CONDENSING EJECTOR FOR SECOND STEP COMPRESSION IN REFRIGERATION CYCLES

Mark J. BERGANDER[1], PhD, P.E., Prof. David P. SCHMIDT[2], David A. HEBERT[2],
Dr. Jerzy WOJCIECHOWSKI[3], Mateusz SZKLARZ[3],

[1] Magnetic Development, Inc.
Madison, CT, USA, Ph.: 203-421-3562, E-mail: mark@mdienergy.com

[2] University of Massachusetts
Amherst, MA, USA, Ph.: 413-545-1393, E-mail: Schmidt@ecs.umass.edu

[3]AGH University of Science & Technology
Cracow, Poland, Ph. 48-12-617-3078, E-mail: jwojcie@agh.edu.pl

ABSTRACT

This paper describes a novel approach to the Rankine vapor compression cycle for cooling and refrigeration. The specific innovation is the application of a two-phase device known as a "condensing ejector" (CE) for a second step of compression. The innovation has the potential of increasing the efficiency of the standard single-stage vapor compression cycle through a reduction of mechanical compression at the expense of harnessing kinetic energy of gas in the ejector device. In addition it will reduce the greenhouse gas emission by providing the same amount of cooling with less electric energy consumption. This is the continuation of the developmental work performed under the funding from the NSF and US Dept. of Energy.

1. INTRODUCTION

The main objective of this project was to explore the possibility of using a condensing ejector as a second step compression in a vapor compression cycle (reversed Rankine cycle) for refrigeration and air-conditioning applications. The entire research consisted of synergistic theoretical analysis, numerical simulation and laboratory experiments. The main research activities were to: 1) develop the methodology for the design of a condensing ejector using refrigerants as working mediums, 2) confirm the theoretical model by practical experiments on the laboratory stand. The results indicated the potential for using a two-phase condensing ejector in refrigeration cycles. Our approach of using the ejector in refrigeration cycle significantly differs from prior attempts:

1) The ejector is placed after the compressor discharge to increase the final cycle pressure, while all to-date designs used ejectors for increasing the compressor suction pressure.

2) In all previous ejectors, the outlet pressure was intermediate between the pressures of motive and suction streams. Our design produces the outlet pressure higher than both inlet pressures. This is achieved by the creation of a "condensation shock", when the vapor phase is quickly condensed onto the liquid stream, producing rapid transformation from two-phase into single-phase flow with a resulting rise in pressure.

2. THEORETICAL ANALYSIS AND NUMERICAL SIMULATION OF THE CE

2.1 Principle of the Condensing Ejector

The condensing ejector is a two-phase jet device in which a sub-cooled refrigerant in a liquid state is mixed with its vapor phase, producing a liquid stream with a pressure that is higher than the pressure of either of the two inlet streams. In our new cycle, the compressor compresses the vapor to approximately 2/3 of the final pressure and additional compression is provided in an ejector, reducing the amount of mechanical energy required by a compressor and improving the efficiency theoretically by up to 35%. The theory of the condensing ejector has been known since 1970's but all previous work was conducted on open systems. Specifically, the theory was described by Levy and Brown (1972), Van Wijngaarden (1972) and Fisenko (1986). This research is believed to be the first attempt to use such device in a refrigeration system, where inlet and outlet parameters must fall within constraints of a closed thermodynamic cycle.

Figure 1 below presents the predicted behavior of a condensing ejector based on the results of literature study and our initial computer simulation. The high relative velocity between vapor and liquid streams produces a high value of heat transfer and the rate of condensation of the vapor phase is high. This condensed vapor adds to the momentum flux of liquid stream. The remaining vapor and liquid jet (original plus condensed vapor) enter the constant area mixing section where a condensation shock may occur such that a completely liquid state exists downstream of the shock. The pressure rises due to lowering the velocity of the stream and the mass flow balance is maintained by a sudden change of density (condensation).

