Computational Materials Science 90 (2014) 7–15

Contents lists available at ScienceDirect

Computational Materials Science

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

Effect of injection pressure on particle acceleration, dispersion and deposition in cold spray

Shuo Yin [a,b,*], Qi Liu [b], Hanlin Liao [b], Xiaofang Wang [a]

[a] School of Energy and Power Engineering, Dalian University of Technology, Dalian 116024, Liaoning, China
[b] LERMPS (Laboratoire d'Etudes et de Recherches sur les Matériaux, les Procédés et les Surfaces), Université de Technologie de Belfort-Montbéliard, Site de Sévenans 90010, Belfort Cedex, France

ARTICLE INFO

Article history:
Received 2 January 2014
Received in revised form 17 March 2014
Accepted 25 March 2014
Available online 22 April 2014

Keywords:
Cold spray
Computational fluid dynamics (CFD)
Numerical simulation
Injection pressure
Coating
Deposition window
Dispersion

ABSTRACT

The effect of injection gas pressure on particle acceleration, dispersion and deposition in cold spray process was investigated by both numerical and experimental methods. A computational fluid dynamics (CFD) model was developed which exactly matches the real nozzle in experiment to predict the supersonic gas flow field and particle velocity prior to the impact. Based on the simulation results, it is found that injection pressure significantly affects the flow field of the driving gas. Higher injection pressure leads to higher injection flow rate as well as powder injection rate, producing thicker coating on the substrate. Besides, the particle footprints on the substrate surface at different injection pressures were predicted and compared with the experimental measurements of the single track coating width. The results indicate that particles disperse more widely at higher injection pressure. In addition, the parameter η which refers to the ratio of particle impact velocity to critical velocity is used to evaluate the effect of injection pressure on coating characteristics. With increasing the injection pressure, both η and deposition range have a downward trend, which implies the deterioration of the deposition efficiency and coating bonding strength. For the purpose of validation, the deposition efficiency of copper coatings was also experimentally measured at different injection pressures, which confirms the numerical analysis.

1. Introduction

Cold spraying is a relatively new coating technique developed in the mid-1980s and has been rapidly developing during the past two decades. In this process, powder particles (typically <50 μm) are accelerated to a high velocity ranging from 300 to 1200 m/s by a supersonic gas flow and then impinging onto a substrate in solid state without significant fusion, undergoing intensive plastic deformation. The 'low temperature' in cold spray process can minimize the adverse effect brought by molten or semi-molten state, providing a possibility to coat oxygen-sensitive materials [1,2]. It has been widely accepted that there exists a material-dependent critical velocity for a given condition (e.g. specific particle size, temperature and material properties), only above which bonding at the particle/substrate interface can take place and the cold spray coating can be formed on the substrate surface [3–7].

Therefore, much effort was devoted to investigate the particle acceleration behaviour and the consequent particle in-flight velocity. It has been widely accepted that particle in-flight velocity is highly dependent on the gas flow field inside and outside the nozzle. In this respect, a number of studies have been conducted to explore the main factors influencing the flow field of the driving gas. The results revealed that operating parameters and nozzle geometry are two dominant causes that determine the flow field of the supersonic driving gas [8–10]. Therefore, a large body of works regarding to the optimization of nozzle geometry and operating parameters were further carried out in order to achieve the optimum particle impact velocity during the coating build-up process [8,11–17]. Many meaningful conclusions were drawn from those studies. Specifically, at the given working condition, nozzles with optimal expansion ratio significantly reduce the shockwaves outside the nozzle and maximize the particle kinetic energy [8,10,11]. Also, nozzle cross-section shape was found to significantly influence the particle velocity and dispersion. Circular and square cross-sections result in higher particle velocity while elliptical cross-section makes the particles more dispersed [12,13]. Besides, the investigation on the optimization of operating

---

* Corresponding author at: School of Energy and Power Engineering, Dalian University of Technology, Dalian 116024, Liaoning, China. Tel./fax: +86 411 84707905.

E-mail address: s.yin1984@hotmail.com (S. Yin).

http://dx.doi.org/10.1016/j.commatsci.2014.03.055
0927-0256/© 2014 Elsevier B.V. All rights reserved.

parameters indicates that increasing the inlet pressure, temperature or using helium as the driving gas is able to increase the particle impact velocity [8,9,14–17].

Although there have been sizable literatures concerning the optimization of the nozzle geometry and operating parameters, only few studies focused on the significant influence of powder injection on the particle acceleration and deposition. One of the relevant studies was carried out by Han et al., which suggested that an improved injector configuration can avoid the clogging at the injector tunnel and thus enhance the final coating formation [18]. Lupoi and O'Neill reported that injector with small diameter results in narrow powder beam. Also, they pointed out that turbulence is the main reason for the particle dispersion [19]. Additionally, injection position was also found to affect the particle velocity and temperature [13]. Most recently, a numerical study further revealed that injection position even has some effects on the particle preheating temperature [20]. All the mentioned studies have showed the particular importance of injector and injection conditions in the cold spray process. It is known that, in cold spray practice, the injection pressure must be controlled to be higher than the main gas to guarantee the successful injection of powders. However, so far, the complex influence of injection pressure during the cold spray process is not clarified and a systematic study is still lacking. In this study, therefore, a comprehensive investigation on the effect of injection pressure on the particle acceleration, dispersion and deposition was carried out. CFD technique was employed as the main method due to its less economic and timing consumption. The experimental observations against the numerical results were also performed to further provide some convincing evidences.

