International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878, Volume-8 Issue-4, November 2019

Splat Behaviour Under Substrate Temperature Of 673K Via Coupling Simulation

Mohd Hafiz Mohd Noh, Ahmad Hussein Abdul Hamid, Kochi Mori

Abstract*: *The process of collision flattening is difficult to evaluate via experiment because is normally under rapid conditioned (in µsec). Via simulation, under compressible flow condition we propose a moving particle semi implicit method (MPS) coupling with finite volume method (FVM) to examine the deformation, splitting and conjoining of liquid. The substrate temperature has been constant at 673K. The result shows that the phenomena of splat behavior have been successfully being observed. The validation of droplet flow field also being confirm visually. Besides, the substrate temperature also influences the shape of the splat.

Index Terms*: *collision flattening, substrate temperature, moving particle semi implicit, finite volume method, droplet flow field

I. INTRODUCTION

There are three main process which has been involve in thermal spraying film formation process which are heating acceleration, collision flattening solidification and lastly, lamination process. All this process is difficult to view experimentally. Hence, the numerical analysis via simulation is considered to be good option to visualize these phenomena. The process involves the droplets exit from the nozzle, the initial contact of the high speed droplet into the substrate and the droplet collide, flattened and stick to the substrate.

Then, this process has attracted interest to be investigated. But there are several challenges in order to produce an accurate simulation of all this process. This entire process has taken very rapid, this explain why the experiment is difficult to be done. Besides, the droplets flattening process is a sudden deformation of the interface, liquid splitting and coalescence [1]. Volume of Fluid method is not suitable to visualize this condition instead this method is popular in multiphase flow analysis.

In order to capture an accurate behavior of droplets during hitting the substrate (initial and after), it's necessary to use the entire domain between nozzle and the substrate. This obviously requires very fine simulation grids and hence will also increase the computational time.

Second process of film formation called flatness process. The particles flatness trend was found was equivalent with $Re^{0.2-0.3}$[2]. The droplet after contact with the substrate becomes a splat. Fukumoto et. al., has done an experiment in order to evaluate the verge temperature between the splash and the splats [3].

The purpose of this research is to analyze droplet collisions. Therefore, we handle complicated phenomena such as large deformation, splitting, and phase change of the free surface of liquid. Therefore, we adopt the Moving Particle Semi-implicit (MPS) method, which is a typical particle method that makes it easy to handle the free interface. In the Navier-Stokes equation that describes the motion of a fluid, it is represented by the first derivative of the space for the convection term and the second derivative of the space for the viscosity term. It is common practice to differentiate such governing equations using a grid. On the other hand, in the particle method, the fluid is represented by a collection of particles. In the macro model, each particle is considered to represent a mass of fluid. However, since the fluid equations are written in partial derivatives, in order to perform discretization without using a computational grid, a concept different from the conventional finite difference method using finite grids and finite element method is required.

The MPS method is a method used by Koshizuka [4] which discrete calculations of continuum are performed by an inter-particle interaction model. So, we make a model of the inter-particle interaction corresponding to the partial differential operator in the equation and replace each term of the equation with the equivalent inter-particle interaction. This will lead for calculation based on the interaction between particles given the change of the variable that particles have. The particles discretized by the MPS method at this time do not represent actual molecules as in molecular dynamics, but are Lagrangian calculation points and represent one mass with fluid.

Therefore, the inter-particle interaction in the MPS method is a computational model, not only for solving fluid equations, but also for general discretization of partial differential equations. The MPS will couple together with finite volume method (FVM) to solve the interface calculation and we expect the visualization for full process of splat behavior can be achieved.

Manuscript published on November 30, 2019.
* Correspondence Author

Mohd Hafiz Mohd Noh*, Faculty of Mechanical Engineering, Universiti Teknologi MARA, 40450 Shah Alam, Selangor, Malaysia. Email: mhafiz_noh@yahoo.com

Ahmad Hussein Abdul Hamid, Faculty of Mechanical Engineering, Universiti Teknologi MARA, 40450 Shah Alam, Selangor, Malaysia. Email: a.hussein.uitm@gmail.com

Kochi Mori, Department of Aerospace Engineering, Nagoya University, Nagoya, Japan.

