INTERNATIONAL
ENERGY AND ENVIRONMENT
FOUNDATION

Computational Fluid Dynamics Modeling in Development of Renewable Energy Applications

Editor: Maher A.R. Sadiq Al-Baghdadi

www.IEEFoundation.org

Chapter **

OPPORTUNITIES FOR CFD IN EJECTOR SOLAR COOLING

M. Dennis

School of Engineering, The Australian National University, ACT 0200, Australia.

Abstract

In recent times, there has been a rise in popularity of comfort cooling systems, mainly based on electrically driven heat pumps. The accompanying peak loading on electricity grids is proving to be particularly problematic and expensive for electricity utilities, notwithstanding the greenhouse gas emissions associated with the electricity consumption of these het pumps. An alternative is to generate cooling effect using waste heat or solar heat. The means to achieve this now coincides with the imperative to do so. Heat driven cooling technologies have existed for some time but only recently has research effort been applied to comfort cooling. One such technology is the ejector heat pump. An ejector is a thermally driven compressor that can substitute for an electrically driven compressor in a heat pump cooling system, thereby alleviating peak electricity consumption and associated emissions. Although ejectors have been used as steam driven vacuum pumps for almost a century, they are usually designed empirically and there is little understanding of the flow mechanisms within an ejector. Ejector heat pumps offer further advantages of exceptional reliability, potential for low cost and ease of control. Since their use in the early 1900s, researchers have grappled with analytical analyses of ejectors with limited success, hampered somewhat by the complexity of the flow mechanisms within the ejector. Ejector flows are characterised by supersonic jets, turbulent shear mixing and metastable thermodynamic states. Over the last ten years, a number of researchers have endeavoured to use Computational Fluid Dynamics (CFD) to elucidate these mechanisms and thereby improve ejector design. The volume of scientific literature in ejector CFD has been steadily increasing, indicating a growing interest and confidence in the application of CFD to ejectors. As yet, there are only a limited number of experimental datasets for CFD model validation, but early studies report good agreement at ejector design conditions albeit with a

ISBN xxxxxxxxxxxxx

range of possible flow structures. However, off-design modelling is generally poor. There is a need for improved turbulence modelling of jet and shear mixing layer and a need for improved real gas modelling. Furthermore, there is currently insufficient knowledge in this application of CFD although progress is accelerating. A critical point in the application of CFD in ejector design comes when researchers have sufficient confidence in the CFD models to rely primarily on CFD to design better ejectors, thereby enabling a new range of environmentally friendly heat pumps to be marketed.

Keywords: CFD; Cooling; Ejector; HVAC; Solar.

1. Introduction

The air conditioning industry faces a number of difficult challenges over the coming decade, the most prominent being substituting for banned fluorocarbon refrigerants, coping with carbon costing and reducing water consumption. Furthermore, electricity grids in some countries are under severe stress from air conditioner peak load in summer.
Solar air conditioning would seem to be an obvious approach: demand for cooling coincides with availability of sunshine in a causal manner. However, there are currently no commercially competitive solar air conditioning systems for residences or industrial applications.
A number of demonstration systems have been deployed with total installations of solar cooling equipment now approaching 1000 units worldwide. Furthermore, several nations, including France, Germany and Australia are now active in developing standards for solar cooling systems and looking at ways to encourage their uptake.
There are a number of methods of making these units more commercially viable. Since the system cost is dominated by the cost of the solar collector, one might consider using this collector to also provide space heating and water heating in a complimentary fashion. This approach is being widely adopted in Europe with the development of Combi systems that provide water and space heating. Combi systems have recently been extended to Combi+ systems, providing space cooling in addition to the other services. In this way the solar collector utilisation is much greater and its cost is spread over the three services.
Other approaches to boost commercial viability of solar cooling systems are to increase the cooling system thermal efficiency through improvements to component design and system integration. Alternatively, one might seek to reduce the lifecycle costs of the solar cooling equipment. Indeed all of these aspects are being addressed, depending somewhat on the nature of the solar cooling equipment selected.

2. Approaches to solar cooling

Solar air conditioning may be achieved using solar heat or solar electricity. A recent study has highlighted that there is little difference in cost effectiveness of each of these approaches and that the costs are changing rapidly in both sectors [1]. However, thermal systems have potential to offer solar space and water heating from the solar collector, thereby providing a more holistic thermal service solution.

For this reason, the thermal approach to solar cooling has been the primary interest of researchers. Thermally driven cooling cycles can be classified into several generic methods (Figure 1):

- Absorption technologies – a continuous heat pump cycle whereby compression is achieved by absorption of one chemical into another, with accompanying heat transfer.
- Adsorption technologies – also a chemical heat pump whereby compression is achieved as a refrigerant is adsorbed into the surface of solid material which must later be regenerated by applying heat.
- Ejector (jet pumps) technologies – a mechanical heat pump that uses a supersonic jet to induce refrigerant flow and compression using no moving parts.
- Desiccant technologies – used to dry air which can subsequently be used in an evaporative cooling cycle. Heat is used to regenerate the desiccant.

All of these technologies have reached proof of concept stage with a limited number of demonstration systems from fledgling manufacturers or research institutions.

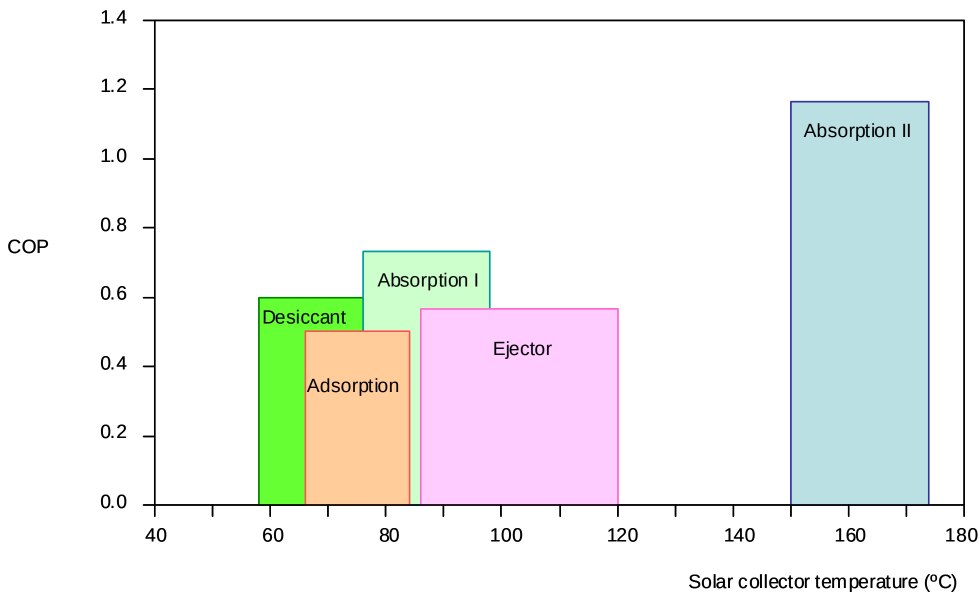


Figure 1. Typical operating parameters of solar thermal cooling technologies

The dominant technology by quantity installed is the absorption system, benefitting from intense development over the last 30 years. Single effect absorption systems (Absorption I) have a Coefficient of Performance (COP)[1] of 0.7. This COP can be increased to around 1.2 if higher solar collector temperatures are available by cascading two single effect absorption chillers to form a double effect absorption system (Absorption II). The main drawback of these systems is their high initial cost, relating to the poor performance and

[1] COP is defined as the ratio of cooling effect produced to the energy input to the cycle

subsequent requirement for a large solar collector. There are also costs associated with maintenance of these systems.

These concerns have led to a revival of interest in ejector technology. Although lower still in performance, ejectors are very simple and reliable, can be designed to have no moving parts, no electricity or water consumption, non-fluorocarbon refrigerant and it should be cheap to manufacture. These features make the system commercially attractive.

Thus the main focus of ejector research is to increase the annual cooling yield from an ejector system. This can be achieved by improvements to system integration and to the ejector itself. A target specific collector area of $4m^2$/kW of cooling effect and specific system cost $US2000/kW of cooling effect, matched to a solar fraction exceeding 50%, would be an ambitious target.

3. Introducing the ejector and its characteristics

Ejectors have been in use prior to 1900 where they found use in evacuating air from leaky low pressure steam condensers. An ejector in this application acts as a vacuum pump, driven by low pressure steam which was readily available in such environments. The ejector's role was characterised by steady state conditions and empirical design. Efficiency was not as important as reliability.

Within 20 years, ejectors found widespread use as vacuum pumps in industrial settings. It was a small step to form a vapour compression heat pump using the ejector as a heat driven compressor. Steam driven ejector heat pumps became common in air conditioning, particularly of hotels and ships during the early 20th century; wherever there was a ready supply of low pressure steam or a steam boiler. Ejector systems were found to be low cost, very reliable and maintenance free.

