21st Australasian Fluid Mechanics Conference
Adelaide, Australia
10-13 December 2018

Detached Eddy Simulation of an Adjustable Radial Ejector

R. Al-Rbaihat[1,2], R. Malpress[1], D. Buttsworth[1] and Kh. Saleh[1]

[1]School of Mechanical and Electrical Engineering
University of Southern Queensland, Queensland 4350, Australia

[2]Department of Mechanical Engineering
Tafila Technical University, Tafila 66110, Jordan

Abstract

A high-performance Adjustable Radial Ejector (ARE) might be capable of achieving optimum performance over a wide range of operating conditions by changing the primary nozzle and ejector duct throat areas during operation by altering the separation of the disk-like surfaces. Previous results show that the simulations of a prototype radial ejector using a variety of RANS turbulence models have not achieved consistently good agreement with the experimental data across the range of ejector operating conditions. The present work describes new simulations of an ARE using Detached Eddy Simulation (DES) in ANSYS FLUENT in conjunction with the DES k-ω SST turbulence model. The influence of varying both the nozzle throat separation (d = 0.39, 0.49 and 0.59 mm) and the duct throat separation (D = 2.3, 2.6, 3.0 and 3.5 mm) on the performance of an ARE is assessed for different operating conditions. The results show that smaller nozzle separations increase the entrainment ratio, but decrease the critical back pressure. Larger duct separations do not always increase the entrainment ratio, but do always yield a lower critical back pressure.

Introduction

Ejectors are regarded as a promising technology, because they can be powered by a low-grade energy, such as solar energy and waste heat [1]. One area of study that may improve the performance of ejectors is the adjustable radial ejector (ARE) [2, 3]. In this configuration, the nozzle and duct geometry can be simply changed during operation and only marginal additional pressure losses should occur as a consequence of adjustment. AREs could be particularly useful as thermo-compressors in heat powered refrigeration and air conditioning systems exposed to fluctuating temperature of both the heat source and cooled space.

An ejector with fixed geometry only works with high performance in a narrow range of operating conditions. In axial ejectors, the concept of adjustable geometry has been used to compensate for the restricted performance at off-design conditions by adjusting the nozzle axial position and/or changing the area ratio using a spindle to change the axial position of a flow blockage feature in the ejector duct. Many researchers have demonstrated the potential benefits of variable geometry ejectors compare to fixed geometry ejectors over varying operating conditions [4, 5]. However, ejectors incorporating an additional flow blockage feature must surely be inducing underwanted pressure losses.

Many authors have employed computational fluid dynamic (CFD) analyses of ejectors, with ANSYS FLUENT software being predominantly used over the past two decades. Studies have shown the importance of the turbulence model used in CFD, and no general agreement has been achieved on the best turbulence model for simulating ejector flows [6, 7]. A wide range of the relative error between CFD results and experimental results for the entrainment ratio - between 3% and 25% - can be observed in prior studies (see Refs. [6-10]).

The concept of a radial ejector was first proposed by Ng and Otis [11] where the flow passages were defined by a spool, a primary diffuser plate, and a secondary diffuser plate. A rotary radial ejector was introduced by Garris et al. [12]. Tacina et al. [13] investigated the performance of a supersonic radial ejector with a rotary nozzle. Several issues for the rotary concept including vibration, mechanical failures and the high costs of the high precision components have been identified by Ababaneh et al. [14]. Rahimi et al. [2, 3] carried out CFD analysis using RANS turbulence models to investigate and optimise the performance of a radial ejector. Simulations were carried out for the radial ejector with different duct separations over different operating conditions. However, the CFD results did not achieve consistently good agreement with the experimental data across the range of ejector operating conditions for the entrainment ratio using RANS turbulence models.

The purpose of this work is to identify a CFD model offering the best agreement between simulations and experiments and then in future, to optimise the performance of ARE working with air. The present work describes new simulations of an ARE using DES turbulence models in ANSYS FLUENT. This comparison takes into account global parameters (entrainment ratio and critical back pressure), and five different RANS turbulence models have been compared with experimental results to assess the range of agreement. This study focuses on the influence of both nozzle separation and duct separation on the performance of ARE using DES turbulence models. The main parameters of the base prototype radial ejector used were 0.49 mm, 2.3 mm and 72 mm for the nozzle separation, the duct separation and the flow path length, respectively. Figure 1 shows the ARE configuration.