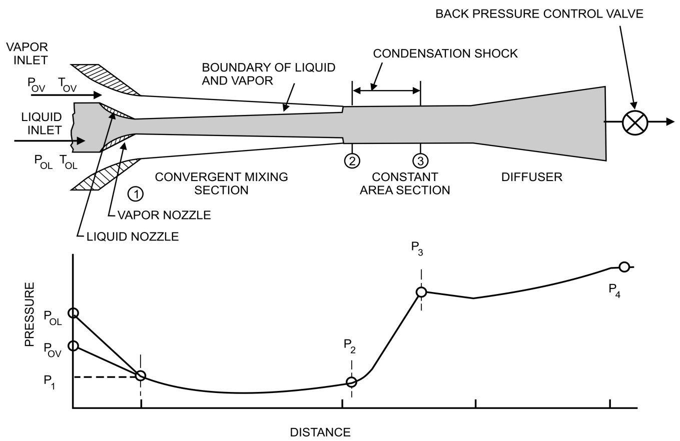

Figure 1: Condensing Ejector Processes and Pressure Distribution

2.2 Modeling the Condensing Ejector

Theoretical modeling was conducted to develop the design methodology of the condensing ejector. Two types of simulation were performed in order to better understand functioning of this device. The basic problem considered was, given the inlet conditions, **what will be the exit pressure of the condensing ejector.** A conventional mathematical analysis, which consists of equations of mass, momentum and energy, results in the determination of the discharge pressure; however it does not explain the character nor the location of a pressure shock. Therefore in order to gain the full understanding of these phenomena, we used computer simulation together with experiments. Consequently, we have created a three-dimensional CFD implementation based on a pseudo-fluid approach where the mixture is not in thermodynamic equilibrium, but instead returns to equilibrium over an empirical time scale. The Homogenous Relaxation Model was used as the basis for the rate of return to equilibrium.

2.3 Control Volume Analysis

A control volume representing the condensing ejector is shown in Figure 2.

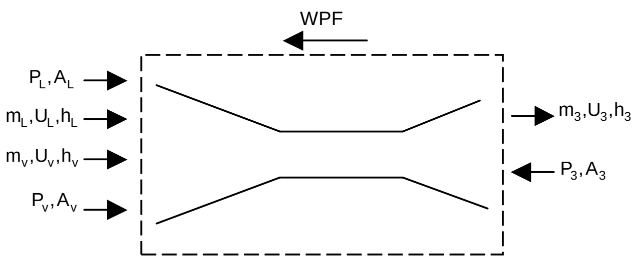

Figure 2: Control Volume for CE analysis. Point 2 (not shown in the picture) relates to the constant-area mixing chamber – between converging and diverging areas

From this control volume the continuity, momentum, and energy equations are obtained:

$$-\dot{m}_v - \dot{m}_L + \dot{m}_3 = 0 \tag{1}$$

$$-\dot{m}_v U_v - \dot{m}_L U_L + \dot{m}_3 U_3 - P_v A_v - P_L A_L + P_3 A_3 + WPF = 0 \tag{2}$$

$$-\dot{m}_v \left(h_v + \frac{U_v^2}{2} \right) - \dot{m}_L \left(h_L + \frac{U_L^2}{2} \right) + \dot{m}_3 \left(h_3 + \frac{U_3^2}{2} \right) = 0, \tag{3}$$

where the subscripts $v, L, 3$ signify vapor, liquid, and outlet, $\dot{m}$ is the mass flow rate, U is axial velocity, P is pressure, A is cross-sectional area, h is enthalpy, and WPF is the wall pressure force. WPF is evaluated by assuming the pressure is constant and equal to the vapor phase pressure at the inlet and the end of the convergent mixing section:

$$WPF = P_V (A_1 - A_2) \tag{4}$$

where A_1 is a sum of inlet areas for vapor and for liquid: $A_1 = A_v + A_L$

By re-arranging formulas (1) through (4) above, and calculating nozzle efficiency using generally accepted calculation methodology, the following formulas for final enthalpy and pressure were obtained:

$$h_3 = \frac{m_L}{m} \left(h_L + \frac{U_L^2}{2} \right) + \frac{m_v}{m} \left(h_v + \frac{U_v^2}{2} \right) - \frac{U_3^2}{2} + z \frac{U_3^2}{2} \tag{5}$$

$$P_3 = P_L \frac{A_L}{A_3} + P_v \frac{A_v}{A_3} + \frac{m_L U_L}{A_3} + \frac{m_v U_v}{A_3} - \frac{m_3 U_3}{A_3} - \frac{P_V (A_1 - A_2)}{A_3} - \frac{P_2 (A_2 - A_3)}{A_3} \tag{6}$$

