ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Issue 10, October 2017

Intention and Recital CFD Analysis of Thermocompressor

Maharshi Singh[1], L.Natrayan[1], T.Amalesh[2]

[1] School of Mechanical and Building Sciences, VIT University, Chennai, Tamilnadu, India.
[2] Department of Mechanical Engineering, SSN College of Engineering, Chennai, Tamilnadu, India.

Abstract: In this work designates the flow performance inside a considered prototypical of a thermocompressor exhausting the CFD code and FLUENT. Solver type charity remained density constructed. Tempestuous prototypical was designated with reckoning energy happening. Solution approaches are rehabilitated into second order exposed for improved precision. This expedient routine liquid vapours as the operational fluid and works at 6 bar drive pressure, 0.5 bar force pressure. The throat diameter, nozzle inlet diameter and nozzle position are superior for obtaining exciting entrainment ratio and leaving pressure.

Keywords: Thermocompressor, Ejector, Performance evaluation, entrainment ratio, Converging- Diverging Nozzle, Computational fluid dynamics (CFD)

I. Introduction

Significant quantities of low pressure steam often vent by industries to the atmosphere, wasting energy and water. This low pressure steam can be used in various areas like, driving evaporation and distillation process, chilling water, producing vacuum, producing hot water. But sometimes the pressure of the steam may be too low for these applications [1-4]. Thermocompressor is used to boost the pressure and temperature of a low pressure steam in to high pressure steam, for this device uses a high pressure steam to entrain a low pressure steam. The thermocompressor can mix two different pressure fluids, and forming a mixing fluid with an intermediate pressure [5]. Thermocompressor does not contain any moving parts, so it can be said that they are an interesting alternative to conventional compressors [6]. Also lack of moving parts makes it easy to maintain. The operating principle of thermocompressor is based on the transformation of potential energy into kinetic energy by its favourable geometry and which promotes the suction of the low pressure fluid through one entrance, mixing it with the motive fluid with high pressure and compression of the mixture in the mixing chamber device [7-9]

In this study thermocompressor is designed using CFD because of the ability of the CFD to explain the flow field inside the complex geometries. The required data are collected from a plywood industry in Cochin and then its performance is optimized by choosing the best result.

Literatures in this area present a lot of work. Bartosiewicz and Aidoun [2] developed a one dimensional theory on this area but that did not take the effects of ejector geometry on its performance. Riffat and Gam [11] analysed the efficiency of a methanol driven ejector.

They used different primary nozzle to test how they affect the performance of the device. Natrayan.L [1] investigated the effect of convergence, divergence, length and diameter of throat section, nozzle position. Effect of geometry on performance of a nozzle design and nozzle exit locations are carried out by Natrayan.L [1, 3, 5, 6].

II. Theory

The three important parts of a thermocompressor are nozzle, suction chamber and diffuser. The nozzle and diffuser are of converging diverging type. According to various journals the high pressure steam which enters through the nozzle is termed as motive steam and low pressure steam is termed as secondary fluid or suction steam and discharge steam is the steam that exit from the diffuser.

A schematic view of a typical thermocompressor based on one dimensional theory is shown in Fig.1. From the figure it is clear that, as the motive fluid (high pressure) expands and accelerates through the nozzle, (i), it comes out with supersonic speed to create a very low pressure region at the nozzle exit plane (ii), which means a low pressure fluid can be entrained in to the mixing chamber. At some point along this duct the speed of the secondary fluid rises to sonic and chokes. After the secondary fluid chokes the mixing process begins, which in turn causes the primary flow to repeat while the secondary flow is accelerated. By the end of the mixing chamber, the two flows are completely mixed. The static

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Issue 10, October 2017

pressure is considered to be constant until it reaches the throat section (iv). A normal shock is induced at the mixing chambers throat due to a region of high pressure (v). This shock creates a sudden drop in the flow speed from supersonic to subsonic; as it is brought to stagnation through a subsonic diffuser a further compression of the flow is achieved.

