International Journal of Multidisciplinary Engineering in Current Research
Volume 6, Issue 1, January 2021, http://ijmec.com/

CFD ANALYSIS OF STEAM EJECTOR WITH DIFFERENT NOZZLE DIAMETER

PaheerathanVijitha*

*Senior Lecturer, Unit of Siddha Medicine,
TrincomaleeCampus, Konesapuri, Nilavelli, SriLanka

ABSTRACT:

Devices like steam ejectors use the momentum from high-speed main jets to entrain and accelerate a medium that is stationary or moving at a low rate of velocity. Maintaining vacuum, eliminating air from condensers, boosting thrust, and raising vapour pressure are some of the most significant duties of an ejector. By shrinking the size of a traditional multi-stage evaporator, this steam ejector is transformed into a thermal compressor. Analysis of the impacts on the main fluid pressure, mass flow rate and Mach number was carried out The mixing process inside the ejector may be better understood using the Mach number contours. The nozzle diameter of the steam ejector is varied in this project because Determined pressure drop, velocity, and other parameters by varying the pressure rate of the steam ejector. The main fluid's temperature, mass flow rate, and heat transfer rate are calculated using CFD. The Mach number, pressure drop, heat transfer rate, and mass flow rate rise when the nozzle diameter and condenser pressure are increased, as shown by the CFD study.

Keywords: Steam Ejector, Mach Number, and Computational Fluid Dynamics (CFD) Analysis.

I INTRODUCTION

A steam ejector is a device which utilizes the momentum of a high- velocity primary jet of vapor to entrain and accelerate a medium in still or at a low speed. The important functions of an ejector include maintaining vacuum in evaporation, removing air from condensers as a vacuum pump, augmenting thrust, and increasing vapor pressure as a thermal compressor. The thermal compressor is a steam ejector, but it utilizes the thermal energy to

augment the performance by reducing the size of a conventional multi-stage evaporator. The

effects on the primary fluid pressure, mass flow rate and Mach number were observed and analyzed. The Mach number contour lines were used to explain the mixing process occurring inside the ejector. In this thesis, we modeled steam ejector changing with different nozzle diameters and Analyzed the steam ejector with different mass flow rates to determine the pressure drop, Mach number, velocity and heat transfer rate for the primary fluid by CFD

International Journal of Multidisciplinary Engineering in Current Research
Volume 6, Issue 1, January 2021, http://ijmec.com/

technique. This work focuses on the numerical simulation of the working of a steam ejector in order improve the performance. Computational Fluid Dynamics (CFD) was employed for the numerical simulation. In this work the effect of operating conditions on the performance of the steam ejector operating in conjunction with an ejector refrigeration cycle was considered along with the effect of geometry parameter. The model and meshing is done with GAMBIT and FLUENT solver is used for the analysis. The simulations are performed with different operating conditions and geometries. The entrainment ratio is found to increase with the decrease of boiler saturation temperature for the same condition of superheat, evaporator temperature and condenser pressure. The entrainment ratio is also found to increase with increase of evaporator temperature keeping the boiler temperature and condenser pressure constant. The entrainment ratio does not vary much with

the condenser pressure until the critical condenser pressure. It is also found that the entrainment ratio increases with decrease of throat diameter of the primary nozzle. The increase of entrainment ratio can be found out from the moving downwards of the effective position. But, a larger mass of secondary fluid causes the momentum of the mixed stream to decrease. The decrease of momentum can be determined from the moving upstream of the shocking position. The movement of shocking position upstream can cause the ejector to operate at a lower critical condenser pressure.

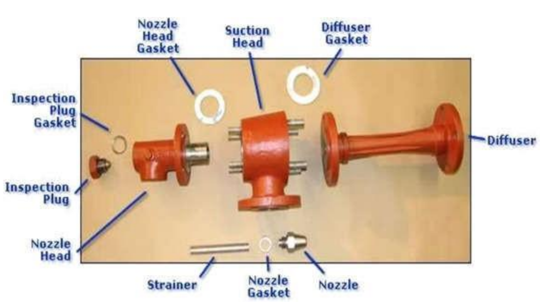


Figure 1: Components of a steam jet ejector

II.SURVEY OF LITERATURE

In a research by Natthawut Ruangtrakoon et al. (2013), the influence of steam jet refrigeration cycle ejector nozzle shapes on performance was examined using CFD. Only one fixed-geometry mixing chamber and eight distinct mixing chambers are used in all circumstances.

