Nawed Aqbal Khan, 2024, 12:3
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

International Journal of Science, Engineering and Technology

An Open Access Journal

CFD Analysis of Ejector Use in Refrigeration System

Scholar Nawed Aqbal Khan, Professor Dr. Ajay Singh, Nitin Barodia
Department of Mechanical Engineering, RITS Bhopal

Abstract- The refrigeration system plays an important role in various applications from households to industries. An ejector refrigeration system is one of the thermal driven refrigeration systems. In this study, the performance and optimization of ejectors predicted by the RANS-based CFD model were examined. The effects of primary nozzle throat diameter and primary nozzle discharge diameter were investigated and analyzed to better understand the phenomena inside the ejector. Findings indicate that a critical condenser pressure of 236585 Pa was observed at optimum level of primary nozzle throat diameter of 1.5 and primary nozzle discharge diameter of 9.6. The application of RSM method improves the effectiveness of ejector. Best result is achieved from applying of RSM method.

Keywords- CFD analysis, ejector use, refrigeration system

I. INTRODUCTION

The simplest form of a refrigeration cycle (the Carnot cycle) consists of a refrigerant cycling through four components. First, the refrigerant flows through an evaporator where it absorbs energy and changes phase from liquid to vapour. Second, a compressor is used to increase the pressure of the refrigerant.

Third, a condenser is used to reject heat from the refrigerant. Fourth, an expansion device is used to decrease the pressure of the refrigerant to the pressure desired in the evaporator where the cycle begins again. In most modern refrigeration systems, a mechanical compressor is used as the second component of the Carnot cycle. Supersonic ejectors can provide an alternative method of providing the required compression.

Where conventional vapour compression systems generally use electrically powered mechanical compressors, thermal energy from existing processes can be recovered to drive the ejector. A schematic illustration of an ejector refrigeration system is shown in Fig. 1.1.

As illustrated in Fig. 1.1, ejectors generally consist of a primary fluid that flows through a converging-diverging nozzle, exhausts into a chamber in which there are secondary inlets and a single outlet. In regular operation, the primary fluid is vapourized in the generator and accelerated to supersonic velocities in the converging-diverging nozzle. The supersonic primary flow exits the nozzle at low pressure and entrains a secondary flow of vapour from the evaporator.

The two streams mix, and a series of shock waves result that reduce the flow to sub-sonic velocities. The combination of the shock waves and of the sub-sonic flow through the diffuser increase the pressure of the flow to that found at the condenser.

