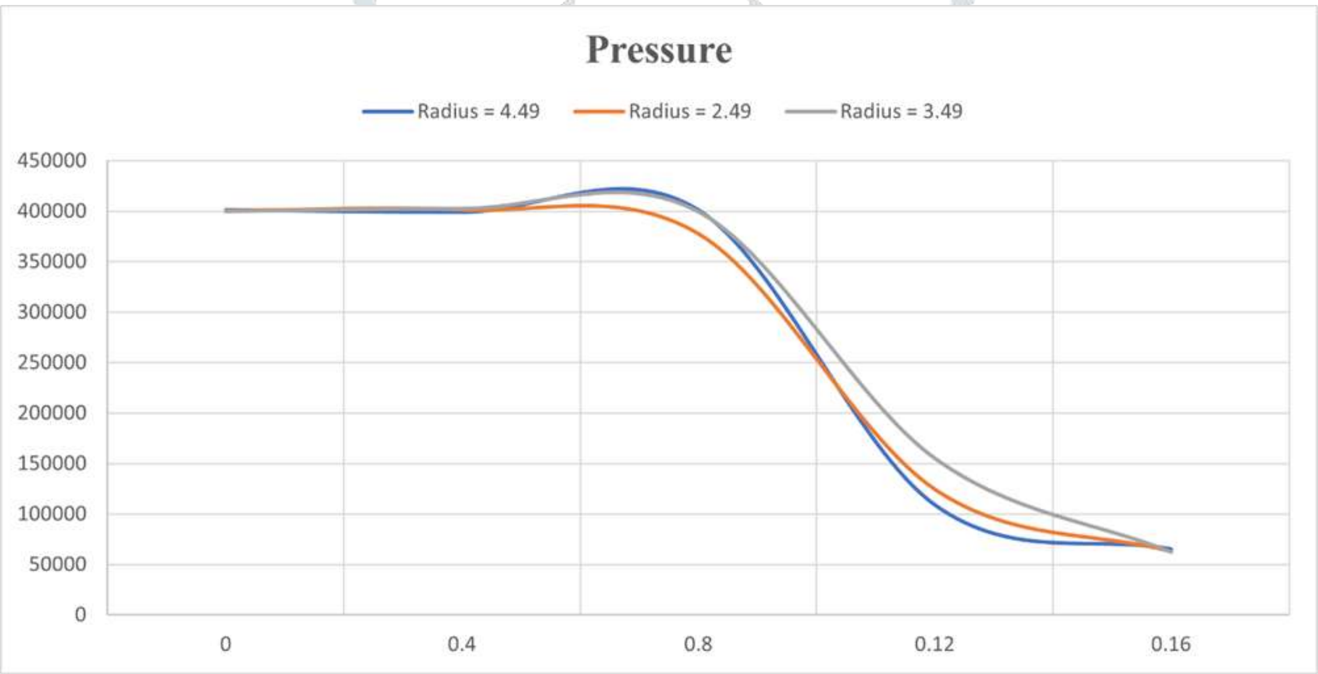
Throat radius is 2.49:



Throat radius is 3.49:

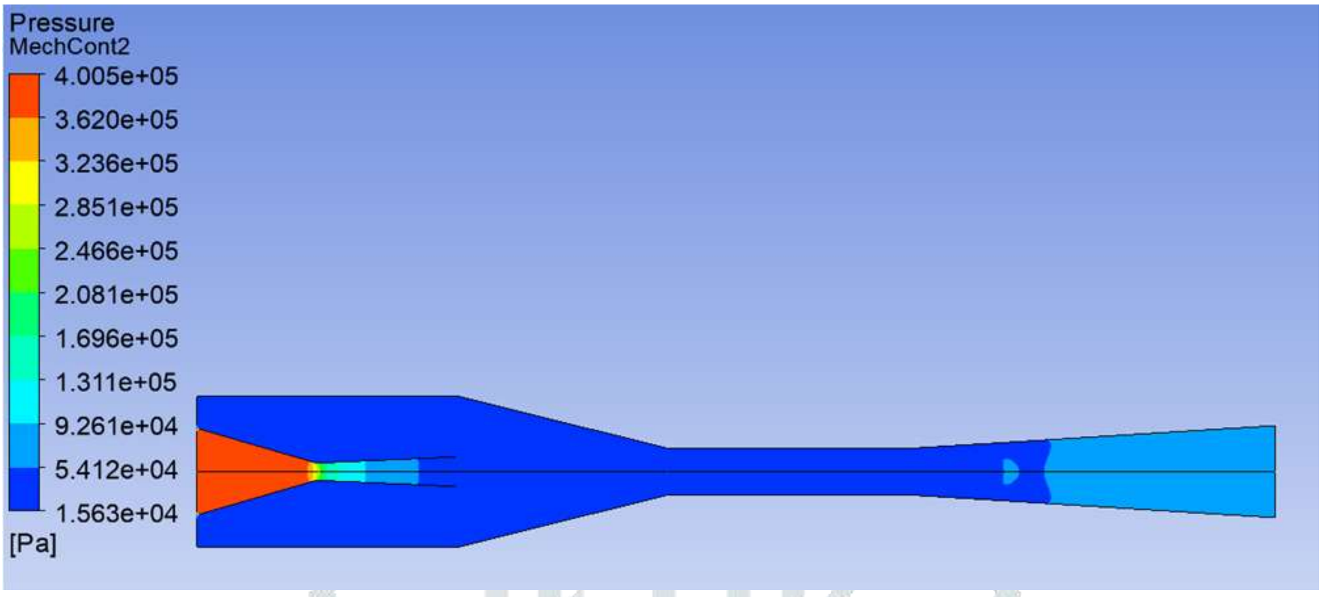


Graph of Pressure to compare change in throat radius:

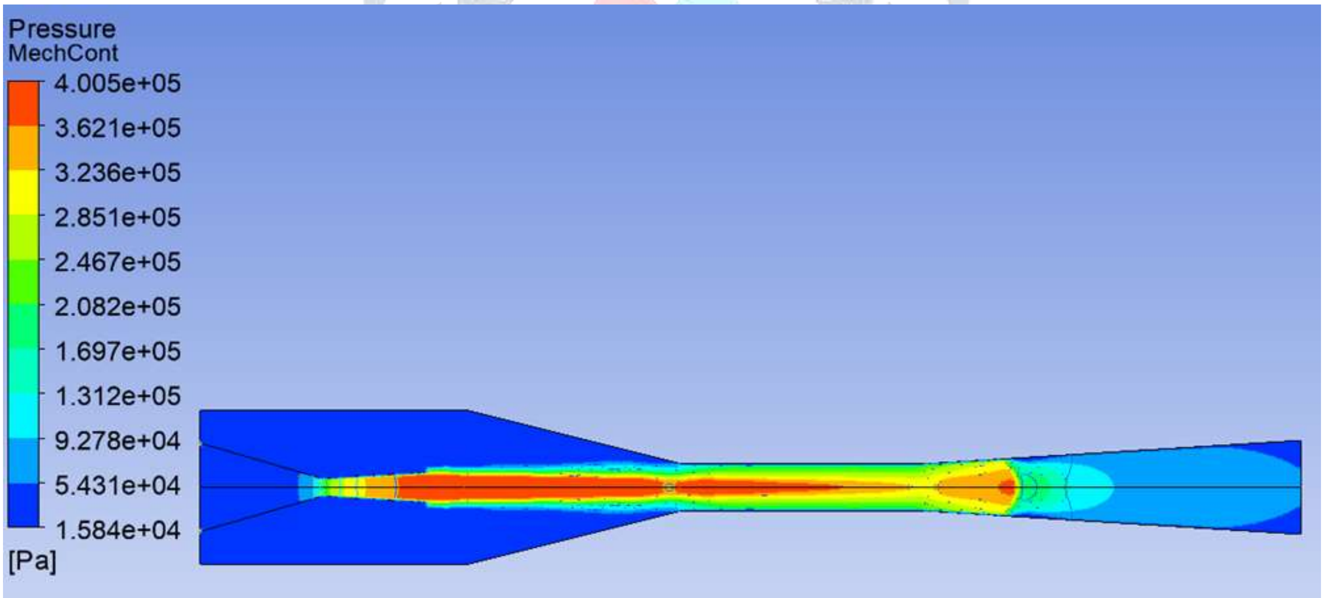


The Figure shows variation of pressure with increase in axial length of the Ejector. Now as seen from the graph, the pressure decreases with increases in Ejector Throat Radius however in the radius of 4.49 variation is more, so maximum value at 0.7 or minimum value at 0.13. The least variation in the radius of 2.49 in which minimum value at 0.16 and maximum value at 0.6 in secondary nozzle.