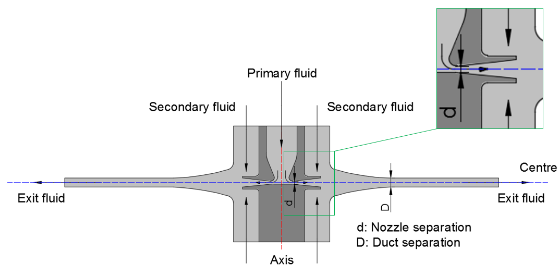

Figure 1: Schematic diagram of a prototype radial ejector showing the relevant flow paths [2].

Detached Eddy Simulation (DES)

Hybrid RANS/LES approaches provide promising options for improving the prediction of engineering separated flows at reasonable computational cost. One of the most popular RANS/LES methods is Detached Eddy Simulation (DES) proposed by Spalart et al. in 1997 [15]. DES models have been particularly designed to treat separated flows and to accommodate high Reynolds number wall bounded flows, where the cost of a near-wall resolving LES would be prohibitive. The difference with the LES model is that it depends only on the required RANS resolution in the boundary layers. The computational costs of DES models is less than LES models, but greater than RANS [16].

A major issue for optimal use of a DES approach is that the interface between the RANS and LES regions depends on grid spacing. RANS/LES transition within the boundary layer is achieved by setting a fine mesh with a grid spacing much smaller than the boundary layer thickness. The benefits of DES over RANS is evident in two points: (i) RANS can be adjusted to treat boundary layers and their separation well, but not in the case of large separation regions; and (ii) time-resolved simulations are often useful for engineering analysis in areas including noise and vibration. The DES turbulence model in ANSYS FLUENT offers five different options: DES Spalart-Allmaras, DES Transition SST, DES BSL k-ω, DES Realizable k-ε, and DES SST k-ω [16-18].

ANSYS FLUENT 18.1, was adopted to simulate the performance of ARE. CFD simulations have been carried out for 2D unsteady models with air as a working fluid in a compressible axisymmetric model. The conservation equations were solved using the density-based implicit solver method, and the controlling equations of mass conservation, momentum conservation and energy conservation were in unsteady forms. The second order upwind scheme was used to discretise the convective terms. Five DES turbulence models and different five RANS turbulence models (SST k-ω, BSL k-ω, Realizable k-ε, Spalart-Allmaras, Transition SST) were tested under different conditions to assess their performance in simulating ARE.

The primary/secondary flow inlets are defined as "pressure inlet", while the mixing flow outlet is set as a "pressure outlet". The fine mesh density was between 70,000 and 85,000 elements for all models. The flow is computed by setting the time step size of 5×10^{-5} seconds; chosen after several attempts to achieve stable solution. The number of time steps was set to be 10,000 and the number of iterations per time step reporting interval was set to be 100 to collect the unsteady statistics for all models.

In order to better simulate the internal flow, a fine mesh density is employed at locations with significant gradients such as in the primary nozzle and mixing region by setting the mesh face sizing. The boundary layer was defined by setting the mesh inflation to attain the Y-plus value that was between 0.4 and 0.6 for both RANS and DES turbulence models, as shown in Figure 2.

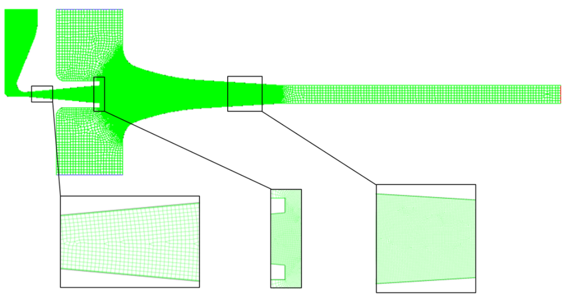

Figure 2: Detail of the numerical grid for the ejector simulations.