During the 1930s, Freon refrigerants were developed and vapour compression heat pumps based on these new refrigerants were far superior in performance to ejector systems. Ejector air conditioning fell from favour for 50 years until the Montreal protocol of 1987 highlighted a link between Freon use and atmospheric ozone depletion.

This rekindled an interest in ejector technology and at about this time, two important improvements in ejector design were made. Firstly, refrigerants other than water were tested and found to improve performance. Secondly, researchers began to look at system integration issues and to compose systems incorporating solar energy and hybrid designs.

The modern era of ejector research combines supersonic thermodynamics, computational fluid dynamics and experimental work. Despite this effort, the inner workings of the apparently simple ejector are not fully understood, but are reasonably well modelled.

The ejector has no moving parts, bestowing virtues of simplicity and reliability which make it attractive for commercial production. However, the thermal efficiency of the ejector is low which implies that an ejector requires a large solar collector and large condenser to operate in a heat pump application. Thus the savings in electricity consumption must be compared with the additional cost of the solar collector. Trading capital cost for operating cost is a feature common to most solar systems.

4. Description of the ejector system

The ejector is a thermally driven compressor that operates in a heat pump cooling cycle. In a heat pump system, the ejector takes the place of the electrically driven compressor, but uses heat rather than electricity to produce the compression effect (Figure 2).

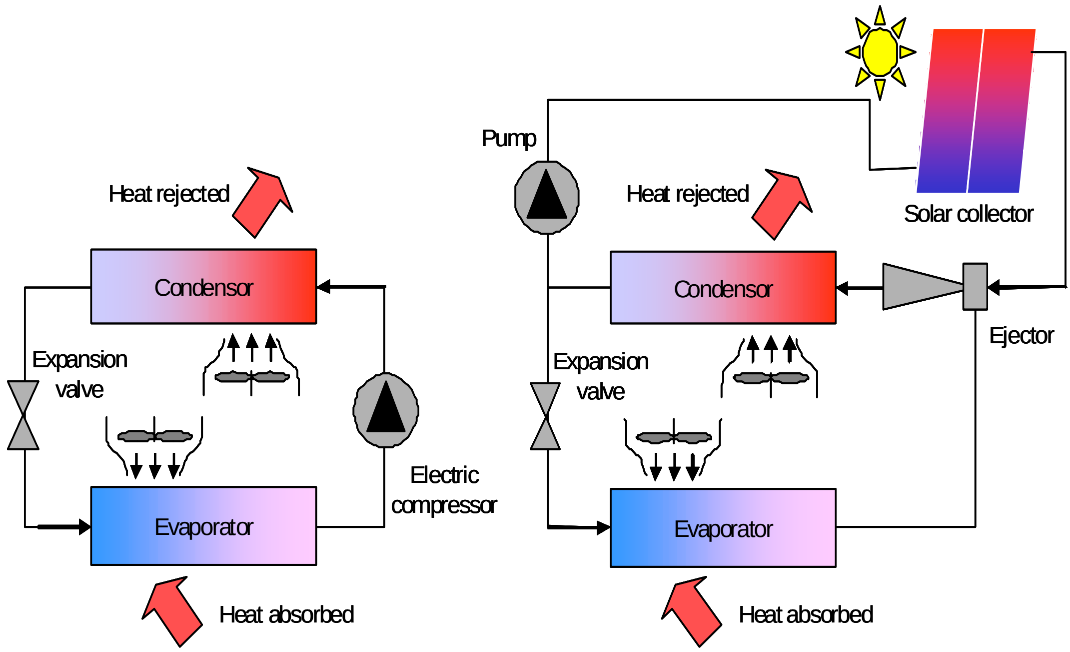


Figure 2. An ejector circuit compared to a conventional heat pump

A liquid pump is required to generate a pressure difference for the ejector heat pump to operate, but since liquid rather than vapour is being compressed, the electricity required is relatively small. With the exception of the ejector, all other components in the heat pump circuit are conventional.

The ejector cycle consists of high and low temperature sub cycles. In the high temperature sub cycle, heat that is transferred to the ejector cycle from the solar collectors causes vapourisation of the ejector cycle working fluid in the solar collector at a temperature slightly above the saturation temperature of the refrigerant. This heat transfer is usually achieved using a heat exchanger not shown in Figure 2. Superheated vapour then flows to the ejector where it is accelerated through a converging-diverging nozzle (Figure 3).

This nozzle accelerates the vapour to supersonic speed. In doing so, much of the vapour enthalpy is converted to kinetic energy, whereby conservation of energy suggests that the vapour temperature and pressure will be very low. The low pressure in the vicinity of the nozzle exit entrains vapour flow from the evaporator. This is the suction effect of the ejector compressor. Optimal operation is defined by the entrained flow reaching sonic velocity in a converging virtual duct formed between the expanding nozzle flow and the ejector mixing chamber wall.

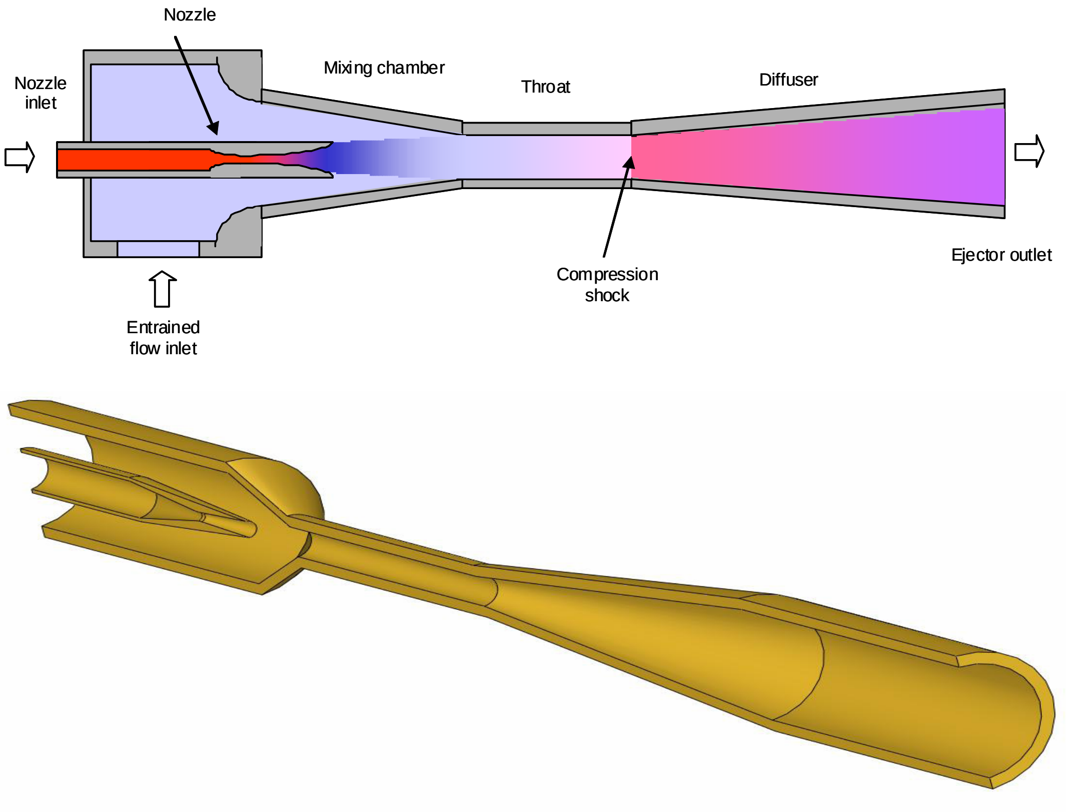


Figure 3. A typical ejector in cross-section, showing the main components

The nozzle flow and entrained flow mix in the ejector mixing chamber and the supersonic combined flow undergoes a transverse compression shock in the ejector throat. Thus thermal compression replaces the electrical compressor in a conventional heat pump. Further compression takes place in the diffuser such that a subsonic stream emerging from the ejector then flows into the condenser. Thus the net effect of the ejector is to create a compression effect between the evaporator and condenser.
At the condensor, heat is rejected from the working fluid to the surroundings, resulting in a condensed refrigerant liquid at the condenser exit. The ejector needs to provide sufficient exit pressure such that the saturation temperature of the refrigerant at this point is greater than the condenser cooling medium, otherwise heat cannot be rejected and the cycle ceases to operate. This is the malfunction mode of the ejector, caused by excessive condensing backpressure. Malfunction can be overcome by suppling greater solar collector pressure and temperature.
Liquid refrigerant leaving the condenser is then divided into two streams; one enters the evaporator after a pressure reduction through the expansion valve, the other is routed back into the solar collector after undergoing a pressure increase through the refrigerant pump. The fluid is evaporated in the evaporator, absorbing heat from the air-conditioned environment, and then it is entrained back into the ejector completing the cycle.

