International seminar on ejector/jet-pump technology and applications

Louvain-la-Neuve, Belgium
September 7-9, 2009

Paper No. 34

MULTIDIMENSIONAL MODELING OF CONDENSING TWO-PHASE EJECTOR FLOW

David Schmidt and Michael Colarossi
University of Massachusetts Amherst

Mark J. Bergander
Magnetic Development Inc.

ABSTRACT

Condensing ejectors utilize the beneficial thermodynamics of condensation to produce an exiting static pressure that can be in excess of either entering static pressure. The phase change process is driven by both mixing and interphase heat transfer. The complete details of the fluid flow are not well understood and cannot be calculated directly. However, semi-empirical models could be used in conjunction with computational fluid dynamics (CFD) to gain some understanding of how condensing ejectors should be designed and operated. The current work describes the construction of a preliminary multidimensional simulation capability that is built around an Eulerian pseudo-fluid approach. The transport equations for mass and momentum treat the two phases as a continuus mixture. The rate of mass transfer is assumed to be limited by heat transfer, and a modified form of the Homogenous Relaxation Model (HRM) is employed. This model was originally intended for representing flash-boiling, but the authors hypothesize that with suitable modification, the same ideas could be used for condensing flow. The computational fluid dynamics code is constructed using the open-source OpenFOAM library, which facilitates object-oriented fluid flow calculations. The resulting code is a transient, finite-volume code that can operate in either two or three dimensions, though two-dimensional calculations appear to be much more numerically stable. The simulations are parallelized using a standard message passing interface (MPI) combined with domain decomposition. The simulation can use general polyhedral cells, for flexibility in geometry. Fluid properties are evaluated using the REFPROP database from NIST, which includes many common fluids and refrigerants. Development is currently underway for handling multi-component fluids. The results of the simulations will allow the study of the flow within the two-phase condensing ejectors. Of particular interest are the stability of the condensation process and the permissible boundary conditions. The model can also be used to study flow separation and consequent diffuser stall that may occur in the diverging portion of the ejector. Due to the strong adverse pressure gradients that coincide with condensation, extreme care is warranted in avoiding such recirculations.

NOMENCLATURE

h	Enthalpy (J/kg)
p	Pressure (Pa)
p_{crit}	Critical pressure (Pa)
p_{sat}	Saturated pressure (Pa)
t	Time (s)
U	Velocity (m/s)
x	Vapor mass fraction
$\bar{x}$	Equilibrium vapor mass fraction
α	Vapor volume fraction
Θ	Timescale (s)
ρ	Density (kg/m^3)
τ	Stress tensor (Pa)
φ	Mass flux (kg/m^2*s)
ψ	Dimensionless pressure difference

INTRODUCTION

The main objective of this project is to model the flow inside a condensing ejector, including the condensation shock that occurs, 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.

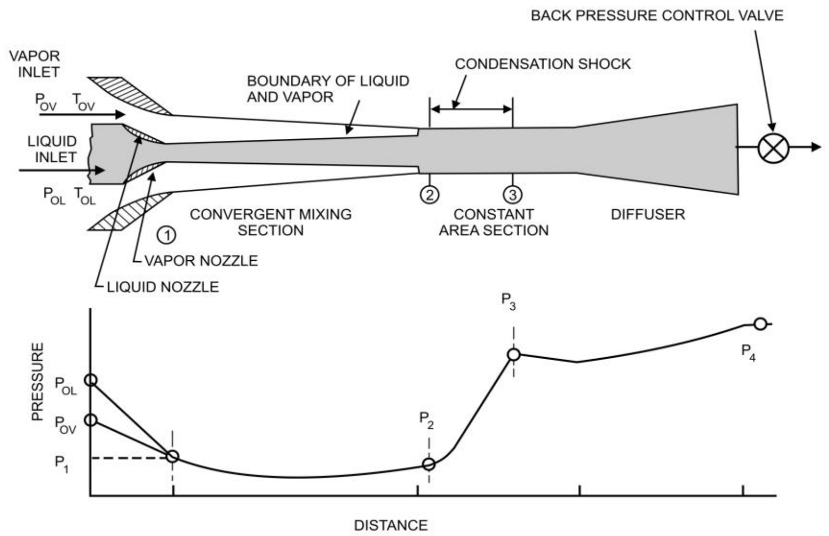

Figure 1. Principle of a condensing ejector

The condensing ejector is a two-phase jet device in which a sub-cooled fluid 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. Figure 1 above presents the predicted behavior of a condensing ejector. 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).