The following convergence criteria were adopted to ensure that the solution results were accurate: (a) the relative residuals were stable and less than 10^{-4}; (b) the relative difference of mass flow rates at the inlet and at the outlet was less than 10^{-6} kg/s; (c) the area-weighed-average value for inlet pressure of primary and secondary flow is constant, this point was discussed by Besagni et al. [6, 10].

Figure 3 shows the results of a mesh independence test for ARE at primary, secondary and outlet pressures of 200 kPa, 1.8 kPa, and 2.84 kPa, respectively. Different mesh sizes consisting of between 8,000 and 150,000 elements were produced using DES k-ω SST. Figure 3 indicates a significant change in the entrainment ratio as the number of elements in the mesh increase until 70,000 elements and only a very slight change in the entrainment ratio as the elements of the mesh increase beyond 70,000 elements. Consequently, to conserve computing time, all of the simulations were performed with approximately 70,000 to 85,000 elements.

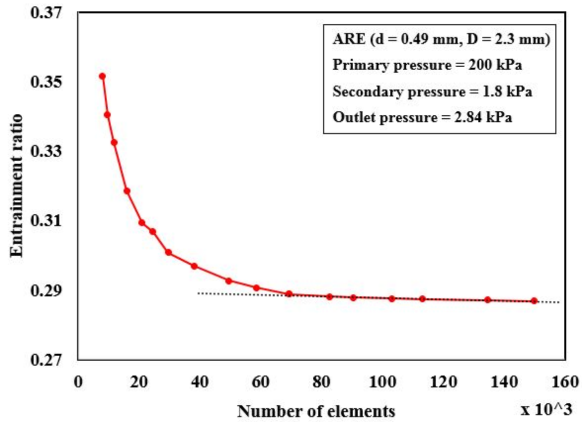

Figure 3: Variation of entrainment ratio with number of mesh elements.

Results and Discussion

The validation of the numerical model has been performed for the base prototype radial ejector by using experimental data obtained by Rahimi et al. [2, 3]. The prototype radial ejector produced an experimental entrainment ratio of 0.29 ± 0.012 for the primary, secondary and outlet pressures of 200 kPa, 1.8 kPa, and 2.84 kPa, respectively. The CFD results shown a large relative error for the entrainment ratio using RANS turbulence models: relative errors for the entrainment ratio were approximately between 30% and 50%. Simulations using DES turbulence models have achieved consistently good agreement with the experimental results for the entrainment ratio. The relative error for the entrainment ratio was approximately less than 2% for all DES turbulence models except for the DES k-ε Realizable, where the relative error was about 48%.

Figure 4 shows the Mach number contours of ARE obtained by RANS and DES in conjunction with the k-ω SST turbulence model. The simulation results show that the maximum Mach number is around 4.5 for both models. However, comparing contours of Mach numbers for both models, it is observed that the DES k-ω SST simulates a subtantially different shock-expansion structural in the free jet than the k-ω SST. The errors for the simulated entrainment ratio relative to the experimental data are approximately 30% and 0.8% for k-ω SST and DES k-ω SST respectively. It is evident that the DES k-ω SST is less symmetric than k-ω SST. The DES k-ω SST provides a better simulation of the ejector performance which depends heavily on the correct simulation of the mixing in the free shear layer, relative to the k-ω SST. The improved simulation of separated flow with a DES approach is consistent with that reported in Refs. [16-19].

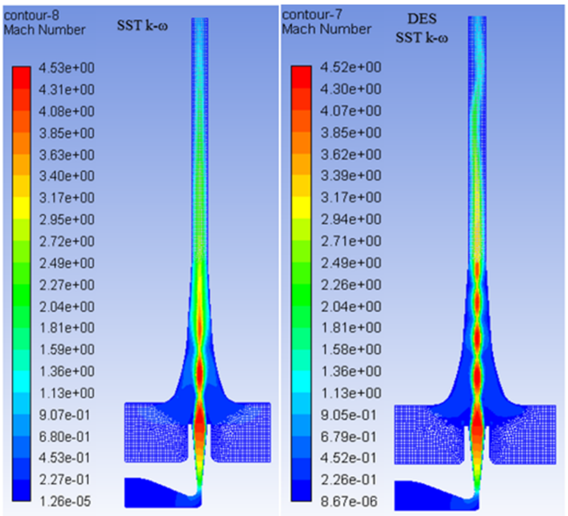

Figure 4: Contours of Mach number for SST k-ω (left) and DES SST k-ω (right) turbulence models at primary, secondary and outlet pressures of 200 kPa, 1.8 kPa, and 2.84 kPa, respectively.

