Numerical approach for the estimation of the deposited material on the laser material deposition process

A. Lamikiz,*, J.I. Arrizubieta, E. Ukar, S. Martínez, J.E. Ruiz, M. Cortina

*Dept. of Mechanical Engineering, University of the Basque Country, Plaza Torres Quevedo 1, Bilbao, 48013, Spain

* Corresponding author. Tel.: +34-946-014-221; fax: +34-946-014-215. E-mail address: aitzol.lamikiz@ehu.eus

Abstract

The Laser Material Deposition (LMD) is an additive manufacturing technique that has become one of the most noteworthy methods for coating and repairing components. It is also a key process in hybrid manufacturing systems that combine LMD and machining operations. One of the main difficulties of LMD is the large number of relevant parameters and the complex interdependencies that must be considered. Thus, process fitting is usually based on trial and error techniques. The present work proposes a series of developments based on the numerical modeling of the LMD process, with the aim of reducing the experimental tests to set up the process.

© 2018 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license

Keywords: Laser Material Deposition; LMD; Powder; CFD; Numeric Model; Additive Manufacturing

1. Introduction

Laser Metal Deposition (LMD) is an Additive Manufacturing (AM) technique based on the generation of new geometry by adding filler material over a substrate [1]. The filler material is directly injected through a nozzle into a melt pool that has been previously generated with a laser beam. The resulting part is a near-net-shape component that needs to be finished. Although the density does not reach values of 100%, the mechanical properties of the parts can be compared to those of other manufacturing processes such as casting.

LMD can be used for the generation of entire parts starting from zero, but it is more likely used for the addition of some substructures or details. The LMD process offers the possibility to repair damaged parts [2] or even increase the wear or corrosion resistance properties of the bulk material by adding a coating layer with specific properties [3].

Regarding to the deposition rate, the typical values for LMD ranges 5-30 g·min⁻¹, whereas other AM processes such as Powder Bed based processes present typical values of 2-3 g·min⁻¹ [4]. On the contrary, since the dimensional accuracy of the LMD parts is worse than 0.1mm, critical features need to be machined afterwards in order to achieve the desired tolerances and dimensional accuracy.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>P</td>
<td>Laser power</td>
</tr>
<tr>
<td>Δt</td>
<td>Time step</td>
</tr>
<tr>
<td>ρ</td>
<td>Density of the material</td>
</tr>
<tr>
<td>c</td>
<td>Specific heat</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
</tr>
<tr>
<td>k</td>
<td>Conductivity coefficient</td>
</tr>
<tr>
<td>u, v, w</td>
<td>Velocity components in x, y, z directions respectively</td>
</tr>
<tr>
<td>d0, y0</td>
<td>Distance from an element to the laser beam,</td>
</tr>
<tr>
<td>r</td>
<td>Laser beam radius</td>
</tr>
<tr>
<td>L</td>
<td>Latent heat</td>
</tr>
<tr>
<td>α</td>
<td>Absorptivity</td>
</tr>
<tr>
<td>γ</td>
<td>Surface angle regarding the laser beam</td>
</tr>
<tr>
<td>T_ref</td>
<td>Surrounding room temperature</td>
</tr>
<tr>
<td>h</td>
<td>Convection coefficient</td>
</tr>
<tr>
<td>ε</td>
<td>Radiation coefficient</td>
</tr>
<tr>
<td>σ_b</td>
<td>Stefan Bolzmann coefficient</td>
</tr>
</tbody>
</table>
Part of the energy of the LMD process is absorbed by the substrate, whereas the rest is reflected and therefore lost. The heat absorbed by the substrate is transmitted towards its interior by means of conduction and a melt pool is generated. Simultaneously, filler material is added into the melt pool (both in powder or in wire form) and it is also molten by the laser beam. During the inflight time, powder particles or wire interact with the laser beam and they get heated as well as the laser beam is attenuated. Consequently the real power of the laser that reaches the surface is lower than the programmed. Laser beam attenuation values up to the 20% of the nominal laser power are obtained in the LMD process [5]. A significant fraction of attenuated energy heats the powder particles that contribute to the power balance, so it does not represent an energy loss.

