Chapter 1

Introduction

1.1 Background

The research presented here is directed at the ejector expansion refrigeration cycle (EERC), a method of increasing the performance of the vapor compression cycle by using an ejector instead of a throttling valve. This work was part of a project funded by the Dept. of Commerce - Advanced Technology Program to develop and commercialize the EERC. The Mechanical Engineering Dept. at Virginia Tech, Calmac Corp., a manufacturer of refrigeration equipment, and NIST participated in the several phases of this project. It grew out of earlier work on ejector cycles that had already been done at Virginia Tech by Kornhauser [1] and Menegay [2].

1.1.1 The Ejector Expansion Refrigeration Cycle (EERC)

Vapor compression refrigeration systems typically employ expansion valves or other throttling devices to meter refrigerant into the evaporator. They also provide sufficient pressure drop so that the refrigerant will evaporate at some desired low temperature. It is well known, however, that throttling wastes available energy. The process is isenthalpic, which implies that the kinetic energy developed as the fluid pressure is reduced is eventually dissipated. A method of recovering this kinetic energy is clearly desirable.

One solution to this problem is the EERC, a modification of the standard vapor compression cycle in which an ejector is used instead of a throttling valve to recover some of the kinetic energy of the expansion process. As shown in Figures 1.1 and 1.2, saturated liquid from the condenser enters the motive nozzle of the ejector and expands to a high velocity two-phase jet. The suction stream is also expanded in a nozzle since doing so decreases slip between the two streams and consequent losses. The motive and suction streams combine in a mixing section to produce an outlet velocity between that of the entering jets. The mixed stream enters the diffuser where its kinetic energy is converted into a pressure boost. At the diffuser outlet a separator ensures that liquid returns to the evaporator and gas to the compressor. Through the action of the ejector the compressor suction pressure is higher than it would have been in a standard cycle. Thus the compressor does less work, resulting in an energy savings.

Analysis of EERC performance by Kornhauser [1] indicates that COP improvements of up to 21 percent over the standard cycle are possible. This result was calculated assuming ideal components, R-12 as the working fluid, and a standard evaporating temperature of 5 F (-15 C) and condensing temperature of 86 F (30 C). Although 21 percent is an upper bound for these conditions, the performance gain using non-ideal components is still good if the ejector efficiency is reasonable. For instance, using typical single-phase efficiencies, a COP improvement of ~10 percent can be expected. Unfortunately current design methods have not produced ejectors of high efficiency and the best experimental COP improvement seen is only 3.8 percent (Menegay and Kornhauser [3]).

1.1.2 Ejector Design and Flow Characteristics

The current method of ejector design (see Kornhauser [1]) for EERC service is based on the one-dimensional homogeneous equilibrium model (HEM). Since this model yields a motive nozzle throat area which is oversized, an empirical model which accounts for metastability is subsequently used to correct the throat to a smaller size. The motive nozzle outlet area is unaffected by this correction and retains its HEM size. The design

