Available online at www.sciencedirect.com

ScienceDirect

Physics Procedia 56 (2014) 429 – 438

Physics Procedia

8th International Conference on Photonic Technologies LANE 2014

# Continuous coaxial nozzle design for LMD based on numerical simulation

J.I. Arrizubieta[a,*], I. Tabernero[b], J. Exequiel Ruiz[b], A. Lamikiz[b], S. Martinez[b], E. Ukar[b]

[a] *"La Caixa" Foundation Scholarship holder. Department of Mechanical Engineering. University of the Basque Country UPV/EHU. Alameda Urquijo,s/n, 48013-Bilbao-Spain*

[b] *Department of Mechanical Engineering. University of the Basque Country UPV/EHU. Alameda Urquijo,s/n, 48013-Bilbao-Spain*

**Abstract**

The LMD technology is becoming one of the most important emerging manufacturing technologies in the modern industry, due to its benefits when building-up geometries, repairing damaged parts or the creation of coatings to improve material properties and behaviour. One of the most relevant parameters in LMD process is the efficiency of the trapped powder into the melt pool, since metallic material powders use to be very expensive.

With the aim of improving the ratio between the trapped powder in the deposited area and the total injected powder, the work presents a new methodology for continuous coaxial nozzle design for the LMD process based on a complete CFD model. The numerical model can predict particle flow, speed, powder concentration, etc. and design can be optimized using this input data.

The model has been validated and then, it has been used for the design of two different nozzles: one discrete and one continuous coaxial nozzle.


Peer-review under responsibility of the Bayerisches Laserzentrum GmbH

*Keywords:* Laser; LMD; Coaxial Nozzle; Simlation based design

* Corresponding author. Tel.: +34-946013932; fax: +34-946014215 .
*E-mail address:* joninaki.arrizubieta@ehu.es


Peer-review under responsibility of the Bayerisches Laserzentrum GmbH
doi:10.1016/j.phpro.2014.08.146

## 1. Introduction

The geometry of the nozzle and the operation conditions are of major importance in the Laser Metal Deposition process (which from now will be called LMD), not only to increase the efficiency of the process, also to ensure the quality of the clads, Lee (2008) and Cheickh et al. (2012) .This is why, the objective of the present paper is the design of a continuous coaxial nozzle based on a commercial CFD model. For this purpose, a methodology previously validated by Tabernero et al (2010) have been followed, in which have been proved that the results obtained from the simulation of the nozzle resemble to the real LMD process, Tabernero et al (2014).

The design of the nozzle is a multidisciplinary process, in which many phenomena have to be taken into account, Balu (2012), Tabernero (2014). The powder flow in the spot is important in order to obtain the maximum efficiency of the process, but not only the size and shape of the melt pool need to be controlled. Also the velocity of the particles have to be among reasonable values, because as it was shown in Oliveira (2005) too high particle speeds implies huge amounts of laser power in order to melt the powder fast enough so that melt pool is stable. Another important point, especially if the laser is going to be used for long periods of time, is the heating of the nozzle due to the reflexions of the laser beam in the part. Therefore, in order to consider this heating and its effect in the process, a thermal simulation is necessary. Furthermore, the nozzle is under the influence of different variables that may change during the LMD process, and they may have a great influence in the powder distribution. As it can be seen in the point 4.1, one of the most critical variables is the inclination angle of the nozzle in relation with the vertical, what modifies the influence of the gravitational forces and consequently the powder distribution.

Before starting with the preliminary design of the nozzle, a review of the state of art and the work done previously have been carried out, Lamikiz (2010). From the originally designed robust and coarse nozzles, they have become finer, with narrower tracks for the powder flow in order to obtain a smaller focusing point.

In the year 2004 a compact nozzle was proposed, specially designed for its use in zones where the geometry of the piece is restricted, Wen Guo (2004). It contains also an external cooling ring to avoid excessive heating of the nozzle. Despite the more stylized shape of the nozzle in comparison with the previous designs, the powder injection angle was very high. This means that the focal plane is near to the lower surface of the nozzle, small focusing distance, so that the heat that reaches the nozzle via radiation from the heated part is very high. In order to avoid this overheating and increase the accessibility of the nozzle, the focusing distance of the powder can be increased. But this has also its drawbacks, because also increases the dispersion of the powder stream, resulting directly in a lower efficiency.