II. METHODOLOGY

A single block structured grid of 7.82×10^6 total grid points is being used in this calculation shown in Fig. 1. A jet from the nozzle to the substrate is given by the Riemann boundary for calculation of the gas using FVM. The specifications of this nozzle and jet are summarized in Table-I. In this study, we simulate the conditions of flame spraying, which is the most common thermal spraying method among the actual thermal spraying methods.

Table-I: Gas Jet Specification

Jet Velocity (m/s)	100
Nozzle diameter (mm)	5
Nozzle – substrate distance (mm)	50

Initially, the MPS method is calculated without considering the influence of the jet. In flame spraying, the velocity at which the spray droplets collide with the substrate changes depending on the distance between the nozzle and the substrate.

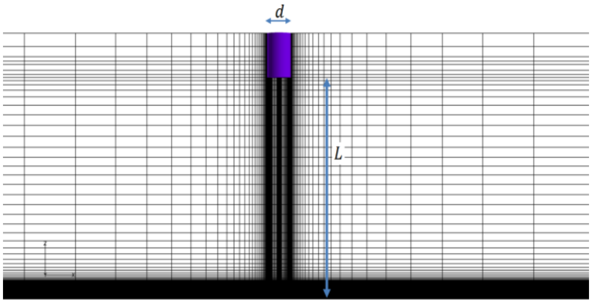


Fig. 1: Simulation grid (close up from nozzle to substrate)

Therefore, in the calculation of only the MPS method, the droplet velocity is given with reference to the literature [5].

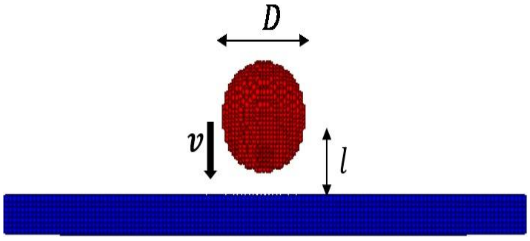


Fig. 2: Initial droplets placement

A. Finite Volume Method (Space Discretization)

The three-dimensional unsteady compressible Navier-Stokes equation is solved. The air is assumed to be an ideal gas, and the viscosity coefficient is evaluated by using Sutherland equation. The non-viscous flux is calculated by using Simple Low-Dissipation AUSM [6]. The advection term in the ξ direction of the compressible Navier-Stokes equation converted to the general coordinate system is discretized as follows.

$$\frac{\partial E}{\partial \xi} = \frac{E_{i+1/2,j,k} - E_{i-1/2,j,k}}{\Delta \xi} \tag{1}$$

Here, the grid spacing in the calculation space is set to Δξ = 1 for simplicity and

$$E_{i+1/2,j,k} = E(Q^L_{i+1/2,j,k}, Q^R_{i-1/2,j,k}) \tag{2}$$

The inviscid numerical flux has been achieved by using Simple Low-Dissipation AUSM with using (+) on the right and (-) for the left,

$$E_{i+1/2,j,k} = \frac{\dot{m} + |\dot{m}|}{2}\Phi^+ + \frac{\dot{m} - |\dot{m}|}{2}\Phi^- + p\breve{N} \tag{3}$$

Where

$$\Phi = \begin{bmatrix} 1 \\ u \\ v \\ w \\ h \end{bmatrix}, \quad N = \begin{bmatrix} 0 \\ x_n \\ y_n \\ z_n \\ 0 \end{bmatrix}, \quad h = \frac{e+p}{\rho} \tag{4}$$

B. Inter-particle Interaction Model (Weighting Function)

Weighting function was a value that indicates the influence of each particle after discretization in MPS method Introduce. And we use this weighting function in the inter-particle interaction model.

$$w(r) = \begin{cases} \frac{r_e}{r} & 0 \le r < r_e \\ 0 & r_e \le r \end{cases} \tag{5}$$

Here r is the interparticle distance. Therefore, the particles interact only if the interparticle distance is shorter than the parameter r_e. (Fig. 3) In this study, we set $r_e = 2.4$. As w gets closer to r = 0, w increases to infinity, which has the effect of making non-compression conditions work well and making the distribution of particles less likely to be uneven.