Therefore, the application of the LMD in a particular environment depend on a large number of variables involving not only laser parameters, but also substrate and added material characteristics, powder shape and mass flow, gas flows, kinematics of the machine, etc. Because of this complexity, it is necessary to adjust the process experimentally by laborious trial and error techniques. In addition, LMD process is commonly applied in unique parts and algorithms for monitoring the LMD process and enhance the reliability and reproducibility of the process. There are different approaches since LMD process can be monitored focusing on different variables. Up to now, the most promising methods are those based on measuring the size or the temperature of the melt pool and acting instantaneously on the laser power in order to keep the measured signals within a desired range. The control is usually based on a conventional PID algorithm that adapts the laser power to the new conditions of the process [6]. On the other hand, many efforts are focused on the development of numerical models of the LMD process. The application of LMD in the industry would become more direct and efficient with reliable numerical tools for the estimation of the parameters and, which is more important, the estimation of the final geometry and quality of the deposited material.

Thus, the present work presents a numerical model for the LMD process with powder injection that considers the most relevant parameters and give an estimation of the result of the deposition process. The model has been also experimentally validated by measuring melt pool size and layer geometries.

2. Numerical Approach for the LMD process

Since the LMD process involves different physical phenomena and relationships among variables, the numerical model approach must be multiphysic. In order to develop an affordable numeric algorithm, which could be used in complex LMD operations, the first idea of including all the variables in the same model and solve multiple coupled algorithms has been discarded. Therefore, analyzing the LMD process, it can be divided into two major phenomena that arise simultaneously:

- A powder flow injection through the nozzle.
- The melt pool formation onto the substrate.

Thus, the numerical model is based on overlapping a thermal field model and the powder flow estimated with a CFD model based on [5]. The estimated powder flow is considered to calculate the laser attenuation effect and, in addition, the powder heating before reaching to the melt-pool. Finally, both thermal and CFD models are overlapped in order to obtain the final geometry of the deposited material. Figure 1 shows schematically the methodology and the inter-relation between the different algorithms.

![CFD simulation of the LMD nozzle](image1)

![Thermal simulation](image2)

![Material addition model](image3)

![Powder particle heating model](image4)

2.1. CFD powder simulation module

A CFD simulation system is proposed to predict the mass flow and the velocity of the powder particles. Due to the complexity of the model, commercial software are usually applied in these cases. In this model, Fluent from Ansys has been used. The results of the CFD simulation is used as input for the laser beam attenuation, powder particle heating during the inflight time and the estimation of the filler powder concentration at the surface of the substrate.

The CFD model simulates the powder particle flow through the nozzle and the inflight trajectories towards the substrate. Hence, the model needs to solve two coupled problems. First, the gas flow through the nozzle conducts. Second, the powder particle motion due of the drag effect of the gas. The gas and powder particle mixture flow is governed by the Navier-Stokes equations. Based on the work of previous authors [7] and considering the geometry of the coaxial nozzles, a turbulent regime is considered.

Since the first task is to solve the turbulent gas flow inside the LMD nozzle, the governing equations need to be solved. Following equations are represented by the Einstein summation convention:
Mass conservation

\[ \frac{\partial}{\partial x_i} (\rho \cdot u_i) = 0 \]  

(1)

Momentum conservation

\[ \frac{\partial}{\partial x_j} (\rho \cdot u_i \cdot u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho \cdot g \]  

(2)

\( \tau_{ij} \) is the component related with the viscosity. The viscosity related component is defined in Eq.(3) where \( \mu \) is the molecular viscosity and \( \mu_t \) is the turbulent viscosity. Finally, Kronecker Symbol \( \delta_{ij} \) is dimensionless and has the unit value when \( i=j \) and in the rest of the cases \( \delta_{ij}=0 \).