With the aim of solving this problem, Whitfield (2008) proposed a triple nozzle, in which an extra protection gas flow is used as an outer guidance cone. This outer flow creates also a protective atmosphere, especially useful when very reactive materials, such as Titanium alloys, are processed. Furthermore, it has an inner vacuum channel to optimize the powder flow. One year later, Sato (2009), patented an enhanced internal cone, which helps to orientate the powder flow and reduce its dispersion.

Based on the review of the different designs that exist nowadays in the market, taking into account the characteristics of the machine in which will be mounted and the requirements of the processes it will be used for, a modular nozzle have been proposed. In it, the different parts will be assembled using the minimum amount of threaded joints and building of conic tighten. This way, if the nozzle suffers a collision or any other breakage, will be easier to change just the damaged part.

The following points should be taken into account in the its design: The nozzle will be used for 5 axis LMD operations, consequently it has to be as sharp as possible in order to gain accessibility. However, the powder stream has to attenuate as little as possible the laser beam before it reaches the part. The velocity of the powder particles is limited in the melt pool, establishing a limit of 2m/s, Oliveira (2005). Furthermore, in order to avoid too high consumable costs, the drag and protection gas flows have been limited to a maximum of 7-15l/min.

## 2. Simulation of the powder flux

In this point all the details regarding to the powder simulation are explained and the evolution of the nozzle is shown. For the analysis of the powder flux, the outer cone has been left out, because it has no influence over it and would increase the computational cost.

### 2.1. Materials

The nozzle is composed of five bodies, the ones that have threaded joints, will be manufactured in AISI 316 stainless steel: The outer cone (1) and the joint ring (4), whereas the others: the intermediate cone (2), the inner cone (3) and the conical ring (5), will be manufactured in Aluminium 7075, which thanks to the high conductivity and its low density together with a high machinability is the suitable material for the nozzle. The cooling channel and the powder distribution channel are represented with the letters C and D respectively.

As the roughness of the inner face of the inner cone has no influence in the process, the hole is needed just to let the laser beam pass throw, it will be manufactured in a stepped way in order to make its manufacture easier (even if for the simulations it has been taken as a conical smooth surface).

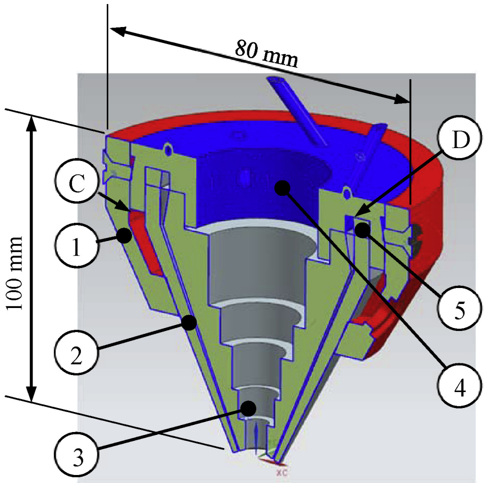


Fig. 1. Cross section of the nozzle. (1) Outer cone; (2) Intermediate cone; (3) Inner cone; (4) Joint ring; (5) Conical ring.

The gas used, both for protection, as well as for dragging the powder particles is Argon, which is widely used for this kind of applications due to its inert behaviour. So that, besides dragging the powder, it can protect the melt pool from the atmospheric oxidation and avoid unwanted reactions.

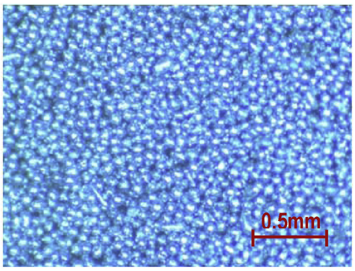


Fig. 2. Photograph of the Inconel 718 powder obtained with an optical microscope at x5 magnification.

The powder used for the simulations is the Inconel 718, which is an austenitic nickel-chromium based superalloy. Its composition is detailed in Table 1. The grain size of the powder used is contained between 45 and 100 µm, with an average diameter of 90 µm. The powder particles real distribution was analysed in Tabernero et al (2010), where was concluded that they have a Rosin-Rammler distribution. During the CFD simulation the interaction between the continuous phase (Argon) and the discrete phase (Inconel 718 powder) was taken into account, what improves the accuracy of the results.