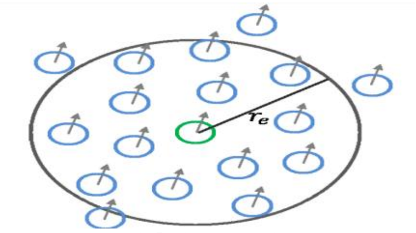


Fig. 3: Inter-particle interaction range

The heat conduction equation was solved to simulate the cooling process between the droplet and the substrate after the contact:

$$\frac{\boldsymbol{Dh}}{\boldsymbol{Dt}} = \nabla \boldsymbol{j} \tag{6}$$

where *h* is the enthalpy of each particle of the droplet and *J* is the heat flux from droplet to the base material. The heat flux is given as follows,

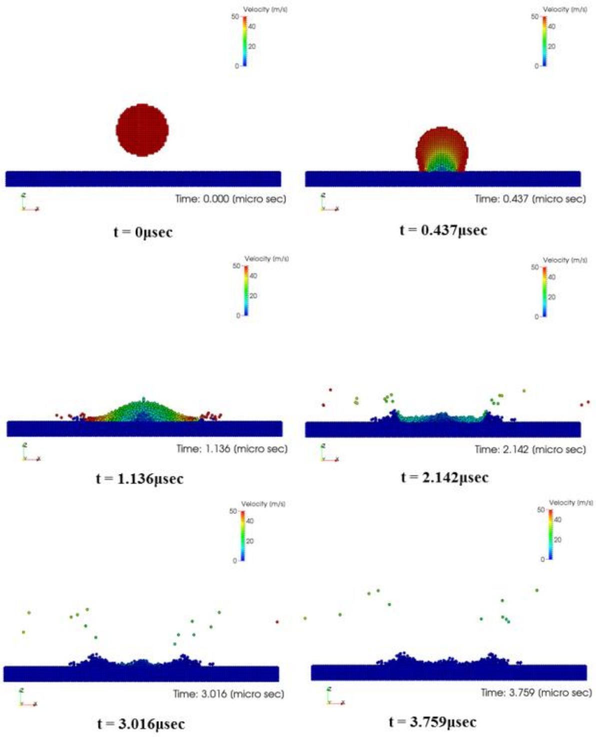
$$J = \frac{2k_1k_2}{k_1+k_2}\nabla j \tag{7}$$

where k_1 is the thermal conductivity of liquid droplet, and k_2 is the thermal conductivity of solid substrate. The thermal conduction inside the droplet and the substrate are calculated by solving Eq. (7) putting $k_1=k_2$. The present model for the thermal conduction is quite simplified ignoring the contact thermal resistance across the interface, and is much simpler than, for example, in the VOF simulation [7].

III. DISCUSSION AND RESULTS

Since the objective of this research is to look into behavior of the splat, so we look into the splat shape change condition in term of speed, temperature, heat transfer and state change. Fig. 4 shows the velocity of the change of the collision process of the thermal spray droplet when the droplet velocity is 50 m/s and the initial temperature of the substrate is 673K.

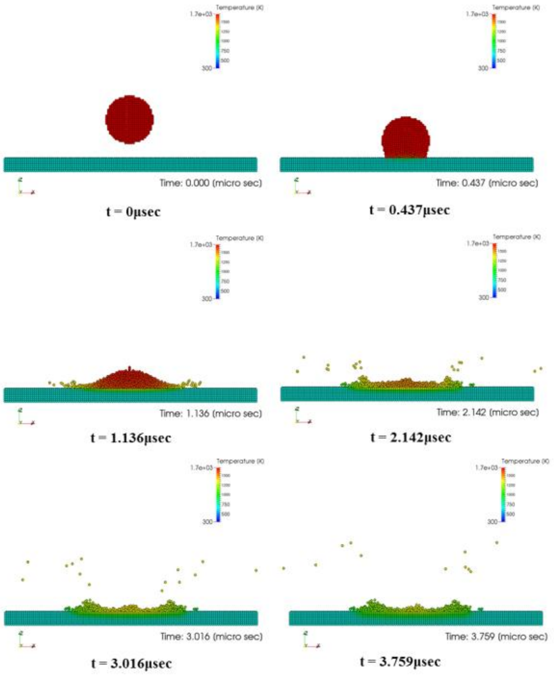
The study also extends to the investigation of the temperature change from nozzle to the substrate. The result has shown in Fig. 5. The substrate temperature is 673K, this consider as high temperature. As the droplets initial contact with the substrate surface, the outer edge portion spreads at a high speed, and as a result, it is cooled and solidified from the outer edge. From the result we can also observed the angle formed by the outer edge after solidification is slower than when the substrate temperature is low (published in other result), which believe due to the cooling rate difference.