In addition to the apparent advantages of simplicity and reliability of the ejector, the compression mechanism is not complicated by the need for compressor lubricant compatibility and offers freedom of refrigerant choice. Also, the ejector is tolerant of liquid in the compressor since both solar collector port and evaporator port are essentially open tubes.

5. Ejector operational characteristics

The operational characteristics of ejectors are somewhat different to conventional compressors. The ejector performance is typically very sensitive to refrigerant gas properties and off-design operation.

Despite the complexity of internal operation, the ejector may still be considered to be a compressor and its performance may be defined conventionally by its compression ratio and its effective isentropic efficiency.

Ejector performance is noted by a constant capacity region, a critical operating point and a malfunction region, for a given evaporation and condensing temperature (Figure 4). Ideal operation of the ejector, indicated by maximum entrainment of the evaporator flow, is indicated by the knee of each curve in the figure. This point is very close to the malfunction condensing temperature where the entrainment falls to zero so that there is no cooling effect. Indeed the ejector is so sensitive to backpressure (itself related to ambient temperature), that complete malfunction occurs with several degrees of condensing temperature from the optimum operating point.

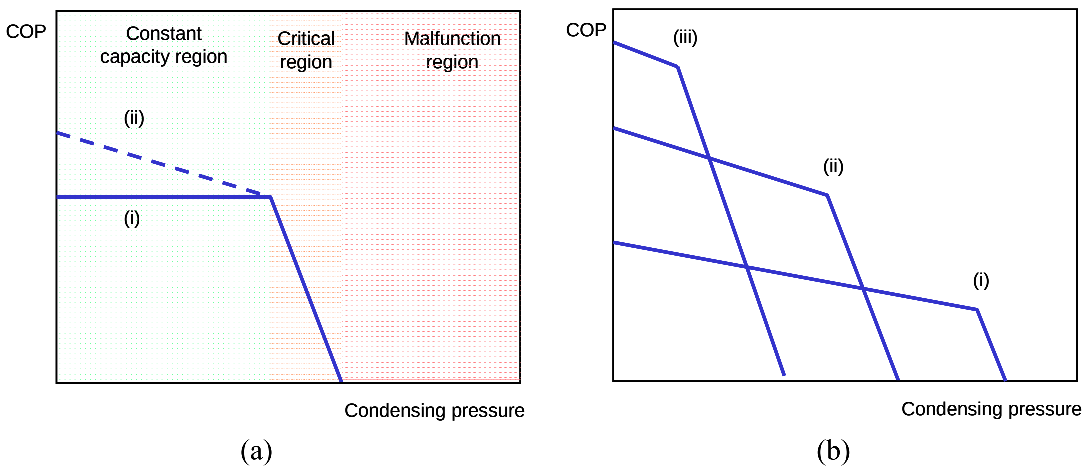


Figure 4. Typical operating characteristics of an ejector showing constant capacity operation (a)(i) and relieved constant capacity operation (a)(ii), the effect of decreasing backpressure on performance is also shown (b)(i) to (b)(iii)

When the condensing pressure is below the critical condensing pressure, ejector cooling capacity stubbornly refuses to increase (Figure 4 (a)). This is contrary to the behaviour of a conventional heat pump and arises from choking of the entrained evaporator flow. Choked flow refers to a condition in supersonic flows whereby a flow cannot exceed sonic velocity inside the converging space, in this case formed between the expanding nozzle flow and the mixing chamber wall. However, if the solar collector pressure and temperature are relaxed, the nozzle flow is weakened such that this converging duct is larger and more entrained flow can be accommodated. Thus the solar collector may be operated at lower temperatures with increased efficiency at milder condensing temperatures. The ejector will then have greater cooling effect (Figure 4 (b)). Thus, optimal selection of the solar collector pressure is necessary to realise best performance from an ejector.

A second important observation is that an increase in solar collector temperature will allow continued operation at elevated condensing temperature but at the expense of COP (Figure 4 (b)). This is because the mass flow of the choked nozzle decreases with increasing driving temperature and thus, there is less motive power to combat the increased condenser backpressure.

An ejector design map may be composed from consideration of many combinations of solar collector, evaporator and condenser temperatures. Ejector design, comprising the various geometric parameters of the ejector, is based on selecting one critical operating point from Figure 5.

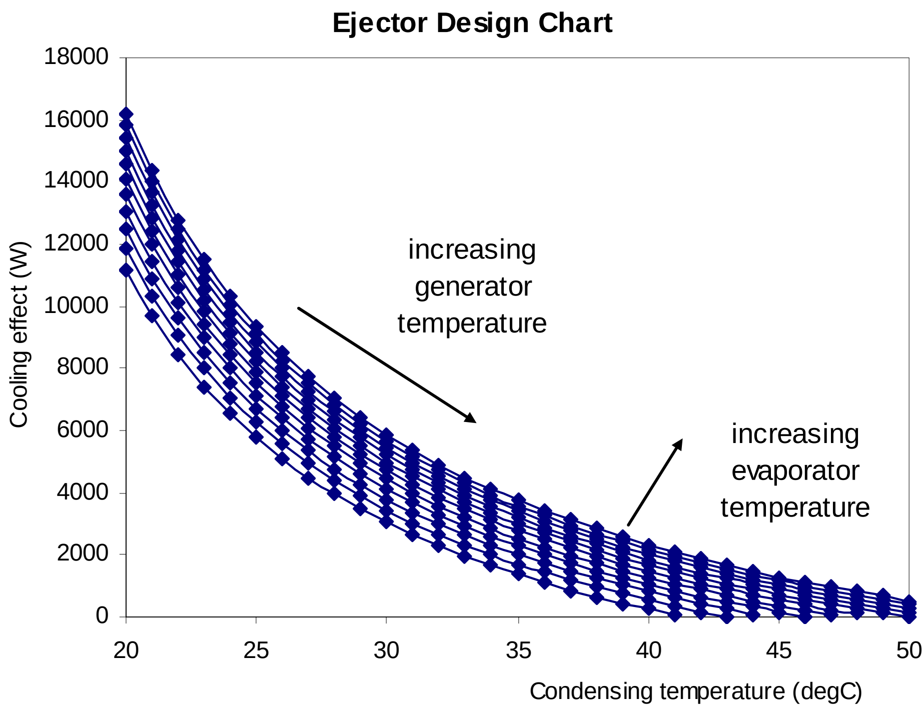


Figure 5. Design chart for a family of ejectors operating at various generating, evaporating and condensing temperatures

The operational characteristics of the ejector are then separately calculated and mapped for control purposes (Figure 6). The ejector design depicted in Figure 6 is designed to produce 3.5kW of cooling effect at a condensing temperature of 32ºC. The cooling effect can be

controlled to some extent by varying the generating and evaporating temperature but is limited to some degree by the need to condense and reject heat to ambient environment. Thus the control strategy for an ejector is to manipulate these three degrees of freedom available to maximise the cooling effect with the amount of solar energy available.
The low COP and the performance characteristics of ejectors have so far prevented market acceptance of ejector based cooling systems, notwithstanding other advantages previously noted.

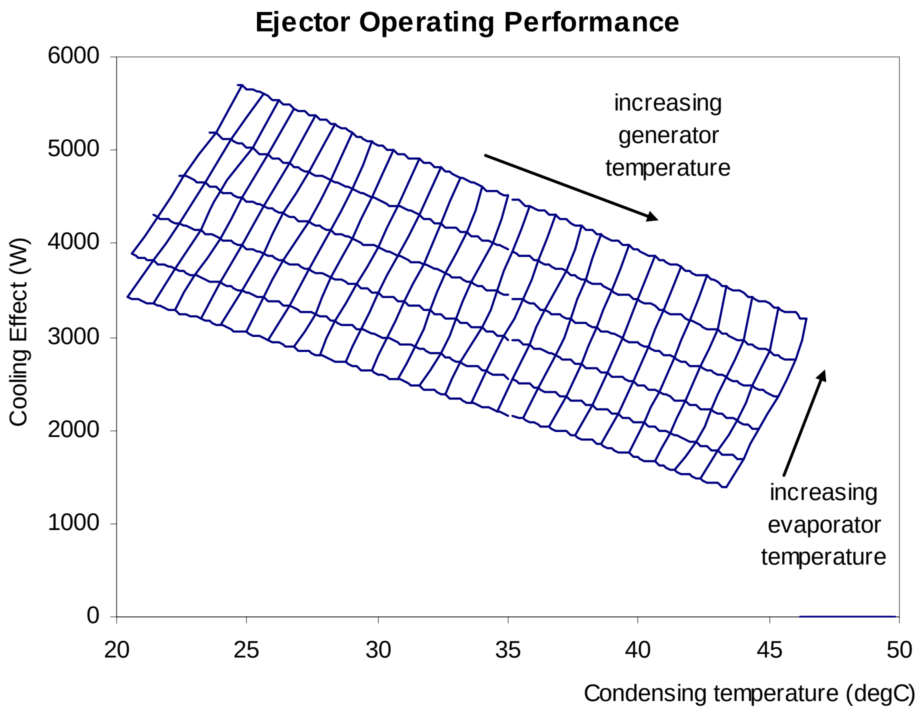


Figure 6. A plot of operational characteristics for a 3.5kW cooling effect design

