Applied Thermal Engineering 95 (2016) 24–34

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

Simulation of spray direct injection for compressed air energy storage

C. Qin *, E. Loth

Department of Mechanical and Aerospace Engineering, University of Virginia, Charlottesville, VA 22904, USA

HIGHLIGHTS

- Novel CAES can help provide levelized wind energy power output with high efficiency.
- Water spray was employed during compression to promote heat and mass transfer.
- Multi-phase CFD is implemented using a spray discharge at various mass loadings.
- Injected mass loading of 1.6 yielded efficiency as high as 93% in compression cycle.

ARTICLE INFO

Article history:
Received 8 September 2015
Accepted 6 November 2015
Available online 14 November 2015

Keywords:
Spray injection
Droplet heat transfer
Isothermal compression
Energy storage
Simulation

ABSTRACT

Integrating Compressed Air Energy Storage (CAES) to a variable and unsteady energy source can help provide a levelized power output. This is particularly attractive for off-shore wind turbines integrated with the energy storage that has high efficiency. Such efficiency is possible if the compression portion can be isothermal, and a novel approach has been developed to achieve this by employing water spray during compression to promote heat transfer. This concept has been previously investigated with one-dimensional simulations that indicated spray cooling with droplet heat transfer over a large total surface area allows high-efficiency compression. However, the actual application is more complicated, and therefore the present study examines this concept with detailed two-dimensional unsteady flow simulations. In particular, multi-phase computational fluid dynamics is implemented in an axisymmetric domain to investigate compression in a cylinder for first-stage and second-stage compression using a spray discharge within the cylinder at various mass loadings. The spray is based on a single pressure-swirl nozzle directed along the centerline and operating at the maximum liquid mass flux possible while retaining a mean droplet diameter of no more than 30 μm. The two-dimensional simulations uncovered flow characteristics such as vortex formation for the air-flow near the cylinder head and strong spatial variations in droplet size and concentration. Despite these effects, the overall two-dimensional efficiency was similar to that of one-dimensional predictions. The results also indicated that a single pressure-swirl nozzle injection resulted in an injected mass loading of 1.6 and yielded efficiency as high as 93% for a first-stage compression cycle. However, a second-stage compression cycle (with an intake pressure of 10 bar) using this same single nozzle resulted in reduced overall work efficiency indicating that a multi-nozzle configuration should be considered.

1. Introduction

Compressed Air Energy Storage (CAES) can bring levelized power when integrated with a variable and unsteady energy source [1]. Taking wind energy as an example, off-peak or excess electricity can be used to drive compressors to pressurize ambient air, which is then stored in an accumulator. When user's electricity demands are high but the wind is not strong, the compressed air is released.

In current practice, the stored energy is extracted by burning the compressed air with fuel and expanding through a turbine to produce shaft work, which can be converted to electrical energy through a generator. Thus, a smooth electrical output can be supplied to the grid despite the variability of the wind energy. However, this wind-turbine/gas-turbine CAES system can be inefficient when based on adiabatic compression for the storage, because the built-up of thermal energy during this process is later dissipated to the surrounding atmosphere when in long term storage, before it can be utilized with the gas turbine [2]. For realistic conditions that use an underground cavern as storage accumulator, in each cycle, some of the injected energy is lost by heat transfer into the rock [3]. To avoid such problem, thermal energy storage units have to be added to absorb the heat released from air compression [4]. Additionally, this wind-turbine/gas-turbine CAES system becomes problematic for off-shore wind turbines because these gas turbines are difficult to locate, fuel and operate off-shore. Considering the trend to off-shore wind turbines in recent wind energy development, a new

* Corresponding author. Tel.: +1 (434) 872 3343.
E-mail address: chaoqin@virginia.edu (C. Qin).

http://dx.doi.org/10.1016/j.applthermaleng.2015.11.008
1359-4311/

type of CAES technology is needed to gain the advantages of energy storage.

Recently, a novel approach to compressed air energy storage was proposed for off-shore wind turbines based on a liquid piston for a compressor and expander combined with an open accumulator for storage [5]. Unlike the traditional piston, a liquid piston uses a liquid column on top of the piston surface to compress gas and this allows a shaped chamber for the portion above the mechanical piston stroke. In particular, the top part of the cylinder need not be a cylinder but can be a tapered cone to help minimize dead volume, decrease seal leakage and increase overall pressure ratio. Based on this liquid piston concept, Qin and Loth [6] modeled a spray enhanced liquid piston compression to allow near isothermal compression (to avoid wasteful adiabatic compression). This is achieved through droplet heat transfer with a large total surface area and this conceptual study showed that a high-efficiency compression process could be established with a mass loading of unity and small droplets of about 30 μm in diameter. In another study, the proposed CAES system with spray-cooled liquid pistons and multiple stages for compression was simulated for a 5-MW off-shore wind turbine baseline [7]. These one-dimensional simulation results showed that spray cooling may effectively improve the overall performance of an energy storage system, which can bring benefits to off-shore wind turbine systems.