t = 0μsec t = 0.437μsec
t = 1.136μsec t = 2.142μsec
t = 3.016μsec t = 3.759μsec

Fig. 4: Time history for velocity

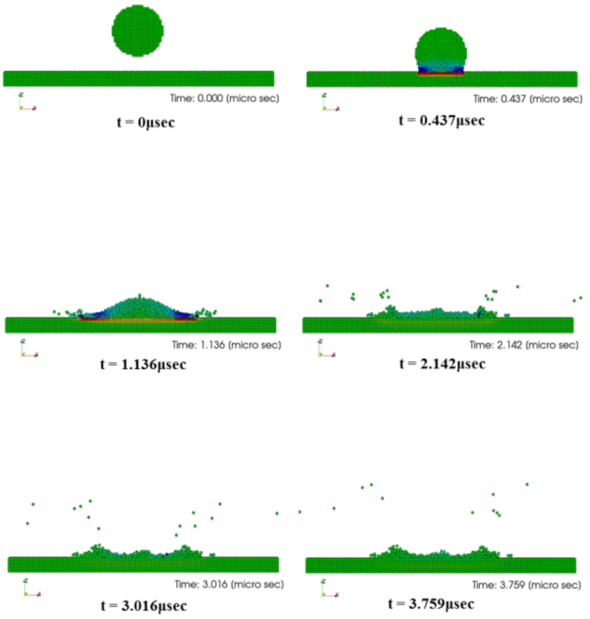
Fig. 6 shows the heat transfer simulation. At this time, the solidifying process before the speed for flattening becomes higher, the entire respective liquid cannot obtain enough speed in the flat direction, and cannot spread beyond the solidified portion and accumulates. When the substrate temperature is low, cooling occurs rapidly from the outer edge after droplet collision.



t = 0μsec t = 0.437μsec
t = 1.136μsec t = 2.142μsec
t = 3.016μsec t = 3.759μsec

Fig. 5: Time history for temperature

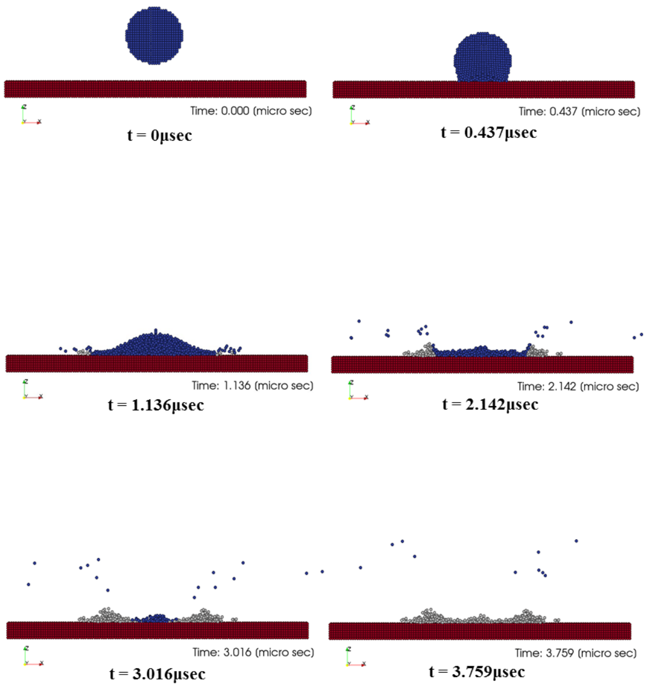
It can be said that it has become a steep outer edge by doing so.



t = 0μsec t = 0.437μsec
t = 1.136μsec t = 2.142μsec
t = 3.016μsec t = 3.759μsec

Fig. 6: Time history for heat transfer

On the other hand, when the substrate temperature is high, the area where the outer edge starts to solidify is the area away from the center (Fig. 7).



t = 0μsec

t = 0.437μsec

t = 1.136μsec

t = 2.142μsec

t = 3.016μsec

t = 3.759μsec

Fig. 7: Time history for state change

Since this causes the liquid to have a large velocity in the direction of flattening, adhesion to the substrate occurs beyond the solidified part of the outer edge, and it is considered that a splat end with a loose angle is formed. This swelling phenomenon of the outer edge of splats obtained in this calculation is similar in actual thermal spray splats, and it is known that dense splats can be obtained when the substrate temperature is high. [8], it can be said that they are consistent with these.

