

PAPER • OPEN ACCESS

## Effect of ejector design parameters on flow structure inside the mixing chamber

To cite this article: Ahmed Al-Manea and Thaer Al-Jadir 2021 *IOP Conf. Ser.: Earth Environ. Sci.* **779** 012033

View the [article online](#) for updates and enhancements.

### You may also like

- [Thermodynamic Analysis of Two-Phase Ejector as Expansion Device with Dual Evaporator Temperatures on Split Type Air Conditioning Systems](#)  
M E Arsana, I G B Wijaya Kusuma, M Sucipta et al.
- [Exergy analysis of heat assisted ejector refrigeration system using R1234yf](#)  
Bharat Sharma, Gulshan Sachdeva and Vikas Kumar
- [A Study of a Fluid Flow in the Jet Ejector System Used in Industrial Applications](#)  
I Olaru



The Electrochemical Society  
Advancing solid state & electrochemical science & technology

243rd Meeting with SOFC-XVIII

Boston, MA • May 28 – June 2, 2023

Early registration discounts end **April 24!**

**Accelerate scientific discovery!**

Learn More & Register



# Effect of ejector design parameters on flow structure inside the mixing chamber

Ahmed Al-Manea<sup>1</sup> and Thaer Al-Jadir<sup>2</sup>

<sup>1</sup>Al-Samawah Technical Institute, Al-Furat Al-Awsat Technical University, Iraq.

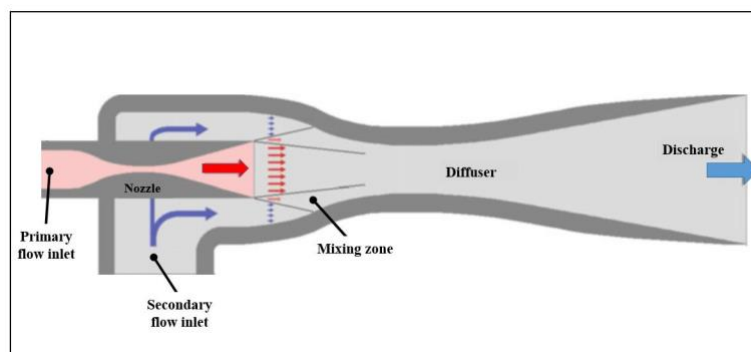
<sup>2</sup>Environment Research Center, University of Technology, Baghdad, Iraq.

E-mail: Dr.Ahmed.Almanea@atu.edu.iq

**Abstract.** Renewable energy is a sustainable source of energy that never ends. Ejectors are reliable devices as they rely on solar energy to run. Ejectors can be used as an alternative to compressors in refrigeration systems or provide vacuum zones in various applications. Ejectors have no moving parts, compact, and stable operation. However, they have lower efficiency at the current time. There may be opportunities to improve ejector efficiency under some conditions and develop a thorough understanding of the impact of their design parameters. This paper presents the numerical simulations CFD study to investigate the effect of various design parameters on the flow structure inside an ejector. This was achieved by modelling a variable ejector geometry with the ideal gas flow. This is a CFD study with new approaches. The numerical model has been validated by conducting different meshing levels. Three mesh size has been suggested, and the medium size was considered. The results showed that the secondary inlet with a diameter of 90 mm resulted in a reduction with the generated vortices compared to the Model I result. Also, the effect of changing the diffuser throat to 45 mm in Model II, has shown improvement in reducing vortices.

## 1. Introduction

Ejectors have no moving parts and are preferable to mechanical compressors in thermal refrigeration systems using low-grade heat sources, but ejectors typically have relatively low efficiency. An ejector is a pumping or a compression device in which a higher-pressure-flow (called the primary flow) is used to induce a lower pressure-flow (called the secondary ow) into a mixing section. These two streams mixes and discharge to a pressure that lies between these two fluids' initial pressures. The compression process's efficiency depends on the efficiency of the mixing between the two streams and the deceleration of the mixed streams. An ejector typically consists of five main sections: (i) the primary inlet; (ii) the secondary inlet and suction chamber; (iii) the primary nozzle, (iv) the mixing chamber; and (v) the subsonic diffuser section, as illustrated in Figure 1.