One of the important parameters to describe the performance of a thermocompressors is the Entrainment Ratio (Rm) which is defined as the ratio of the recovered Suction steam quantity to motive steam quantity. This is the ratio of mass flow rate (kg/h) of suction steam to flow rate (kg/h) of motive steam. The high entrainment ratio for a thermocompressor signifies the high quantity of steam which can be recovered and so, high performance thermocompressor has a high entrainment ratio

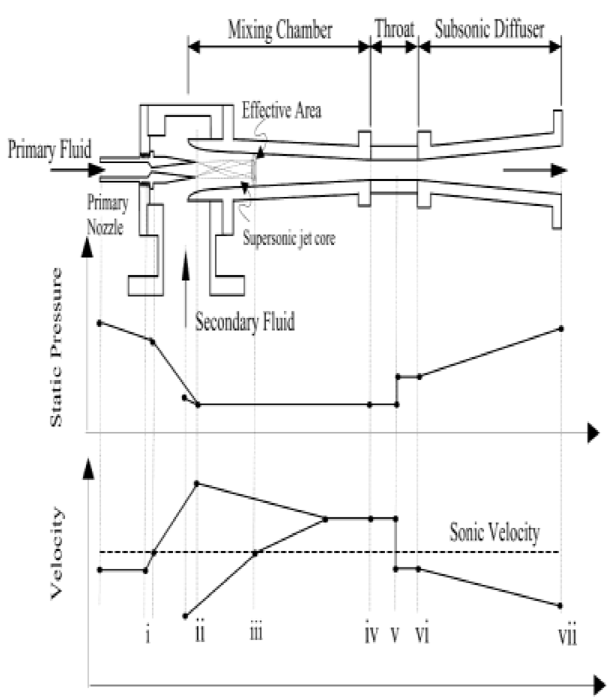

Fig. 1. Schematic view and the variation of stream pressure and velocity as a function of location along a steam ejector

Fig.2. shows the mass flow rate through the nozzle. Pb is back pressure, Pa is the upstream pressure of the convergent divergent nozzle and m. is the mass flow rate which enters the nozzle. When Pb is equal to Pa there is no mass flow rate entering the nozzle. When Pa is constant the back pressure decreases, the mass flow rate starts to enter the nozzle and when back pressure is further decreased more mass flow rate enter the nozzle.

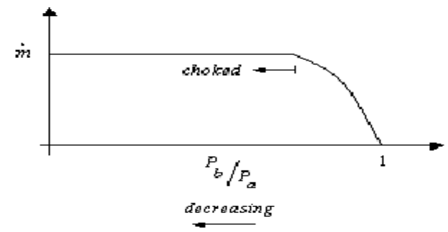

Fig 2: Mass flow rate through the nozzle

III. CFD MODEL

The problem solved numerically by using ANSYS FLUENT, which is a commercial CFD software pack. The working fluid is selected as water vapour, although it seems unrealistic, it has been proved by many researchers in the case of thermocompressor [1,11,13,15,17]. The properties of working fluid are selected from the fluent data base and ideal gas is selected for density. Convergence of the solution is considered when two criteria are satisfied: every type of the calculation residual must be less than 10-3 and mass flow rate through the outlet face must be stable.

Table 1: Properties of working fluid used in CFD simulation

Properties	Values
Viscosity, μ	1.34×10^{-5} $kgm^{-1}s^{-1}$
Thermal conductivity, k	0.0261 $Wm^{-1}K^{-1}$
Specific heat capacity, C_p	2014.00 $Jkg^{-1}K^{-1}$
Molecular weight, M	18.01534 kg $kmol^{-1}$

IV. GEOMETRIES AND MESH DETAILS

The model is designed in the ANSYS design modeller and it is created as 3D. Only the fluid part is taken in to account since the body part does not have any effects and in order simplify the design side. The dimensions of the thermocompressor designed are given below in a table. The mesh was made of 102348 quadrilateral elements as shown in fig.3.

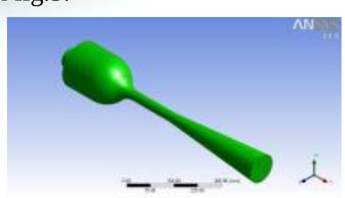

Fig 3: 3-D model of thermocompressor

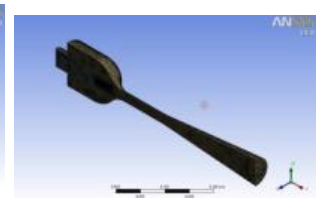

Fig 4: Mesh

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Issue 10, October 2017

V. CASE SETUP

Fluent 14.5 is a commercial CFD software package. Here in this problem the flow is turbulent compressible flow. The realizable k-ε model was selected to govern the turbulence characteristics. This model is the advanced version of the standard k-ε model. The realizable k-ε model can predict spreading rate of round jet more accurately. The nonlinear governing equations were solved using the coupled implicit solver.