Table 1. Composition of Inconel 718 in weight percentages (commercial name MetcoClad-718, source Sulzer).

| | Ni | Cr | Mo | Fe | Nb | Ti | Si |
|---|---|---|---|---|---|---|---|
| Inconel 718 powder | Bal. | 19 | 3 | 18 | 5 | 1 | 0,2 |

### 2.2. Boundary conditions and simplifications.

In order to simulate the fluid flow in the most possible realistic conditions, the following boundary conditions are applied at the faces of the simulated volume:

- Fluid intake: At the inflow sections, the velocity value is defined.
- Fluid exit: At the flow exist section; an outflow condition is defined, what does not enforce a fixed value of pressure nor velocity. An exit ratio of 1 was defined.
- Wall condition: A zero velocity condition is established at those faces, so that the fluid cannot go through them.

Furthermore, some simplifications are needed in order to simulate the fluid flow inside the nozzle and make the problem approachable but with realistic results.

- ✓ At the flow inlet sections, the flow is supposed to be perpendicular to the section and with a constant velocity value.
- ✓ The problem is solved as a static problem, where the powder distribution is not time dependent.
- ✓ As there are not expected great density variations of the fluid phase, a segregated scheme is used, in which per DPM iteration, 5 continuous phase iterations are carried out.
- ✓ Regarding to the viscosity model, a κ-ε standard turbulence model is used.
- ✓ A Rosin-Rammler distribution is used for the powder particles diameters.
- ✓ No interaction between the different particles is taken into account. Only the gravity, drag and inertial forces are simulated. A spherical drag law scheme is used.
- ✓ No interaction between the laser and the powder particles occurs. They are not heated up.

### 2.3. Evolution of the design

After the first simulations, two problematic zones were found inside the nozzle related to the particle paths. On the one side, when the powder was injected into the distribution chamber, due to the centripetal component of the velocity, the powder accumulates in the outer side of the distribution chamber, instead of flowing out of the nozzle. In order to solve it and guide the powder towards the exit of the nozzle, a chamfer was designed. The chamfer has been manufactured in another part (the conical ring), so that, in addition to the function of guiding the powder, it works as an adjustment part that makes easier the final assembly.

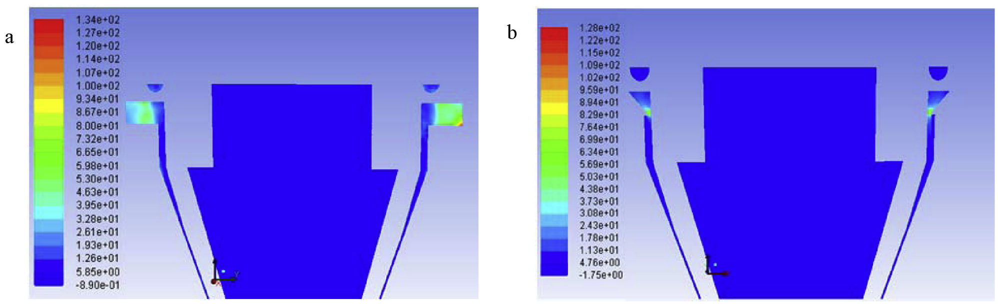


Fig. 3. (a) Distribution of the powder in a XZ section; (b) Distribution of the powder after the employ of a chamfer.

On the other side, there appeared some problems at the exit of the nozzle regarding to the powder concentration. If the tangential component of the particles velocity is not completely eliminated in the distribution chamber, as the nozzle narrows, the curvature radius decreases and the centripetal force increases ($\overrightarrow{F_{ZP}} = m.\vec{a} = m.\vec{v}^2/r$). As a consequence of this, at the nozzle exit appears a spray effect, where is impossible to direct the powder into the focus point. This phenomena is shown in the Figure 4.