**Figure 1.** General configuration of an ejector system [1]



Efforts to understand the dominant factors which control the mixing-induced entrainment of the low-pressure stream inside ejectors are on-going research topic. The entrainment ratio (the ratio of the secondary mass flow rate  $m_s$  / primary mass flow rate  $m_g$ ) is mainly used as an indication of an ejector performance,

$$E = \frac{m_s}{m_g} \quad (1)$$

Studies in the literature were relied on changing the ejector geometries at constant operating conditions. Chang et al. in 2000 Conducted an experimental work to investigate the effect on changing the primary nozzle geometry using two different shapes of nozzles [2]. However, Rao in 2014 claimed that only 30 % of enhancement was obtained from using supersonic nozzle types [3]. Other studies were used different techniques to study the flow structure inside an ejector [4, 5, 6]. In this paper, the effect of changing two essential parameters: the secondary inlet diameter and diffuser throat have been carried out to understand their impact on the flow vortices generated inside the mixing chamber.

## 2. Computational Fluid Simulation

The commercial CFD package ANSYS FLUENT V 18.1, was used to solve the governing equations numerically. The flows were considered ideal gas (the thermodynamic properties of the primary and the secondary are based on the ideal gas assumption). A density-based, steady-state, 2D axisymmetric solvers and the realizable k- $\epsilon$  turbulent model was used, as this model has been used for the ideal gas simulations because it appears to be more accurate than k- $\omega$  SST in the case of homogeneous flows of ideal gases and it has demonstrated good results in some ejector studies [7].

A first-order upwind scheme was used for turbulent kinetic energy. The energy equation was enabled. The convergence criteria used for the residual analysis were set to be  $10^{-6}$  for continuity, X-velocity, Y-velocity and Energy. Implicit formulations were used considering the accuracy and stability. The wall was set to have a stationary, non-slip condition, and the energy equation was enabled.

The turbulence model adopted in all of the CFD simulations is the realizable k- $\epsilon$  turbulence model because it has more accurate to describe the spreading rate of both planar and round jets [8], and its capability to solve the flow inside ejector [9]. Therefore, this model was adopted to investigate the mixing and jet development downstream of the primary nozzle exit. The realizable k- $\epsilon$  turbulence model is based on model transport equations for the turbulence kinetic energy  $k$ , and its dissipation rate  $\epsilon$  and are described by [10]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\psi_k} \right) \frac{\partial k}{\partial x_j} \right] + Src_k \quad (2)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\psi_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + Src_\epsilon \quad (3)$$

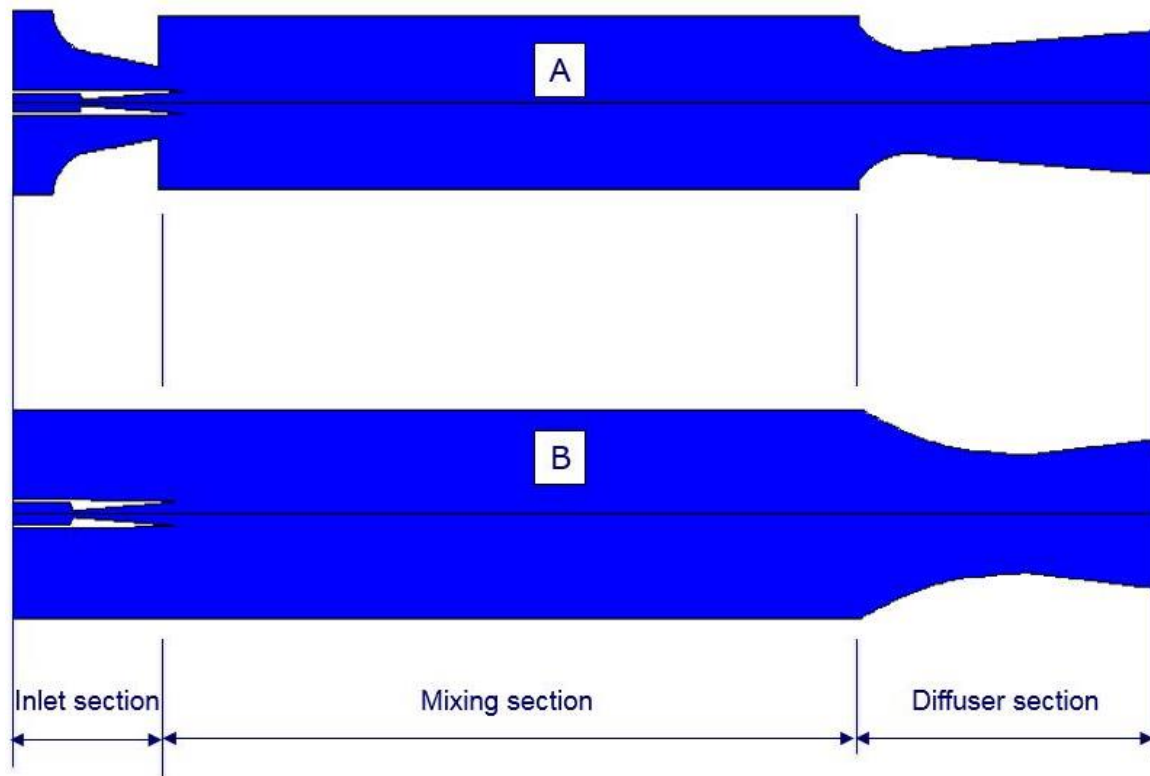
Where  $Src$  is the source term,  $\psi$  is the turbulent Prandtl number and  $\mu_t$  is the turbulent viscosity.