A computational fluid dynamics (CFD) analysis was employed to overcome the limitations of a conventional control volume analysis and to provide an input to design methodology of the condensing ejector. The simulation method is needed to predict the behavior of the condensing ejector in various operating conditions and for different refrigerants. This provides the basis for establishing fundamentally new engineering and design methods for a broad spectrum of equipment operating on two-phase flow, not only condensing ejectors but also pumps, heat exchangers, jet compressors, supersonic nozzles and diffusers.

MODELING APPROACH

For a condensing flow simulation, the basic laws of conservation are followed as for a single fluid. This is known as the "pseudo-fluid" approach. The equations for the conservation of mass, momentum, and energy are given below. In these equations, ϕ is the mass flux and τ is the stress tensor.

$$\frac{\partial \rho}{\partial t} + \nabla \phi = 0 \qquad (1)$$

$$\frac{\partial \rho U}{\partial t} + \nabla(\phi U) = -\nabla p + \nabla \vec{\vec{\tau}} \qquad (2)$$

$$\frac{\partial \rho h}{\partial t} + \nabla(\phi h) = \frac{\partial \rho}{\partial t} + \vec{U} \cdot \nabla p \qquad (3)$$

These three equations are not a closed system. For a fluid in single-phase, an equation of state would be necessary. In the case of two-phase flow, the condensation process involves mixing and heat transfer between the phases, and there is no direct method to calculate this. Also, the two-phase mixture is not in thermodynamic equilibrium. To bring the mixture towards equilibrium, a modified form of the Homogenous Relaxation Model (HRM) is used. The model was originally used for flash-boiling cases, but with some modifications the same idea can be applied for condensing flow.

The Homogeneous Relaxation Model is based on a linearized expansion proposed by Bilicki and Kestin [1]. The equation for the total derivative of quality, the mass fraction of vapor, is shown below. The derivative shows the relaxation of the quality, x, compared to the equilibrium quality, $\bar{x}$, over the timescale Θ.

$$\frac{Dx}{Dt} = \frac{\bar{x} - x}{\Theta} \qquad (4)$$

The formula suggested by Downar-Zapolski et al. [2] for the timescale, Θ, is shown below in Eqn. 5.

$$\Theta = \Theta_o \alpha^a \psi^b \qquad (5)$$

The values for Θ_0, a, and b were determined by best-fit values to flashing experiments [2]. The values are $\Theta_0 = 6.51 \cdot 10^{-4}$ [s], a = -0.257, and b = -2.24. The variable α is the vapor volume fraction and the variable ψ is a dimensionless pressure difference represented in Eqn. 6. Though the current application is far beyond the intended use of this empiricism, it serves as a starting point for further investigation.

$$\psi = \left| \frac{p_{sat} - p}{p_{crit} - p_{sat}} \right| \tag{6}$$

The HRM model solves for the pressure that satisfies the chain rule and utilizes the continuity equation. Because a non-equilibrium fluid is being considered, the chain rule takes the place of an equation of state. In the chain rule, the pressure reacts to the change in compressibility and density caused by the phase change. Eqn. 7 describes the chain rule as a total derivative of density.

$$\frac{D\rho}{Dt} = \left.\frac{\partial \rho}{\partial p}\right|_{x,h} \frac{Dp}{Dt} + \left.\frac{\partial \rho}{\partial x}\right|_{p,h} \frac{Dx}{Dt} + \left.\frac{\partial \rho}{\partial h}\right|_{p,x} \frac{Dh}{Dt} \tag{7}$$

As shown in the above equation, density is a function of pressure, quality, and enthalpy [1]. The first and last terms on the right-hand side of Eqn. 7 are currently not included.

The HRM model is an empiricism that has been tuned to represent flash boiling. The current application to rapid condensation represents a stretch of the model beyond its original intent, and quite possibly, beyond its validity. One should certainly expect that some adjustment of the model is required. In the current application, the following changes were made to the HRM model:

- A turbulent mixing model based on a single length scale was added to the transport equations for momentum, energy, and density (Eqn. 7) [3]
- An extra term was added to Eqn. 5 to provide symmetry of phases. According to Eqn. 5, the timescale of phase change becomes infinite at zero void fraction. This behavior represents the effect of nucleation. A similar behavior should be present for the case of unity void fraction.
- The timescale was bounded, both above and below, in order to avoid numerical instability and floating point exceptions.