However, the above investigations were limited to one-dimensional laminar flow simulations, and it was recommended that two- or three-dimensional effects and inclusion of turbulence should be considered to provide more realistic predictions. These can lead to effects such as turbulent dispersion and cylinder vortices that drive radial gradients of temperature and droplet concentration. Such gradients may adversely affect the net heat transfer and degree of compression efficiency and the overall performance. In the present study, multi-phase computational fluid dynamic (CFD) simulations are implemented in an axisymmetric domain with turbulent flow conditions. These simulations can help to understand the impact of droplet dispersion and spray cooling in liquid piston compression and help to characterize the predictive performance of the previous one-dimensional simulation results. This characterization can provide guidance in preliminary piston design or linear control system design that use 1D model.

The objective of this work is to employ an axisymmetric unsteady Reynolds-Averaged Navier–Stokes approach to simulate the compression portion of a single cycle for a liquid piston with spray cooling. The flow field was chosen such that axisymmetry was a reasonable approximation. In particular, the intake and exhaust valves were not included since they would require a three-dimensional approach due to their off-center location at the chamber head. Additionally, a single spray nozzle was used and placed asymmetrically. To the author's knowledge, the present study is the first to simulate spray-enhanced liquid piston compression two-dimensionally and the first to investigate effects of flow turbulence, vortex formation and radial non-uniformity on droplet concentration and overall compression efficiency.

2. Methodology

2.1. Governing equations and numerical methods

The present method employs an Eulerian numerical approach for the gas phase and the Lagrangian approach for the liquid phase. The Lagrangian approach is based on that developed by Berlemont et al. [8] for droplet vaporization in turbulent fields, with two-way coupling between phases. This type of approach was preferred over Eulerian distribution of droplet concentration (e.g. Ref. 9) since droplet collisions were included in the simulations. To ensure both numerical stability and low computational cost, the continuous phase (ambient gas) is solved by unsteady Reynolds-averaged Navier–Stokes (RANS) equations using the standard k–ε model, while the dispersed phase (droplet) is solved by trajectory discretization with a discontinuous random walk (DRW) model to produce synthetic turbulent fluctuations.

The gas-phase is treated with a standard k–ε model [10] whereby the turbulent kinetic energy (k) and its dissipation rate (ε) are solved in two transport equations:

$$\frac{\partial}{\partial t}(\rho_g k)+\frac{\partial}{\partial x_i}(\rho_g k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu_g+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho_g\varepsilon-Y_M \quad (1)$$

$$\frac{\partial}{\partial t}(\rho_g \varepsilon)+\frac{\partial}{\partial x_i}(\rho_g \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu_g+\frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho_g\frac{\varepsilon^2}{k} \quad (2)$$

In Eqs. (1) and (2), u_i refers to the gas velocity in tensor notation, x_i refers to the Eulerian coordinates in tensor notation, ρ is the density, and the subscript g indicates a gas-phase property. Herein, the constants and non-dimensional numbers have the following values: $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$. In the above expressions, turbulent viscosity (μ_t) is calculated by:

$$\mu_t = C_\mu \rho_g \frac{k^2}{\varepsilon} \quad (3)$$

The turbulent kinetic energy generation and distraction terms (G_k, G_b, and Y_M) are described in Ref. 10. In this model, two-way coupling on the turbulent kinetic energy is neglected so that none of these terms are related to the presence of the droplets. However, the two-way coupling is included between continuous phase and discrete phase momentum. In particular, interphase interaction forces act as a sink term for the mean gas momentum and interphase heat transfer acts as a sink term for the mean gas temperature.

To describe the trajectory of a discrete phase particle (droplet), the equation of motion for droplets with centroid velocity v_i is expressed in a Lagrangian reference frame [11]:

$$\frac{dv_i}{dt} = F_D(u_i - v_i) + \frac{(\rho_d - \rho_g)}{\rho_d} \quad (4)$$

In this expression, subscript d indicates a droplet property and the forces applied to the droplet are assumed to be dominated by drag and gravity forces (which is reasonable given the high ratio of droplet density to gas density). The first term on the right hand side is drag force per particle mass per relative velocity and F_D is defined as

$$F_D = \frac{3\mu_g C_D \mathrm{Re}}{4\rho_d d^2} \quad \text{for} \quad \mathrm{Re} < 2 \quad (5)$$

In this expression, Re is the drop aerodynamic Reynolds number defined along with the magnitude of relative velocity (w) and the component of relative velocity in the i direction (w_i) as

$$\mathrm{Re} = \frac{\rho_g d w}{\mu_g} \quad (6a)$$

$$w = \sqrt{w_i w_i} \quad (6b)$$

$$w_i = v_i - u_i \quad (6c)$$

A spherical droplet assumption is reasonable since the present droplet sizes are small so that their Weber numbers are much less than unity [11]. Thus, the drag coefficient C_D for spheres is used [12]. In the limit of small Reynolds numbers, typical for present conditions, this drag coefficient can be approximated as $C_D = 24/Re_p$, taken from Ref. 13.