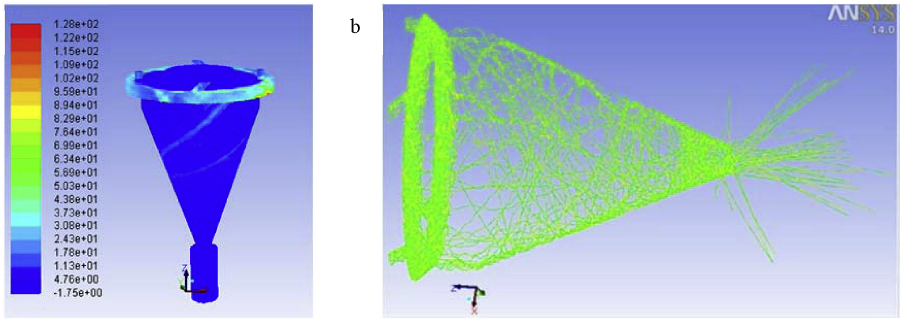


Fig. 4. (a) Distribution of the powder in an isometric view; (b) Powder particle paths.

The solution for this problem is the manufacture of some small groves, 0,5x1mm, in the periphery of the internal cone (it is much easier their accomplishment in the external face of the inner cone rather than in the internal face of the outer cone). This way, when the powder particles exit the distribution chamber, the tangential component of their velocity is almost zero and they can be directed properly, what enables their focusing in a small area. The resulting nozzle can be seen in the Figure 5.

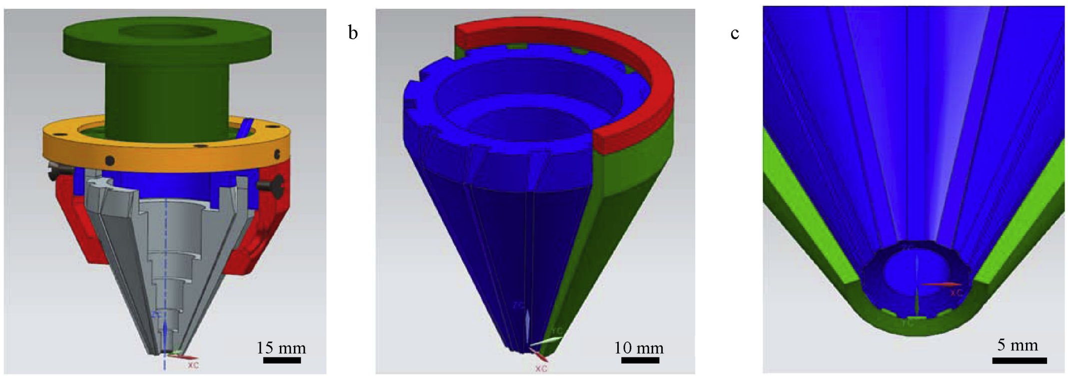


Fig. 5. (a) Geometry of the nozzle with the grooves in the inner cone; (b) and (c) detail of the grooves: Inner cone together with the half section of the intermediate cone and the conical ring.

## *2.4. Analysis of the optimum gas pressure*

Once the design of the nozzle was finished, the influence of the drag and protection gas flows have upon the other parameters have been analysed in order to determine their optimal values. The analysed output parameters are the internal pressure of the nozzle, which shows a linear increase with the increase of the protection gas flow; the flow velocity in the spot, which is influenced by both protection and drag flows; the powder flux maximum in the focal plane and the powder spot diameter.

As an example, the results for the output "flow velocity in the spot" are shown in Table 2. The velocity of the particles in the focal plane exhibits almost a linear increase when the drag and protection flows increase. In order to fulfil the requirements established in the introduction, flow velocity in the spot has to be comprised between 1-2m/s. Therefore, the optimum $v_{drag}$ and $v_{protection}$ are 1,75 and 1,5m/s respectively. Similar analyses have been carried out for the rest of the outputs parameters mentioned in the previous paragraph.

Table 2. Influence of the $v_{protection}$ and $v_{drag}$ over the velocity flow in the spot.

| Flow velocity in the spot [m/s] | | V drag [m/s] | | | |
|---|---|---|---|---|---|
| | | 1,75 | 2,5 | 3 | 4 |
| | 1,5 | 1,68 | 2,12 | - | - |
| V protection | 2 | - | 2,35 | - | 2,89 |
| [m/s] | 2,5 | - | 2,9 | - | - |
| | 3 | - | - | 3 | - |

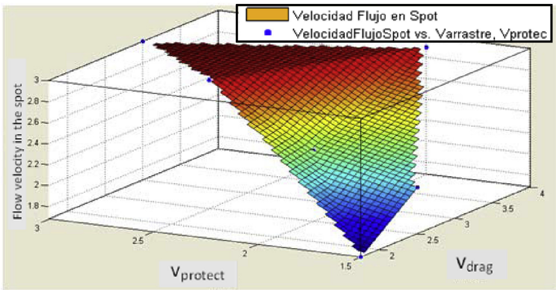


Fig. 6. Variation of the powder flow velocity based on the protection and drag flow velocities.