In addition to the above equations of motion, the system must satisfy the second law of thermodynamics; entropy generation must be greater than or equal to zero. In an adiabatic process with no work applied, the entropy balance equation becomes:

$$-\dot{m}_v s_v - \dot{m}_L s_L + \dot{m}_3 s_3 \geq 0, \tag{7}$$

where s is the entropy at the specified location. Knowing the input entropies and mass flow rates and the exit mass flow rate from (1), the inequality sign is replaced with an equals sign, and (7) can be solved for s_3. Once s_3 is known, and using h_3 from (3), the output pressure P_3 can be obtained and compared with the output pressure when entropy is not taken into account. These equations are solved using an iterative method with the Engineering Equation Solver (EES) or F-Chart Software.
The inlet operating conditions used in the analysis are those obtained from experiments, and are shown in Table 1. The properties used in the equations are for R-22.

Table 1. Condensing Ejector Operating Conditions

$T_v = 63.7\,^oC$	$P_v = 1.574\,MPa$
$T_L = 32.4\,^oC$	$P_L = 1.641\,MPa$
$\dot{m}_L = 0.0487\,kg/s$	Fluid = R22

For the operating conditions outlined in Table 1, the ratio of the theoretical output pressure to the input vapor pressure, P_3/P_v, vs. the ratio of the vapor to gas mass flow rates, m_v/m_L was investigated. The analysis indicates that the second law is an active constraint in the design of condensing ejectors and suggests that in a perfect system with zero entropy generation m_v/m_L must be less than approximately 1.25 to obtain a gain in output pressure. The quality of R22, $x = m_v/(m_v + m_L)$, at the output of the control volume should be liquid for maximum efficiency. In order for the output to be liquid phase at the operating temperature shown in Table 1, the ratio m_v/m_L must be less than 0.15.

While observing an experimental run, it was noted that extreme caution was taken to prevent the liquid phase inlet pump from cavitating. Upon further inspection it was noticed that the condensate temperature was very close to the vapor temperature for R22. This prompted an investigation to see what effect the input liquid phase temperature has on the output pressure. In order to examine this, m_v/m_L was set to a typical operating value of 0.3, T_v was set to 64°C, and the liquid phase temperature was varied from 10-33^{o}C. The results of this analysis are shown in Figure 3, where a large increase in output pressure is seen when T_L is decreased. The location of the current operating temperature is marked with an "X" and is seen to be very close to the right edge where the performance of the CE is weakest. This result indicates that the condensate should be cooled before being used in the condensing ejector.

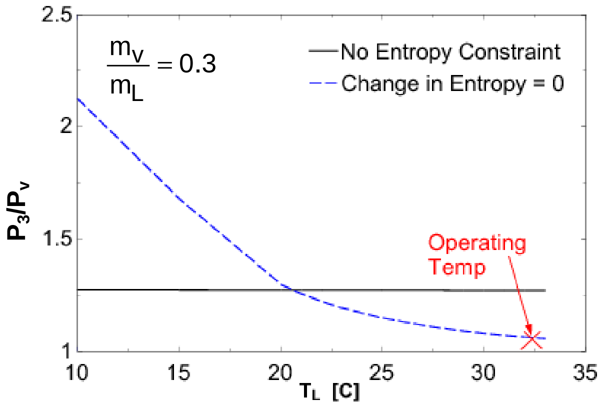

Figure 3: Effect of liquid temperature on output pressure, as constrained by the Second Law. The dashed line represents an upper bound on pressure ratio.

The foregoing discussion is a sample of the analysis achieved with the control-volume assessment of the condensing ejector. The low computational cost and the convenience of this type of calculation have greatly informed our operation of the condensing ejector. For example, we have identified optimum mass flow ratios of liquid to vapor and the optimum area ratio of inlet nozzle to throat area.

2.4 Results of Control Volume Analysis for Closed Refrigeration System

Originally, the mathematical model described above did not consider the fact that the condensing ejector had to work within the constrains of closed refrigeration cycle. In order to further verify the model, we inserted into the formulas values which are achievable in such a closed system, taken from the initial experimental run as shown in Table 2 below. The results were then plotted on an h-p diagram for R22 to determine if full condensation took place and if the pressure rise was indeed achieved. Table 2 shows the inlet data as measured during the experiments on the laboratory stand and further, the resultant final enthalpy and pressure calculated from formulas (5) and (6).