By subtracting the conservation of mass, Eqn. 1, from Eqn. 7, the left side gives an expression for the velocity divergence at the new time step.

$$-\rho \nabla \cdot U = \left.\frac{\partial \rho}{\partial p}\right|_{x,h} \frac{Dp}{Dt} + \left.\frac{\partial \rho}{\partial x}\right|_{p,h} \frac{Dx}{Dt} \tag{8}$$

To make an equation for pressure, a discretized form of the momentum equation is coupled with the chain rule. A generalized form of the momentum equation is shown below in Eqn. 9, where a_p is the coefficient term of the matrix of velocity equations and H represents convection and diffusion as discretized equation coefficients multiplied by neighboring velocities. The pressure equation, Eqn. 10, does not include the neglected terms from Eqn. 7.

$$a_p U_p = H(U) - \nabla p \tag{9}$$

$$\rho \nabla \cdot \left(\frac{H}{a_p} \right) - \rho \nabla \frac{1}{a_p} \nabla p + \left.\frac{\partial \rho}{\partial x}\right|_{p,h} \frac{Dx}{Dt} = 0 \tag{10}$$

A benefit of this pressure equation is that most of the terms are linear in relation to pressure. In the case of constant density, an incompressible form is established. Schmidt et. al [4] used this idea in a two-step projection method on a staggered mesh approach. This method was used for their two-dimensional structured grid solver. In order to apply this model to a three-dimensional solution and for an unstructured, general polyhedral mesh, a collocated variable approach is used. The numerical details can be found in Gopalakrishnan and Schmidt [5].

Fluid properties are obtained using REFPROP (Reference Fluid Thermodynamic and Transport Properties Database) from the NIST (National Institute of Standards and Technology) database. Given two thermodynamic inputs, REFPROP calls its necessary subroutines to calculate the thermodynamic and transport properties requested by the user. The user selects the two thermodynamic properties to use as inputs and selects an appropriate range and step size. The properties calculated at these specified points are arranged in a table format. Properties can be calculated for many fluids, including mixtures; in this case the working fluid is water.

Typically, five to ten PISO/secant iterations are employed, and the solution of the pressure equation is necessary for every iteration. For the full pressure equation, a diagonal incomplete LU preconditioned biconjugate gradient is used. Once the pressure equation is solved, the pressure field is used to correct the fluxes and the time step is finished.

The structure for solving these equations is provided by OpenFOAM, which allows rapid construction of CFD codes in an object-oriented framework [6]. OpenFOAM also provides a message passing interface (MPI) that can run decomposed cases in parallel.

LEVY-BROWN CONDENSING EJECTOR

The nozzle being modeled in the CFD simulations is one that has been used in experiments by E.K. Levy and G. A. Brown [7], and is shown below in Figure 2.

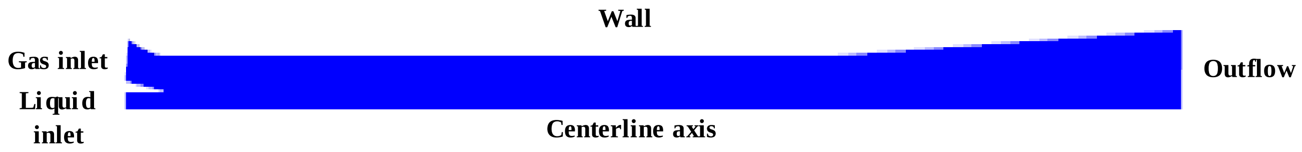

Figure 2: Two-dimensional representation of nozzle used by Levy and Brown [7]

This nozzle acts as a condensing ejector for two-dimensional flow of a fluid. In the figure above, the image can be mirrored around the centerline axis.

For these simulations, saturated vapor enters through the annular channel and saturated liquid enters near the centerline. As the saturated liquid and saturated vapor flow past the splitter, the two streams should start to mix. Around the diverging part of the nozzle there is a shock wave that leads to more mixing and circulation of the two streams. The location and size of the shock wave is difficult to determine at this time. Due to the unstableness of the flow that occurs during condensation, the validity and accuracy of the mixing prediction both before and after the throat are in question.

Choosing appropriate boundary conditions are essential to running a meaningful simulation. Because of the strong pressure gradients and general instability of the flow, finding acceptable boundary conditions can be difficult. The velocity at the gas inlet is set to a value much lower than what was used in Levy and Brown [7]; 75 meters per second compared to approximately 472 meters per second in the experiments. Zero pressure gradient boundary conditions and specified densities, 0.5 kg/m^3 at the gas inlet and 996.7 kg/m^3 at the liquid inlet, are used at the inlets. At the outlet, a specified pressure is imposed at an arbitrary distance, creating a partially non-reflection boundary condition. The outlet boundary condition on velocity is zero gradient. All walls are adiabatic.