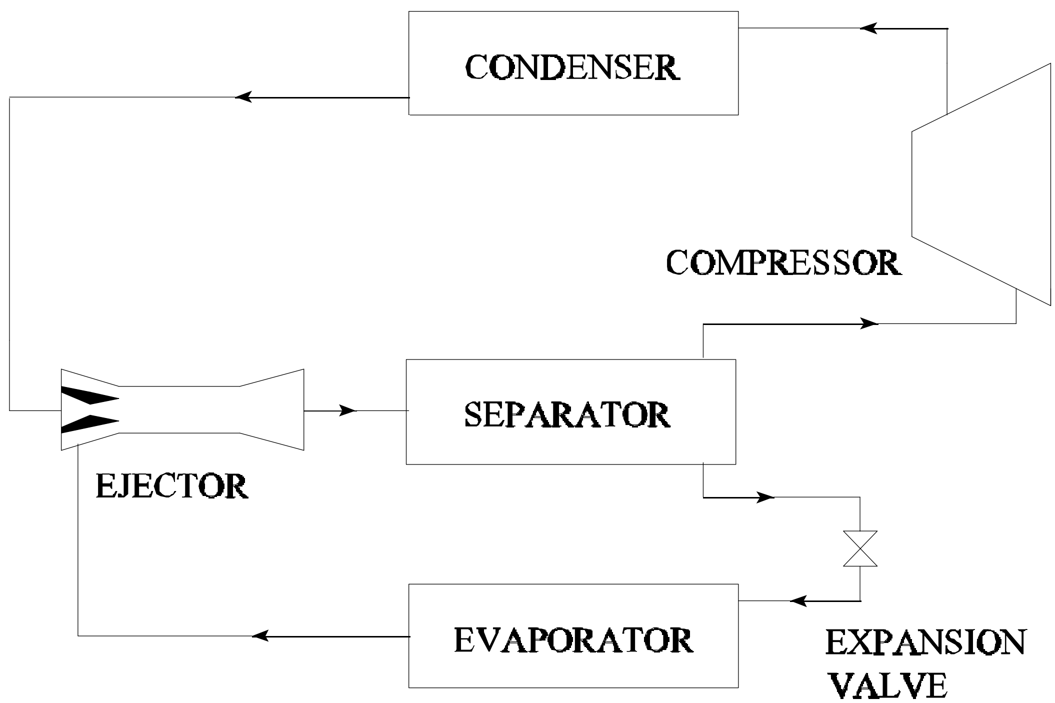

Figure 1.1 - Schematic of EERC

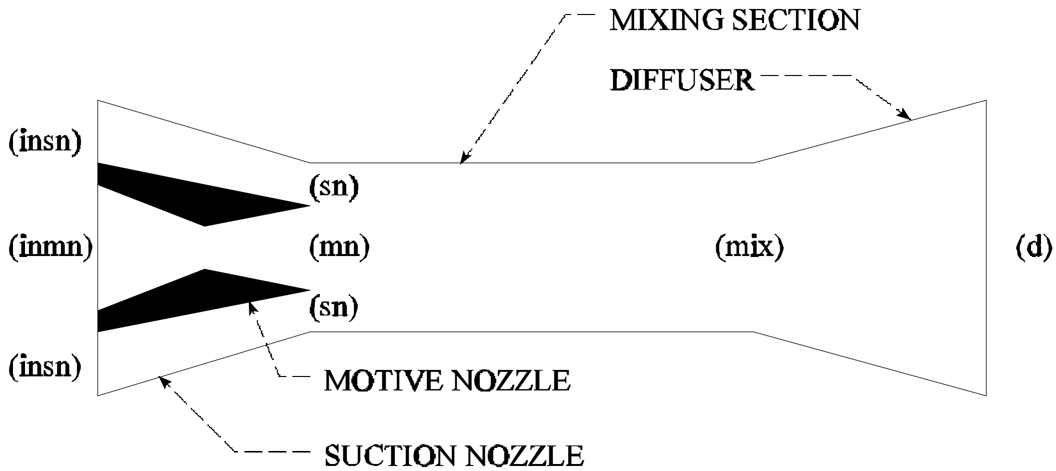

Figure 1.2 - Close-Up View Of Ejector

proceeds by sizing a constant area mixing section. Another option, constant pressure mixing, is not used because, according to HEM modelling, it gives slightly lower performance. The HEM model is thus used to size all ejector components except the throat area of the motive nozzle.

The HEM model is chosen primarily for its simplicity although it is well known that it does not accurately represent the complex flow within the ejector. This idea is confirmed by experimental results. In one case, for example, a laboratory vapor compression system retrofitted with an HEM designed ejector yielded a COP improvement of 3.6 percent over the standard cycle. A theoretical calculation reveals that for the same conditions using an ideal (isentropic) ejector, a 16.2 percent improvement is possible. Since ejector inefficiency is the only reason for the difference in these results, it appears that the HEM design method is inadequate.

As noted above, this inadequacy rests with the fact that the HEM model poorly represents the actual ejector flow. To illustrate this point, and to provide a scope for any modeling effort, it is instructive to list a step by step account of the flow within the ejector:

1. Slightly subcooled liquid from the condenser is throttled a small amount into the two-phase region. Downstream of the valve, a small diameter tube induces enough turbulence to mix the two-phase stream into a highly dispersed bubbly flow. The bubbles provide nucleation sites to reduce subsequent metastability in the motive nozzle. This idea has been shown experimentally to yield better performance despite the penalty associated with throttling upstream of the motive nozzle.
2. The bubbly flow enters the motive nozzle and expands rapidly with flashing taking place and the bubbles growing as the pressure decreases. Calculations using flow regime maps reveal that at some point the flow changes from bubbles in liquid to

droplets in vapor, with some transitional regime in between. The motive nozzle outlet is thus a high velocity jet of such droplets. Because of the speed at which the expansion takes place it is expected that a high degree of thermal non-equilibrium is present.