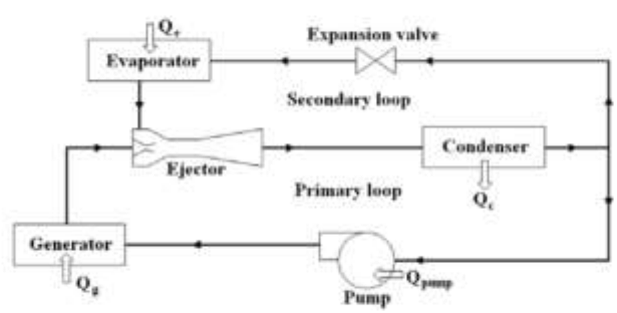

Figure 1: A basic ejector refrigeration cycle

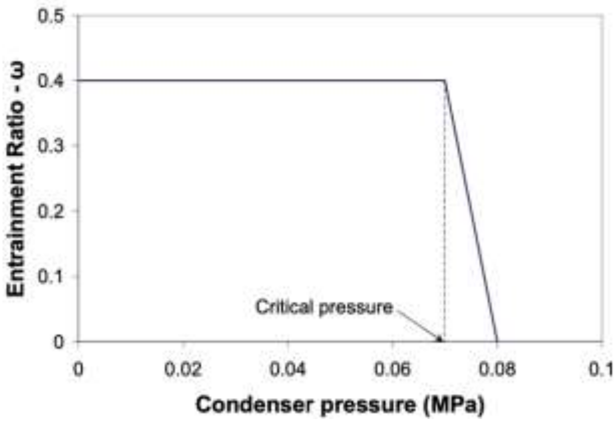

Figure 2: Ejector characteristics

History of the Ejector

Of the multiple components that make up the entire Ejector Refrigeration System (ERS), the ejector is the heart of the system and the ERS efficiency is tightly coupled with the efficiency of the ejector itself. The ejector was first developed in 1901 [1] for removing air from a steam engine's condenser. Just nine years later, the development of the first ERS is attributed to Maurice Leblanc [1]. From there, the ERS was popular for conventional heating purposes, mainly in air conditioning large buildings [1]. When the vapor-compression refrigeration cycle hit the market, it dominated up to present day, sending the ERC concept on the backburner of the scientific research due to the low COP. As environmental and energy issues become a larger concern, the research focus has begun to realign with the ERC.

Conceptual Ejector Operation

Figure 2.1 illustrates what a typical ejector might look like, however many variations on geometry throughout research are evident. Referring to this figure, the high-pressure primary fluid (PF) expands and accelerates through the primary nozzle to fan out with supersonic speed and create a lower pressure region than what is present in the low temperature evaporator. The lower pressure region and the high-speed primary fluid entrain a secondary fluid (SF) through a duct connecting to the low temperature evaporator. The two flow streams come into contact; however, do not actually begin the mixing process until some particular length along the length of the ejector. The SF further accelerates through the convergent area formed by the fanned out conical jet stream of the PF. As the local velocity of the two flow streams converge, the mixing process takes place and it is assumed to be completed by the end of the constant-area mixing chamber. The mixing process is assumed to take place at some cross-section known as the effective area [1].

At the end of the mixing chamber, where the mixing is assumed to have been completed, a shock wave of infinitesimal thickness is typically present. This is a direct result of the higher backpressure of the diffuser exit, which is the condenser pressure. The shock wave causes a compression effect and the flow stream after the shock wave is of uniform temperature and pressure. The flow stream is then further compressed as velocity decreases in the subsonic diffuser, and some pressure is recovered [1].

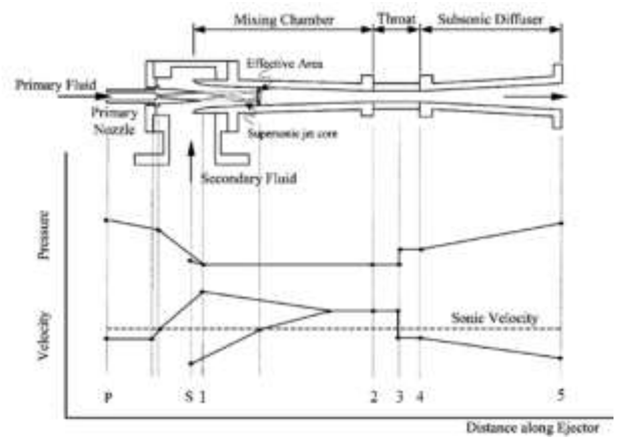

Figure 3: Typical ejector geometry, pressure and velocity profiles along ejector length [1]

In order to describe the ERS performance, the entrainment and compression ratio as defined by,

$$\omega = \frac{refrigerant\ mass\ flow\ rate}{primary\ fluid\ mass\ flow\ rate} \quad (1)$$

$$CR = \frac{condenser\ saturation\ pressure}{static\ pressure.secondary\ flow\ inlet} \quad (2)$$

respectively, are employed. The entrainment ratio is directly coupled to the energy conversion efficiency of the ejector, while the compression ratio directly limits the maximum saturation temperature at which the condenser can reject heat to the environment. These two metrics can be directly used to evaluate a given ejector design.

II. METHODOLOGY ADOPTED

Performing a computational fluid dynamics (CFD) analysis of an ejector typically involves the following methodology:

1. Geometry Creation
The first step is to create a 2D geometric model of the ejector. This can be done using a CAD software.

