Chapter 6
Conclusions and Recommendations

6.1 Conclusions

The objective of this project was to produce a CFD code for non-equilibrium two-phase ejector flow. The code was to be used as a design tool to improve ejector performance in EERC service. This objective has been only partially fulfilled due to time constraints and the realization that the flow is more complicated than was originally thought. However, a number of conclusions can be drawn about both ejector design and two-phase modelling which are useful to future workers in either of these fields. Beginning with two-phase modelling, the conclusions which are most significant are the following:

1. The two-fluid approach to solving this problem has yielded promising initial results, and appears to simulate correctly the basic physics of the flow. It works well in predicting pressure profiles for non-recirculating conditions in the ejector mixing section. It also gives qualitatively good velocity and void fraction profiles, although there is no experimental data to quantitatively verify these results. Furthermore, it can be used to determine if recirculation is likely, and it appears to be able to approximate the location at which recirculation begins.
2. The interfacial drag model used works well when the droplet diameter is properly adjusted.
3. The droplet diameter, through its influence on the interfacial drag force, was the single most important variable used in matching the pressure rise in the mixing section. Unfortunately, no experimental data was available to confirm the size used here. Furthermore, semi-empirical equations from the literature for

determining droplet size yield highly inconsistent results, so the droplet size used for this project was essentially just a guess.

4. Assuming a single droplet size does not work because it leads to unrealistic liquid velocity profiles and is physically inconsistent with the negligible amount of liquid present in the annular region. Assuming that the droplet size varies linearly with liquid volume fraction produces more realistic liquid velocity profiles throughout the radial domain, although the linear relationship used was made more for the sake of convenience than for any physical reason.
5. The mixing length turbulence model works well in this application, despite its simplicity compared to newer methods. However, the turbulence constants used in this model had to be modified from the textbook values in order to match the shape of the experimental pressure rise curve. These constants were the second most important variable in obtaining a match to the experimental pressure profile.
6. The use of wall functions for reducing the required fineness of the computational grid works very well in single-phase flow and appears to work well for two-phase flow. It reduces computer time requirements by several orders of magnitude over simply using a uniform fine grid throughout the domain. It needs, however, to be tested against results for a fine grid, non-wall function method to be certain of its applicability. It's accuracy in the presence of high interfacial drag should be especially scrutinized.
7. The parabolic approximation is justified, at least in the mixing section, since pressure and velocity gradients in the radial direction are much smaller than in the axial direction.
8. The IPSA method works but produces a difficult and slow convergence for the vapor void fraction.
9. The code, in general, experiences difficult convergence in many cases. This may have to do with the large number of iteration loops present in the computer program. Adjustments of underrelaxation factors, error tolerances, and some

physical variables such as turbulence constants, needed to be made to produce convergence. For several practical cases the code did not converge at all.

The conclusions pertaining to ejector design rely heavily on the work of Harrell [23], who produced extensive experimental data on ejector performance. These conclusions are:

1. Slip at the motive nozzle outlet was higher than expected. This led to the importance of modelling correctly the interfacial drag, which in turn required an accurate estimate of droplet diameter. High slip justifies the use of the two-fluid approach, rather than a mixture modelling approach, because much of the pressure rise comes from the equilibration of the unequal velocities between the phases.
2. Recirculation occurs in the mixing section of the ejector, unless the section is artificially reduced in diameter to increase suction velocity. The tendency toward recirculation is much higher in two-phase flow than single phase flow. This poses both a modelling and design problem since the code cannot handle recirculation and because this effect is so detrimental to ejector performance. If recirculation occurs in the mixing section it will, without doubt, also occur in the diffuser.
3. It is possible that a shock-like phenomenon is occuring inside the motive nozzle known as the flow regime transition shock. This theory gives a physical argument for why high slip is occuring at the nozzle outlet. The code would need modifications to handle this phenomenon.
4. The spread of liquid from the droplet jet core emanating from the motive nozzle proved to be almost negligible, according to CFD code results. This result is consistent with Harrell's experimental observations of the jet mixing region through a glass section. The minor amount of wall wetting that Harrell also noticed may have been due to recirculation.

6.2 Recommendations

The recommendations for future work stem mostly from the conclusions listed above. Some consist, however, of tasks which were original objectives of this project which went unfulfilled but are deemed still worth pursuing.

1. An experimental investigation should be undertaken to find out what the droplet size is and, if possible, what its distribution is. This would possibly involve using high speed photography through a glass mixing section.
2. In conjunction with the above recommendation, a droplet size model should be developed using local turbulence levels, shear forces, etc. The model should be incorporated into the CFD code after testing and tuning it against experimental droplet size observations.
3. The code's capabilities should be extended to handle small regions of recirculation with the current parabolic method. Certain inverse techniques are available (Anderson, et al. [52], p.365) to compute through small recirculation zones using parabolic methods. This would allow comparison of the code with experimental data for recirculating cases. It would also allow CFD analysis of the diffuser, a component which most probably has exhibited recirculation in every experimental run taken.
4. The energy equation, compressibility, and interfacial mass transfer should be incorporated into the code. Run the code for the motive nozzle and check results against the experiments of Alexandrian [10] and Bunch [11] and the results of one-dimensional codes for the flow regime transition shock.
5. The modifications required to handle the flow regime transition shock should be studied. This would possibly involve time marching techniques to handle supersonic flow upstream of the shock and the inclusion of the energy equation into the computer algorithm.

6. The use of a more efficient numerical solution method should be investigated. The present routine is slow and cumbersome since it contains a large number of iteration loops. It is possible that a more direct solution is available by placing the discretized equations into a large matrix which is then solved numerically.
7. The use of the wall function for two-phase flow should be studied further. In particular, it is recommended that it be compared to the fine grid case where no wall function is used to ensure that the assumed velocity profile near the wall is accurate. This study should assume substantial amounts of liquid near the wall, not the negligible amounts actually found here.
8. The code should be tested against the large body of experimental data now available with a small diameter, non-recirculating ejector. Only one case was run and shown here because of time constraints. Determine, through this testing, if the droplet diameter and turbulence parameters are constant, or must be varied in order to match the pressure profile.
9. Code results should be obtained using a radial velocity profile at the outlet of the motive nozzle which accounts for the diverging walls of the nozzle. The effect of the radial profile on liquid distribution in the mixing section should be investigated.
10. The turbulence model should be modified to handle non-uniform inlet velocity profiles. This is because the routine that determines boundary layer thickness can only do so for flat inlet profiles. The effect of non-uniform profiles on the liquid distribution should be checked.