6. Limitations of analytical ejector modelling

The chemical engineering industry in particular has developed empirical design rules for steam based ejectors mainly for use as vacuum pumping applications. Much of this experience was captured in a British post-war program called the Engineering Sciences Data Unit (ESDU) and these rules continue to be refined. The emphasis of the empirical design is to provide predictable functionality through robust design.
For heat pump applications where energy efficiency of the ejector is crucial, and where working fluids other than steam are used, more sophisticated models are usually developed. Early attempts to thermodynamically model the steam ejector were carried out by Keenan [2]. Keenan was able to reasonably predict the ejector performance characteristics for several classes of ejector geometries. Further clarification of the mixing mechanism was provided by Munday and Bagster [3]. Perhaps the most important improvement in

understanding was provided by Eames [4] and Huang et al [5] with descriptions of a one dimensional design methodology. These advances provided researchers with a means to design an ejector, including the effect of supersonic shock and three calibration constants such that model and experimental data generally agreed within about 10%.

Since Huang's method, there has been a flurry of activity leading to several modifications that give refined analytical descriptions of ejector operation. Rogdakis and Alexis [6] used the thermodynamic and transportation properties of real gas. Sherif [7] and Cizungu [8] developed homogeneous two phase models to account for partial condensation of the expanding nozzle flow. Selvaraju [9] corrected for the friction loss in the mixing chamber. Zhu [10] developed the Shock Circle Method to better account for the radial velocity profile in the mixing chamber prior to mixing, resulting in errors of less than 5% compared to published experimental data. A synthesis of these approaches would represent the current state of conventional 2-D thermodynamic modelling of ejectors.

These one dimensional thermodynamic models attempted to explain ejector phenomena through analytic consideration of the compressible flows present in the ejector nozzle and mixing chamber. When compared to experimental data, researchers report varying agreement with the models. This is partly due to large experimental errors associated primarily with measurement of the nozzle and entrained flows and partly due to the assumptions of the models. While most researchers report an average deviation between model and experimental results, perhaps the most disturbing finding is the variability in this deviation. One of the great difficulties of ejector modelling is the large degree of freedom in the ejector design. For this reason, it is difficult to globally optimise an ejector and difficult to compare recommendations regarding ejector design given in the literature.

Many researchers assume ideal gas behaviour in the ejector which seems most unlikely [4, 5, 11, 12]. Researchers universally use constant ratios of heat capacities in the compressible flow calculations and usually use constant correction factors for primary nozzle efficiency, momentum mixing efficiency and diffuser efficiency. In fitting model efficiency coefficients to experimental data, several studies have demonstrated that these coefficients should not be assumed to be constant [13-16].

There are other factors arising from experiments that are not well captured by analytic models. Experimental evidence suggests that the nozzle position relative to the mixing chamber has a large influence on mixing efficiency and entrainment ratio. In practical design the nozzle position should be controlled to match operating conditions, dictated by condensing pressure. Vapours flows are assumed to be single phase and isotropic which is clearly contradicted by experimental evidence [17] as is the existence of a normal shock in the ejector throat [18].

Nevertheless, even a comprehensively parameterised analytic model will struggle to properly represent boundary layer behaviour, vortex formation and complex oblique shock behaviour, particularly the dynamic and localised behaviour of these phenomena. It would not be unreasonable to suggest that analytical models are approaching their zenith of appropriate development. Further advances in this area through calibration constants may provide improved matching to experimental data but are not likely to provide useful insights into ejector processes.

Recent advances in numerical modelling leads to the notion that CFD would appear to offer advantages for the modelling of ejectors. While more effort might be expended on analytic

models by dedicated researchers, the emergence of CFD as an alternative and more comprehensive modelling environment for ejector design provides an opportunity for a step up in understanding of flow behaviour. Already, several researchers have investigated 3-D flow structures in the ejector including vortex shedding, oblique shocks and non-symmetric boundary layer separation. The level of understanding gained through these studies has potential to surpass conventional experimental data and flow visualisation experiments.
A measure of success for ejector research using CFD is to develop a universal CFD methodology for ejector model development such that model outputs show excellent correlation with high quality experimental data and flow visualisation experiments and, in so doing, gain the trust of ejector practitioners.

7. Ejector CFD model design

Computational Fluid Dynamics has matured over the last decade with the advance in hardware computational capability. This is allowing researchers to investigate the ejector processes in far greater detail including supersonic shock effects, boundary layer flow, flow separation, vortex formation and the like.
The results of CFD studies are now producing close agreement in entrainment ratios with experimental data, provided that the ejector is operating at its design state. Despite this apparent success, models with differing turbulence handling may disagree at off- design conditions and on the flow structure within the ejector.
Insights into real ejector flows are provided by advanced visualisation techniques involving transparent ejectors. Few researchers are involved in this activity and results are limited at this stage. This technique will become important as a means to verify CFD predictions.
The total number of CFD studies in literature relating to ejectors for cooling purposes is rather low, perhaps several dozen. Consideration of this literature informs the following discussion on progress in CFD with ejector modelling. In this chapter, only ejectors relating to cooling or refrigeration applications are considered.
The general procedure for CFD modelling consists of 8 steps:

- Model formulation
- Mesh generation
- Selection of a turbulence model
- Setting of boundary and initial conditions, solution method and fluid properties
- Selection and running of a solver, selection of convergence criteria
- Presentation of results

The generation of a computer model of an ejector and the complexity of the mesh generated to represent it are steadily growing more detailed in line with available computational capacity. In early studies, researchers were limited to moderate resolution 2-D CFD models. Over the last decade, axis-symmetric 2-D and full 3-D models have been increasingly used to provide greater insights into flow behaviour. Although some researchers have noted no significant differences between axis-symmetric 2-D and 3-D models [19], it would be reasonable to assume that behaviour of boundary layer separation and vortex formation near the nozzle exit are 3-D phenomena. Perhaps the time averaging of the CFD code hides this detail. Nevertheless Bouhanguel [18] noted that3-D meshes provided greater detail in the flow structure (Figure 7), particularly for real ejectors where

ISBN xxxxxxxxxxxxx

the entrained flow inlet is typically not axis symmetric although Pianthong [19] specifically disagrees with this hypothesis.
Perhaps the greatest impediment to a full understanding of ejector operation is that only a limited set of operating conditions and design criteria are usually modelled, making it difficult to draw general conclusions. Furthermore, very few researchers, with some exceptions [20], have conducted sensitivity analysis within the CFD program itself to test the robustness of the models to assumptions in the model configuration.

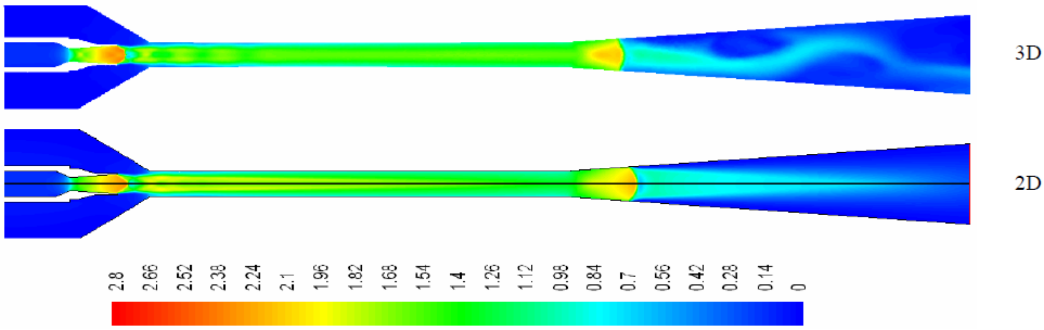


Figure 7. Iso-Mach number fields for 2D and 3D models using the RNG k-epsilon turbulence model (Source: Bouhanguel [18])

A most important consideration for ejector CFD is that of the thermodynamic properties, transport properties and equation of state of the primary and entrained fluids. Some studies considered only ideal gas models which are now known to be a remote resemblance to observed behaviour [19, 21, 22]. Noting that both nozzle and entrained flows reach sonic or supersonic velocities inside the ejector, these flows must be considered compressible. Thus Favre averaged Navier-Stokes equations are regarded as appropriate.
Visualisation experiments have noted condensation near the nozzle exit with some refrigerants as the driving vapour rapidly expands and cools. Indeed, the boundary conditions for an ejector CFD simulation often require several degrees of superheat on the inlet vapours to ensure stable operation of the solver. The most susceptible fluids have a positive sloped saturation curve in the expansion region of a T-s plot, but this is not well predicted by either an analytical model or an ideal gas model. Indeed, even a real gas model may not be able to accurately represent the metastable vapour states present near the nozzle exit and in the mixing region although some attempts have been made [21]. Nevertheless, the use of a real gas model to represent the refrigerant behaviour must now be considered mandatory.
Recently, time dependent CFD analysis has been performed to investigate shock, boundary layer and vortex behaviour in more detail [23]. Very little research has been presented in this area, yet it is essential to understanding the unsteady flow nature of the ejector.
Generation of a mesh representative of the ejector geometry is usually generated automatically by a pre-processing program. The meshes are usually tetrahedral with 50,000 to 100,000 cells (Figure 8). It is common for meshes to be adapted dynamically by pressure or velocity gradient and researchers almost universally report mesh dependence tests.