3. The motive jet entrains the slower suction jet and mixes with it. This mixing process is characterized mainly by hydrodynamic non-equilibrium since the droplets are initially moving much faster than the entrained vapor. Eventually momentum transfer between the two streams tends to equalize their velocities, and results in pressure rise. The suction nozzle itself simply expands superheated vapor and so is not expected to pose any difficulties in modelling. Evaporation of the droplets continues in the mixing section.
4. The mixed droplet-vapor stream enters the diffuser where a pressure rise takes place. Evaporation is also expected in the diffuser so some lesser amount of thermal non-equilibrium may be present here as well.

Given these generally unusual (and poorly understood) flow characteristics, a two-phase flow model is required which can handle unequal temperatures and velocities between the phases. Thus, it should also predict values for interphase heat, mass, and momentum transfer. It must incorporate a method to find the size of bubbles and droplets, and be able to predict the volume fraction of each phase. It should also account for two other important effects, compressibility and turbulence.

1.1.3 Project Objective: The Two-Fluid Model

The two-fluid model appears better suited to these requirements. The model is an application of CFD in which each fluid of the two-phase problem is treated separately. Thus, the two-fluid model inherently handles thermal and hydrodynamic non-equilibrium.

The two-fluid problem is solved by first writing the two-phase versions of the continuity, momentum, and energy equations in differential form. Equations for the interfacial transfer of mass, momentum, and energy are also derived and added to the basic conservation equations in the form of source terms. In addition to the basic equations, models for turbulence, type of flow regime, droplet/bubble size, and physical properties of the refrigerant are used. After appropriate transformation of coordinates, the equations are discretized and programmed into a computer routine.

The resulting program can then be used to conveniently carry out numerical simulations. By varying ejector geometry and inlet flow conditions the user can easily generate a wide array of output such as pressure at the diffuser outlet, losses due to wall friction, velocity profiles, etc. This output can provide guidelines for future ejector designs.

It should be noted from the outset that the objective of this project, as stated above, has not yet been fully achieved. Time constraints and the realization that the flow is more complicated than expected have delayed the project. Most importantly, this means that the energy equation is not solved by the code. Stemming from this limitation, interfacial heat and mass transfer, and compressibility are also not present. Thermo-physical properties are assumed constant and the flow regime is constant (droplet flow) since the code is only capable of solving the flow in the mixing section. Furthermore, the droplet/bubble size is guessed although simple algebraic models were attempted which failed to give consistent results.

Although the objective of this project has not been fully achieved, the work shown in this dissertation provides the foundation for its eventual completion. The present work contributes in two ways to the literature on two-phase flow:

1. It provides a code which for some limited cases can be used to simulate ejector flow, particularly in the mixing section. Furthermore, it is structured in a manner well suited to the addition of new features, such as the energy equation. The basic models for most of the new features have already been developed or adapted from the literature, as will be seen in later chapters. Thus, the remaining work consists mostly of programming subroutine additions to a working code.
2. It gives details about several decisions which were made regarding two-fluid modelling which would be useful to anyone attempting to write such a code for his/her own specialized application. The remainder of this work will elaborate on both of these contributions.

This dissertation describes the modelling, programming, and results of a two-fluid model for two-phase, non-equilibrium ejector flow. It is based on a great deal of past work on both the EERC concept and the general two-phase modelling problem. It is useful to review the history of these topics first.

1.2 Previous Work

1.2.1 Work on EERC

The ejector cycle as shown in Fig. 1.1 was invented by Gay [4] in 1931. Modifications to the basic cycle were patented by Kemper et al. [5] in 1966. Two patents by Newton [6,7] in 1972 were made for control systems for the ejector cycle and obviously made use of a detailed physical understanding of the fluid mechanics in the ejector itself. Unfortunately, this early work is only available in the form of the patent documents and no experimental or theoretical background seems to have been published.

Following up on this early work, Kornhauser [1] did a theoretical analysis in which he showed the ideal 21 percent improvement (mentioned above) of the ejector cycle over a standard cycle. Encouraged by these results, Menegay [2] built and tested an experimental ejector cycle but found that actual performance was much less than the ideal. Menegay's ejector was designed using the HEM model with a simple two-phase flow model by Henry and Fauske [8] to size the motive nozzle throat. The control system for the cycle consisted of bypassing hot gas around the condenser and injecting it upstream of the motive nozzle, an approach which controlled the system but lowered its COP. Kornhauser and Menegay [9], realizing that seeding the flow with a highly turbulent bubbly flow could reduce motive nozzle metastability (suspected as one reason for low performance), patented a valve and small tube arrangement to accomplish this. Experimental tests of an ejector equipped with this invention yielded better performance but still not as good as was expected (Menegay and Kornhauser, [3]). Apparently, more accurate design methodologies for two-phase ejectors were required.

