Influence of Nozzle's Exit Mach Number on the Steam ejector's Performance by Using Computational Fluid Dynamics

Tongchana THONGTIP[1], Natthawut RUANGTRAKOON[2], Thanarath SRIVEERAKUL[3]
And Satha APHORNRATANA[2,*]

[1] Joint Graduate School of Energy and environment, King Mongkut's University of Technology Thonburi, Bangkok, Thailand 10140
[2] Sirindhorn International institute of technology, Thammasat University, Patumthani, Thailand 12121.
[3]Department of Mechanical Engineering, Ubon Ratchathani University, Ubonratchathani, Thailand 34190
*Corresponding Author: Tel: (662) 9869009 Exit2210, Fax: (662) 9869009 Exit. 2201, E-mail: satha@siit.tu.ac.th

Abstract

In this study, the computational fluid dynamics (CFD simulation) was used to describe the effect of the primary nozzle's exit Mach number to the steam ejector performance. Three primary nozzles with difference area ratio were used. They produce the exit Mach number of 3, 4 and 5.5. The boiler saturation temperature and the evaporator saturation temperature were fixed at 150°C and 7.5°C, respectively. The physical model and grid structure of steam ejector used was created by commercial software package, Gambit2.3 whilst the mathematical model was applied by commercial software package, FLUENT6.3. Simulation results were validated with the experimental values.

***Keywords**: Ejector, Steam jet refrigeration, computational fluid dynamics*

1. Introduction

A steam jet refrigeration system can be considered as a tool which utilizes waste heat or low grade heat to produce a useful refrigeration effect. These heats can be found from industrial processes, geothermal energy, or thermal solar collector. Thus, steam jet refrigeration is suitable for currently energy situation. In addition, water can be used as working fluid, thus it is environment friendly.

The major disadvantage of a steam jet system is its relatively low COP compared with the other type of refrigeration systems. Therefore, its performance should be improved. From the literatures [1-3], COP of jet refrigeration system was dependent on the ejector equipped.

Recently, computational fluid dynamics technique or CFD simulation was rapidly developed. It could be used to explain the phenomenon inside the ejector, [1,2]. The distinctive point of CFD simulation is that the flow behavior inside the ejector can be presented and explained the mixing process. It can be used to predict the system performance accurately, [1].

In this study, CFD technique was used to study the effect of the primary nozzle's exit

Mach number to the ejector performance. The simulation's results were compared with that obtained experimentally. The CFD simulation was also used to explain the process inside the ejector.

2. Experimental ejector

Basically, the ejector composed of four principle components which are the primary nozzle, the mixing chamber, the constant area throat and the subsonic diffuser. All these components have affect to the ejector performance. However, the main objective of this study is to investigate the effect of the nozzle's exit Mach number. Thus, to avoid the influences of other component, one fixed geometries ejector was used as shown in fig. 1. Three difference primary nozzles were used as shown in table. 1. All of them have equal throat diameter of 1.4 but produce difference exit Mach number, 3, 4 and 5.5. The nozzle exit position (NXP) is located at +23mm.

Table. 1 The dimension of primary nozzle

Nozzle	d	d:D	Mach number
D1.4M3	1.4mm	1:7	3
D1.4M4	1.4mm	1:20	4
D1.4M5.5	1.4mm	1:85	5.5

3. CFD model setup1

3.1. Geometries and grid structure

In this study, a CFD software package, FLUENT 6.3 was used. Normally, using the CFD simulation is divided as two main parts which are creating physical model and solving.

The physical model was divided as grid elements by Gambit 2.3 commercial software package. In the part of solving, FLUENT 6.3 was used to define mathematical model for calculation.

In the part of creating the model, Gambit 2.3 commercial software package was used to create the geometries of steam ejector. The result from simulation of 2-D axisymetric model was similar to the 3-D model [2]. It requires less time for calculation than that for 3-D model.

The grid elements were created of 80,000 structured quadrilateral elements. The concentration of grid density was focused on the area where significant phenomena were expected. It was expected that, the calculation at that area should provided high accuracy results.