Fig. 8 shows a graph of adhesion efficiency. The deposition efficiency is a measure of the efficiency of thermal spraying, and indicates the rate at which the thermal spray droplets collide with the substrate and become splats and remain attached to the substrate. In addition to the low adhesion efficiency originally compared with other surface treatment methods, it is known that the adhesion efficiency is greatly deteriorated depending on the operating conditions, and the construction of this evaluation method is important [9][10].

In the adhesion efficiency graph in Fig. 8, the horizontal axis is the substrate temperature, and the vertical axis is the adhesion efficiency, and the initial velocity of each droplet is described. The graph of deposition efficiency shows that the drop velocity is 30 and 70m/s, and after the peaks of high efficiency and low peak, respectively, it fluctuates greatly. On the other hand, when the drop velocity is 50m/s, the result is monotonously increasing.

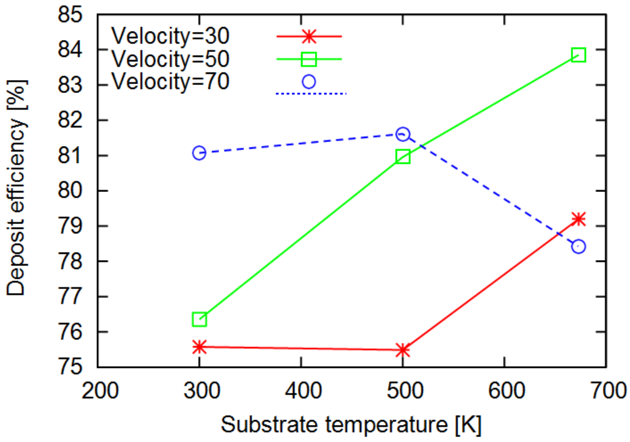
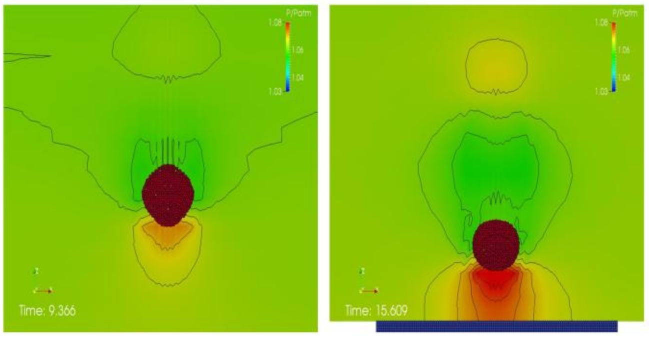


Fig. 8: Deposition efficiency with substrate temperature and particle velocity

Close-up of velocity and pressure near the droplet at 9.366μsec and 15.609μsec are shown in Fig. 9. At t=9.366μsec, the droplet is deformed forward along the flow surrounding the droplet. The head part of the droplet is compressed by the high pressure just before the droplet collides with the substrate, and the droplet shape is deformed into the shape crushed forward and backward when viewed in the traveling direction of collidation. In this way, by considering the gas-liquid interference, we analyzed the shape of the droplet just before the collision and the surrounding flow.



Flying Droplet Behavior at t = 9.366μsec

Flying Droplet Behavior at t = 15.609μsec

Fig. 9: Flying droplet behavior at t = 9.366μsec and t = 15.609μsec

IV. CONCLUSION

In this study, multi-phase flow analysis was performed by combining the finite volume method using a grid and the particle method using particles in the process of droplet flight and collision in the thermal spraying method. The results are summarized as the interaction between the particle method and the finite volume method was modeled to construct an analysis method for gas-liquid solid three-phase flow with large deformation. Droplet particles are strongly cooled and solidified from the outer edge, causing deposition to become a factor in shape determination. It was confirmed by analysis that the substrate temperature is an important factor of the flat shape.