## 3. Simulated mixing chamber geometry

In order to study vortices development within ejector, two geometries models were simulated:

- (1) The first model was based on [11] which was deficient in that large zone of recirculation developed in the mixing chamber;
- (2) The second model was developed during the current study to minimise the impact of recirculation zones ejector entrainment ratio. Both of the designs are shown in Figure 2. The key dimensions of each configuration are shown in Table 1.

The design of the two models is similar to an ejector, with some exceptions. The mixing chamber (the zone after the primary nozzle) has been extended to increase the cross-section area in which the vortices can develop.



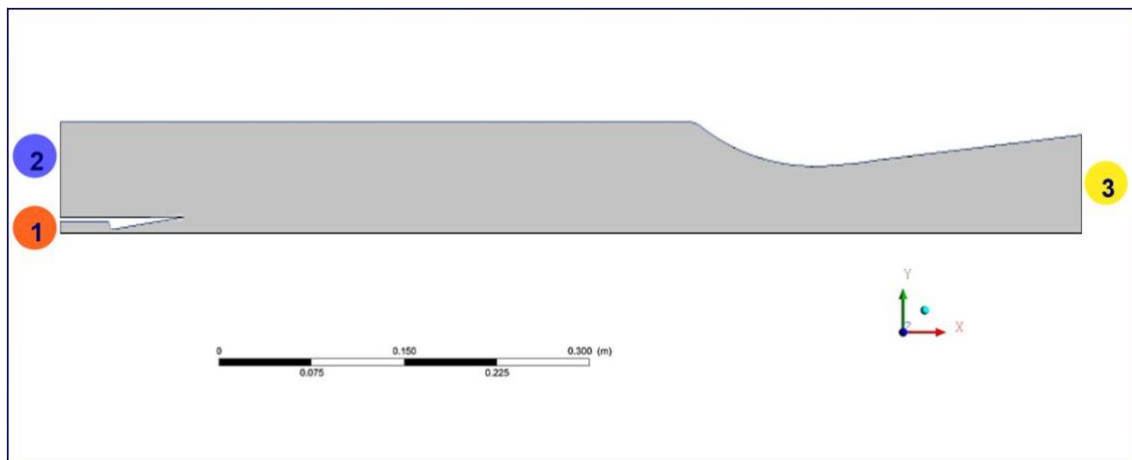
**Figure 2.** Mixing chamber configurations: (a) Model I; (b) Model II.

**Table 1.** Geometry design dimensions.

Geometric parameters	(a) Model I in mm	(b) Model II in mm
Diameter of the secondary inlet	37	90
Nozzle Exit Position (NXP)	+15	+15
Mixing chamber length	420	420
Mixing chamber diameter	90	90
Diffuser inlet diameter	80	90
Diffuser throat diameter	22	28
Diffuser outlet diameter	80	94
Diffuser convergent length	20	153
Diffuser divergent length	270	169

#### 4. Simulated boundary conditions and gas models

Boundary conditions were applied for the two inlets as shown in Figure 3: the primary steam inlet, labelled as 1, and the secondary inlet, labelled as 2. The outlet flow from the diffuser also specified boundary conditions, labelled as 3 in Figure 3. In the current study, the simulations were carried out using two different models. The boundary conditions are presented in Table 2. A stationary adiabatic wall was applied for the nozzle and the mixing chamber walls.