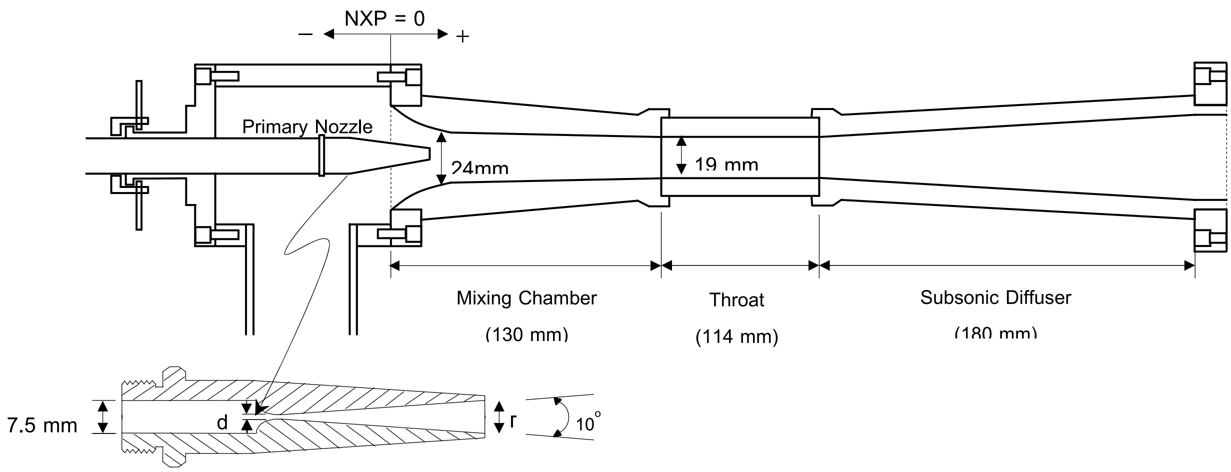


Fig. 1 Show the dimension of steam ejector used

In the part of solver setup, the CFD commercial software package, FLUENT 6.3 was used to specify the mathematical model for the physical model. The flow inside the steam ejector was considered as a turbulence compressible flow. Thus, the non-linear governing equation as “density-based implicit” was selected as a solver. This solver was combined with viscosity turbulence model for solving the problem.

3.2 Working fluid properties

Actually, the properties of working fluid, water vapor, should be considered as real gas. However, it required very long time for calculation and difficult to converge to convergence criterion. This cause may be the error of problem.

However, normally, the steam ejector was mostly operated at the relatively low pressure. At this situation the properties of real gas could be assumed to be ideal gas [1]. The density of working fluid is evaluated by using ideal gas relation. The properties of working fluid were shown in table 2.

Table. 2 Working fluid properties

Properties	value
Viscosity, (kg/m.s)	1.34×10^{-5}
Conductivity, (W/m.K)	0.0261
Specific heat, (J/kg.K)	2014.00
Molecular weight, (kg/kmol)	18.01534

3.3. Boundary conditions

There are two type of boundary condition that suitable for turbulence compressible flows which are pressure-inlet type and pressure-outlet type [5]. For the upstream of ejector (primary fluid inlet and secondary fluid inlet), the pressure-inlet type was applied to this face. For the downstream of ejector (mixed fluid outlet or subsonic diffuser), the pressure-outlet type was applied to this face. The values of operating conditions were assigned as the saturation properties (pressure and temperature).

3.4 Convergence criterion and solution

During the simulation, it was considered as converged when the following two criteria were satisfied. Firstly, the mass fluxes through all faces in the model were stable (the conservation of mass must be satisfied). The difference in mass flow between of the inlet and the outlet was less than 10^{-7} kg/s. Secondly, every type of the calculation residual must be reduced lower than the specified value (less than 10^{-6}). This ensured that the solution from simulation was accurate. The number of iteration for simulation was greater than 100,000 iterations.