RESULTS

The simulations were run for over 0.3 s of simulated time. During this period, the flow never achieved a steady state due to diffuser stall. Vortices were created due to flow separation, as shown in Fig. 3. These vortices create oscillations in pressure and convolute the condensation shock.

The upper half of Fig. 3 shows that, by the beginning of the diffuser section, the high enthalpy of the gas mixes with the low enthalpy of the liquid. This mixing predisposes the vapor to condense by lowering its temperature. In Fig. 4, the mass fraction of vapor declines just after the end of the high enthalpy zone. Pressure shows a modest increase, as shown in the upper half of Fig. 4. However, the vortices created by the flow separation cause locally low pressures that reduce the effectiveness of the condensing ejector. This separation is exacerbated by the strong adverse pressure gradient in this part of the flow.

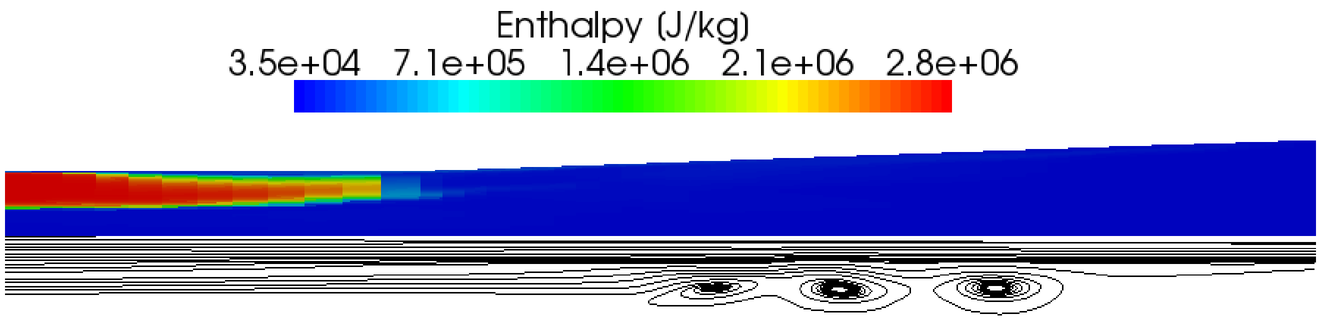

Figure 3. The enthalpy near the diffuser is shown in the top half of the figure. In a mirror image, the streak lines, indicating the instantaneous velocity field, are shown in the bottom half of the figure.

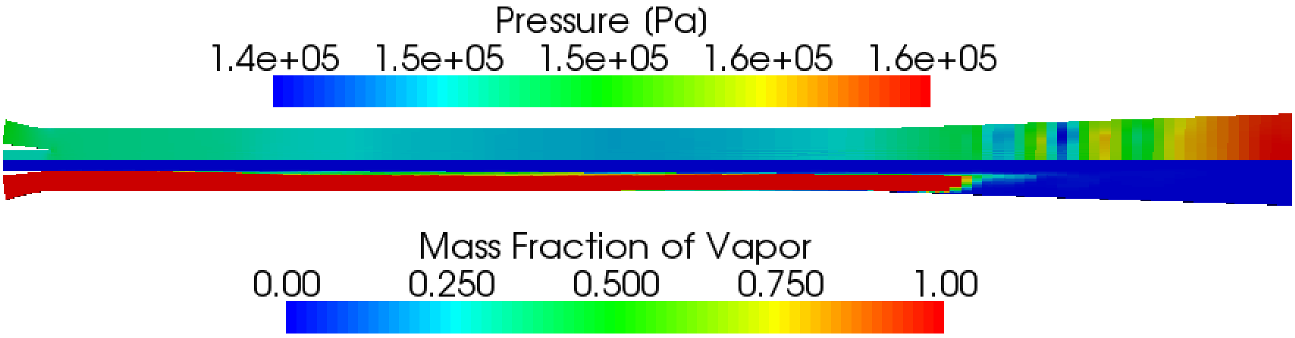

Figure 4. The pressure field in the computational domain is show in the top half of the figure while the mass fraction of vapor is mirrored in the bottom half of the figure.