Table 2. CE operating conditions in the closed refrigeration system

Input Data		**Calculated Data**	
Inlet liquid pressure	$P_L = 2.1\ MPa$	Outlet pressure	$P_3 = 2.36\ MPa$
Inlet vapor pressure	$P_V = 1.7\ MPa$	Outlet enthalpy	$h_3 = 273.5\ kJ/kg$
Liquid flow	$m_L = 0.077\ kg/s$		
Vapor flow	$m_V = 0.016\ kg/s$		
Inlet liquid enthalpy	$h_L = 240\ kJ/kg$		
Inlet vapor enthalpy	$h_V = 420\ kJ/kg$		

Plotting the obtained values on h-p graph for R22 it is possible to determine that the fully condensed liquid at elevated pressure appears on the outlet from the ejector as shown in Figure 4 below. Outlet point (P_3, h_3) is located very close to the saturated liquid line. The pressure rise in this case might not be that substantial (12%) but it depends mainly on selected inlet parameters and higher results are possible.

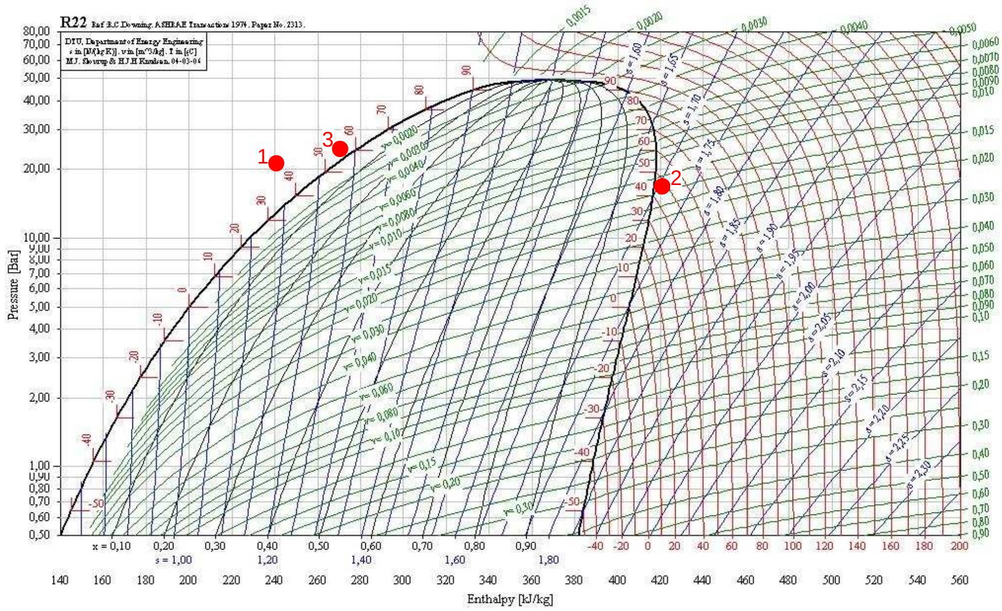

Figure 4. Inlet and Outlet operating points plotted on h-p graph for R22. 1 – liquid inlet, 2 – vapor inlet, 3 – ejector outlet

4.4 Multi-Dimensional CFD Analysis

An entirely new computational fluid dynamics (CFD) code was developed to simulate the condensing ejector. When modeling thermal non-equilibrium, one must first decide whether to employ a full two-phase solution with separate transport equations or a pseudo-fluid approach where the mixture of phases is represented by a continuous density variable. The former approach offers generality, including separate velocities for each phase, while the latter approach offers relative simplicity and expediency. For the initial investigation, the latter approach was employed. The pseudo-fluid approach still allows relative velocity between the phases on the resolved scales. For example, an annular flow might have low speed vapor in an outer sheath around a high speed liquid core. Secondly, no explicit model for interphase drag is required. The numerical problems of very high-drag rates and tight coupling between phase velocities, such as high computational cost and problems with numerical instability, are mostly avoided. The main risk of using the pseudo-fluid approach is that that interphase momentum transfer will be over-predicted and mixing will be suppressed.