4. Result and discussion

4.1. Phenomenon within the steam ejector

At first, CFD simulation is used to explain the phenomenon within the steam ejector. The understanding the flow behavior and mixing process are necessary. The filled contour of Mach number from CFD simulation as shown in fig. 2 is used to explain this phenomenon.

Referring to fig. 2, as high temperature, high pressure fluid known as “primary fluid” enters the primary nozzle; the fluid is accelerated in the converging portion of primary nozzle (1). At the nozzle’s throat, the Mach number is unity. The flow is choked. The flow is

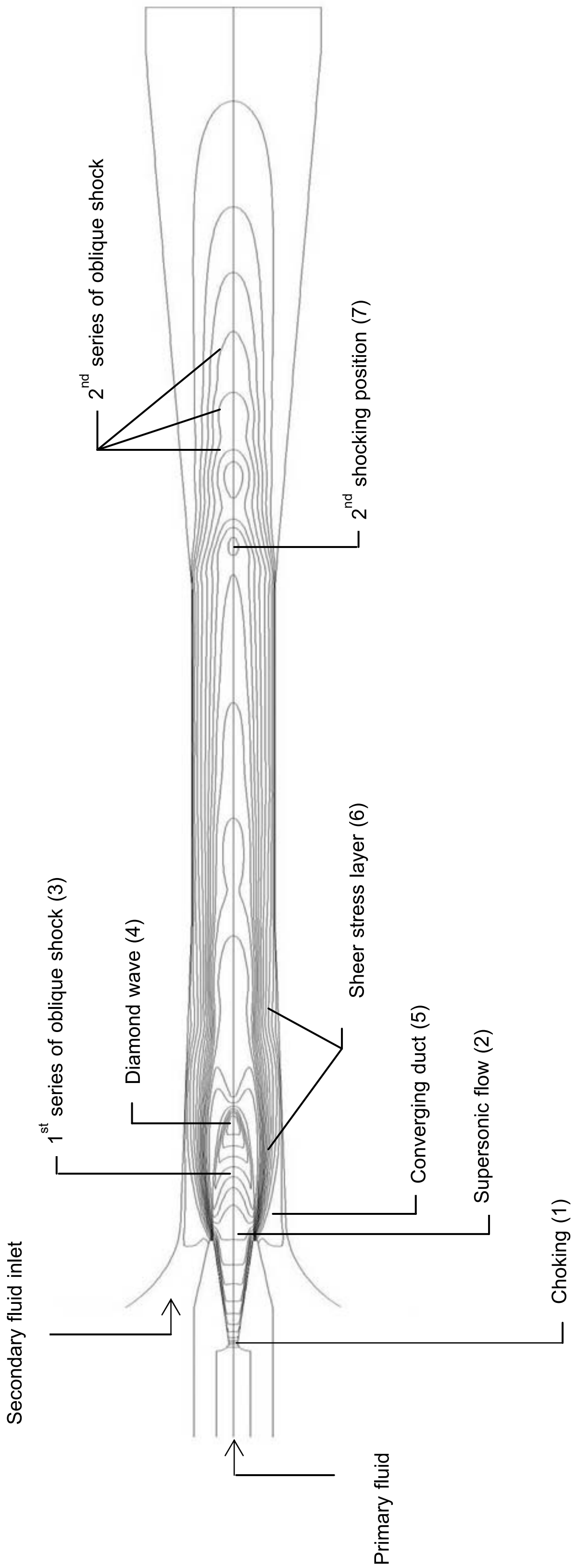


Fig. 2 Filled contour of mixing process within the steam ejector

further accelerated to supersonic level in the diverging part of the nozzle (2). At exit, the primary leaves the primary nozzle as supersonic stream which flows under the free boundary pressure condition, the expanded wave is formed [2,4] with some value of expansion angle. This expanded wave is classified as two characteristics which are "under-expanded wave" and "over-expanded wave" [4]. As a result of the occurrence expanded wave, the oblique shock (3) is simultaneously formed [2] which both are called "diamond wave" (4). The expanded wave is classified as under-expanded (the expansion angel is divergence after leaving from the primary nozzle) and the oblique shock wave is called "1st series of oblique shock" or "1st shocking". The flow form is semi-separation between the primary and secondary fluid and these fluids are not mixed. Therefore, the "converging duct" (5) for entraining the secondary fluid is formed [2]. At the interface of these fluids (primary and secondary fluids), due to the large velocity difference, the "shear stress layer" (6) is created. This causes the secondary fluid to accelerate to sonic velocity and choked at some section. The two flows (primary and secondary fluids) are then mixed together. The cross sectional area where the secondary fluid choked is called "an effective area".