ACKNOWLEDGMENT

This research has been supported by the Grant Lestari (600-IRMI/Dana KCM 5/3/LESTARI (196/2017).

REFERENCES

1. T. Owatari, Hakushinki, M. Fukuki., “Flat and solidifying behavior of various metal material sprayed particles on SUS304 steel substrate”, J. Japan Inst. Metals, Vol. 59, No. 11, pp. 1178-1184, 1995
2. K.Shinoda, T.Koseki and T.Yoshida, “Influence of Impact Parameters of Zirconia Droplets on Splat Formation and Morphology in Plasma Spraying”, Journal of Applied Physics, Vol. 100-074903, (2006).
3. M.Fukumoto, S.Kato and I.Okane, “Effect of Substrate Temperature on Splatting Behavior of Plasma Sprayed Ni Particles”, (In Japanese), Journal Japan Inst. Metals, Vol. 58, No. 10, (1994), pp. 1191-1195.
4. S Koshizuka and Y Oka. Moving-particle semi-implicit method for fragmenta- tion of incompressible fluid. Nuclear science and engineering, Vol. 123, No. 3, pp. 421–434, 1996.
5. Seokkwan Yoon and Dochan Kwakt. Implicit Navier-Stokes solver for three- dimensional compressible flows. *AIAA journal*, Vol. 30, No. 11, pp. 2653–2659, 1992.
6. E.Shima and K.Kitamura, “Parameter-Free Simple Low-Dissipation AUSM-Family Scheme for All Speeds”, AIAA Journal, **49**-8, (2011), 1693-1709.
7. Toyonobu Yoshida, Tomohisa Okada, Hideki Hamatani, and Hisashi Kumaoka. Integrated fabrication process for solid oxide fuel cells using novel plasma spray- ing. *Plasma Sources Science and Technology*, Vol. 1, No. 3, p. 195, 1992.
8. S.Kamnis and S.Gu, “Numerical Modeling of Droplet Impingement”, Journal Physics D: Applied Physics, **38**, (2005), 3664-3673.
9. S.Koshizuka, “A Particle Method for Incompressible Viscous Flow with Fluid Fragmentation”, Computational Fluid Dynamics Journal, **4**, (1995), 29-46.
10. J.C.Martin and W.J.Moyce, “Part IV. An Experimental Study of the Collapse of Liquid Columns on a Rigid Horizontal Plane, Philosophical Transaction of the Royal Society of London”, Series A, Mathematical and Physical Sciences, **244**-882, (1952), 312-324.

AUTHORS PROFILE

Mohd Hafiz Mohd Noh is an academic staff in the Faculty of Mechanical Engineering, Universiti Teknologi MARA, Malaysia. He received his Bachelor degree in Mechanical Engineering from University of Technology Tun Hussien Onn Malaysia (UTHM) in 2005 and Master Degree in Master of Science (Aerospace Propulsion) from Cranfield University, UK in 2008. Mohd Hafiz graduated his PhD in Aerospace Engineering from Nagoya University, Japan. His research interest is in CFD, Turbomachinery and Heat Transfer.

Ahmad Husein Abdul Hamid is an academic staff in the Faculty of Mechanical Engineering, Universiti Teknologi MARA, Malaysia. A. H. A. Hamid received his M.Sc. degree in 2008 from the same university. In 2016, A. H. A. Hamid graduated his PhD in Mechanical Engineering from Monash University, Melbourne and specialised in the field of magnetohydrodynamics and heat transfer. His research interests include wake flow of a bluff body, magnetohydrodynamic flow, heat transfer, liquid atomization and renewable energy

Koichi Mori received his Master Degree from University of Tokyo, Japan and PhD from University of Tokyo, Japan. He is a member of Japan Society of Aeronautics and Astronautics Japan and Japanese Applied Physics Society. At present, he is working as Associate Professor at Fluid Dynamics Lab, Department of Aerospace Engineering, Nagoya University Japan. His research interest is in Fluid Dynamics, Advanced Space Propulsion and Laser Technology.