2. Meshing
The next step is to create a mesh of the geometry, which divides the volume into small cells or elements. The quality of the mesh can significantly impact the accuracy and stability of the simulation, so it's important to use appropriate meshing techniques.

3. Boundary Conditions
Boundary conditions define the behavior of the fluid at the inlet, outlet, and other surfaces of the ejector. For example, the inlet may be specified as a uniform flow with a given velocity or mass flow rate, while the outlet may be specified as a pressure boundary.

4. Solver Setup
The CFD solver software is used to solve the equations that govern the fluid flow in the pump. The solver setup involves selecting appropriate numerical schemes, defining convergence criteria, and specifying other parameters.

5. Running the Simulation
Once the solver is set up, the simulation can be run. During the simulation, the solver iteratively solves the governing equations until the solution converges to a steady state or time-dependent solution.

6. Post-Processing
After the simulation is complete, the results can be post-processed to extract the desired quantities such as velocity, pressure, and turbulence. Visualization tools can be used to analyze and interpret the results.

7. Validation
Finally, it's important to validate the simulation results by comparing them with experimental data or other benchmark solutions. This helps to ensure the accuracy and reliability of the CFD analysis.

RSM optimization and finding of optimum results.

CFD Modelling
As shown in Figure 3.1, a steam ejector refrigeration system consists of six components: boiler, steam ejector, condenser, evaporator, expansion valve, and pump. Low-grade heat is delivered to the boiler for vaporization of the water. The high-pressure vapor out from the boiler enters into the ejector nozzle and draws low-pressure vapor from the evaporator. The two flows undergo mixing and pressure recovery inside the ejector. The mixed flow is then fed into the condenser, where condensation takes place by rejecting heat to the heat sink. The liquid from the condenser is divided into two parts. One goes through the expansion valve to the evaporator, where it evaporates and produces a cooling effect. The remaining liquid is pumped back to the boiler by the pump, and completes the cycle.

Boundary Condition
On all the walls (including the roughened one) of the rectangular duct, no-slip boundary conditions were assigned. Constant heat flux of 1000 W/m2 was decided to be the boundary condition at the upper wall of the absorber plate.

Fig. 4: Boundary condition for CFD analysis

At the inlet, uniform velocity with an inlet temperature of 300 K and at the exit, invariable pressure (atmospheric pressure) boundary conditions were assigned. All the other edges were assigned as walls with insulated boundary conditions, as shown in Fig. 3.3. The upper and other two side walls were assumed to be adiabatic.

Construction of Geometry

The geometry was constructed in commercially available software ANSYS Design Modeler v15.0. Firstly, an outline of the geometry without ribs was created in x-y plane with appropriate dimensions (in mm) and then surface was generated from the "built sketches" option. Then another sketch that involved the interface between absorber plate and fluid was developed. The surface initially created was split into two faces with the help of "face-split" option by choosing the second sketch as the tool geometry. The face-splitting option was followed by the generation of surfaces from the faces with the help of "create surface from faces" option. Finally, all the edges and surfaces were named accordingly.

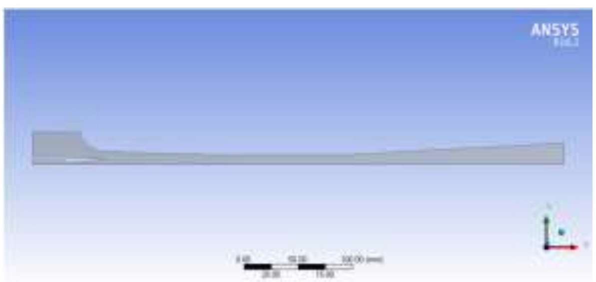

Fig. 5: CFD model

Meshing of the Domain

The meshing work was accomplished on commercially available ANSYS meshing software. The geometry created was imported in ANSYS meshing.