During the mixing process, the momentum of primary fluid is transferred to the secondary fluid. As the mixed fluid flow passes through the constant area section, due to the large difference pressure between the mixed fluid and ejector's downstream pressure, the oblique shock is induced (7). This is called "2nd series of oblique shock" or "2nd shocking". As a result of occurrence this shock wave series, the static pressure of mixed fluid is increased rapidly. The flow form is changed from supersonic to subsonic.

In the subsonic diffuser section, the mixed flow is further slow down to almost stagnation in order to recover the static pressure before discharging to the condenser.

4.2. An ejector's performance

The performance of a steam ejector is normally defined by an entrainment ratio:

$$Rm = \frac{\text{mass flow of the primary fluid}}{\text{mass flow of the secondary fluid}} \qquad \text{Eq. (1)}$$

Fig. 3 shows a performance curve of a steam ejector. The curve is classified as three regions which are choked flow, unchoked flow, and reversed flow region.

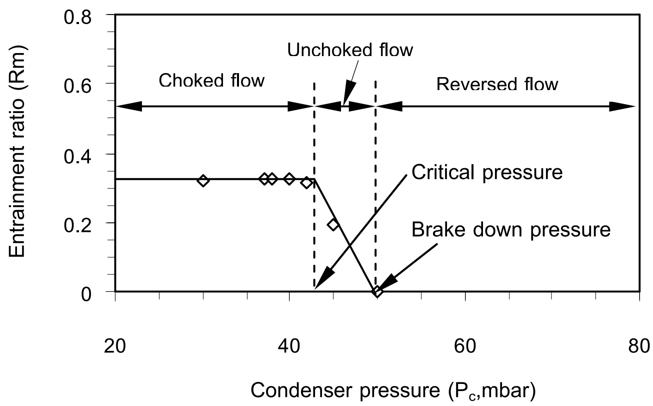
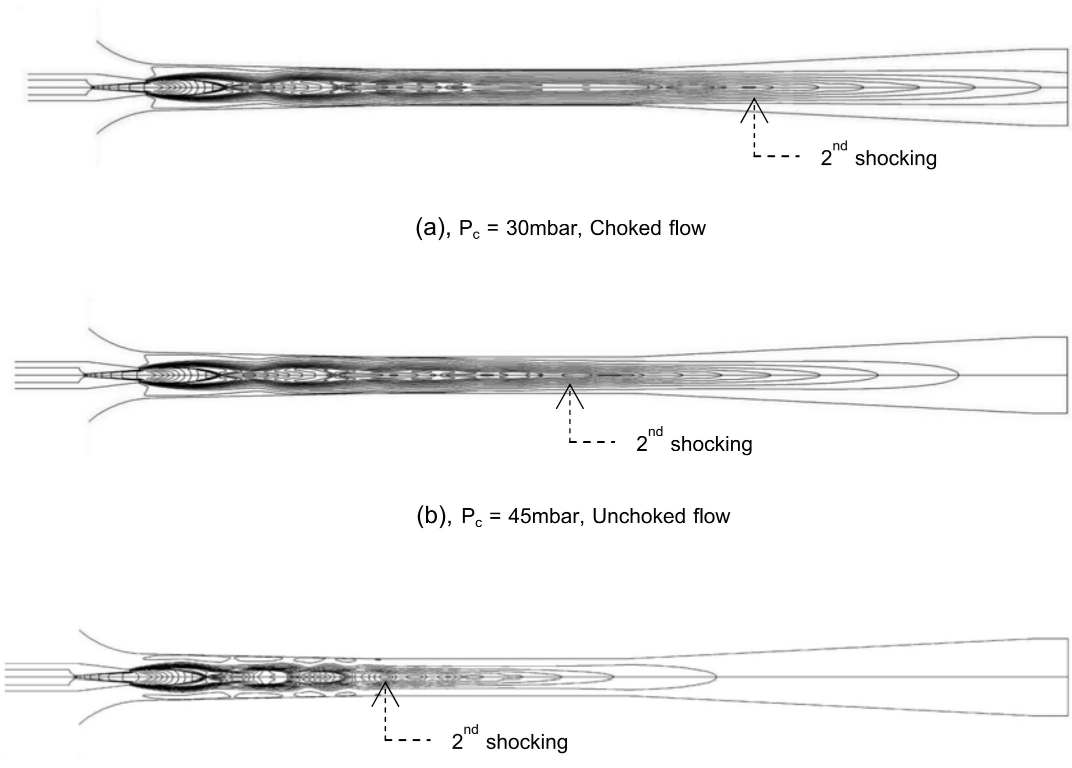