Based on the results obtained, after the analysis of all the output parameters, it has been established that the optimum flow values for the protection and drag gases are those flows that enables a flow velocity of $v_{protection}$=1,5m/s and $v_{drag}$=1.75m/s at the entrance of the nozzle.

## 2.5. Results

Once all the parameters have been determined, a last simulation has been carried out in order to check whether the design and the established gas flow values are the correct ones. In Figure 7.a can be seen the mesh used for the simulation. The quality of the mesh have been analysed by means of the skewness parameter, lower than 0.89 in all cases, what ensures the convergence of the CFD simulation and the reliability of the results.

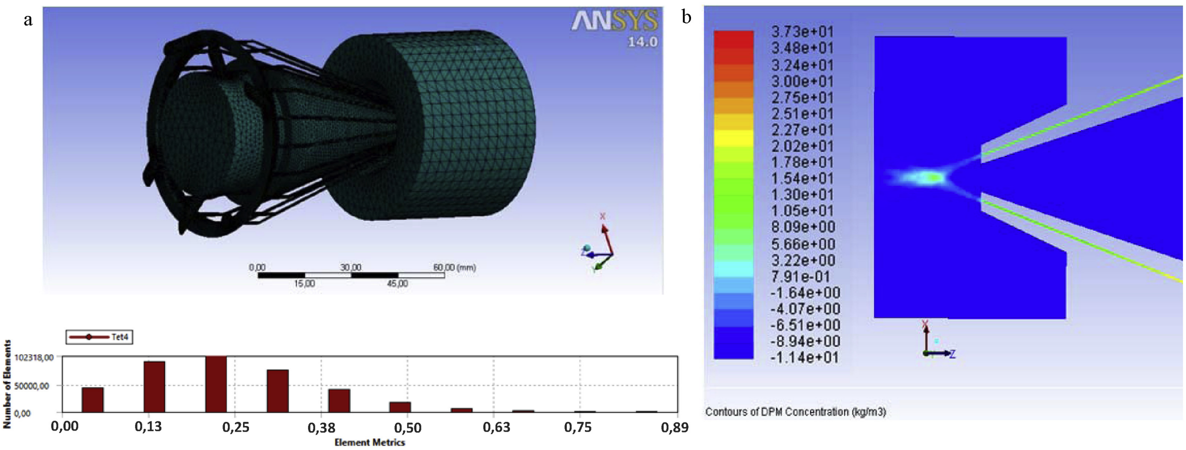


Fig. 7. (a) Mesh employed and the skewness of its elements; (b) Particle concentration [kg/m$^3$] after the CFD simulation.

The particle flux (kg/m$^2$s) have been analysed on the negative Z axis, in order to find the maximum powder concentration plane. For the nozzle analysed and the protection and drag flows employed, that plane, called powder focal plane position, is located 11mm below the lowest face of the nozzle, Figure 8.a. Furthermore, it can be seen that in a 3mm upper plane the powder flux decreases below the 30% of the maximum in the focal plane, Figure 8.b. This characteristic is really important for the LMD process, because it means that in a 3mm upper plane the Gaussian stream is not formed yet, so the absorption of the laser beam is minimal.

For the determination of the powder spot diameter, all the area in which the powder concentration is higher than the 13,6% of the maximum value ($1/e^2$) has been considered as the powder spot. So the diameter of the powder spot is around 3,5mm.