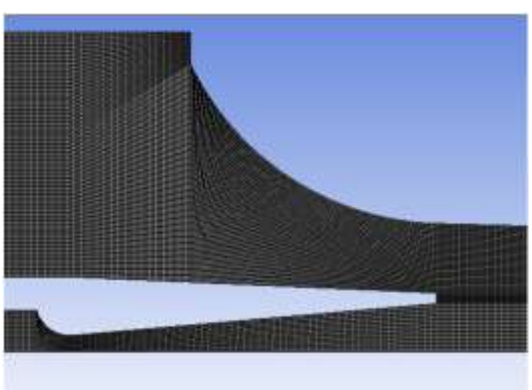

Fig. 6: CFD mesh model

The required number of divisions and the type of "bias" were assigned to each edge. In order to obtain regular rectangular shaped mesh cells with the best orthogonal quality, mapped facing option was activated. Finally, mesh was generated by clicking on "Generate Mesh" button. Fig. 6 shows the meshed domain for different cases.

III. RESULTS AND DISCUSSION

1. Contours after Simulation Results

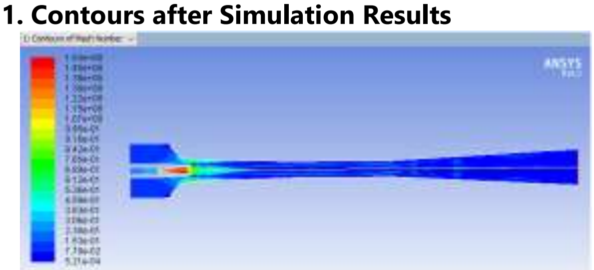

Fig. 7: Contour of Mach number at simulation 1

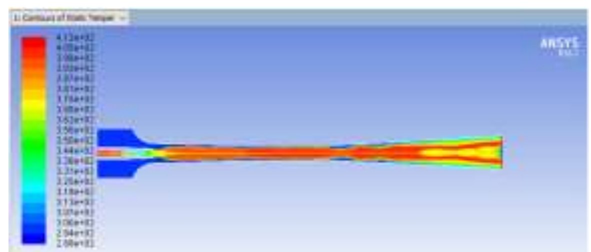

Fig. 8: Contour of static temperature at simulation 2

2. Optimization Using RSM Method

The behavior of 3D response surfaces in terms of critical condenser pressure are demonstrated in Fig. 9. To be specific, Fig. 9 shows the response surface plot of critical condenser pressure with respect to primary nozzle discharge diameter and primary nozzle throat diameter was shown.

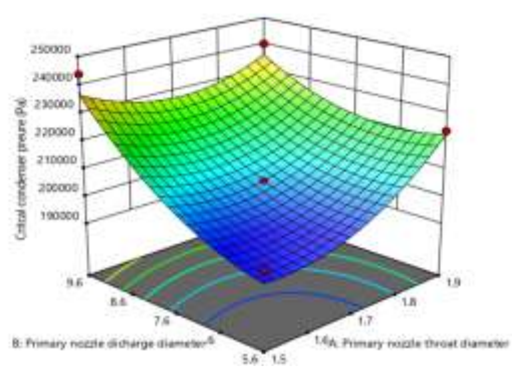

Fig. 9: 3d Surface plot

Optimization by RSM Method

Following the mathematical modeling and statistical analysis of the performance indices, a multi-objective optimization has been performed. The quality characteristics, used for the optimization modeling is the transmission losses and this work has been performed by the desirability approach.

Desirability is as an objective function, D, introduced by Myers and Montgomery [40] and its desirable value (di) ranges from 0 to 1, least to most desirable, respectively. The function searches a point in the specified design space within the constrained levels of factor settings and according to weights and importance (ri) that exaggerates the function value, where di = 1 is the target value, by sufficing all the goals those are addressed.

During the optimization process, the aim was to achieve the optimum levels of factor settings which yield the maximum transmission loses. Combination of one or more goals which may apply to either factors or responses is optimized by numerical optimization. The possible goals are: maximize, minimize, target, within range, none (for responses only) and set to an exact value (for factors only).