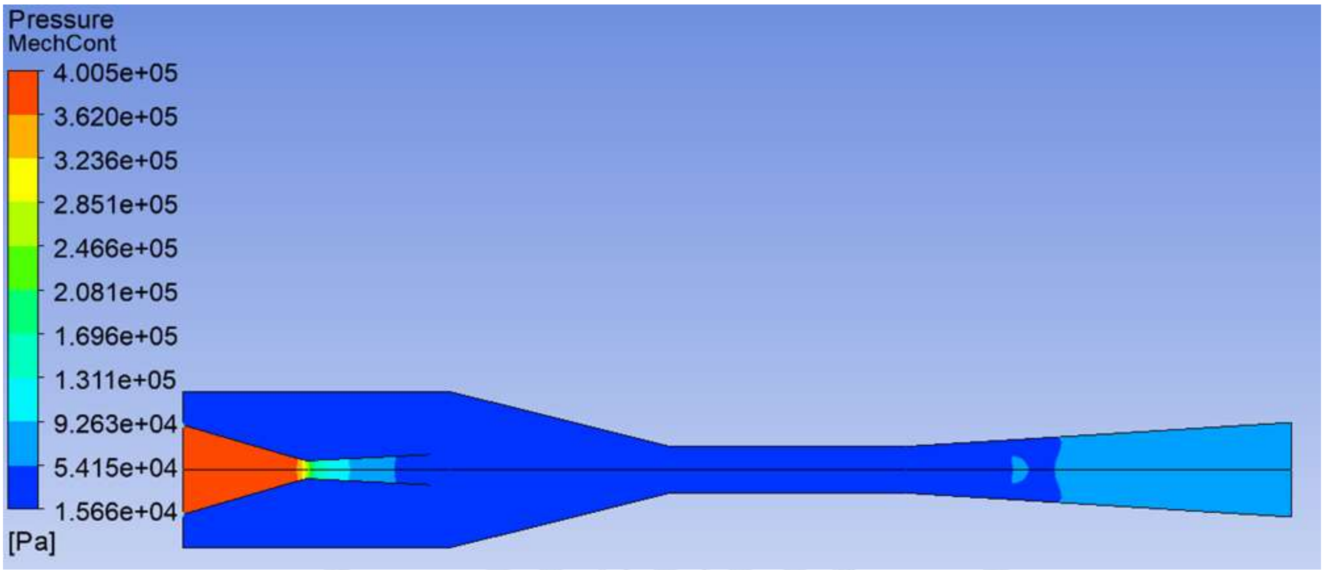
Nozzle exit position is 21.32:



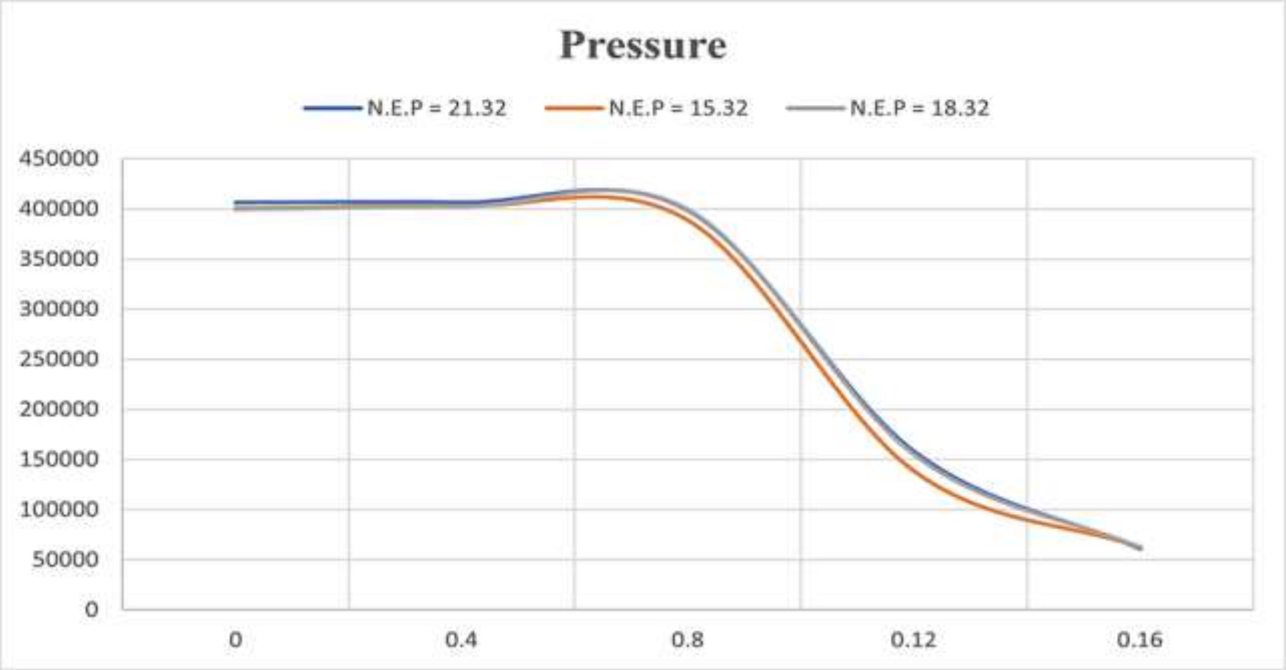
Nozzle exit position is 15.32:



Nozzle exit position is 18.32:



Graph of Pressure to compare change in length of nozzle exit position:



As seen from Figure, with increase in axial length of the Ejector, pressure of refrigerant R134a is increases in exit of primary nozzle. When the length of exit position is 15.32, then the mean of pressure is minimum and at length of exit position 21.32 or 18.32 the variation is maximum in primary nozzle and throat.

5. Future Scope

the present work focuses on effect of change in nozzle exit position and throtte radius on the performance of ejector using R134a refrigerant. The result show that both of these geometrical parameters affect the performance of ejector the study can be further extended in the following ways

1. Varying or changing the geometric parameter
2. Changing the operating parameter
3. Changing the mesh method

6. Conclusion

In the present study the CFD analysis of ejector using R134a as a working fluid has been presented the ejector has been modelled and analyzed using ANSYS 19.2 the results shows that there is change in performance of the ejector when geometric parameters like throat radius and nozzle exit position changes. it can be concluded that change in geometric parameter affects the performance of ejector.

Acknowledgment

The authors express their heartfelt gratitude and thankful to Dr. D V Singh, Associate Professor and Mr. Lokesh Aurangabadkar Assistant Professor, IIST Indore for guiding the project and encouraging to publish the paper.

References

Dr. D V Singh; Lokesh Aurangabadkar, Gajendra Singh Tomar, Adarsh Partap Singh Bais, Ankit Patidar, Devanand Patel, “CFD Analysis of Ejector Using Different Refrigerants: R141b, R152a, R135a” (2022) Journal of Emerging Technologies and Innovative Research (JETIR) Januray 2022, Volume 9, Issue 1

Scott, David; Aidoun, Zine; Bellache, Omar; and Ouzzane, Mohamed, "CFD Simulations of a Supersonic Ejector for Use in Refrigeration Applications" (2008). International Refrigeration and Air Conditioning Conference. Paper 927.

Honra, Jaime; S. Berana, Menandro; M. Danao,Louis Angelo; E. Manuel, Mark Christian; “CFD Analysis of Supersonic Ejector in Ejector Refrigeration System for Air Conditioning Application”(2017) Proceedings of the World Congress on Engineering 2017 Vol II.

Scott, David; Aidoun, Zine; Bellache, Omar; and Ouzzane, Mohamed,(July 14-17, 2008)"CFD Simulations of a Supersonic Ejector for Use in Refrigeration Applications". International Refrigeration and Air Conditioning Conference. Paper 927. http://docs.lib.purdue.edu/iracc/927

Kostic, O., Stefanovic, Z., & Kostic, I. (2015). CFD modeling of su-personic airflow generated by 2D nozzle with and without an obsta-cle at the exit section. FME Transaction, 43(2), 107–113. https://doi.org/10.5937/fmet1502107k.

Rao, A. V. R., & Akhil, I. (2016). CFD analysis of CD nozzle at two divergent angles for prediction of Mach number (M< 1, M=1 & M>1) and flow parameters of newtonian fluid by using ANSYS-FLUINT 14.0. International Journal of Engineering Sciences and Re-search Technology, 5(5), 413–428.

Su, L. (2015). CFD Simulation and Shape