Fig. 3 The performance curve of steam ejector

Fig. 4-a to 4-c show the filled contour of Mach number from CFD simulation. It can be seen that an increasing of the condenser pressure causes the 2nd shocking position to move backward into the constant area throat section. If the back pressure does not exceed the critical pressure value or operates in the choked flow region (fig. 4-a), the shock will not affect the mixing behavior of the two fluid



(a), P_c = 30mbar, Choked flow

(b), P_c = 45mbar, Unchoked flow

(c), P_c = 50mbar, Reversed flow

Fig. 4 the effect of condenser pressure (P_c) on the 2nd shocking position streams. Therefore, the secondary fluid remains choked at the effective area section when the ejector is in the choked flow region. As the back pressure is increased and the ejector is in an unchoked flow region (fig. 4-b), the 2nd shocking moves closer to the region where the mixing process occurred. In this case, the effect of the shock disturbs the mixing process. The secondary fluid is no longer choked. This causes the ejector to entrain less secondary fluid; therefore, the entrainment ratio drops sharply in this unchoked flow region. If the back pressure increase to the value above the breakdown point or reversed flow region (fig. 4-c), the shock will move toward the primary nozzle and disturb the mixing process until the secondary fluid cannot be entrained, the entrainment ratio drops to zero.

4.3 Effect of nozzle's exit Mach number

In this case study, the boiler temperature and the evaporator temperature were 150°C and 7.5°C respectively. The condenser pressure was between 25 to 60 mbar. Three primary nozzles were used. They produced an exit Mach number of 3.0, 4.0 and 5.5.

Fig. 5 and 6 show results from experiments and CFD simulations, respectively. By comparing these results the error of entrainment ratio and critical condenser pressure are listed in table 3. The entrainment ratio in the choke flow region is independent from the change of the Mach number at the nozzle exit. All nozzles entrain the same amount of the secondary fluid. The critical condenser pressure is increased with the Mach number. However, maximum nozzle's exit Mach number is limited by the diameter of the nozzle exit and the boiler pressure as shown in table 4. For the case of Mach number of 6.0, the nozzle exit diameter is 16 mm which is as large as the mixing chamber throat diameter (19 mm). This will block the secondary flow at the mixing chamber inlet. Therefore, in practice, the primary nozzle should be designed so that the exit Mach number is between 4.0 and 5.5. Nozzle with exit Mach number greater than 5.5 will has a large exit area which will obstruct the secondary flow at the mixing chamber inlet.