ISBN xxxxxxxxxxxxx

Nevertheless, rigorous selection of the time and spatial resolution is so far lacking and while the results indicate that the predicted ejector entrainment is reasonably consistent with experimental data, the flow structures indicated by CFD may not be. This could be due to numerical diffusion as a result of an inappropriate choice of mesh.

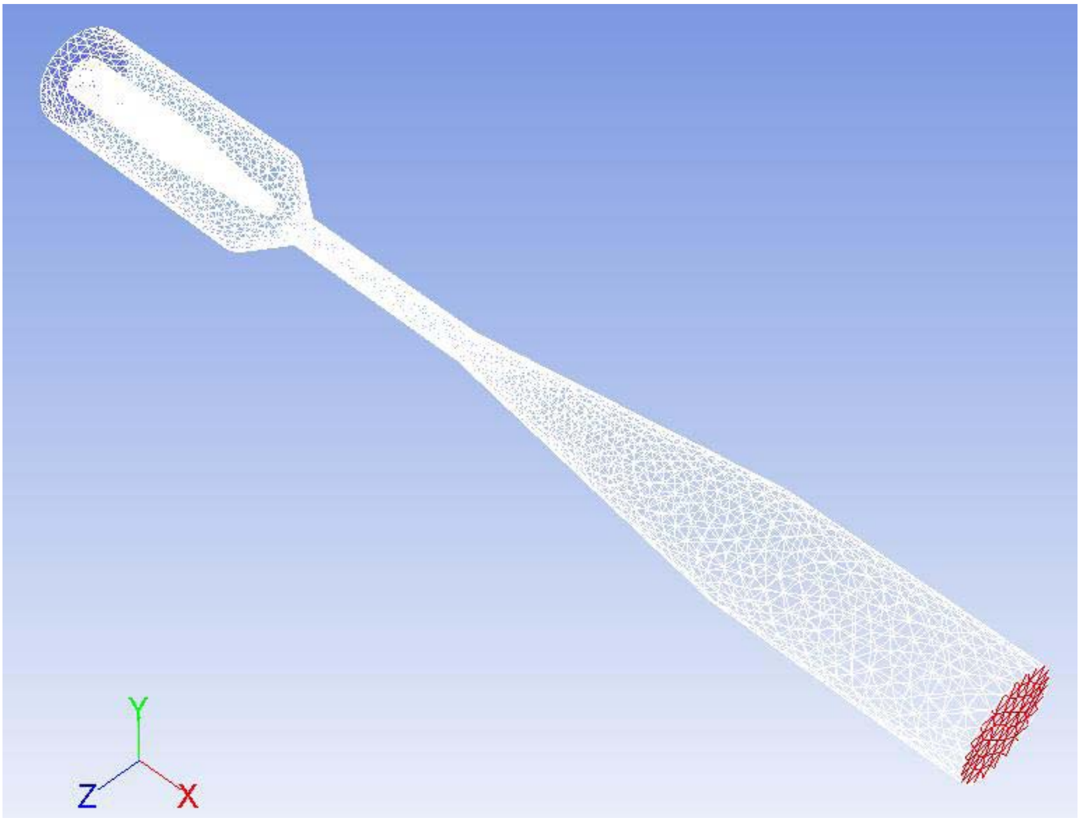


Figure 8. A typical tetrahedral mesh for an ejector CFD analysis, containing 137,857 elements (Source: Marina [20])

Boundary conditions are universally chosen to be total pressure and total temperature for the nozzle and entrained streams. The stream inlet velocities are usually low and have little effect on the results. Wall conditions, including wall friction and heat transfer, are also modelled but are less reported in the literature. Noting that most of the internal flow is of high velocity and separated from the wall by a laminar sub-layer, the wall condition assumptions were not found to be influential by Marina [20]. The total outlet pressure at the ejector diffuser is the final boundary condition.

The initial condition for the model is usually a pressure and temperature field throughout the ejector. Intelligent selection of the initial condition may shorten the convergence time. This might simply be a linear pressure rise between entrained flow entry and diffuser exit, along the ejector axis.

CFD modelling of turbulent flow is only as good as the validity of the turbulence model and indeed careful consideration of the turbulence models has been shown to be important to accurate representation of ejector behaviour.
Flow features at small scales are unsteady and three dimensional. Attempts to resolve all features at every scale using Direct Numerical Simulation (DNS) is a most challenging task and not practical with ejector flows using contemporary computing capacity. The DNS technique can be approximated by resolving large turbulent eddies and modelling smaller scale features whereby the turbulent viscosity is derived directly from the velocity field. This approach is known as Large Eddy Simulation (LES) but has been shown to be inaccurate for ejector modelling, underestimating the entrained flow by around 20% [24]. Perhaps the studies have not persisted sufficiently in this area and a hybrid RANS/LES turbulence model could be tried. The LES approach would be useful to compare to flow visualisation data because it resolves large features in the flow.
Thus most approaches aim to model rather than resolve all of the turbulence. In this case the steady Navier Stokes equations are replaced with Reynolds Averaged Navier-Stokes (RANS) equations to account for the turbulent flow. For compressible turbulent flows, the more complex Favre Averaged Navier Stokes (FANS) approach is best suited.
Within the ANS approaches exists a turbulence model. Most commonly applied to ejector CFD are the two equation models k- ω and k- ε. One restricting assumption of these turbulence models, arising from the Boussinesq hypothesis, is that the turbulence structure is isotropic.
It is generally accepted that the standard k- ε model is not well suited to high shear stresses present in free jet flows in the ejector. However, a modified version of this model known as the Renormalised Group (RNG) k- ε model has been used with apparent success [15, 18]. The RNG approach relies upon modifying the turbulence viscosity in response to strain rate within the fluid and is thus a theoretical improvement over the standard k- ε model. Swirl turbulence is modelled and the inclusion of Prandtl number allows heat transfer to be included.
The Realisable k-ε model uses a different model for turbulence viscosity and a new transport equation. This results in improved modelling of the nozzle jet in ejector applications and better representation of separation and complex flows. The Realisable k-ε model is the most common turbulence model and has been used by several ejector researchers with good results [19, 25-28].
The k- ω turbulence model reduces the turbulence viscosity in response to low turbulence Reynolds number and is thus a good representation of the turbulence structure near the ejector wall. However, it is not sufficiently representative of the entire turbulent flow and is usually blended with the k- ε model to form the Shear Stress Transport (SST) turbulence model. Differences in the flow structures are clearly visible in Figure 9.
Bartosiewicz [29] carried out a comprehensive review of turbulence models in ejector CFD. The best turbulence model was found to be the k-ω-sst, which represented a good compromise between free jet modelling (k-ε) and near wall modelling (k-ω). The authors noted that most turbulence models correctly predicted the entrainment ratio but disagreed on the flow structure and performance at off-design operating conditions. However Bouhanguel [18] found that the k-ω-sst model under-predicted the entrainment ratio and preferred the RNG k- ε model although this work was based on air as the working fluid so

ISBN xxxxxxxxxxxxx

that the CFD predicted flow structures could be compared to published laser tomography results. In this study, all ANS turbulence models predicted entrainment within 8% of experimental data but showed different flow structures, particularly regarding shock behaviour.
A more detailed approach is to discard the notion that momentum transfer in turbulent regions can be modelled as an eddy viscosity and directly compute the Reynolds turbulence stress by solving a transport equation for each stress. This is the principle of the Reynolds Stress Model (RSM), a three dimensional anisotropic turbulence model. Bartosiewicz [30] found no improvement over k-ω-sst turbulence models but the CPU cost was much higher due to the seven extra equations to be solved. The RSM model is based on the same assumptions for the ε equation as the k-ε method and is thus may not be well suited to axis-symmetric modelling of jet flows found in ejectors.