One of the problems with Menegay's [2] experimental apparatus was that it could only measure an overall efficiency for the ejector and could not distinguish between efficiencies of the motive nozzle, mixing section, and diffuser. Alexandrian [10] and Bunch [11] (see also Kornhauser, et al. [12]) began correcting this problem by calculating the efficiency of the motive nozzle alone by measuring how much thrust it produced. Although their results indicated apparent efficiencies well below single-phase nozzles, Bunch believed that slip between the phases was likely, which would make the actual efficiency higher than what was predicted from thrust measurements. Thus the basic fluid mechanics inside the motive nozzle were still not known with certainty.

The significance of slip in the motive nozzle was investigated by Harrell [13] who developed a dedicated test rig for measuring ejector performance independently of the constraints imposed by a connected refrigeration cycle. He has found the best ejector performance to date although there was probably no design improvement per se in this ejector since the design methodology remained the same as what was used earlier. It is believed that his ejector yielded better performance mainly because it is physically larger than the one used previously and thus has less frictional loss and less thermal non-equilibrium. Harrell has also supported Bunch's assertion that slip must exist in the motive nozzle, based on comparisons between his data and a one-dimensional computer model of the mixing section that he has developed (see below).

1.2.2 Two-Phase Flow Modelling

Although the problem of two-phase ejector flow for EERC systems has been known for years, no model has been found which deals with it specifically. Indeed it is believed that the lack of such a model and the consequent design methodology which has kept the EERC concept from successful commercial development. However, a number of models for other applications and for general use exist that deal with non-equilibrium two-phase flows.

A great many early two-phase flow models are available, motivated mainly by a desire to find the maximum flowrate of water after a loss of coolant accident (LOCA) in a nuclear reactor. One example is Starkman et al.'s work [14] in which three models (HEM, frozen, and slip flow) were developed to predict the expansion of low quality water in a converging-diverging nozzle. A review by Saha [15] contains detailed information about these models, as do reviews by Wallis [16] and Ardron and Furness [17]. Although many of them account for thermal and hydrodynamic non-equilibrium, their lack of generality makes them difficult to apply accurately to conditions other than the ones for which they were tested. Indeed, one such model (Henry and Fauske [8]) that was used to size the throat of the motive nozzle in Menegay's work [2] still resulted in an area that was too large. In general, these models rely heavily on empirical data for a specific problem rather than fundamental principles and are hence not generally applicable.

Later, still motivated by the LOCA scenario, more computationally intensive models appeared. Richter [18] wrote a computer code based on one-dimensional flow through a short tube or nozzle. He used a separated flow model in which the conservation equations were written separately for each phase (similar to the two-fluid model used here). Similar models were developed by Dobran [19], Ardron [20], and Schwellnus and Shoukri [21]. Many of the basic principles of one-dimensional two-phase flow used in these models were explained in a textbook by Wallis [22]. However, none of the models discussed by these authors was considered for use in the present investigation because they were one-dimensional. Their value lies in the information they provide about modelling interfacial interaction terms.

Harrell [23], as part of the overall EERC project, wrote a computer program based on a one-dimensional model for the flow in the ejector mixing section. The model divides the flow into a jet core of droplets and vapor, and an annular suction flow. Friction and heat transfer are allowed between these two regions. His model predicts the pressure rise in the mixing section and diffuser.

The basic equations for general, multi-dimensional two-fluid models have been derived in several sources. Ishii and Mishima [24] wrote two-fluid differential conservation equations using time averaging. They also derived several interfacial terms for various flow regimes of interest in nuclear reactors. In addition, Ishii [25] wrote a lengthy, mathematically rigorous derivation of the conservation equations for two-phase flow. Lahey and Drew [26] performed a similar two-fluid derivation using ensemble averaging. They paid particular attention to satisfying the second law of thermodynamics, a principle which they claim is ignored in other models. Dobran [27,28] presented a theory of multiphase mixtures and derived conservation equations using a volume averaging procedure.