Figure 5 displays the experimental entrainment ratio compared to the entrainment ratio computed from the CFD model, for the different DES turbulence models. According to the results, DES turbulence models are able to simulate the ejector performance in terms of entrainment ratio with acceptable errors that are consistent with those reported in the literature [6-9]. The relative error for the entrainment ratio is approximately less than 2% except for the DES k-ε Realizable where the relative error is about 48%. The entrainment ratio determined from CFD results is very close to the experimental results with less than 0.8% relative error for DES k-ω SST. It is globally the most accurate model in terms of mass flow rate simulation with about 0.3% relative error for choked flow conditions. Therefore, DES k-ω SST has been chosen to assess the performance of radial ejectors. At off-design conditions, the situation is more complex because the entrainment of the secondary stream depends more strongly on the quality of simulations of the mixing [20, 21]. This results in increased difference between numerical and experimental results for off-design operation.

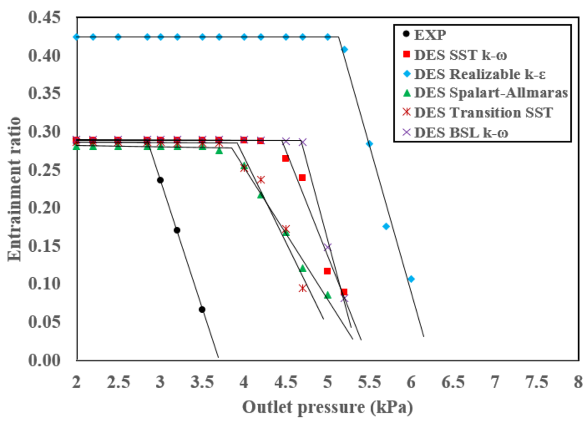

Figure 5: Variation of entrainment ratio with outlet pressure for DES turbulence models for nozzle separation of 0.49 mm and duct separation of 2.3 mm at primary and secondary pressures of 200 kPa and 1.8 kPa respectively.

Figure 6 presents the effect of nozzle separation (d = 0.39, 0.49 and 0.59 mm) on entrainment ratio using the DES k-ω SST turbulence model. The entrainment ratio increases by about 34% when the nozzle separation decreases from 0.49 mm to 0.39 mm, while the critical back pressure decreases by around 13%. The entrainment ratio decreases by about 17% when the nozzle separation increases from 0.49 mm to 0.59 mm, but the critical back pressure increases by around 15%. The ARE can achieve higher entrainment ratio by decreasing the nozzle separation, but yields a lower critical back pressure.

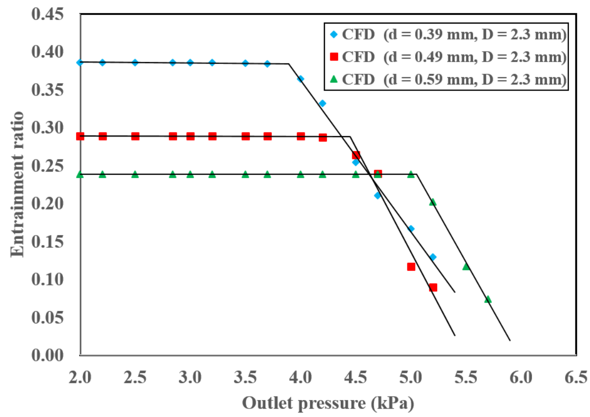

Figure 6: Effect of nozzle separation on ejector performance for DES k-ω SST turbulence model at primary and secondary pressures of 200 kPa and 1.8 kPa respectively.