\[ \tau_{ij} = \left[ (\mu + \mu_t) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu_t \cdot \frac{\partial u_i}{\partial x_j} \cdot \delta_{ij} \right] \]  

(3)

The turbulent viscosity is defined by Eq. 4. In the k-ε standard method, “k” is the kinetic energy of the turbulence and “ε” is the turbulent energy dissipation. The viscosity coefficient “\( \mu_t \)” is set to a constant 0.09 value.

\[ \mu_t = \rho \cdot C_{\mu} \cdot \frac{\varepsilon^2}{\varepsilon} \]  

(4)

In the case of the k-ε standard turbulence method, two additional transport equations are required for the determination of the kinetic energy of the turbulence “k” and the rate of dissipation “ε”. The first one is the turbulence energy conservation:

\[ \frac{\partial}{\partial x_i} \left( \rho \cdot k \cdot u_i \right) = \frac{\partial}{\partial x_j} \left( \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \cdot \varepsilon \]  

(5)

The second additional equation is the turbulence energy dissipation.

\[ \frac{\partial}{\partial x_i} \left( \rho \cdot \varepsilon \cdot u_i \right) = \frac{\partial}{\partial x_j} \left( \frac{\partial \varepsilon}{\partial x_j} \right) + C_{2\varepsilon} \cdot \frac{\varepsilon^2}{k} \cdot \left( G_k + G_b \right) - C_{\varepsilon} \cdot \rho \cdot \varepsilon \]  

(6)

The values of the constant parameters introduced in the last equation: “\( C_{\varepsilon} \), “\( C_{2\varepsilon} \), “\( \sigma_k \)” and “\( \sigma_{\varepsilon} \)” are determined experimentally. Besides, “\( G_k \)” is defined as the turbulence kinetic energy generation due to the average gradient of the velocity and “\( G_b \)” is defined as the turbulence kinetic energy generation due to the buoyancy. They are defined by means of the following equations, where “\( Pr_t \)” is the dimensionless Prandtl number for the turbulent regime flow.

\[ G_k = \mu_t \cdot \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \cdot \frac{\partial u_i}{\partial x_j} \]  

(7)

\[ G_b = -g_l \cdot \frac{\mu_t}{\rho \cdot Pr_t} \cdot \frac{\partial p}{\partial x_i} \]  

(8)

Discrete phase modeling

Regarding the simulation of the powder particles, the Euler-Lagrange approximation is the classical approach. The Euler-Lagrange approximation is adequate for disperse phases, where the discrete phase does not exceed the 12% of the total volume but represent a relevant fraction of the total mass. This approximation allows considering the gas phase as a continuous and homogeneous medium in which the Navier-Stokes equations are solved in every iteration, while the discrete phase is solved every “n” iterations. Powder particles can be considered spherical, and therefore, the spherical drag law is used for determining the drag force of the particles. Besides the drag force, CFD software considers the gravity and inertial forces. In the following equation, the different forces acting on a particle in the “i” axis direction are defined.

\[ \frac{\partial u_{pi}}{\partial t} = F_B \cdot (u - u_p) + g_i \cdot \left( \frac{\rho_p - \rho}{\rho_p} \right) + F_i \]  

(9)

2.2. Heat transfer simulation module

The geometry to be modeled is divided in equally distanced elements and the finite elements method is used with 0.2 mm element size. The proposed model is based on the SIMPLE algorithm developed by Patankar for solving the pressure-velocities coupled problem [8]. When solving the transient phenomena during the LMD process, a fully implicit formulation is adopted (f=1) and a linear variation of the variables is supposed between two consecutive steps.

\[ \int_{t}^{t+\Delta t} \phi_p \cdot dt = [f \cdot \phi_p + (1-f) \cdot \phi_p^0] \cdot \Delta t = \phi_p \cdot \Delta t \]  

(10)

In each time step, the new velocity field values are obtained for each element-interface based on a staggered-grid and the pressure values for each element are calculated iteratively, until a 10⁻² Pa error is obtained.