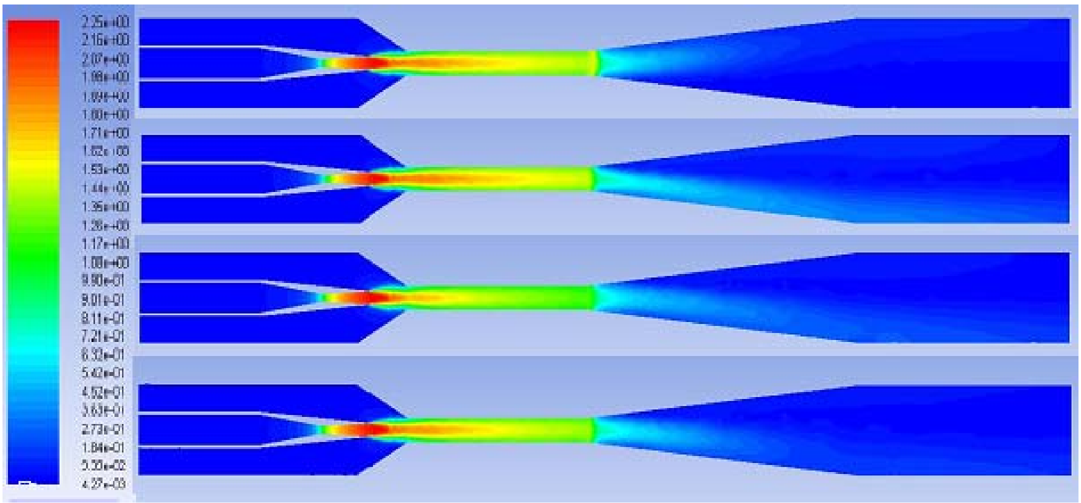


Figure 9. The effects of turbulence model on flow structure is evident in these plots; from the top, the turbulence models are standard k- ε, RNG k- ε, k- ω and k-ω-sst respectively (Source: Marina [20])

Once a suitable turbulence model has been determined, the equations are discretised using a finite volume method. This is usually done with an upwind technique to capture flow direction and to a second order approximation. The equations are solved using standard techniques such as the Gauss-Siedel iterative elimination method. Somewhat faster techniques such as the Alternating Direction Implicit (ADI) by Peaceman [31] and the Strongly Implicit Procedure (SIP) by Stone [32] have not been reported in ejector literature but offer computational advantages.
Solution stability and convergence are promoted by the careful selection of relaxation factors in the solving process. No special recommendations for these factors have been noted for ejector CFD modelling, so their selection is determined by trial and error.

8. CFD research findings

Some of the pioneer studies in ejector CFD modelling must be considered unreliable due to the assumptions and approximations used. In particular, ejector studies using the ideal gas model, coarse grids and simple k- ε turbulence models should be interpreted with caution.

Nonetheless, a small number of helpful studies have been conducted. Riffat [33] conducted a CFD study into the nozzle exit position (NXP) for a constant pressure mixing methanol ejector operating at three possible condensing temperatures. This work provided some design recommendations for this type of ejector and was confirmed by experiments. Zhu [34] conducted a more comprehensive CFD mapping of the ejector nozzle position and inlet convergence angle for a constant pressure mixing R141b ejector, reporting off design performance variations of up to 26%. Zhu's results differ in recommendation of NXP and highlights high sensitivity to small changes in the convergence angle of the constant pressure mixing chamber. One might deduce that CFD cannot provide generalised guidelines but is useful in optimising the design of a specific ejector.

Al-Ansary [21] studied two phase flow in ejectors using CFD. The study examined the use of atomised water particles in the primary flow to increase mixing efficiency. The author noted limitations to the CFD model of the particle homogeneity and ideal gas behaviour, but also worthwhile improvements to ejector entrainment when the secondary flow was not choked. These CFD results were confirmed experimentally.

Perhaps the most extensive CFD ejector studies have been conducted by Bartosiewicz and co-workers. In the first of a series of studies [22], the authors compared CFD results to visualisation experiments of Desevaux [17] using air as the working fluid and treating it as an ideal gas. Nonetheless, using appropriate turbulence models, good agreement was shown in the flow structures at design and off-design conditions, providing the researchers with confidence to proceed to a two phase real-gas model. This was completed and compared to separately published experimental data for a R142b ejector [30], once again showing good agreement and providing insights to flow structures including local interactions between shock waves and boundary layers, their influence on mixing and recompression (Figure 10). This landmark study took into account shock–boundary layer interactions in a real refrigerant for the first time.

Sriveerakul [27] also studied flow structures in the ejector, suggesting that much of the compression effect of an ejector arises from oblique shocks rather than a normal shock as has often been assumed in the past. The study of real time shock behaviour was only recently presented by Al-Doori [23], providing great insight into actual ejector mixing behaviour (Figures 11 and 12) that can not be perceived from analytical ejector models.

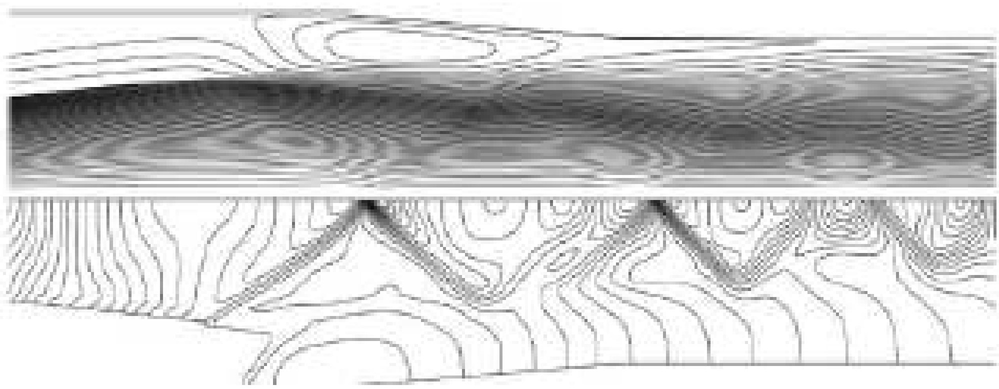
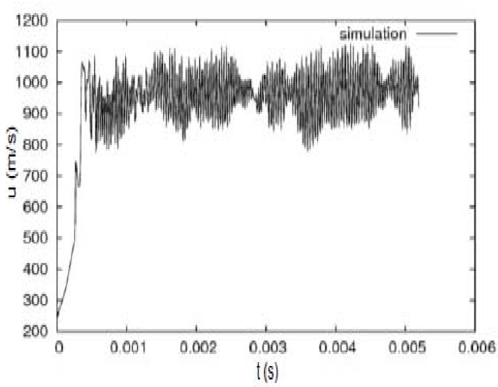
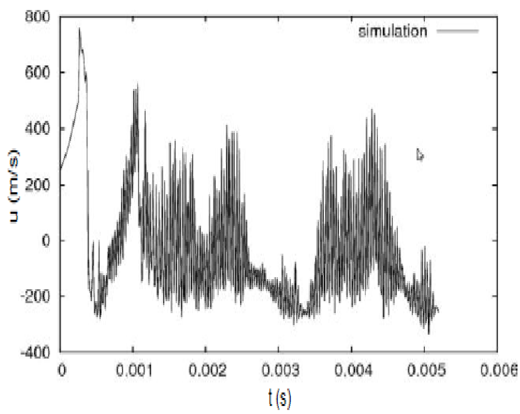


Figure 10. CFD stream function and density fields showing strong oblique shocks originating near the nozzle exit and flow recirculation opposite the nozzle exit, prior to entry of the mixing chamber (Source: Bartosiewicz [30])



(a)



(b)

Figure 11. A time-resolved velocity of the axial flow in the mixing chamber of an ejector near the centreline (a) and near the wall of the ejector (b) (Source: Al-Doori [23])

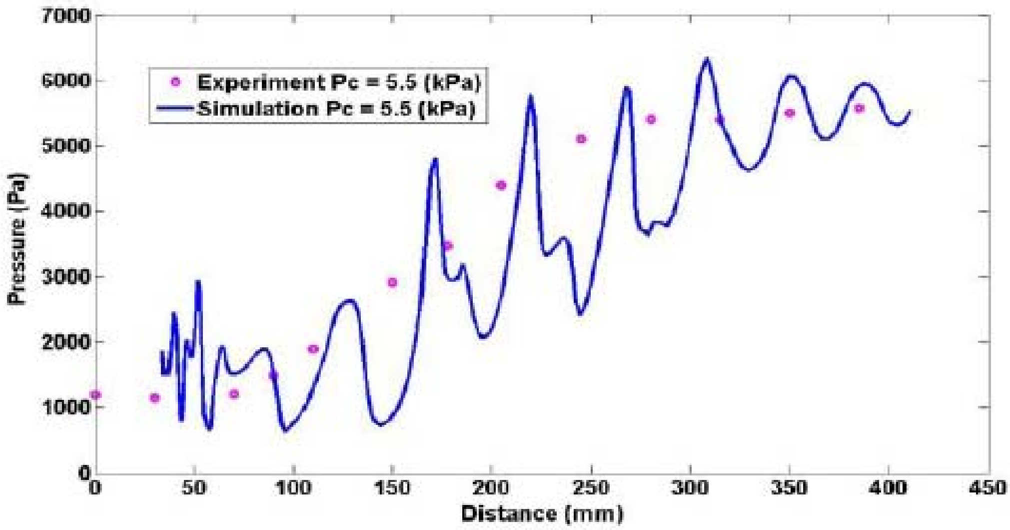


Figure 12. CFD reveals instantaneous behaviour in the flow not visible in analytical models or resolved by instruments (Source: Al-Doori [23])

Pianthong [19] used CFD to evaluate geometric properties of ejector design using a realisable k- ε turbulence model, concluding that CFD was a most useful tool in evaluating ejector geometric design and understanding flow structures in the ejector.