A great many CFD models have been published which deal with multi-dimensional, multiphase flow problems. None, though, is available which deals specifically with two-phase ejector flow. However, several multi-purpose CFD codes exist which could conceivably be used with modifications to solve the problem of interest here.

1.2.3 Currently Available Computer Programs

Given the effort required to numerically model two-phase, non-equilibrium ejector flow, it seemed prudent to explore the use of an already available code. Although such an approach was ultimately rejected, the information learned from considering this option provides a review of these codes. As noted in the following discussion, these codes were rejected primarily because of technical deficiencies and costs.

For the EERC ejector flow problem, three general classes of codes/models can be listed which could be used for a solution: (1) Commercially available general purpose computational fluid dynamics (CFD) codes which have two-phase capability, (2) Codes which model loss-of-coolant accidents (LOCA) in nuclear reactors, and (3) Other two-fluid models for specific applications which could be modified for this application. A description of each class and why it is not being used for this project will be described.

A number of commercial CFD codes are now available and claim the ability to solve a wide range of flow problems, including multiphase flows. For instance CFD-TWOPHASE, a code developed by CFD Research Corporation, is advertised as being capable of any manner of interphase interaction (heat, mass, and momentum transfer), a wide range of flow regimes, several turbulence models, compressibility, and two or three-dimensional coordinates. Although these features are attractive, its $17,400/yr. licensing cost (Sukumar [29]) is prohibitive. Universities can license for $1,000/yr. but the commercial customer of the EERC project (Calmac Corp.), who would eventually need the code, would have to pay the full amount. The code is also expensive computationally since it solves an elliptic set of equations. Furthermore, since only an executable format and not the source code is available, any modifications required for this problem would be difficult to make. Similar difficulties exist for other commercial CFD codes such as PHOENICS, CFDS-FLOW3D, FIDAP-7, FLUENT [30], etc. Although not mentioning any code in particular, some reviewers are in general very suspicious of the claim that a code can handle multiphase flow (Foumeny and Benyahia [31]). It appears that these codes are good investments for an engineering group with a sizable number of diverse and more conventional projects.

Among the codes which model LOCA's, some well known examples are RELAP5, TRAC, WCOBRA/TRAC, and COMMIX-2. Although they handle non-equilibrium two-phase flow, there are a number of problems with them. Arnold et al. [32] finds that they do not model turbulence effects, the energy equations neglect interfacial work, and viscous effects are neglected. Also, these codes are written especially for nuclear reactors and particularly for the problem of determining the maximum flowrate from a ruptured coolant line, a problem somewhat different from ejector flow. The LOCA codes, like the commercial CFD codes, also suffer from large size and long computational run times.

A number of two-fluid models are available in the literature for the solution of specific problems. Two such models which were studied in some detail are described below and serve to illustrate the general applicability of other models.

Dobran [33] created and programmed a two-fluid model for a two-phase, nonequilibrium free jet. The model neglects viscosity (although includes interfacial drag) and seems to neglect turbulence. The code which Dobran wrote based on this model is not available and, even if it were, would require significant modification to handle the ejector geometry. It was therefore dropped from further consideration, although it yielded some insights into the two-fluid model formulation.

Spalding [34] wrote a two-fluid code named GENMIX2P in 1978. This code is of reasonable size and is parabolic, requiring much less time and computational resources than the codes described above. It was designed to handle combustion problems involving a motive jet of fuel entraining air, so its geometric capability is very similar to that of an ejector. Furthermore the source code listing is available in the open literature. GENMIX2P initially seems like an ideal solution, but closer inspection reveals some serious problems. First, it is unable to handle interfacial mass transfer. Although this feature could be added to the code, doing so is complicated by its use of streamline coordinates. Also, the code has many bugs since it was never fully tested. An attempt by this writer to make the code work for other than its default case proved time consuming and led to the general conclusion that if a ready made code cannot be used as is or easily modified, it is probably not worth considering.

Spalding has since replaced GENMIX2P with PHOENICS, one of the major commercial CFD packages. Although this code was not used directly for the reasons mentioned above, Spalding's interphase slip algorithm (IPSA) [34] was used as the basis for handling the coupling between the phases in this work. The IPSA procedure, as implemented here, will be described in detail later.

Based on this information, an in-house program to model ejector flow would be a more effective approach than relying on an already developed model or code. Such a program would also be tailored to the particular problem of EERC ejector flow and would thus be optimized for efficiency and accuracy.