Figure 7 presents the effect of duct separation (D = 2.3, 2.6, 3.0 and 3.5 mm) on entrainment ratio using the DES k-ω SST turbulence model. The entrainment ratio increases by about 39% when the duct separation increases from 2.3 mm to 3.0 mm, but the critical back pressure decreases by around 35%. The entrainment ratio decreases by about 16% when the duct separation increases from 3.0 mm to 3.5 mm, but the critical back pressure decreases slightly by around 2%. Therefore, the performance reduces significantly when the duct separation is larger than 3.0 mm. The entrainment ratio increases by about 16% when the duct separation increases from 2.3 mm to 2.6 mm, but the critical back pressure decreases by around 11%. This affirms that larger duct separations do not always increase the secondary flow rate to improve the entrainment ratio. This allow a change in the secondary flow rate in both on-design and off-design operation through controlling duct separation. The ARE can achieve higher entrainment ratio by increasing the duct separation, but yields a lower critical back pressure.

Conclusion

Numerical simulation using ANSYS FLUENT - CFD models has been investigated for ARE. Simulations of a prototype radial ejector were performed by RANS and DES methods. By assessing the simulations against the experimental results for ejector performance, DES k-ω SST was the most accurate model in terms of mass flow rate predictions with about 0.3% relative error, for choked ejector operating conditions.

This study focused on the influence of both nozzle separation (d = 0.39, 0.49 and 0.59 mm) and duct separation (D = 2.3, 2.6, 3.0 and 3.5 mm) on ARE performance. The results showed that the maximum improvement of simulated entrainment ratio relative to the experimental data, was about 34% with a nozzle separation of 0.39 mm, while the critical back pressure decreased by around 13%. The entrainment ratio increased by about 39% for the duct separation of 3.0 mm, but the critical back pressure

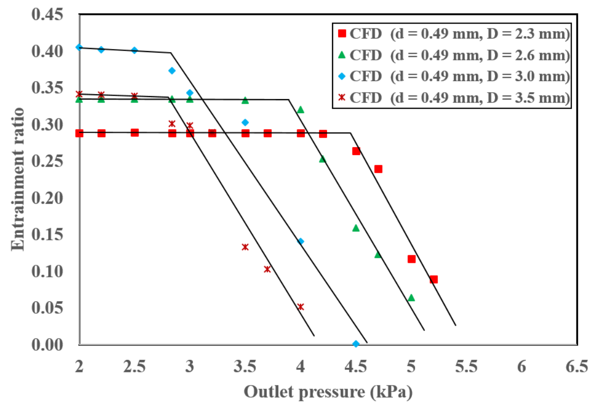

Figure 7: Effect of duct separation on ejector performance for DES k-ω SST turbulence model at primary and secondary pressures of 200 kPa and 1.8 kPa respectively.

decreased by around 35%. The CFD results indicated that the ARE has relatively high entrainment ratio and low critical back pressure for both small nozzle separation of 0.39 mm and large duct separation of 3.0 mm.

The results pointed out that the ejector performance using DES turbulence models was reasonably predicted at on-design operation, whereas the numerical models poorly predicted the performance of the ejector at off-design operation. Efforts are still needed to further refine CFD investigations to design and optimise ARE using DES turbulence models for optimal performance.

References

[1] Chen, W., Chen, H., Shi, C., Xue, K., Chong, D. and Yan, J., A novel ejector with a bypass to enhance the performance, *Applied Thermal Engineering*, **93**, 2016, 939-946.

[2] Rahimi, H., Malpress, R. and Buttsworth, D., Investigation of radial flow ejector concept through CFD analysis, *20th Australasian Fluid Mechanics Conference*, Perth, Australia, December 5-8, 2016.

[3] Rahimi, H., Buttsworth, D. and Malpress, R., CFD study of variable flow geometry radial ejector, *Proceedings of the 4th International Conference of Fluid Flow, Heat and Mass Transfer (FFHMT'17)*, Toronto, Canada, Paper No.: FFHMT 123, ISSN: 2369-3029, August 21 23, 2017.

[4] Varga, S., Oliveira, A.C. and Diaconu, B., Influence of geometrical factors on steam ejector performance A numerical assessment, *International Journal of Refrigeration*, **32**, 2009, 1694-1701.