2. Numerical methodology

2.1. Computational domain and boundary conditions

Numerical simulations were performed by using ANSYS-FLUENT 14.5 to predict the gas flow field and particle velocity in cold spray [21]. The commercial MOC nozzle (CGT GmbH, Germany) was used in this study, which exactly matches the real gun in the experiment. The dimensions of the nozzle are listed in Table 1. The two-dimensional axi-symmetric model was employed to save the computational time. The dimensions of the computational domain and boundary conditions were provided in Fig. 1. The computational domain was meshed into 126690 quadrilateral cells in order to achieve a grid-independent solution. Fig. 2 illustrates the local grid at two important regions. The grids at the nozzle throat region and impinging jet region were refined to accurately capture the rapid variation of flow properties due to the highly compressible character of the supersonic driving gas. The grid at the near-wall region was also refined to deal with the viscosity-affected region. The detailed boundary conditions of the computational model are listed in Table 2, and the operating parameters are listed in Table 3.

Table 1
Dimensions of the MOC nozzle.

Configuration	Dimensions (mm)
Throat diameter	2.7
Inlet diameter	18.2
Outlet diameter (inner wall)	6.4
Outlet diameter (outer wall)	8.4
Injector diameter (inner wall)	2
Injector diameter (outer wall)	3
Injection position from inlet	20
Divergent length	120
Convergent length	52.4

2.2. Gas phase and solid phase

Air was chosen as the working gas and the gas properties are listed in Table 4. The ideal gas law was used to calculate the density in order to take the compressibility effects into consideration. The governing equations for a two-dimensional steady compressible flow in the rotating coordinate system can be written as follows:

Continuity equation:

$$\frac{\partial}{\partial z}(\rho u_z) + \frac{1}{r}\frac{\partial}{\partial r}(\rho r u_r) = 0 \tag{1}$$

z-Momentum equation:

$$\frac{\partial \rho u_z u_r}{\partial z} + \frac{1}{r}\frac{\partial \rho r u_z u_r}{\partial r} = -\frac{\partial P}{\partial z} + 2\frac{\partial}{\partial z}\left(\mu\frac{\partial u_z}{\partial z}\right) + \frac{1}{r} \times \frac{\partial}{\partial r}\left[\mu r\left(\frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z}\right)\right] \tag{2}$$

r-Momentum equation:

$$\frac{\partial \rho u_z u_r}{\partial z} + \frac{1}{r}\frac{\partial \rho r u_r u_r}{\partial r} = -\frac{\partial P}{\partial z} + \frac{2}{r}\frac{\partial}{\partial r}\left(\mu r\frac{\partial u_r}{\partial r}\right) + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z}\right)\right] - \frac{2\mu u_r}{r^2} \tag{3}$$

Energy equation:

$$\frac{\partial \rho u_z h}{\partial z} + \frac{1}{r}\frac{\partial \rho r u_r h}{\partial r} = \frac{\partial}{\partial z}\left(\frac{k}{C_p}\frac{\partial h}{\partial z}\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(\frac{k}{C_p} r\frac{\partial h}{\partial r}\right) \tag{4}$$

Equation of state:

$$P = \rho R T \tag{5}$$

where ρ, u_z, u_r, P, T, h, μ, k, C_p, R are gas density, axial component velocity, radial component velocity, pressure, temperature, enthalpy, dynamic viscosity, thermal conductivity, constant-pressure specific heat and gas constant, respectively. Density-based implicit solver was used to solve the steady flow inside and outside the nozzle due to its great capability to solve the strong compressible flow, such as high subsonic or supersonic problem. In order to accurately capture the turbulent flow features, the re-normalized group (RNG) k–ε turbulence model was utilized instead of the commonly used standard k–ε turbulence model because this model has an additional term in its ε equation that improves the accuracy for rapidly strained flow sand streamline curvature flow. Besides, the high-order QUICK discretization scheme was applied for turbulence terms to further improve the accuracy. The non-equilibrium wall function was chosen for the near-wall flow treatment due to its great capability to deal with the large pressure gradient in the impinging jet.

Copper was used as the substrate and particle materials. All the particles have the spherical shape. The trajectory of the particles was computed using discrete phase modeling (DPM) method which requires the discrete to be present at sufficiently low volume fraction. Particle–particle interactions and the effect of particles on the gas phase were neglected due to the low volume fraction of powder particles during the cold spray process (solid phase volume fraction <10%). The high-mach-number drag law was applied to compute the particle drag force. This drag law accounts for a particle Mach number greater than 0.4 at a particle Reynolds number greater than 20. The detailed description of this drag law can be found elsewhere [22]. The drag force balance equation can be expressed as:

$$\frac{du_p}{dt} = F_d(u - u_p) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x \tag{6}$$