Computers & Fluids 46 (2011) 212–217

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Computational fluid dynamic analysis and design optimization of jet pumps

J. Fan [a,1], J. Eves [a,*], H.M. Thompson [a], V.V. Toropov [a,b], N. Kapur [a], D. Copley [c], A. Mincher [c]

[a] School of Mechanical Engineering, University of Leeds, UK
[b] School of Civil Engineering, University of Leeds, UK
[c] Parker-Hannifin, Racor Filter Division Europe, Dewsbury, UK

ARTICLE INFO

Article history:
Received 30 April 2010
Received in revised form 21 October 2010
Accepted 31 October 2010
Available online 29 November 2010

Keywords:
CFD
Optimization
Jet pump
Ejector pump

ABSTRACT

Jet pumps have a wide variety of applications and are commonly used in thermal power plants and refrigeration systems. An initial jet-pump design was developed using an analytical approach and its efficiency was improved using an efficient and accurate computational fluid dynamics model of the compressible turbulent flow in the pump, whose predictions agreed well with corresponding experimental data. Parametric studies were performed to determine the influence of the pump's geometry on its performance and the high fidelity CFD solutions were used to build surrogate models of the pump's behavior using the moving least squares method. Global optimization was carried out using the surrogates. This approach resulted in pump efficiency increasing from 29% to 33% and enabled the energy requirements of the pump to be reduced by over 20%.

1. Introduction

Supersonic jet pumps were originally developed in the 19th century to maintain vacuum pressures in the condensers of steam engines and are simple devices that can pump and compress a flow without any moving parts [1]. A typical jet pump consists of two, counter-facing, straight-sided cones separated by a parallel throat section (Fig. 1). Within this section the primary flow, in the form of a high velocity jet issuing from the nozzle, entrains (and mixes with) the secondary flow, thereby generating the pumping behavior. The second cone is divergent and acts to raise the static pressure so that it equals that downstream of the pump, thus minimizing exit losses. Research into jet pumps has seen a resurgence in recent years, since they offer an environmentally-friendly component of refrigeration and air conditioning systems that can be activated by low grade heat release from renewable energy sources such as solar energy. A wide variety of jet pump applications exist, ranging from single-phase gas flows to two-phase flows which employ liquid as the primary fluid and gas as a secondary fluid. Examples of the latter arise in steam-driven jet pumps used in thermal power plants and in refrigeration systems where nucleation and growth of condensing droplets can be influential [2].

This paper is concerned with the analysis and design of supersonic gas flows in jet pumps, and several mathematical models have now been developed to analyze and design such flow systems. The majority are one-dimensional thermodynamic models which obtain state and operating parameters along the jet pump. Eames [1], for example, developed a useful analytical method for jet-pump design where a constant rate of momentum change is prescribed within the diffuser in order to prevent the occurrence of thermodynamic shocks. However the complexity of the flow in supersonic jet pumps, involving compressible flow, shock interactions and turbulent mixing of two streams, has led to increasing reliance on computational fluid dynamics (CFD) as a design tool for improving jet pump efficiencies [3].

Previous CFD studies have highlighted the important role of good grid resolution and an appropriate choice of turbulence model in any accurate CFD analysis of supersonic flow in jet pumps. There is currently some debate about the best choice of turbulence model for such flows, with the accuracy of a range of models, such as the RNG k–ε and SST k–ω, having been analyzed [4]. Very recent comparisons between CFD and experiments have shown that the standard k–ε model is capable of predicting accurately the important global performance indicators, such as the entrainment and pressure lift ratios, and that the main difference between turbulence models lies in their prediction of local flow structure [2].

One of the key design issues limiting the wider exploitation of jet pump technology is the dramatic reduction of efficiency and pressure lift ratios that occur due to the onset of thermodynamic shocks within the nozzle. Although the effects of various aspects of the jet-pump geometry have been considered previously, there is still some disagreement over important issues such as the

* Corresponding author. Tel.: +44 113 343 6360, mobile: +44 7717 327 202.
E-mail addresses: menjfa@leeds.ac.uk (J. Fan), j.s.v.eves@leeds.ac.uk (J. Eves), h.m.thompson@leeds.ac.uk (H.M. Thompson), v.v.toropov@leeds.ac.uk (V.V. Toropov), n.kapur@leeds.ac.uk (N. Kapur), dcopley@parker.com (D. Copley), adrian.mincher@parker.com (A. Mincher).
[1] Permanent address: School of Astronautics, Northwestern Polytechnical University, Xi'an, China.

0045-7930/$ - see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compfluid.2010.10.024

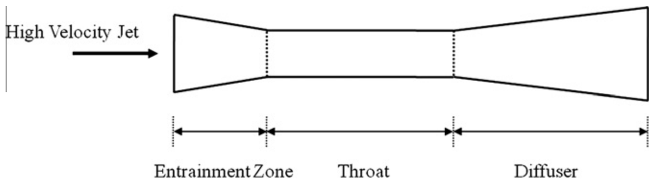

Fig. 1. A typical jet-pump design.

optimal nozzle position, pump dimensions and their dependence on operating conditions [5]. In this paper we combine high fidelity CFD analysis and experimentation within a formal optimization framework in order to improve jet pump efficiency, in terms of pressure lift and entrainment ratios, by optimizing jet pump geometrical and operating parameters.