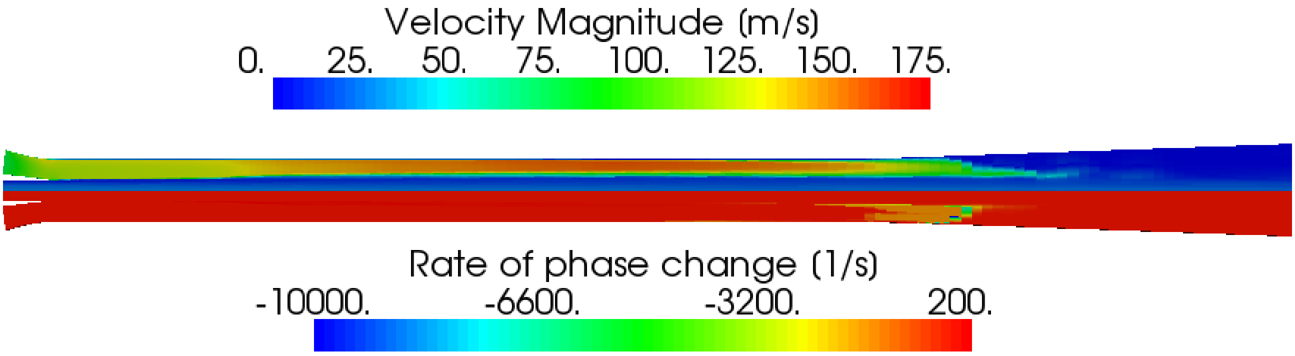

Figure 5. The velocity magnitude in the computational domain is show in the top half of the figure while the time derivative of mass fraction of vapor is mirrored in the bottom half of the figure.

Figure 5 shows the deceleration that accompanies the pressure increase and condensation shock. As can be seen in the upper half of the figure, the shock is oblique. Locally low pressures, corresponding to the locations of vortices, are visible in the diffuser section. The bottom half of Fig. 5 shows the high rate of condensation in the shock. Though not clearly visible in the figure, small amounts of liquid can vaporize in the high-shear zone near the inlet.

Though, in these images, the condensing ejector is producing the desired static pressure increase, the strongly oscillatory nature of the flow caused very large transient variations in pressure. Depending on the time considered, the static pressure at the exit could be below the upstream pressures. Further investigations will include time-averaging of performance and investigations of the acoustic reflectivity of various exit boundary conditions.

CONCLUSIONS

The idea embodied in the Homogenous Relaxation Model was used to construct a CFD model of condensing ejectors. Though further model adjustment and experimental validation is likely required, it is an encouraging early result. To date, the authors are unaware of any existent CFD simulations of condensing ejectors. The current implementation benefits from the flexibility of the OpenFOAM libraries and is fully parallelized.

The preliminary results show the potential for a condensation shock to occur at the beginning of the diffuser section of the condensing ejector, triggering diffuser stall. The separated flow causes a highly unstable flow field and interferes with ejector efficiency. The shed vortices produce locally low pressures and contribute to the oscillations.

Continuing work will use experimental validation to adjust the model to improve its accuracy and check the overall applicability to predicting condensation shocks. It is also likely that a more sophisticated model of interphase turbulent mixing will be required, since the mixing rate strongly affects the performance of the ejector. These areas will be explored in future work.

ACKNOWLEDGEMENT

This material is based upon work supported by the National Science Foundation, STTR Phase II Project No. 0822525 and by the Department of Energy, under Award Number DE-FG36-06GO16049

any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

REFERENCES

[1] Bilicki Z., Kestin J.: Physical Aspects of the Relaxation Model in Two-Phase Flow, Proc. Roy. Soc. London A., **428**(1990), 379-397.

[2] Downar-Zapolski P., Bilicki Z., Bolle L., Franco J.: The Non-Equilibrium Relaxation Model for One-Dimensional Flashing Liquid Flow, IJMF, **22**(1996), No. 3, 473-483.

[3] Prandtl, L.: About Fully-Developed Turbulence (Über die ausgebildete Turbulenz), Z. Angew. Math. Mech., **5**(1925), 136-139.

[4] Schmidt D.P., Corradini M.L., Rutland C.J.: A Two-Dimensional, Non-Equilibrium Model of Flashing Nozzle Flow, 3rd ASME/JSME Joint Fluids Engineering Conference, **208**(1999), No. 616.

[5] Gopalakrishnan S., Schmidt D.P.: A Computational Study of Flashing Flow in Fuel Injector Nozzles, SAE Int. J. Engines **1**(2008), No. 1, 160-170.

[6] Weller H.G., Tabor G., Jasak H., Fureby C.: A Tensorial Approach to Computational Continuum Mechanics Using Object-Oriented Techniques, Computers in Physics, **12**(1998), No. 6, 620.

[7] Levy E.K., Brown G.A.: Liquid-Vapor Interactions in a Constant-Area Condensing Ejector, Journal of Basic Engineering, **94**(1972), No. 1, 169-180.