Ablwaifa [35] compared CFD results with experimental data for a R245fa ejector showing good agreement with critical condensing temperature (3%) and entrainment ratio (10%), providing confidence in the CFD method, although no details of the CFD process were given in the paper.

These studies accumulate confidence in CFD as a tool to design better ejectors reflected by the growing number of publications using CFD to evaluate new ejector designs. So far, few recommendations have been produced to improve conventional ejectors except for optimising nozzle position and some other geometric parameters.

CFD analysis is now able to be applied with confidence to new ejector designs. Varga [28] used CFD to analyse a new nozzle design employing a throttling spindle to modify the nozzle flow. The CFD studies showed that by axially changing the spindle position, the ejector could be made to operate in an improved manner across a range of condensing temperatures, thereby increasing the flexibility and yield of an ejector.

Performance improvement in the ejector is largely concerned with reducing mixing irreversibility and removing the compression shock in the mixing chamber (Figures 13 and 14). Several novel methods have been proposed including the Constant Rate of Momentum Change (CRMC) methodology [36, 37], the porous plug mixing methodology [38] and the Pressure Exchange Ejector [39]. All of these studies used CFD combined with experimental validation to evaluate and confirm large improvements to ejector entrainment ratio and/or pressure ratio.

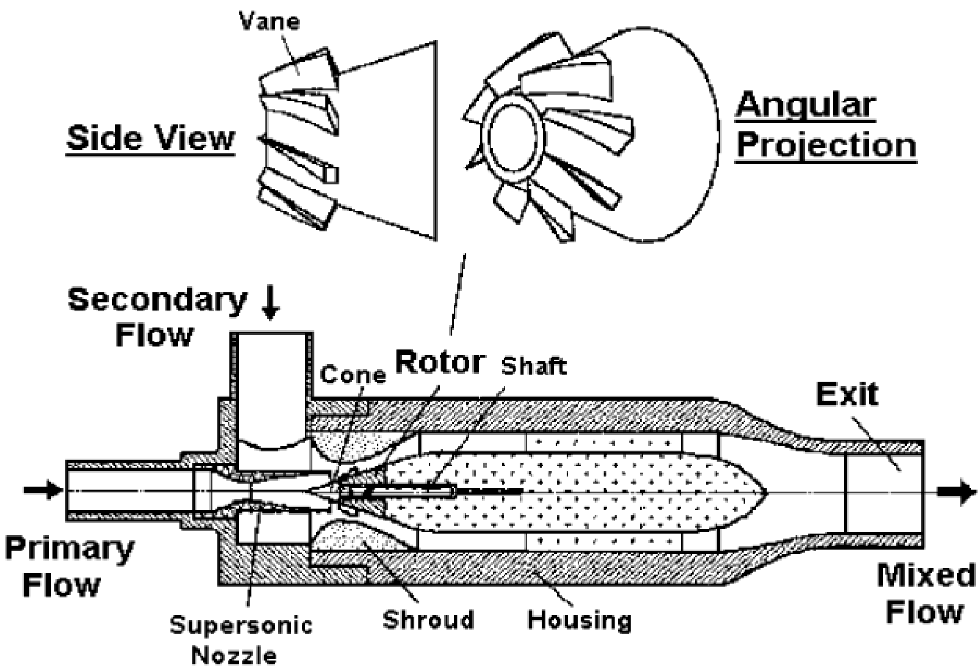


Figure 13. Supersonic rotor-vane pressure-exchange ejector and its rotor (Source: Alhussan [39])

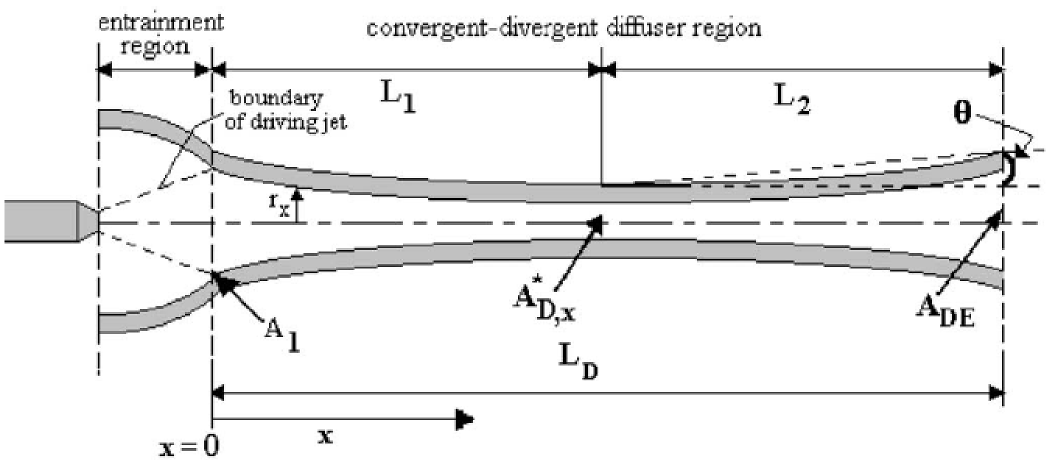


Figure 14. Geometry of a CRMC ejector showing a smooth chamber contour. This design offers increased entrainment and compression ratio through improved mixing efficiency (Source: Eames [36])

9. Future directions and research opportunities

It is clear that any successful future role for heat driven ejectors in cooling applications requires a substantial improvement in performance characteristics in the ejector component and the system within which it operates.

The first step is for ejector researches to determine and universally apply a consistent CFD recipe for ejector analysis. Such an approach might require 3-D adaptive meshes, real gas models and a suitable turbulence model. In published literature and through conferences, there appears to be convergence in the approach to ejector CFD problems. Concommitant to this is the need for comprehensive and high quality experimental data sets to be published by which CFD models may be validated. Given a sufficient volume of experimentally validated CFD models, researchers will gain confidence and trust in the CFD methodology. This is a critical milestone.

The emergence of time dependent CFD analysis, rather than time averaged (pseudo steady state) CFD analysis, will surely provide greater insights into ejector behaviour. A pilot study in this area [23] shows the potential of this approach and perhaps this should be the basis of future ejector CFD studies.

Design optimisation of an ejector is a challenging multivariable problem that is difficult to model by analytical means alone. It is not surprising that researchers analysing a small number of such variables come to different conclusions regarding optimal ejector design and indeed it is difficult to provide generic design rules for ejectors, given that one set of variables that comprises the ejector design is valid only for one set of operating conditions.

This leads to the notion of a variable geometry ejector in which the variables continuously adapt to the prevalent conditions. These variables should include the nozzle exit position, primary nozzle diameter, mixing tube diameter and length, secondary inlet convergent angle. The variable geometry ejector idea was first proposed by Sun [40] and further evaluated by Dennis [41] although no physical design has been published. Future CFD studies will allow optimal geometry for every operating condition to be accurately determined at the design stage. In this way, the model calibration constants are tuned by CFD analysis at each operating condition comprising a solar collector temperature, an evaporating temperature and a condensing temperature. This is an outstanding challenge in the art.

The design mapping of real ejectors is also valuable for the implementation of optimal control of ejectors. Robust and optimal control is required because the ejector performance lags its market competitors and this is the main barrier to market entry.

As computational capacity improves, perhaps the time will come where CFD can be performed in near real-time such that designers will be able to modify ejector designs and directly observe a pseudo steady state CFD response. This "virtual experiment" environment would foster rapid evaluation of novel ejector designs and perhaps enable new variable geometry ejector designs.

Such developments, available only through CFD analysis, may finally enable solar heat driven ejector cooling systems to reach market acceptance.

References

[1] Dennis, M. Solar Cooling Using Variable Geometry Ejectors. Proceedings of the Australian Solar Energy Society Annual Conference, Townsville, Australia, 2009.