VI. BOUNDARY CONDITIONS

In this case there are two inlets and one outlet. The primary fluid enters through the nozzle inlet and secondary fluid through the suction inlet. Both these inlets were set as pressure inlet and the one through where the steam leaves was set as pressure outlet [15,16]. They are set based on the saturation condition of the inlet and outlet steam. For the current model the motive steam pressure is 6 bar with corresponding saturation temperature and suction and discharge steam temperatures are assigned as the saturation properties[16,17]

Table 2: Dimensions of the thermocompressor designed

Dimensional types	Numerical magnitude (mm)
Diameter of working nozzle throat	4.325
Diameter of working nozzle inlet section	32.5
Length of gradually reduction of working nozzle	28.1
Diameter of mixing room inlet section	81.25
Diameter of second throat section	16.75
Length of second throat	201

VII. RESULT

A. Flow Behaviour

Flow behaviour inside the thermocompressor, No matter how complex they are. Can be described easily by the FLUENT and it is one of its main advantage. Figure 4 shows the contours of Mach number and fig 5 shows the static pressure profile along the thermocompressor axis

From fig.5 it can be understood that the high Mach number at the nozzle exit leads to a sudden drop in the static pressure at this area. This pressure is lower than the suction pressure at the suction inlet. So this pressure difference causes the suction steam to enter the suction chamber of this thermocompressor. At the entrance the suction flow velocity is very low, but after mixing with high velocity motive steam it accelerates through the thermocompressor

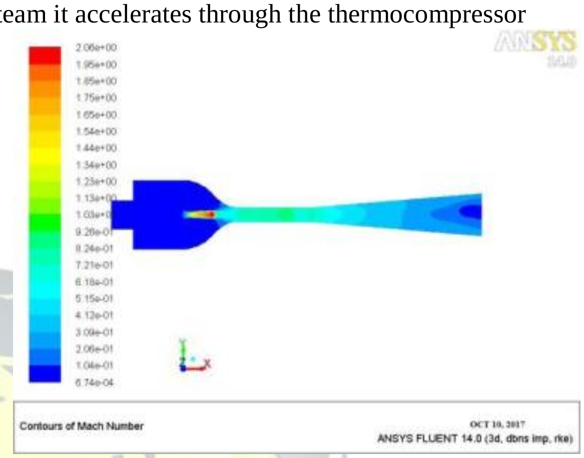

Fig.4. Contours of Mach number

The velocity of the mixed flow reduces at the diffuser. And at this point the static pressure gradually increases with fluctuations. The high velocity difference between the motive fluid and suction steam near the wall creates a separate layer at the converging diverging section of the diffuser

The flow experiences a normal shock at the beginning of the diverging part of the diffuser and the static pressure gradually changes in to the discharge pressure value while the flow velocity decreases to a subsonic level

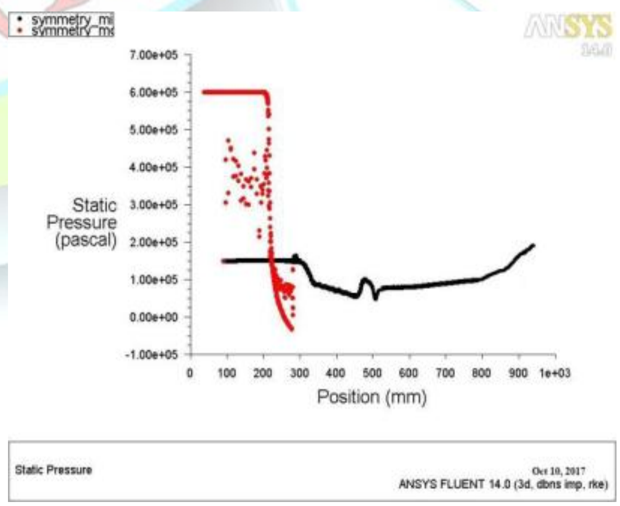

Fig.5. Static pressure profile

B. Entrainment ratio and effect of Motive pressure on entrainment ratio

The variation of entrainment ratio for different motive pressures varied in the range of 5.5 to 6.5 bar has been calculated and is given in the table below. It shows that decreasing the motive pressure causes the entrainment ratio to increase but critical back pressure to decrease. So motive pressure with best entrainment ratio and preferred critical back pressure is selected for the device

Table 3: Entrainment ratio for different motive steam pressure

Motive steam (bar)	Entrainment ratio	Back pressure (mbar)
5.5	1.617	815
6	1.483	850
6.5	1.209	880

VIII. Conclusion

Entrainment ratio and critical back pressure of the thermocompressor were verified by the CFD method and the advantages of CFD model over other conventional models were proposed. Flow behaviours of the designed thermocompressor investigated and effects of different parameters were evaluated. From the obtained result it is clear that for practical purposes the best way to increase the entrainment ratio is by reducing the motive pressure but decreasing the motive pressure will decrease the critical back pressure, which should be taken in to consideration.