The CFD programme was used to model main nozzles numerically. A 2.3 mm throat diameter and an exit Machnumber of 4 were found to be ideal for the main nozzle, which was run at a boiler temperature of 120°C and an evaporator temperature of 7.5°C. The CFD approach may be used to accurately forecast the performance of a steam ejector, as shown in this research. It may also be used to describe the mixing process that can't be explained empirically. The findings reveal that the ejector performance and, therefore, the COP of the system, are strongly influenced by the geometry of the main nozzles utilised and the operating circumstances. Varga and others (2009)

Their research examined the impact of three geometrical parameters on steam jet ejector performance, including the nozzle-to-constant-section area ratio (rA), the nozzle exit position (NXP), and the length of the constant-section area section (Lm). An appropriate area ratio was found, depending on the operating circumstances. Exit location had an impact on

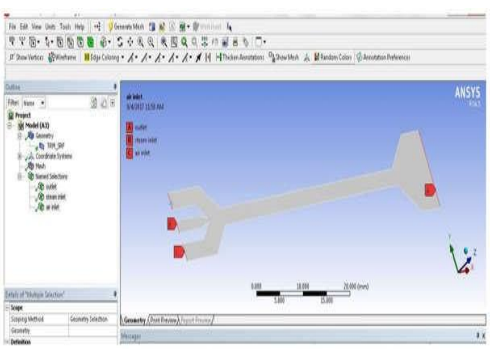
International Journal of Multidisciplinary Engineering in Current Research
Volume 6, Issue 1, January 2021, http://ijmec.com/

critical back pressure as well as entrainment ratio. The optimum nozzle exit distance was determined to be 6 cm from the intake plane of the converging section, resulting in an entrainment increase of between 5 and 12 percent.

in terms of both the crucial back pressure ratio and the ratio. It was found that increasing Lm to 155mm raised the critical back pressure significantly. The current ejector's performance was unaffected by a further increase, therefore it may be regarded ideal.

III METHODOLOGY

Methodology for this study is provided in fig (see below). The first step was to perform a literature review on the area of interest and gather relevant material. After that, the gathered material was analysed to determine the potential scope of future study and the



Plot2. Inlet and outlet model

improvements that may be made. As a result of the review of the relevant literature, a precise diagnosis of the issue was made. A steam ejector's performance was being studied in terms of its geometric characteristics and operating circumstances. GAMBIT 246 was used for geometry modelling and meshing. The steam ejector's axis symmetrical three-dimensional models (each with a distinct main nozzle diameter) were built and meshes were assembled. GAMBIT was used to establish the model's borders, and the resulting mesh was subsequently imported into FLUENT 6326 for further analysis. With varying boundary conditions, FLUENT was used to do the analysis of the data. A comparison was made between the acquired findings and those of previous experiments.

acquired from the reference for purposes of verification As a consequence of doing the analysis for various geometric configurations, optimal geometry and operating conditions may be predicted with confidence.

Analyzing the System III

CFD Model and Analysis of Ejector with Nozzle Diameter 1.7 at Condenser Pressure 20mbar for Ejector CFD Model and Analysis

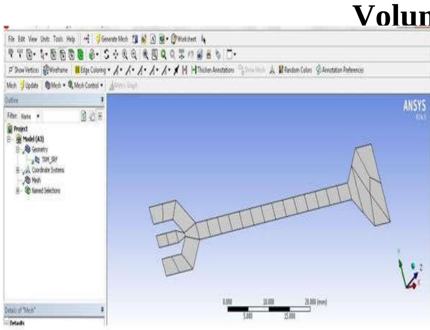
Double-click "Ansys" workbench, then "Fluid flow fluent."

Right-click 'Select geometry' and choose 'New geometry.'"

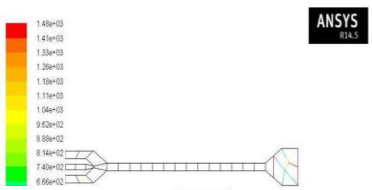
INLET AND OUTLET

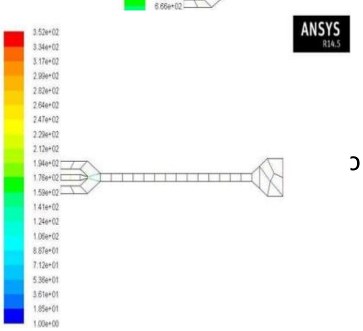
MESHED MODEL

International Journal of Multidisciplinary Engineering in Current Research
Volume 6, Issue 1, January 2021, http://ijmec.com/



Select an ergy equation (on) from the drop-down menu and click OK. Fluid materials may be created or edited under the Materials >





on. Choosing a fluid

TEMPERATURE

HEAT TRANSFER RATE

"Flux Report"

Total Heat Transfer Rate	(w)
air_inlet	41978.523
outlet	2841683.3
steam_inlet	704654.88
wall-_trm_srf	0
Net	3588316.6

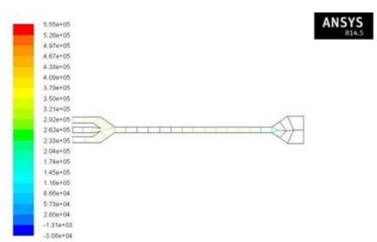
MASS FLOW RATE

"Flux Report"