[5] Pereira, P.R., Varga, S., Soares, J., Oliveira, A.C., Lopes, A.M., Almeida, F.G. and Carneiro, J.F., Experimental results with a variable geometry ejector using R600a as working fluid, *International Journal of Refrigeration*, **46**, 2014, 77-85.

[6] Besagni, G. and Inzoli, F., Computational fluid-dynamics modeling of supersonic ejectors: Screening of turbulence modeling approache, *Thermal Engineering*, **117**, 2017, 122-144.

[7] Mazzelli, F., Little, A.B., Garimella, S. and Bartosiewicz, Y., Computational and experimental analysis of supersonic air ejector: turbulence modeling and assessment of 3D effects, *International Journal of Heat and Fluid Flow*, **56**, 2015, 305-316.

[8] Ariafar, K., Buttsworth, D., Al-Doori, G. and Malpress, R., Effect of mixing on the performance of wet steam ejectors, *Energy*, **93**, 2015, 2030-2041.

[9] Petrovic, A., Svorcan, J., Pejcev, A., Radenkovic, D. and Petrovic, A., Comparison of novel variable area convergent-divergent nozzle performances obtained by analytic, computational and experimental methods, *Applied Mathematical Modelling*, **57**, 2018, 206-225.

[10] Besagni, G., Mereu, R., Chiesa, P. and Inzoli, F., An Integrated Lumped Parameter-CFD approach for off-design ejectorperformance evaluation, *Journal of Thermal Science*, **105**, 2015, 697-715.

[11] Ng, T.T. and Otis, D.R., Experimental Investigation of a Variable Geometry, Radial Ejector, *Journal of Fluids Engineering*, **101**, 1979, 491-494.

[12] Garris Jr, C.A., Hong, W.J., Mavriplis, C. and Shipman, J., A new thermally driven refrigeration system with environmental benefits, *33rd Intersociety Engineering Conference on Energy Conversion*, George Washington University, Washington, DC (US), August 2-6, 1998.

[13] Tacina, K.M., Fernandez, R. and Moody, S.M., Thrust augmentation in an unsteady supersonic ejector, *42nd AIAA Aerospace Sciences Meeting and Exhibit AIAA*, Glenn Research Center, Cleveland, Ohio 44135, USA, Reno, Nevada, January 5-8, 2004.

[14] Ababaneh, A.K., Garris, C.A., Jawarneh, A.M. and Tlilan, H., Investigation of the Mach number effects on fluid-to-fluid interaction in an unsteady ejector with a radial-flow diffuser, *JJMIE*, **3**, 2009, 131-140.

[15] Spalart, P.R, Jou, W.H., Strelets, M. and Allmaras, S.R., Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach, in: C.Q. Liu, Z.N. Liu (Eds.), *Proceedings of the First AFSOR International Conference on DNS/LES*, Greyden Press, Columbus, 1997, 137-147.

[16] Spalart, P.R., Detached-eddy simulation, *Annual Review of Fluid Mechanics*, **41**, 2009, 181-202.

[17] Ruck, S. and Arbeiter, F., Detached eddy simulation of turbulent flow and heat transfer in cooling channels roughened by variously shaped ribs on one wall, *International Journal of Heat and Mass Transfer*, **118**, 2018, 388-401.

[18] Zhou, L., Zhao, R. and Yuan, W., An investigation of interface conditions inherent in detached-eddy simulation methods, *Aerospace Science and Technology*, **74**, 2018, 46-55.

[19] Gritskevich, M.S., Garbaruk, A.V and Menter, F.R., A comprehensive study of improved delayed detached eddy simulation with wall functions, *Flow, Turbulence and Combustion*, **98**, 2017, 461-479.

[20] Lamberts, O., Chatelain, P., Bourgeois, N. and Bartosiewicz, Y., The compound-choking theory as an explanation of the entrainment limitation in supersonic ejectors, *Energy*, **158**, 2018, 524-536.

[21] Lamberts, O., Chatelain, P. and Bartosiewicz, Y., Numerical and experimental evidence of the Fabri-choking in a supersonic ejector, *International Journal of Heat and Fluid Flow*, **69**, 2018, 194-209.