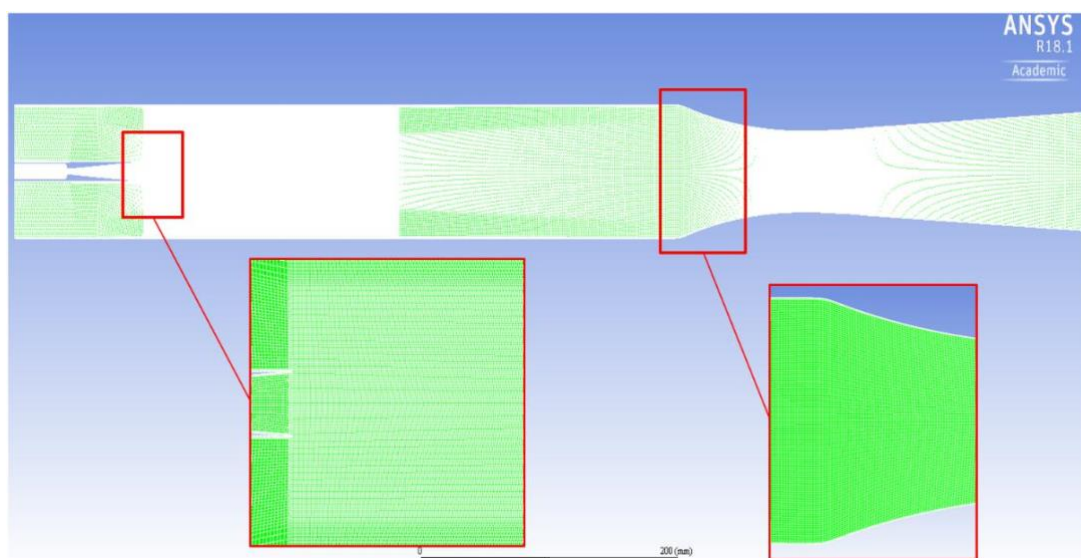
**Figure 3.** Identification of the three flow boundary surfaces for the axisymmetric ejector CFD model.

**Table 2.** Nominated operation conditions for models simulations.

Stream	Pressure (kPa)	Temperature ( $^{\circ}\text{C}$ )	Mass flow (g/s)
Primary stream	150	120	1.8
Secondary stream	3.4	34	5.5-6.5
Condenser	2-4	-	-

## 5. Mesh independence

The meshing was based on structured quadrilateral cells. As shown in Figure 4, refinement was applied to the area from the primary nozzle exit to a distance of 300 mm, which covered the full axial length over which vortices development existed. The mesh independence study used three grid levels: the coarse mesh with 80,000 cells, the medium mesh with 115,000 cells, and the fine mesh with 300,000 cells.