[2] Keenan, J., Neuman, E., Lustwerk, F. An investigation of ejector design by analysis and experiment. ASME Journal of Applied mechanics, Vol 72, 299-309, 1950.
[3] Munday. J., Bagster. D., A new theory applied to steam jet refrigeration. Industrial & Engineering Chemistry Process Design and Development 16:442–9, 1977.
[4] Eames. I., Aphornratana. S., Haider. H. A theoretical and experimental study of a small-scale jet refrigerator. Int J. Refrigeration. Vol 18 378-386, 1995.
[5] Huang, B.J., Chang, J.M., Wang, C.P., Petrenko, V.A. A 1-D analysis of ejector performance. International Journal of Refrigeration Vol. 22, 354–364, 1999.
[6] Rogdakis, E., Alexis, G. Design and parametric investigation of an ejector in an air-conditioning system. Applied Thermal Engineering Vol. 20, 213–226, 2000.
[7] Sherif S., Lear W., Steadham J., Hunt P., Holladay J. Analysis and modelling of a two-phase jet pump of a thermal management system for aerospace applications. International Journal of Mechanical Sciences, 42(2): 185–98, 2000.
[8] Cizungu, K., Groll, M., Ling, Z. Modelling and optimisation of two-phase ejectors for cooling systems. Applied Thermal Engineering Vol. 25, 1979-1994, 2005.
[9] Selvaraju, A., Mani, A. Analysis of a vapour ejector system with environment friendly refrigerants. Int. Journal of Thermal Sciences Vol. 43, 915–921, 2004.
[10] Zhu, Y., Cai, W., Wen, C., Li, Y. Shock circle model for ejector performance evaluation. Energy Conversion and Management Vol 48. 2533-2541, 2007.
[11] Aly, N., Aly, K., Shamloul, M. Modelling and simulation of steam jet ejectors. Desalination 123, 1-8, 1999.
[12] Grazzini, G., Mariani, A. A simple program to design a multi-stage jet pump for refrigeration cycles. Energy Conversion and Management Vol 39, 1827-1834, 1998.
[13] Huang, B., Chang, J. Empirical correlation for ejector design. International Journal of Refrigeration Vol. 22, 379–388, 1999.
[14] Zhu, Y., Li, Y. Novel ejector model for performance evaluation on both dry and wet vapours ejectors. International Journal of Refrigeration Vol. 32, 21–31, 2009.
[15] Zhu, Y., Cai, W., Wen, C., Li, Y. Numerical investigation of geometry parameters for design of high performance ejectors. Applied Thermal Engineering Vol. 29, 898–905, 2009.
[16] Varga, S., Oliveira, A., Diaconu, B. Influence of geometrical factors on steam ejector performance - a numerical assessment. International Journal of Refrigeration Vol. 32, 1694–1701, 2009.
[17] Desevaux, P., Aeschbacher, O. Numerical and experimental flow visualization of the mixing process inside an induced air ejector. Int. J. Turbo Jet Engines 19, 71–78, 2002.
[18] Bouhanguel A., Desevaux P., Gavignet E. 3D CFD simulation of a supersonic air ejector. International Seminar on ejector/jetpump technology and application Louvain-la Neuve, Belgium, 2009.
[19] Pianthong, K., Seehanam, W., Behnia, M., Sriveerakul, T., Aphornratana, S. Investigation and improvement of ejector refrigeration system using computational fluid dynamics technique. Energy Conversion and Management Vol. 48, No. 9, 2556–2564, 2007.
[20] Marina A. Computational Fluid Dynamics Investigation of an Ejector for Refrigeration Systems, A thesis submitted in partial fulfilment for the degree of

Bachelor of Engineering, Department of Engineering, Australian National University, 2010.

[21] Al-Alsary. H. Investigation and improvement of ejector driven heating and refrigeration systems. PhD Thesis, Georgia Institute of Technology, 2004.

[22] Bartosiewicz. Y., Aidoun. Z., Desevaux. P., Mercadier. Y. Numerical and experimental investigations on supersonic ejectors. International Journal of Heat and Fluid Flow 26, 56–70, 2005.

[23] Al-Doori. G., Buttsworth. D., Wandel. A.,Jacobs. P., Dennis. M. Preliminary investigation of a supersonic steam ejector using time-dependent numerical simulation. 17th Australasian Fluid Mechanics Conference Auckland, New Zealand, 2010.

[24] Domel, N., Miller, D., Vermeulen, J. Computational simulations of a pulsed ejector actuation system, 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit 10 - 13 July 2005, Tucson, Arizona, 2005.

[25] Rusly, E., Aye, L., Charters, W.W.S., Ooi, A. CFD analysis of ejector in a combined ejector cooling system. International Journal of Refrigeration Vol. 28, 1092–1101, 2005.

[26] Sriveerakul, T., Aphornratana, S., Chunnanond, K. Performance prediction of steam ejector using computational fluid dynamics: Part 1. validation of the CFD results. International Journal of Thermal Sciences Vol. 46 No. 8, 812–822, 2007.

[27] Sriveerakul, T., Aphornratana, S., Chunnanond, K. Performance prediction of steam ejector using computational fluid dynamics: Part 2. flow structure of a steam ejector influenced by operating pressures and geometries. International Journal of Thermal Sciences Vol. 46 No. 8, 823–833, 2007.

[28] Varga, S., Oliveira, A., Diaconu, B. Numerical assessment of steam ejector efficiencies using CFD. International Journal of Refrigeration Vol. 32, 1203–1211, 2009.

[29] Bartosiewicz, Y., Aidoun, Z., Desevaux, P., and Mercadier, Y. CFD-experiments integration in the evaluation of six turbulence models for supersonic ejectors modeling. Conference Proc., Integrating CFD and Experiments, Glasgow, UK, 2003.

[30] Bartosiewicz. Y., Aidoun. Z., Mercadier. Y. Numerical assessment of ejector operation for refrigeration applications based on CFD. Applied Thermal Engineering 26, 604–612, 2006.

[31] Peaceman, D., Rachford, H. The numerical solution of parabolic and elliptical differential equations, J. Soc. Ind. Applied Mathematics, Vol. 3, No.1, 28-41, 1955.

[32] Stone, H. Iterative solution to the implicit approximations to multidimensional partial differential equations, SIAM Journal of Numerical Analysis, Vol 5, No. 3, 530-558, 1968.

[33] Riffat, S.B., Omer, S.A. CFD Modelling and experimental investigation of en ejector refrigeration system using methanol as the working fluid. Int. J. Energy Res. 25, 115–128, 2001

[34] Zhu Y., Cai W., Wen C., Li Y., Simplified ejector model for control and optimisation, Energy Conversion and Management, 49, 1424-1432, 2008.

[35] Ablwaifa, A., Eames, I., Petrenko, V. Experimental validation of CFD model used to design jet pumps. International Seminar on ejector/jetpump technology and application, Louvain-la Neuve, Belgium, 2009.
[36] Eames, I. A new prescription for the design of supersonic jet pumps: the constant rate of momentum change method. Applied Thermal Engineering, Vol 22, 121-131, 2002.
[37] Seehanam, W., Sahumin, K., Pianthong, K., Behnia, M. Prediction of flow characteristic and performance of steam ejector in refrigeration cycle using CFD. Proceedings of the 8th Asian Symposium on Visualization (8ASV), Chiangmai, Thailand, 2005.
[38] Cao, Y., Ling, J. Performance evaluation and CFD analysis of a new jet ejector employing a conical porous duct. Proceedings of the ASME Fluids Engineering Division Summer Meeting Vol. 1, 139–145, 2001.
[39] Alhussan, K., Garris, C. Computational Study of Three Dimensional Non-Steady steam supersonic pressure exchange ejectors. IASME/WSEAS International Conference in Fluid Mechanics, Corfu Island, Greece, 2004
[40] Sun, D. Variable geometry ejectors and their applications to ejector refrigeration systems. Energy, Vol 21, 919-929, 1996.
[41] Dennis, M., Kohlenbach, P. Opportunity and direction for solar cooling -The case in Australia, International Special Issue, Refrigeration and Air Conditioning – Cooling without Warming, Official journal of Centro Studi Galileo (CSG) and European Energy Centre (EEC), 2010.

P h o t o

M. Dennis and the other authors may include biographies at the end of regular chapters. After the author's name (9 pt Bold), the author's educational background is listed: type of degree in what field, which institution, city, state or country, and year degree was earned. The author's major field of study should be lower-cased. The second paragraph uses the pronoun of the person (he or she) and not the author's last name. Information concerning previous publications may be included. Current and previous research interests end the paragraph. The third paragraph begins with the author's title and last name (e.g., Dr. Smith, Prof. Jones, Mr. David, Ms. Elizabeth). List any memberships in professional societies. The photograph (3 cm width) is placed at the top left of the biography. The text (9 pt justified). Please copy and paste this table for each author.
E-mail address: mail@mail.com