2. Problem formulation

The compressible flow within the jet pump is modeled using the Navier–Stokes equations, based on the conservation of mass, momentum and energy [6]. Flow within the jet pump is turbulent and following several previous studies, e.g. [2], in the present work the standard k–ε turbulence model [7] is adopted. In order to quantify a jet pump's performance, the efficiency can be defined as:

$$\text{Efficiency} = \frac{\text{Power}_{\text{out}}}{\text{Power}_{\text{in}}} = \frac{\dot{V}_{\text{ent}} p_{\text{ent}}}{\dot{V}_{\text{pri}} p_{\text{pri}}} \tag{1}$$

where $\dot{V}$ is the volumetric flow rate and p is the static pressure of the primary (pri) and entrained (ent) flows, respectively. This paper focuses on the optimization of the efficiency of practical jet-pump designs.

2.1. CFD validation

An initial baseline jet-pump design (Fig. 2) was obtained using the analytical method developed by Eames [1] where a constant rate of momentum change is prescribed within the diffuser in order to prevent the occurrence of thermodynamic shocks. This is represented by the following equation:

$$\frac{dM_0}{dx} = m_g(1 + R_m)\frac{dc}{dx} = \beta \tag{2}$$

where M_0 is the momentum of the flow, x is the axial position along the diffuser, m_g is the mass flow rate through the primary flow inlet, R_m is the entrainment ratio (m_s/m_g), c is the velocity at x and β is the constant rate of change of momentum. Inlet conditions for the diffuser are calculated by assuming uniform static pressure throughout the entrainment region and an (arbitrary) low velocity at the outlet. Pressure, temperature and Mach number distributions can then be calculated based on one-dimensional compressible flow theory from which a diffuser radius profile can be determined. This should produce designs which avoid high pressure gradients and reduce pressure losses within the pump.

More detailed analysis of jet-pump behavior was based on a series of higher fidelity CFD solutions of the compressible Navier–Stokes equations governing the axisymmetric motion of the gas flows within the jet-pump geometry using the fluent CFD package [8]. Flow profiles were compared with those from the analytical model.

Results from the initial CFD study showed the flow within the pump to be significantly different from the analytical calculations (Fig. 3). This can be explained by the one-dimensional nature of the analytical method, where it is assumed that the flow entering the diffuser has a uniform Mach number. However as shown in Fig. 4, this is not the case, rather the flow entering the diffuser consists of a central supersonic jet surrounded by low speed entrained flow. This also means that the effects of thermodynamic shocks within the supersonic jet are not captured within the analytical framework. Consequently, improvements in efficiency throughout the optimization of the geometry should be possible using a higher fidelity approach than that available from the one-dimensional model.

Due to the computational expense of each CFD solution, the relative merits of two techniques for solving the compressible flow equations, namely the pressure based Navier–Stokes (PBNS) and density based Navier–Stokes (DBNS) solvers, were compared. Although the mesh independent solutions from either method were found to be indistinguishable from one another the PBNS solver was found to require significantly higher mesh densities in order to resolve detailed flow features such as the chain of shocks that occur in the supersonic jet flow (Fig. 4).

However, the overall solution time required by the algorithms on the grid densities needed to achieve mesh independence are much shorter for the PBNS approach than for the DBNS solver, as shown in Table 1, and for this reason the CFD simulations used in the optimization study carried out here were based on the PBNS approach. A study of turbulence modeling was also carried out in which the k–ε model variants and the Reynolds Stress Model (RSM) were applied to the CFD model of the jet pump (Fig. 5). All k–ε models were found to produce similar results. The RSM generated slightly better predictions of vacuum pressure when compared against experiments carried out on a prototype jet pump at Parker-Hannifin, but not enough to justify the additional

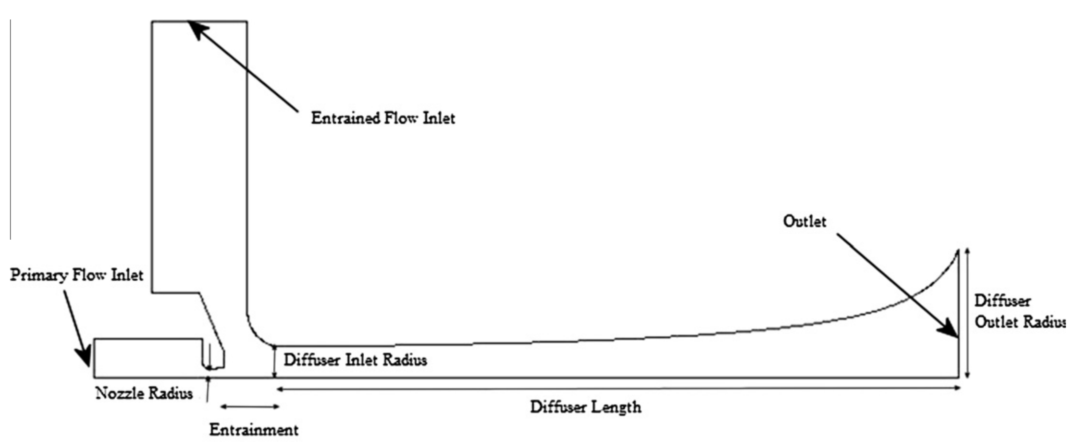

Fig. 2. Axisymmetric model of jet pump with geometrical variables.