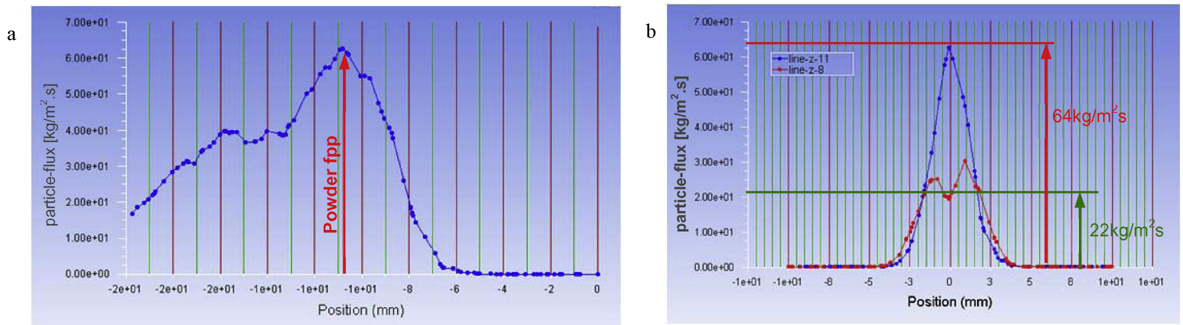


Fig. 8. Particle flux in [kg/m$^2$.s] is shown. (a) Particle flux along the negative Z axis. (b) Powder flux in different Z-levels.

## 3. Thermal analysis

The nozzle can be used continuously for many hours depending on the required LMD operation. So, due to the reflexions of the laser beam in the workpiece and the radiation released by the workpiece because of its high temperature (especially in the melt pool and it surroundings) the nozzle heating cannot be neglected. In order to quantify this heating, a steady thermal analysis has been carried out for the different working conditions and temperatures.

The cooling system has to be independent form the powder flow. For this purpose, a cooling channel has been designed between the outer and the intermediate cones. Because of its high latent heat and low cost, water is used as the coolant fluid.

### *3.1. Simulation conditions*

For the thermal analysis, the whole geometry must be considered, taking into account the different materials of the different parts (see Figure 1) and their conductivities. In order to make easier the thermal simulation, small geometrical details such as the grooves and the powder distribution channel have been omitted, because their influence in the thermal diffusion is almost negligible. Furthermore, a full contact between the different faces is supposed.

Regarding to the boundary conditions, the influence of the convection is neglected and the radiation is taken into account just in the faces numbered as 1 and 2 (see Figure 9). As an initial approach, it has been supposed that the radiation coming from the heated part reaches the nozzle from the faces 1 and 2.

Table 3. Different thermal simulation cases carried out.

| Coolant flow [l/min] | Temperature of the enviroment [K] | |
|---|---|---|
| | $T_{radiation}$_1=1000K; $T_{radiation}$_2=500K | $T_{radiation}$_1=1200K; $T_{radiation}$_2=600K |
| 1,333 | Case 1 | Case 2 |
| 1 | Case 3 | Case 4 |
| 0 (without coolant) | Case 5 | - |

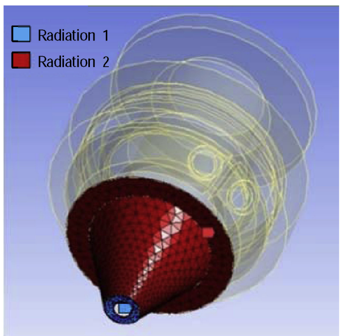


Fig. 9. Faces irradiated by the radiation coming from the heated workpiece.

### 3.2. Results

Two points have been defined to control the temperature variations and plot the results. The first one is located in the underside of the nozzle and the second one in the contact edge between the nozzle and the water. Furthermore, the temperature increase of the coolant has been calculated, because if the temperature difference between the in and out sections is too high, the coolant deposit will heat up and extra equipment should be used to maintain it at room temperature.

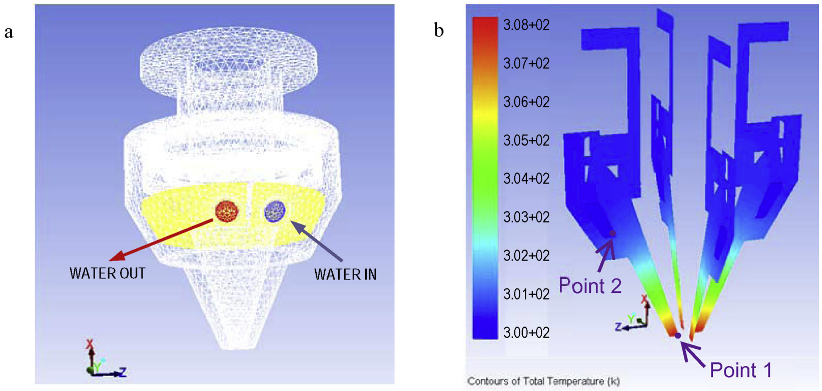


Fig. 10. (a) Mesh employed for the thermal simulations; (b) Points analysed inside the thermal field.