References

[1]. Natrayan.L, M.Prasanth, M.Anbalagan, R.Prabhu, Computational analysis to enhance the jet mixing characteristics using circular grooves in a square nozzle, International Journal of Advanced Research in Basic Engineering Sciences and Technology (IJARBEST), ISSN: 2456-5717, Vol. 3, Special Issue 39, May 2017, Page no: 27-31.

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

[3]. Natrayan.L, G.Selvaraj, N.Alagirisamy, M.S.Santhosh, Thermal Analysis of Engine Fins with Different Geometries, International Journal of Innovative Research in Science, Engineering and Technology (IJIRSET), DOI:10.15680/IJIRSET.2016.0505040, Vol. 5, Issue 5, May 2016, Page no: 6900-6906.

[4]. D.R. Croft and D.G. Lilley, "Jet pump design and performance analysis", AIAA 14th Aerospace Science Meeting , AIAA Paper 76183, New York, 1976.

[5]. Natrayan.L, P.Sakthivel, T.Amalesh, Design and Finite Element Analysis of Industrial Radial Flow Impeller. International Journal of Advanced Research, ISSN 2320-5407, DOI: 10.21474/IJAR01, Page no 1525-1531, Vol 4, Issue 4, April 2016.

[6]. S.B. Riffat, S.A. Omer. CFD modeling and experimental investigation of an ejector refrigeration system using methanol as the working fluid. International Journal of Energy Reservation 25 (2001) 115–128.

[7]. Natrayan.L, R.Prabhu , Vehicle Compartment Heat Control Using Phase Change Materials, International Journal for Research in Applied Science & Engineering Technology - Vol 4 , Issue IX , ISSN : 2321-9653, September 2016.

[8]. M. Senthil kumar, Ajith gopinath & Natrayan.L, Evaluation of mechanical properties of E-Glass and Coconut fiber reinforced with polyester and Epoxy resin matrices, International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), ISSN (P): 2249-6890; ISSN (E): 2249-8001, Vol. 7, Issue 5, Oct 2017, 13-20.

[9]. Natrayan.L, P.Santhakumar, R.Mohan raj and Mohandas, Design and Comparative Analysis of Old & New Model Car Wheel Rims with Various Materials, Journal for Research - Vol 02, Issue 02, ISSN: 2395-7549, Page no 69-73. April 2016.

[10]. S.B. Riffat, G. Gam and S. Smith, "Computational fluid dynamics applied to ejector pumps", Applied Thermal Engineering, 16 (4), 1996, pp. 291-297.

[11]. S.Yogeshwaran, R.Prabhu, Natrayan.L, Mechanical Properties Of Leaf Ashes Reinforced Aluminum Alloy Metal Matrix Composites, International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 13 (2015).

[12]. M.L. Hogarth, "The design and performance of high-pressure injectors as gas jet boosters", Proceedings of the Institution of Mechanical Engineers, 185, 1970, pp. 755-766.

[13]. Natrayan.L, M.S.Santhosh and S.Yogeshwaran, Design and Investigation of Safety Cross Stand For Scrambler, International Research Journal of Engineering and Technology - Vol. 3, Issue 09, October 2016.

[14]. E. Rusly, A. Lu, W.W.S. Charters, A. Ooi and K. Pianthong, "Ejector CFD modeling with real gas model", Mechanical Engineering Network of Thailand the 16th Conference, 2002, pp. 150–155.

[15]. Natrayan.L, K.Sathish Kumar, E.Aravindraj, FEA Analysis of using Para (polyarylamide) Materials in Radiator Pump Impeller, International Journal of Modern Trends in Engineering and Research (IJMTER), Volume 03, Issue 04, [April– 2016] ISSN 2349–9745.

[16]. I.W.Eames, S.Aphornratana and H.Haider, "A theoretical and experimental study of a small-scale steam jet refrigerator", International Journal of Refrigeration, 18 (6), 1995, pp. 378–386.

[17]. K. Chunnanond, S. Aphornratana. Ejectors: applications in refrigeration technology. Renewable and Sustainable Energy Reviews 8 (2004) 129–155.

[18]. E.Aravindaraj, R. Prabhu, S.Kalaimani, Natrayan.L, Experimental Analysis of Biogas from Kitchen Wastes, International Journal of Innovative Research in Science, Engineering and Technology (IJIRSET), Volume 6, Special Issue 8, May 2017, Page no 107-115. ISSN : 2319 – 8753.