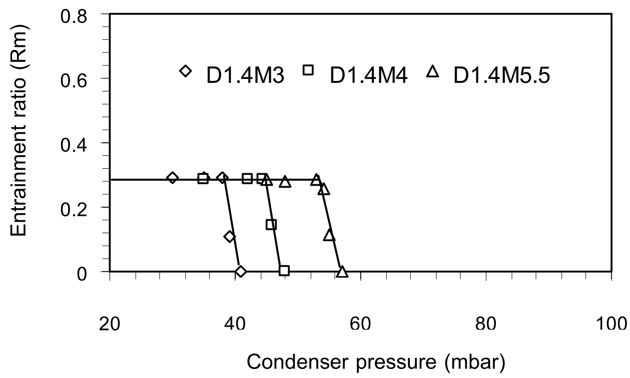


Fig. 5 Show the influences of nozzle's exit Mach number (by experiment)

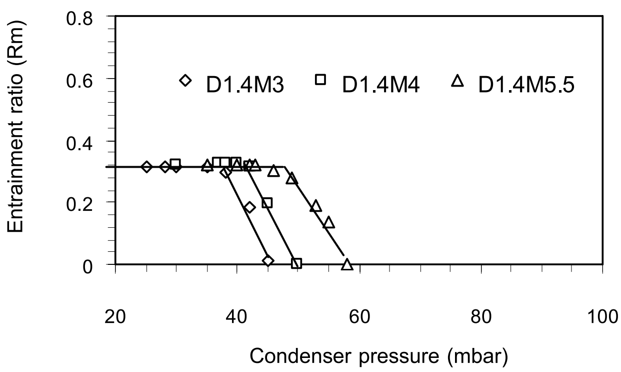


Fig. 6 Show the influences of nozzle's exit Mach number (by CFD simulation)

Table. 3 Comparison of ejector performance from experiment and CFD simulation (at critical point)

	Entrainment ratio			Critical condenser pressure (mbar)		
Nozzle	experiment	CFD	Error[a] (%)	Experiment	CFD	Error[b] (%)
D1.4M3	0.290	0.314	8.27	38	36	5.55
D1.4M4	0.285	0.315	10.52	46	43	6.52
D1.4M5.5	0.287	0.309	7.67	55	47	17.05

[a]Error (%) = 100 × (CFD's entrainment ratio - Experiment's entrainment ratio) / Experiment's entrainment ratio.

[b]Error (%) = 100 × (CFD's critical pressure- Experiment's critical pressure) / Experiment's critical pressure.

Table. 4 Data for the primary nozzles

Mach number	Expansion ratio	Area ratio	Nozzle exit pressure (mbar)	Minimum boiler pressure (bar)	Nozzle exit diameter (mm)
3.0	39	7.2	5.5	0.21 (61.1°C)	3.8
4.0	184	20.8	3.2	0.59 (85.5°C)	6.4
5.5	1386	88.8	2.2	3.05 (134.1°C)	13.2
6.0	2514	137.3	1.9	4.78 (140.2°C)	16.4

From table 4, the nozzle's area ratios were obtained from "Eq. (2)" and the expansion ratios were calculated from "Eq. (3)":

$$\frac{P_{boiler}}{P_{exit}} = \left[1+\left(\frac{k-1}{2}\right)\cdot M_{exit}^{2}\right]^{\frac{k}{k-1}} \qquad \text{Eq. (2)}^{*}$$

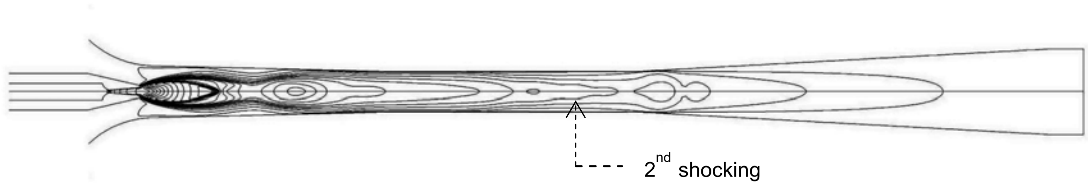
$$\frac{A_{exit}}{A_{throat}} = \frac{1}{M_{exit}}\cdot\left(\frac{2}{k+1}\right)\left[1+\frac{k-1}{2}\cdot M_{exit}^{2}\right]^{\frac{k-1}{2k+1}} \qquad \text{Eq. (3)}^{*}$$

Fig. 7 ("a" to "c") show the filled contour of the Mach number inside the steam ejector. In all cases, the back pressure was set at 30mbar. The figures show that, an increasing of Mach number cause the 2nd shocking position to move forward to the subsonic diffuser exit. This is due to an increase of the primary fluid's momentum. Therefore, the ejector can be operated at a higher critical back pressure.