Finally, we choose not rely on detailed models of interfacial area, convection coefficient, and temperature field in the turbulent, two-phase heat transfer process. Given the nearly intractable complexity of the detailed heat transfer process, it is preferable to rely on an empirical model that encapsulates the physics in a simple correlation. For this purpose, the Homogenous Relaxation model was employed by Bilicki and Kestin (1990) and Downar-Zapolski *et al* (1996). Like most proposed closures for two-phase channel flow, this model has only been previously investigated in one dimension. The Phase 1 work considers the behavior in multiple dimensions.

The Homogenous Relaxation Model is based on a linearized expansion proposed by Bilicki and Kestin (1990). The general model form has been used by numerous others for one-dimensional, two-phase flow. The model represents the enormously complex process by which the two phases exchange heat and mass. The model form determines the total derivative of quality, the mass fraction of vapor. The key physical phenomenon is the relaxation of the mass fraction of vapor to the equilibrium mass fraction, due to the finite rate of inter-phase heat transfer. The timescale is empirically fit to data describing flashing flow in long, straight pipes.

Much of the effort in Phase I was determining how the model equation should be coupled to basic equations of fluid motion. If the continuity equation is used for solving for mixture density and conservation of momentum is used for velocity, then the model is primarily responsible for determining the pressure.

In contrast to incompressible or low-Mach number Navier-Stokes solvers, the current model does not seek a pressure that projects velocity into consistency with the continuity equation. Instead, we solve for the pressure that satisfies the chain rule and employs the continuity equation indirectly. Through the chain rule, the pressure responds to both compressibility and density change due to phase change. The behavior of pressure is seen to be both hyperbolic and parabolic, while the phase change model appears as a source term.

The code must be able to handle phase change in either direction since, during transients, large pressure waves can bounce around the domain. In any given computational cell, at any moment, it is possible for fluid to condense or vaporize. The application of the vaporization capability made the CFD code developed in this project very attractive to sponsors interested in flash vaporization. Publications by Gopalakrishnan and Schmidt (2008) and Lee *et al* (2008) explain the construction and function of the code in detail.

Some examples of the new code results are shown in the figures below. Figure 5 is a simulation in a short venturi nozzle where the entering flow is a mixture of liquid and vapor above the saturated pressure. The fluid entered from the left boundary and was accelerated through a venturi and then re-expanded to the original diameter, exiting to the right. The two phase mixture then condenses and produces a corresponding increase in pressure. The example below is not quite ideal, since the angle of the diffuser section of the venturi is too large and the flow separates. The separation creates large amount of vorticity, which distorts the condensation shock and prevents the maximum pressure recovery.

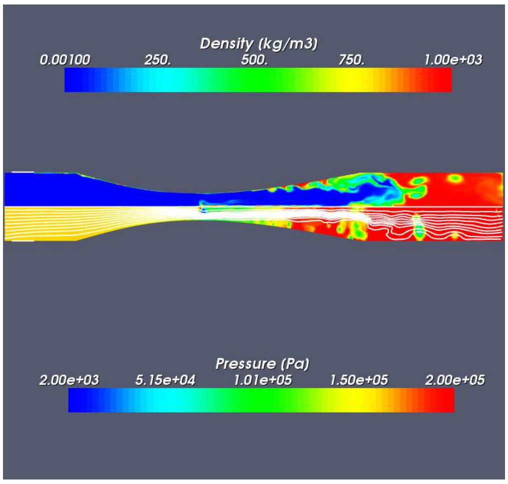

Figure 5. A low density mixture of liquid and vapor accelerates through a venturi and then condenses into liquid, producing a pressure increase. Density is shown in the top half of the nozzle, and the pressure field overlain with streamlines are shown in the lower half. Flow is from left to right.

3. LABORATORY EXPERIMENTS

3.1 Experiments on the Laboratory Test Stand

The design of the two-phase ejector was completed based on theoretical considerations given in Section 2. The previous calculations indicated that under certain conditions the ejector produces a significant pressure rise and a complete condensation. Extensive laboratory experiments were conducted to verify the theoretical model. A special laboratory stand as in Figures 6 was built as a model of the 2.5 Ton refrigeration system. It allowed for varying the flow and thermal parameters in the broad range and comparing the conventional cycle with a cycle equipped with the condensing ejector.