Table 4. Results of the thermal analyses.

| | Case 1 | Case 2 | Case 3 | Case 4 | Case 5 |
|---|---|---|---|---|---|
| Max. Temp. reached (point1) [K] | 307,9 | 3,16,9 | 308,0 | 317,2 | 540,71 |
| Temp. in the interface between the coolant and the nozzle (point2) [K] | 300,9 | 302,5 | 301,1 | 302,3 | 536 |
| ΔTemp coolant [K] | 0,28 | 0,58 | 0,39 | 0,88 | - |

In case that no cooling system is used, the nozzle reaches temperatures above the 250ºC. This may be problematic, because the pipes used to deliver the powder to the nozzle are made of plastics. Therefore, they can melt as a consequence of the high temperatures reached, resulting in the bock of the powder flow and the process interruption.

On the other side, with the employ of a water flow between 1 and 1,4 l/min this heating is minimized and the maximum temperatures reached by the nozzle are below 50ºC. This is why, a cooling system is necessary for long LMD operations.

## 4. The influence of the inclination angle

The designed nozzle will be mounted in a 7 axis articulated robot. Therefore, during the LMD process it can vary its orientation in relation with the vertical. This is why, in this point the influence of that inclination angle have been analysed, in order to determine its maximum value that ensures the correct LMD process.

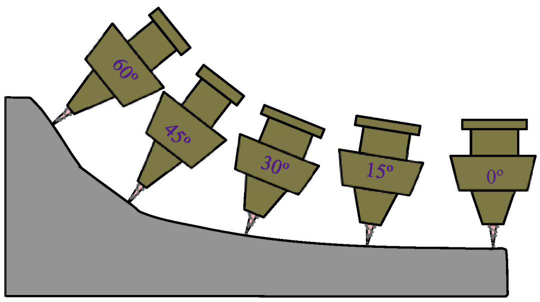


Fig. 11. Different positions and inclinations of the nozzle during the repair of a mould.

The cases of 0º, 15º, 30º, 45º and 60º have been simulated. In each case, instead of spinning the whole mesh, the gravity vector direction has been changed. Thus, for each situation the powder distribution, the position of the fpp and the concentrations reached in the spot at that plane have been analysed.

Table 5. Influence of the inclination angle on the gravitational vector components.

| Inclination | 0º | 15º | 30º | 45º | 60º |
|---|---|---|---|---|---|
| $g_z$ [m/s$^2$] | 9,81 | 9,476 | 8,496 | 6,937 | 4,905 |
| $g_x$ [m/s$^2$] | 0 | 2,54 | 4,905 | 6,937 | 8,496 |

As the inclination increases, the powder distribution becomes asymmetric and the nozzle is not able to distribute the powder uniformly in the distribution chamber.

### 4.1. Results

It has been verified that the inclination angle, is one of the most important parameters regarding to the particle flux distribution, because it influences directly in the correct distribution of the particles among the distribution chamber.

For an angle of 45º, as it can be seen in Figure 12, particles are not distributed in the whole periphery and they do not go down from all the grooves manufactured for that purpose. As a consequence of this, the LMD would be predominantly from one size and instead of having a continuous coaxial nozzle; it would work as a lateral discrete nozzle.

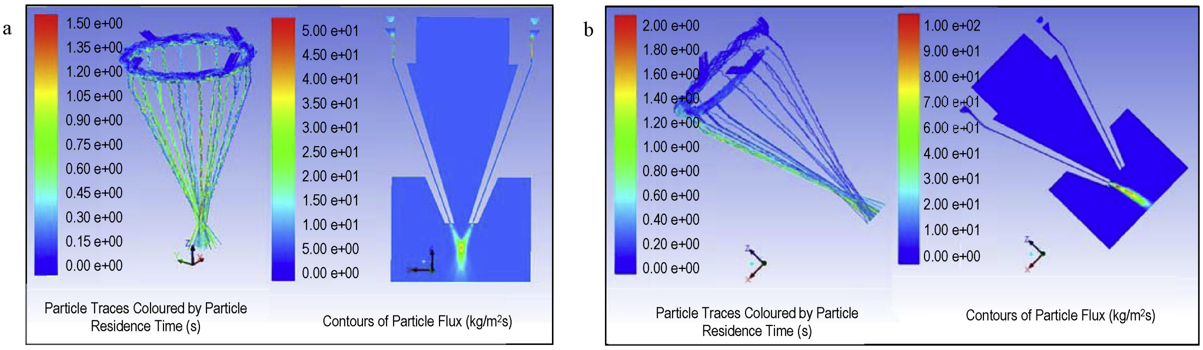


Fig. 12. Powder particle paths coloured according to their residence time and particle flux concentrations for: (a) 0º inclination angle and (b) 45º inclination angle.