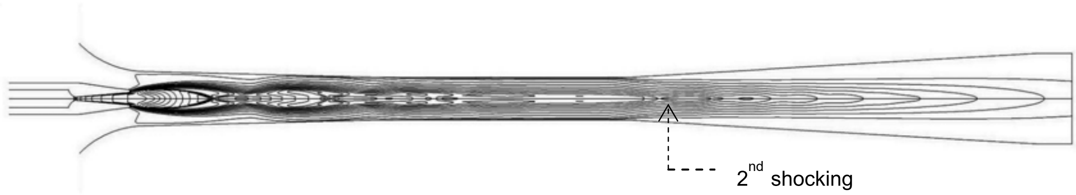
Both experiment and CFD results show that, the entrainment ratio is independent from the change of the primary nozzle's exit Mach number. The possible reason of this phenomenon is that, even the secondary fluid may be obstructed by the larger nozzle size at higher Mach number. This may be supplanted by a larger effective area and a higher velocity of the secondary fluid. Comparing between fig. 7-a, 7-b, and 7-c, it showed that the larger effective area (the area between expanded wave and ejector's wall) results from reducing the expansion angel of primary stream when the nozzle's exit Mach number is also increased.

Eq. (2)*: P_{boiler} is absolute pressure at the boiler; P_{exit} is absolute pressure at nozzle's exit; M_{exit} is Mach number at nozzle's exit; k is specific heat ratio (water = 1.327).

Eq. (3)*: A_{exit} is cross section area of nozzle's exit; A_{throt} is cross section area of nozzle's throat.



(a), D1.4M3



(b), D1.4M4

(c), D1.4M5.5

Fig. 7 The effect of nozzle's exit Mach number on the 2nd shocking position

5. Conclusion

In this study, CFD technique was used to study the effect of the primary nozzle's exit Mach number to the steam ejector performance. The simulation results were compared with those obtained experimentally to ensure the accuracy of the model.

Three primary nozzles were used, the exit Mach number were 3.0, 4.0, and 5.5. In all cases the entrainment ratio was independent from the variation of the nozzle's exit Mach number. The advantage of using the nozzle with high Mach number is that, the ejector can be operated with a higher critical back pressure which is desirable for a steam jet refrigeration system.

CFD simulation was used to explain the process within the ejector; it was showed that the critical back pressure was dependent on the momentum of the primary fluid stream and the location of the 2nd shocking position.

The ejector used in steam jet refrigeration system should be designed so that the primary nozzle's exit is as high as possible.

6. Acknowledgement

This research was financed by joint Graduate School of Energy and Environment (JGSEE), King Mongkut's University of Technology Thonburi and Thailand Research Fund (TRF).

5. References

[1] T. Sriveerakul, S. Aphornratana, K. Chunnanond. (2007). Performance prediction of steam ejector using computational fluid dynamics: Part 1. Validation of the CFD results, *International Journal of Thermal Sciences*, vol. 46(8), August 2007, pp. 812-822.

[2] T. Sriveerakul, S. Aphornratana, K. Chunnanond. (2007). Performance prediction of steam ejector using computational fluid dynamics: Part 2. Flow structure of a steam ejector influenced by operating pressures and geometries, *International Journal of Thermal Sciences*, vol. 46(8), August 2007, pp. 823-833.

[3] Yinhai Zhu, Wenjian Cai, Changyun Wen, Yanzhong Li. (2008). Numerical investigation of geometry parameters for design of high performance ejectors, *Applied Thermal Engineering*, vol. 29(5-6), April 2009, pp. 898-905.

[4] Michel A. Saad. (1985). Compressible fluid flow, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 07623, USA.

[5] FLUENT 6.3 User's guide, FLUENT INC. Lebanon, NH, USA.