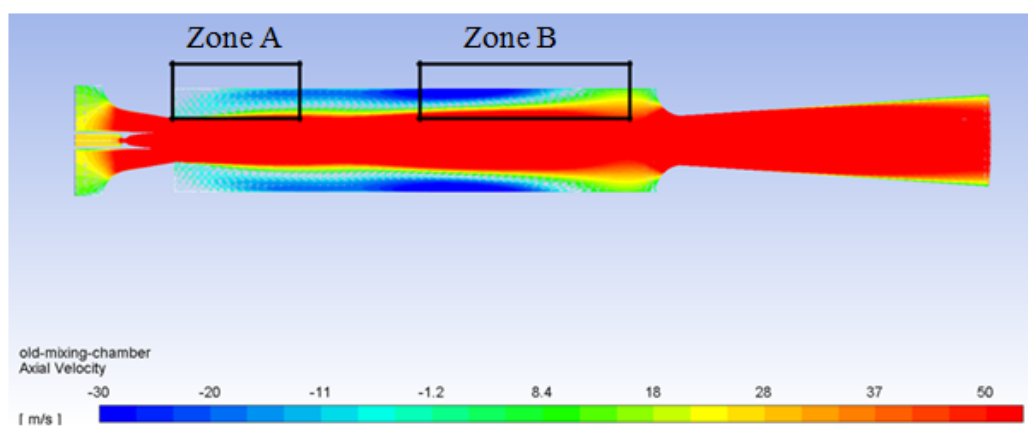


**Figure 4.** Illustration of the medium mesh density used for co-flowing steam jet simulations.

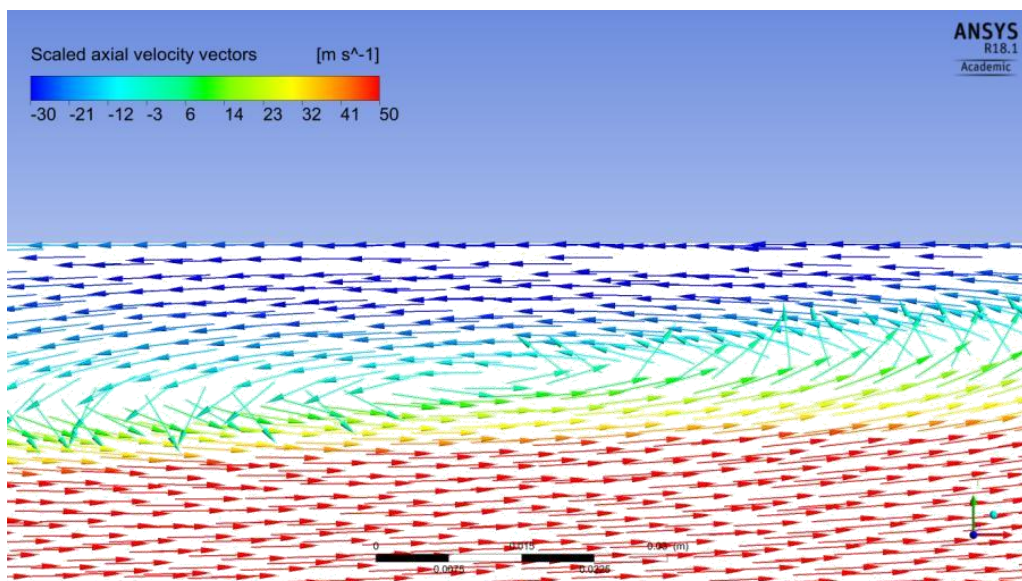
The medium and the fine mesh results are in good agreement with the maximum difference in Mach number typically being less than 2 %. As a result, the medium mesh level was considered adequate for the simulations.

## 6. Results and discussion

Significant recirculation zones were simulated inside the mixing chamber of Model I as shown in Figures 5, 6 and 7. These features caused some of the primary flow streams to reverse toward the secondary inlet. This is potentially a problem because the amount of the mass flow rate coming from the secondary stream will be minimized, affecting the ejector entrainment ratio. For the current study to overcome this problem and improve the jet characteristics, these vortex features must be significantly reduced in size.