I can be concluded that the designed nozzle works correctly until inclinations of 30º. For angles below this value, the focal plane position remains almost constant (±0,5mm) and the particle concentration and spot size are adequate for the LMD process. However, with higher inclination values it cannot be ensured the correct operation of the nozzle.

## 5. Conclusions

In this paper the whole design process of a LMD nozzle is shown. With the CFD model, it has been possible to predict the powder flow inside the nozzle, figure out the problematic zones and correct them in order to obtain de desired results. Furthermore, an optimization of the protection and drag flows have been possible, this way the set up process could shortened and both time and money would be saved up.

In addition to that, a thermal analysis of the nozzle has been carried out. There, the use of a water cooling system has been simulated. It can be concluded that its use is necessary in order to avoid excessive heating during extended periods of use and maintain the nozzle temperatures under a reasonable value. The overheating is especially dangerous for the powder delivery system, because they are made of plastics, so it is important to control it.

Lastly, the CFD model have been used to predict the powder distribution when the nozzle works with different angles regarding to the vertical and the maximum tilt value have been established. Therefore, the maximum inclination angle until which the correct function of the nozzle is ensured has been concluded. If higher values are used the nozzle may work but it cannot be ensured the Gaussian distribution of the powder within the spot and it would not work as a continuous coaxial nozzle. Instead of that, it would work as a lateral discrete nozzle, so it would not be able to add material in all the directions.

## Acknowledgements

Thanks to “la Caixa” Foundation for its financial help. Thanks are addressed to the Department of Education, Universities and Research of the Basque Government for their financial support. Special thanks are addressed to the Industry and Competitiveness Spanish Ministry for the support on the DPI2010-20317-C02-01 SURFACER project. We thank also the UFI in Mechanical Engineering of the UPV/EHU for its support to this Project.

## References

[1] I. Tabernero, A. Lamikiz, E. Ukar, L.N. López de Lacalle, C. Angulo, G. Urbikain, 2010. Numerical simulation and experimental validation of powder flux distribution in coaxial laser cladding. Journal of Materials Processing Technology 210, 2125–34.
[2] A. Calleja, I. Tabernero, A. Fernández, A. Celaya, A. Lamikiz, L. N. López de Lacalle, 2014. Improvement of strategies and parameters for multi-axis laser cladding operations. Optics and Lasers in Engineering 56, 113–120
[3] U de Oliveira, V Ocelik, J.Th.M. De Hassan, 2005. Analysis of coaxial laser cladding processing conditions. Surface and Coatings Technology 197, 127-36.
[4] I. Tabernero, A. Lamikiz, E. Ukar, S. Martínez, A. Celaya. 2014. Modelling of the geometry built-up by coaxial laser material deposition process.l Int. J. Adv. Manufacturing Technology, Vol. 70, 843-51
[5] Wen Guo, 2004. Compact coaxial nozzle for laser cladding. Patent US 7259353 B2
[6] R.P Whitfield, 2009. Laser cladding device with an improved nozzle. US20090095214 A1.
[7] A. Sato, I. Yoshinori, N. Steffen, S. Siegfried, 2009. Powder metal cladding nozzle. US7626136B2.
[8] H.-K. Lee, 2008. Effects of the cladding parameters on the deposition efficiency in pulsed Nd:YAG laser cladding. Journal of materials processing technology 202, 321-27.
[9] H. El Cheickh, B. Courant, S.Branchu, J. Hascoët, R.Guillén, 2012. Analysis and prediction of single laser tracks geometrical characteristics in coaxial laser cladding process. Optics and Laser in Enginering 50, 413-22.
[10] A. Lamikiz, I. Tabernero, E. Ukar, S. Martinez, L.N, López de Lacalle,2011. Current Designs of Coaxial Nozzles for Laser Cladding. Recent Patents of Mechanical Engineering, 4, 29-3629.