In this analysis, the goal for transmission loses were set at the highest level while process parameters involved were set as ‘in range’. Eq. 2 is the desirability function, where n is the number of responses in the measure; if any of the responses or factors fall outside their desirability range, the overall function becomes zero.

A weight value can be assigned to a goal to adjust the shape of its particular desirability function while the “importance” of a goal can be changed in relation to the other goals which will affect the way optimization proceeds.

$$D = (d_1 \times d_2 \times \cdots \times d_n)^{\frac{1}{n}} = \left(\prod_{i=1}^{n} d_i\right)^{\frac{1}{n}}$$

$$D = (d_1^{r_1} \times d_2^{r_2} \times \cdots \times d_n^{r_n})^{\frac{1}{n}} = \left(\prod_{i=1}^{n} d_i^{r_i}\right)^{\frac{1}{\sum r_i}}$$

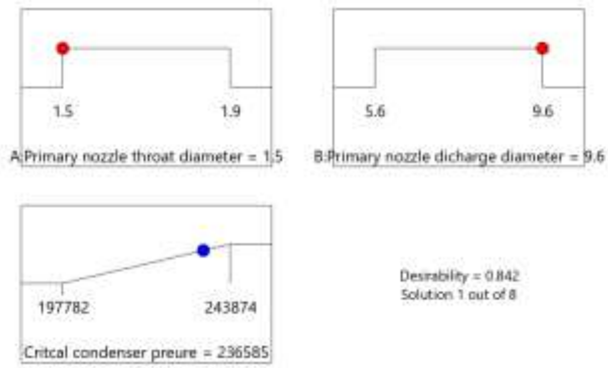

Fig. 10: Optimum parameter plot

A critical condenser pressure of 236585 Pa was observed at optimum level of primary nozzle throat diameter of 1.5 and primary nozzle discharge diameter of 9.6.

IV. CONCLUSION

- F-value of model (=23.36) implies that the generated model is statistically significant where Prob>F is less than 0.0001. Moreover, the terms A, B, are significant model terms as the value of Prob>F is less than 0.05.
- A critical condenser pressure of 236585 Pa was observed at optimum level of primary nozzle throat diameter of 1.5 and primary nozzle discharge diameter of 9.6.
- The application of RSM method improves the effectiveness of ejector.
- Best result is achieved from applying of RSM method.

REFERENCES

1. Yu, M., Wang, C., Wang, L., & Zhao, H. (2022). Optimization Design and Performance Evaluation of R1234yf Ejectors for Ejector-Based Refrigeration Systems. Entropy, 24(11), 1632.
2. Mikielewicz, D., Amiri, M., Klugmann, M., & Mikielewicz, J. (2023). A novel concept of enhanced direct-contact condensation of vapour-inert gas mixture in a spray ejector condenser. International Journal of Heat and Mass Transfer, 216, 124576.
3. Manoj, P. J., & Lijo, V. (2021, March). Recent developments in ejector refrigeration system. In

IOP Conference Series: Materials Science and Engineering (Vol. 1114, No. 1, p. 012046). IOP Publishing.
4. Riaz, F., Yam, F. Z., Qyyum, M. A., Shahzad, M. W., Farooq, M., Lee, P. S., & Lee, M. (2021). Direct Analytical Modeling for Optimal, On-Design Performance of Ejector for Simulating Heat-Driven Systems. Energies, 14(10), 2819.
5. Zhang, K., Chen, X., Markides, C. N., Yang, Y., & Shen, S. (2016). Evaluation of ejector performance for an organic Rankine cycle combined power and cooling system. Applied energy, 184, 404-412.
6. Elbarghthi, A. F., Mohamed, S., Nguyen, V. V., & Dvorak, V. (2020). CFD based design for ejector cooling system using HFOS (1234ze (E) and 1234yf). Energies, 13(6), 1408.