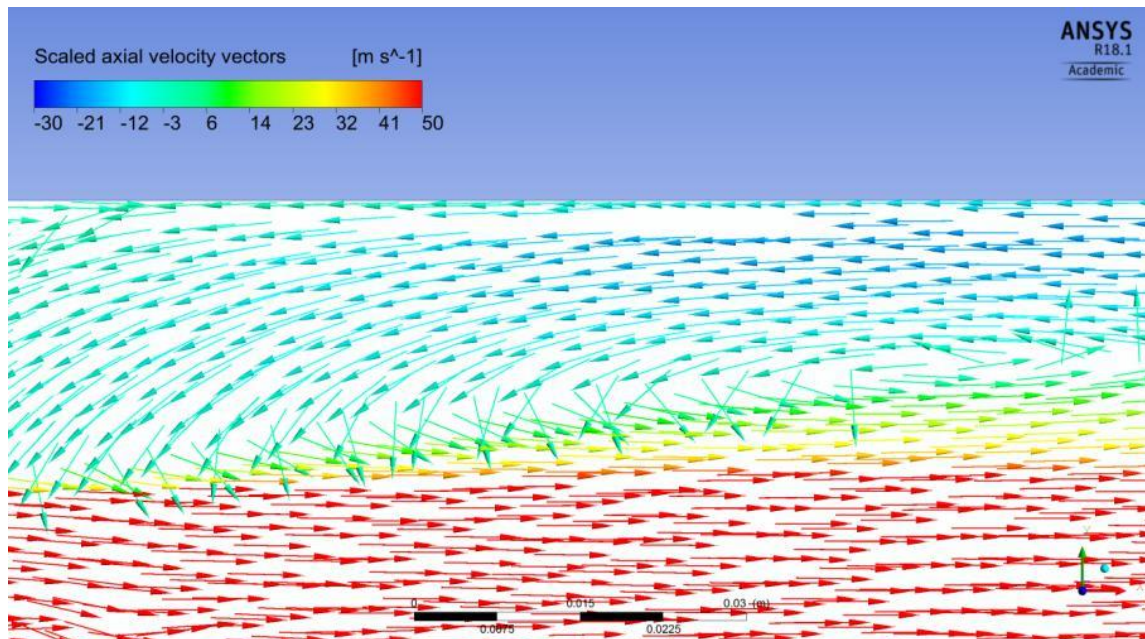


**Figure 5.** Axial velocity vector from CFD simulation of a Model I of mixing chamber at primary stream conditions of 150 kPa, 120 °C with a secondary at 3.4 kPa and condenser pressure of 3 kPa. The velocity scale was adjusted to the given range to emphasise the recirculation zones.



**Figure 6.** Flow recirculation at the secondary stream entrance zone A, for the Model I of mixing chamber and location indicated in Figure 5.

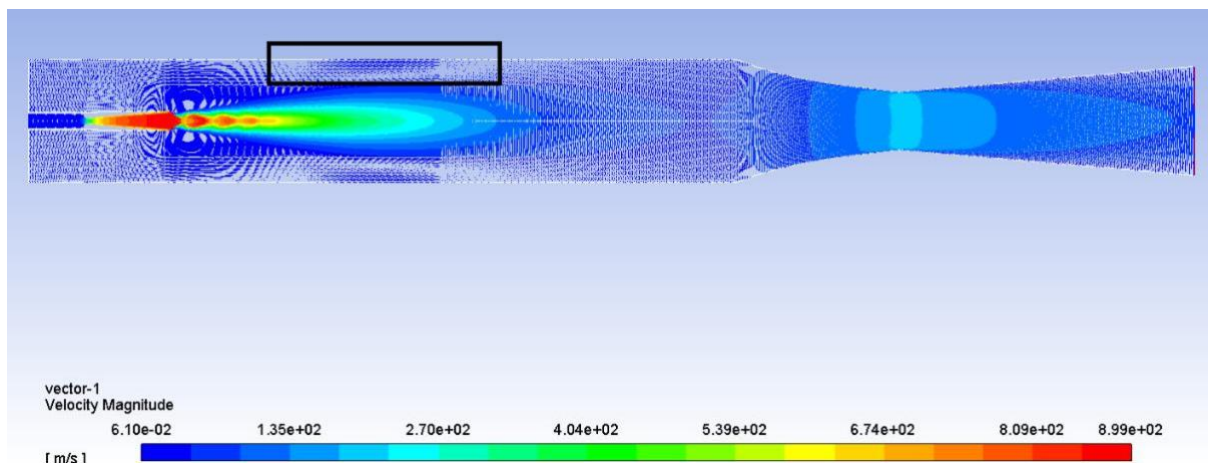




**Figure 7.** Downstream edge of recirculation region zone B, for the Model I of mixing chamber and location indicated in Figure 5.

#### 6.1. Effect of secondary inlet diameter on flow recirculation

In order to design a new configuration with reduced recirculation of the primary jet flow, CFD simulations were performed to study the effect of the inlet area of the secondary stream, and the diffuser throat area. The simulations were performed at the same operating conditions indicated in Figure 5. A larger secondary stream inlet area with the same specified boundary conditions (Table 1) produces a larger secondary mass flow rate. As illustrated through a comparison of Figure 8, the increase of inlet diameter from 37 mm to 45 mm caused a reduction in the mixing chamber's recirculation zone.