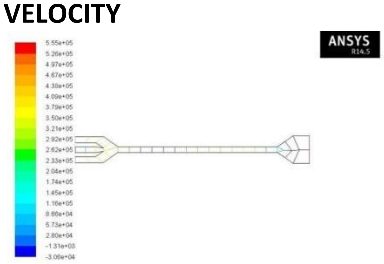
Mass Flow Rate	(kg/s)
air_inlet	0
interior-_trm_srf	-27838.811
outlet	1511.907
steam_inlet	4.5999999
wall-_trm_srf	0
Net	1516.507

CFD ANALYSIS OF EJECTOR CFD MODEL &ANALYSIS OF EJECTOR WITH NOZZLE DIA. 2.4 AT CONDENSER PRESSURE 20mbar

PRESSURE

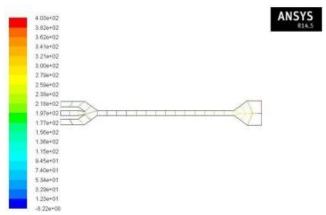


VELOCITY



TEMPERATURE

International Journal of Multidisciplinary Engineering in Current Research
Volume 6, Issue 1, January 2021, http://ijmec.com/



IV RESULTS

RESULTS TABLE NOZZLE DIA 1.7

NOZZLE DIA 2.0

Pressure Rate (mbar)	Pressure Drop (Pascal)	Velocity (m/s)	Temperature (k)	Mass flow rate (kg/s)	Heat transfer Rate (w)
20	1.18e06	1.48e03	3.52e02	1516.507	3588316.6
40	1.14e06	7.38e05	5.00e03	1694.0163	3.3692e14
60	8.65e05	5.76e02	3.81e02	14135	11086.79

Pressure Rate (mbar)	Pressure Drop (Pascal)	Velocity (m/s)	Temperature (k)	Mass flow rate (kg/s)	Heat transfer Rate (w)
20	5.36e05	6.05e02	6.44e02	2.1334	275679
40	5.37e05	6.06e02	6.48e02	2.1255	274124
60	4.78e05	6.29e02	6.50e02	2.27682	300814

NOZZLE DIA 2.4

Pressure Rate (mbar)	Pressure Drop (Pascal)	Velocity (m/s)	Temperature (k)	Mass flow rate (kg/s)	Heat transfer Rate (w)
20	5.55e05	7.06e02	4.03e02	0.300027	589063
40	5.54e04	7.05e02	4.02e02	0.301203	586209
60	5.52e05	7.04e02	4.01e02	0.302922	583292

"Flux Report"

Mass Flow Rate (kg/s)

---

air_inlet	0
interior-_trm_srf	25.597944
outlet	-4.9012032
steam_inlet	4.5999999
wall-_trm_srf	0
Net	-0.30120325

"Flux Report"

Total Heat Transfer Rate (w)

---

air_inlet	14.666726
outlet	-1289226.5
steam_inlet	703002.31
wall-_trm_srf	0
Net	-586209.52

VCONCLUSION

In order to evaluate the pressure drop, velocity, temperature, and heat transfer rate for the main fluid using CFD, I designed and analysed a steam ejector with varying nozzle diameters in my thesis. The Mach number, pressure drop, heat transfer rate, and mass flow rate rise when the nozzle diameter and condenser pressure are increased, as shown by the CFD study. A steam ejector model with an outside circumference of 2.24 millimetres is the best option.

VI REFERENCES

Computational fluid dynamics was used to study the steam ejector of dr.i satyanarayana1

CFD-based Steam Ejector Thermal Analysis Aravind 1; Dr P Ravinder Reddy 2; and S S Baserkoed 3;

Simulation of main nozzle geometries in the refrigeration cycle using CFD with the purpose of studying the influence of nozzle geometry a Natthawut Ruangtrakoon b, Tongchana

International Journal of Multidisciplinary Engineering in Current Research
Volume 6, Issue 1, January 2021, http://ijmec.com/

Thongtip c, Satha Aphornratan d,*, and Thanarath Sriveerakul c.
4. A numerical investigation of the use of a supersonic steam ejector in a steam turbine Miao He and Lin Cai
An investigation of ejector design by analysis and experiment was conducted by Keenan, JH and NeumannEP (1950) in the Journal of Applied Mechanics.
A numerical examination of the performance of a steam evaporator's thermal ejector was published in Applied Thermal Engineering in 2010 (Volume 30, Issue 27, Pages 2708–2717).
Converging Duct Angle Effects on Steam Ejector Performance: A CFD Study by Tony Utomo, Myongkuk Ji, Pilhwan Kim, Hyomin Jeong, and Hanshik Chung (2008), International Conference of Engineering Optimization
9. Imad Alatiqi, Ghada Al-Nuwaibi and Hisham El-Dessouky (2002), Evaluation of steam ejectors in chemical engineering and processing, 41(5), 551–561