

CHAPTER 8

CONCLUSION

An ejector is regarded as a critical component of a steam jet refrigeration cycle. This is because the COP of this cycle is strongly dependent on the ejector. In order to improve the performance of the cycle, the ejector's improvement should be first considered. Therefore, a clear understanding of the flow and process inside the ejector is necessary.

From available survey literatures on the past researches of ejector as provided in Chapter II, it was shown that the flow inside the ejector was complicated. The classical 1-D theory was inadequate to explain the flow inside the ejector and therefore, it was later explained by various methods, i.e., the Schindler photograph, pressure profiles along the ejector, and CFD technique. The CFD technique, from which the flow inside the ejector could be presented graphically, could provide a better understanding than others.

In this study, the steam ejector's performance was investigated experimentally and numerically (using CFD technique) at various operating conditions and various primary nozzle's throat size. The entrainment ratio and critical condenser pressure were used to indicate the steam ejector's performance. In the experimental part, the experimental steam ejector was tested. The detail of experimental steam jet refrigeration cycle and instrumentation were provided in Chapter III. In the part of CFD technique, a commercial CFD software package, Gambit 2.3 and FLUENT 6.3 were used. The detail of CFD model setup was provided in the Chapter IV.

The simulated results provided by CFD technique were validated with actual value obtained experimentally in Chapter V. It was found that the simulated results shown the similarities in the ejector's performance characteristic when compared to the actual values. Moreover, it was also found that, the simulated results based on *k- ω -sst viscosity model* were more accurate than those based on *reliable k- ϵ viscosity model*. Overall, it could be concluded that CFD technique could be used to evaluate performance of the steam ejector effectively.

As ensured that the CFD technique provided the exceptional result. It was used to explain the flow and mixing process inside the ejector. The graphical results obtained from

the CFD simulations were presented in the form of filled contour of Mach number as provided in Chapter VI. The flow behavior and mixing process inside the ejector was explained clearly by the fill contour of Mach number.

In Chapter VII, effects of operating condition and primary nozzle's throat size to the ejector performance were explained by the fill contour of Mach number obtained from CFD simulation. It was found that the entrainment ratio was related to the size of *converging duct* (formed by the primary fluid jet core and the mixing chamber wall) and *effective area*. Meanwhile, the critical condenser pressure was related to the location of 2nd *shocking* of the mixed flow inside the mixing chamber.

The entrainment ratio of the ejector was increased when a larger size of converging duct and effective area were formed and vice versa. The larger size of converging duct and effective area occurred when the ejector was operated at the relatively low boiler saturation temperature, or the relatively high evaporator saturation temperature or using smaller primary nozzle's throat size.

The ejector was able to operate at the higher critical condenser pressure when the 2nd shocking position was moved forward to the subsonic diffuser and vice versa. The moving forward of the 2nd shocking position occurred when the ejector was operated at the relatively high boiler saturation temperature, or the relatively high evaporator saturation temperature or using the larger primary nozzle's throat size.

At the ended of the Chapter VII, it was shown that when adjusting the operating-condition of ejector and primary nozzle's throat size, the entrainment ratio and critical condenser pressure could not be increased together. The only adjustment which could increase both entrainment ratio and critical condenser pressure simultaneously, the most desired point, was the increase of the secondary fluid saturated temperature (evaporator saturated temperature). However, this achievement comes with the undesired point of the refrigerated temperature.

Overall, this study has verified that the CFD technique shows the proficiency of predicting the entrainment ratio and the critical condenser pressure. In addition, it also provides the good understanding the flow behavior and mixing process inside the ejector. This helps to efficiently explain the influences of changed operating-condition and primary nozzle throat size. Lastly, it is hoped that the information provided in this study will lead to novel design of the high performance ejector.

From the study, it can be said that the CFD technique was useful in order to improve the ejector's performance. However, there are still some parts that should be adjusted in order to obtain the more realistic CFD model. These can be presented as follow:

- The real gas assumption should be applied to replace the ideal gas assumption; this is so that the fluid flow inside the ejector was more realistic.
- The heat transfer function at the ejector's wall should be added in the mathematical model.
- The others turbulence viscosity model should be applied to govern turbulence characteristic.
- The others form of grid elements applied to the physical model of ejector should be investigated so that it may provide a more accurate result.