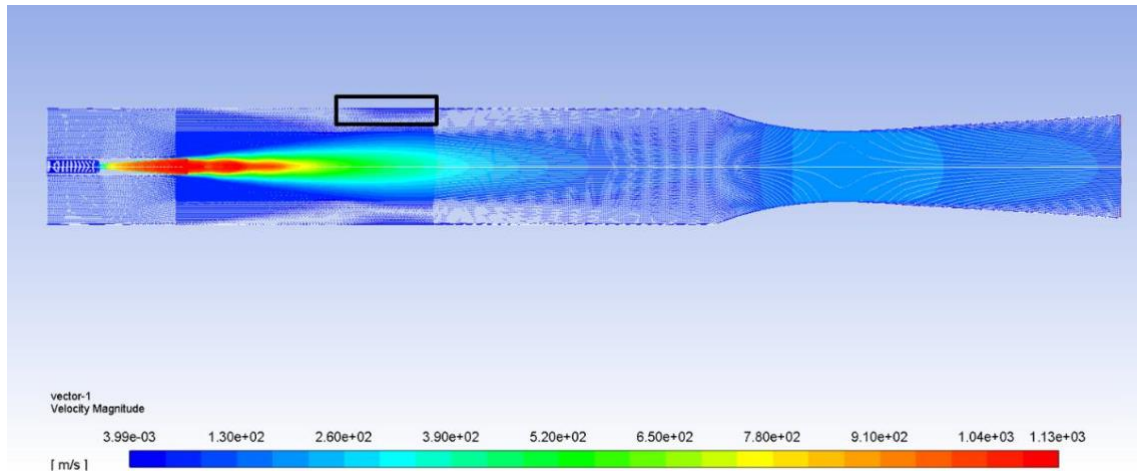


**Figure 8.** Flow recirculation for the new mixing chamber with an inlet diameter of 90 mm, for diffuser throat diameter of 42 mm.

#### 6.2. Effect of diffuser diameter on flow recirculation

Results in Figure 8 and Figure 9 show that increasing the diffuser throat diameter from 42 to 56 mm leads to more of the mixture mass being directly discharged into the condenser, and further reduces the

recirculation inside the mixing chamber. However, there is a limitation of increasing the diffuser throat diameter behind 56 mm as it loses its function to choke the flow at the diffuser's throat.



**Figure 9.** Flow recirculation for the new mixing chamber with an inlet diameter of 90 mm, for diffuser throat diameter of 56 mm.

## 7. Conclusion

This paper has shown the results of the computational fluid dynamic simulations using ANSYS FLUENT 18.1. Ideal gas simulations were used to study the vortex effect inside the mixing chamber and design a new diffuser and secondary inlet configuration to remove these recirculation effects. The simulation results indicate that the ejector's entrainment ratio could be enhanced if more mass from the secondary stream is entrained into the mixing chamber. Increasing the diffuser diameter from 42 mm to 56 mm has shown promising results to reduce these vortices.

## References

- [1] Grazzini G, Milazzo A and Mazzelli F 2018 *Ejectors for Efficient Refrigeration: Design, Applications and Computational Fluid Dynamics*, Springer.
- [2] Chang YJ and Chen YM 2000 Enhancement of a steam-jet refrigerator using a novel application of the petal nozzle, *Experimental Thermal and Fluid Science* **22** (3-4):203-211.
- [3] Rao SM and Jagadeesh G 2014. Novel supersonic nozzles for mixing enhancement in supersonic ejectors, *Applied Thermal Engineering*, **71**(1):62-71.
- [4] Gagan J, Smierciw K, Butrymowicz D and Karwacki J, 2014 Comparative study of turbulence models in application to gas ejectors, *International Journal of Thermal Sciences* **78** (2014) 9-15.
- [5] Al-Manea A, Buttsworth D, Leis J, Choudhury R and Saleh K 2018, Measurement of water vapour in axisymmetric jet development using tdlas, *Measurement* **10** (2018) 13.
- [6] Al-Manea A, Buttsworth D, Leis J and Saleh K 2019 Tunable diode laser absorption spectroscopy in a supersonic steam jet, 2019.
- [7] Grazzini G, Milazzo A and Mazzelli F, 2018 *Ejectors for Efficient Refrigeration: Design, Applications and Computational Fluid Dynamics*, Springer, 2018.
- [8] Bakker A 2002 Lecture 10-turbulence models applied computational fluid dynamics, Power-Point presentation.
- [9] Mazzelli F, Little A, Garimella S and Bartosiewicz Y 2016, Condensation in supersonic steam ejectors: comparison of theoretical and numerical models, in: *International Conference on Multiphase Flow, ICMF*, May 22nd - 27th, Firenze, Italy, 2016.
- [10] Ariaifar K, 2018 Simulation and measurement of condensation and mixing effects in steam ejectors, Ph.D. thesis, University of Southern Queensland (2016).