As it is shown in Figure 2, the temperature field is iteratively calculated, creating a loop that includes the SIMPLE algorithm. The temperature field is considered to be converged when an error below 10⁻¹ K is obtained between two time steps. Besides, conduction and convection heat transfer mechanism are considered within the substrate and filler material, therefore, the following heat transfer conservation equation is considered.

\[ \frac{\partial}{\partial x} (\rho \cdot c \cdot T) + \frac{\partial}{\partial x} (\rho \cdot c \cdot u \cdot T) + \frac{\partial}{\partial y} (\rho \cdot c \cdot v \cdot T) + \frac{\partial}{\partial z} (\rho \cdot c \cdot w \cdot T) = \frac{\partial}{\partial x} \left( k \cdot \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \cdot \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( k \cdot \frac{\partial T}{\partial z} \right) + S \]  

(11)
The energy generation term $S$ represents the energy introduced by the laser ($q_{\text{laser}}$), the phase change latent-heat ($q_{\text{latent}}$) and heat losses due to radiation and convection on the surface ($q_{\text{losses}}$).

$$q_{\text{laser}} = \frac{2 \cdot \alpha \cdot \cos(\gamma) \cdot P}{\pi \cdot r^2} \cdot e^{-2\left(\frac{r_x^2 + r_y^2}{r^2}\right)}$$  

(12)

$$q_{\text{latent}} = \rho \cdot \frac{\partial T}{\partial t}$$  

(13)

$$q_{\text{losses}} = h \cdot (T - T_{\text{ref}}) + \varepsilon \cdot \sigma_b \cdot (T^4 - T_{\text{ref}}^4)$$  

(14)

### 3. Experimental Validation of the model

The validation tests have been carried out in an AISI 304 stainless steel. The composition of the material is shown in the following table.

| Table 1. Chemical composition (wt. %) of AISI 304 [9]. |
| Cr | Fe | S | Si | Mn | C | P | Ni |
| 19.0 | 71.5 | 0.02 | 0.2 | 0.08 | 0.05 | 0.02 | Bal. |

A series of corner tests have been carried out for the experimental validation. The domain dimensions are 50x50x5 mm and a constant 0.2 mm element size is used. In order to validate the LMD as a layer-by-layer process, a number of 10 layers were simulated. In Table 2 the process parameters for the validation tests are detailed.

| Table 2. Process parameters for the corner test. |
| Laser Power [W] | Feed rate [mm·min⁻¹] | Laser beam radius [mm] | Powder spot radius [mm] | Powder mass rate [g·min⁻¹] |
| 19.0 | 3.0 | 5.0 | 18.0 | 1.0 |

Once the tests have been carried out, the geometry of the tracks are measured with a Leica DCM 3D confocal microscope. In Figure 3 the Leica DCM 3D microscope and the measured topography of the deposited material is shown. After the simulation of the deposition of 10 layers, a total height of 1.094 mm is obtained in the straight part of the path. As it can be seen in Figure 3 c), the experimentally measured height of the deposited material is of 1.006 mm. Focusing on the corner, due to the feed rate reduction while maintaining the deposition rate and laser power, an expected over-deposited area can be observed. This effect is also present on the model, which estimates a 0.628 mm over height (total height of 1.722 mm). On the other hand, a 0.651 mm real over-height is measured in the experimental tests. Both the simulated and tested corners are shown in Figure 4 a) and b), respectively.
Acknowledgements

This work has been founded by the project H2020-FoF13-2016 PARADISE project (contract No.: 723440). This work has been also carried out in the framework of the POCTEFA 90/15 Transf3D project.

References