3.2 Test Results

The main objective of our laboratory experiments was to assess the feasibility of non-mechanical compression according to the theory presented earlier. The second objective was to evaluate the behavior of the condensing ejector in a closed refrigeration cycle and to compare the cycle with and without the condensing ejector. In general the results have confirmed the mathematical model – the discharge pressure rise over both suction and motive streams was obtained and the outlet stream from the ejector was a single phase. The key scientific objective was **to obtain the pressure jump on the ejector** and such condition was indeed observed during the first round of experiments.

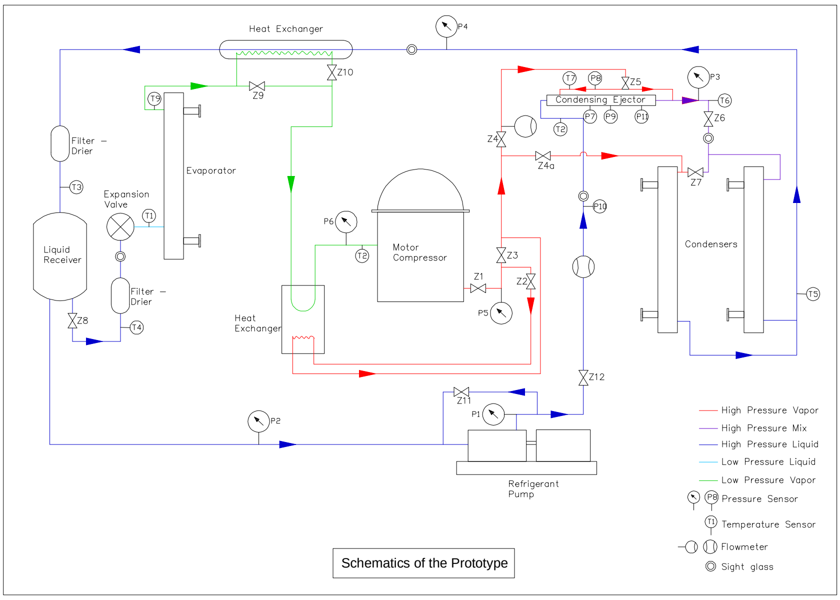

Figure 6. Schematics of modified laboratory stand

As shown in Figure 7, the pressure shock on the outlet of the ejector was achieved when counter-pressure was increased. Such an increase is normally caused in the refrigeration system by temperature rise on the condenser, however in our experiments, we simulated this condition by slowly closing the valve installed on the ejector outlet. In this case, the pressure rise of approx. 7% above the compressor pressure was achieved. During the next round of experiments, we tried to obtain the continuous pressure "shock" by adjusting liquid and vapor flow rates and changing the heat load on evaporator and condensers. This was challenging in our simple experimental stand, but the continuous pressure shock was observed at certain combination of parameters. It is shown in Fig. 8.

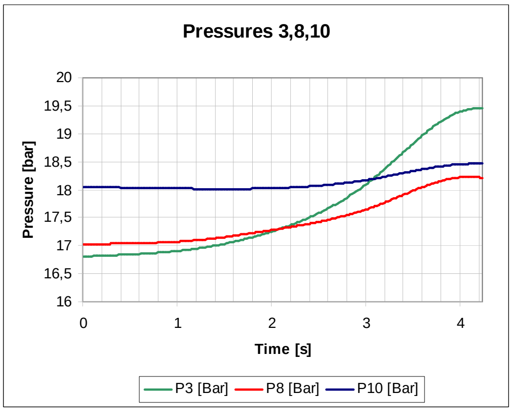

Figure 7. Shows the inlet pressures to the ejector: P8 for liquid and P10 for vapor and outlet pressure from the ejector: P3. The outlet pressure shock is visible when the counter-pressure is

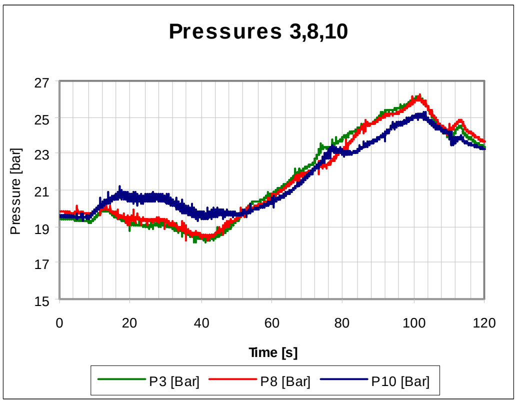

Figure 8. Continuous rise of ejector outlet pressure (P3). As shown, the outlet pressure (green line) is slightly higher than any of inlet pressures and this condition is held for approx. 50 sec.

The condition of Figure 8 was achieved during continuously raising both liquid and vapor inlet pressures. The vapor pressure rise was caused by decreasing flow of cooling water to the condenser and pressure rise of liquid was produced by changing the RPMs of the pump motor. Combinations of pressures and flow rates that produced a rise of outlet pressure were randomly obtained in this stage of research and their reproduction was difficult. However, the fact of achieving such pressure rise confirms the results of the analytical calculations and provides the background for further studies in the next phase of this project.

4. CONCLUSIONS

The objectives of the first phase of this project were met by synergistic theoretical analysis, numerical simulation and laboratory experiments and specifically by: 1) developing the theoretical model that clearly showed the possibility of a pressure rise in two-phase condensing, 2) verifying the model by laboratory experiments and practically demonstrating for the first time the occurrence of a pressure shock and 3) developing the methodology for CE design and successfully applying it in a closed refrigeration cycle.
The reduced-order modeling using control volume analysis was successfully applied and used to perform parameter studies. The calculations were performed with refrigerant properties. The key new observation for proper operation of the condensing ejector was the precise temperature of the incoming liquid in order to be sufficiently cool to provide a sink for the enthalpy of vaporization. Optimum operating conditions for the ratio of vapor to liquid mass flow and optimum nozzle area ratios were also identified.
A new multi-dimensional CFD code has been constructed for simulation of the two-phase flow inside the condensing ejector. The code can operate in two or three dimensions and has been used for simulating both flash-boiling and condensation. The application to flash-boiling is more straight-forward, since macro-scale mixing need not always be resolved. However, a major finding of the Phase I effort is that modeling two-phase mixing will be necessary for correct representation of condensing ejectors. The current model does not discriminate between density changes due to mixing and due to condensation.

REFERENCES

Bergander, M., "Refrigeration Cycle with Two-Phase Condensing Ejector", *11th Intern. Refrigeration and Air-Conditioning Conference at Purdue,* Paper No. R008, Purdue University, July 2006.

Bilicki, Z., and Kestin, J., "Physical aspects of the relaxation model in two-phase flow," *Proc. Roy. Soc. London A. 428* (1990), 379–397.

Downar-Zapolski, P., Bilicki, Z., Bolle, L., and Franco, J., "The Non-Equilibrium Relaxation Model for One-Dimensional Flashing Liquid Flow," *IJMF 22,* 3 (1996), 473–483.

Fisenko, V.V., "S'zhimayemost Teplonosityela i Effektivnost Raboty Konturov Tsirkulatsiy Ya.E.U.", (book in Russian), *Energoatomizdad*, Moscow, 1987, 196 p.

Gopalakrishnan S. and Schmidt David P., "A Computational Study of Flashing Flow in Fuel Injector Nozzles," *to appear in* SAE Congress, 2008.

Lee J. C., Madabhushi R. K., Fotache C., Gopalakrishnan, S., Schmidt D. P., "Flashing Flow of Superheated Jet Fuel," *in review,* Proceedings of the Combustion Institute, 2008.

Levy, E. K. and Brown G. A., "Liquid-Vapor Interactions in a Constant-Area Condensing Ejector," *Trans. ASME,* v. 94, ser. D, n. 1, March 1972.

Van Wijngaarden, L., "One-Dimensional Flow of Liquids Containing Small Gas Bubbles", *Annual Fluid Mechanics*, 1972, vol. 4, pp. 369-396

ACKNOWLEDGEMENT

This material is based upon work supported by the Department of Energy, under Award Number DE-FG36-06GO16049 and by the National Science Foundation, STTR Phase I Project No. 0610613.

DISCLAIMER