The Negative Effect of Turbulences in Supersonic Gas Flow During Cold Spray

William Cusato

August 13, 2026

Contents

1 Summary of Article 1
2 Details of Turbulence Analysis 2
2.1 DNS Solver . . . 2
2.2 Shadowgraphy . . . 4
3 Critical Review of Article 4
4 Impact of Article 6
5 Future Work 7

1: Summary of Article

Cold spraying is a subset of additive manufacturing that has recently gained attention as an alternative to traditional manufacturing methods. Specifically, cold spraying is a coating technique in which metal particles are deposited onto a substrate by accelerating them to supersonic conditions with a pressurized transport gas and a converging–diverging (CD) nozzle. While analytical frameworks based on isentropic flow principles exist to characterize particle motion, no comparable framework addresses the behavior of the gas itself. Raoelison *et al.* focus on this gap by examining the transport gas and highlighting how turbulence within it can negatively impact the overall cold spray system. CFD analysis of cold spray flow has largely disregarded the effects of turbulence within the gas. This omission is problematic, as particle movement depends on instantaneous property variations, phenomena that turbulence strongly influences.

To investigate the effects of turbulence in cold spray technology, Raoelison *et al.* employ a two-front approach combining CFD analysis with high-speed laser visualization. A simplified two-dimensional nozzle geometry is used to examine both the internal flow and the freestream outside the nozzle, with boundary conditions simulating atmospheric release. Notably, a Direct Numerical Simulation (DNS) CFD method is chosen over turbulence modeling for its ability to capture flow behavior in a phenomenological context. Key simulation parameters include a mesh size of 70 µm and a time step of 1 µs. On the experimental side, a De Laval nozzle, featuring a long, gradual diverging section and a short, sharp converging section, is used to accelerate preheated, pressurized nitrogen gas. Typical cold spray copper particles serve as flow tracers. Data are captured using a charged-coupled device (CCD) that records laser illumination with a 1 µs shutter speed at intervals of 1 ms.

Shock waves are known to exist within cold spray De Laval nozzles as a result of the long divergent section reducing static pressure below atmospheric levels. Raoelison *et al.*'s DNS simulation reveals that these shocks create a series of pressure peaks that gradually attenuate as the flow progresses down the nozzle, consistent with previous RANS analyses. However, the pressure fluctuations are far more intense in the DNS results, which the authors attribute to turbulence effects that RANS models cannot capture. Additionally, the DNS simulation shows that the supersonic expansion induces significant turbulence extending into the freestream. High vorticity levels are observed, primarily due to the injection of cooler flow (carrying the cold spray particles) into the main stream. This creates a shear plane between the upper and lower walls, where flow properties are disrupted at the injection point. The resulting vorticity, most evident near the bottom of the nozzle, generates irregular flow instabilities that propagate toward the outlet. These instabilities cause shearing across the nozzle exit and induce the introduction of external fluid into the nozzle, decreasing local temperature and velocity. The outcome is a recurrent, unstable jet movement, oscillating up and down, which contributes to localized thermal confinement near the upper wall, causing repeated heating, softening and increased wear on the upper nozzle wall. Downstream of the nozzle exit, the flow turns from a fairly linear jet to a wake-like regime characterized by alternating zones of vorticity. This flow, which is visually similar to a Karman vortex street, is indicative of turbulence outside the nozzle, which reduces substrate adhesion. These computational findings were confirmed experimentally through shadowgraphy. The experiments showed that particles within the nozzle followed paths consistent with the DNS predictions: a nearly straight, stable jet up to about 25–30 mm downstream of the nozzle exit, followed by a sharp transition to a dispersed, turbulent jet corresponding to the onset of vorticity.

2: Details of Turbulence Analysis

2.1 DNS Solver

To address the need for identifying and analyzing transient turbulence effects within the flow, Raoelison *et al.* employ a DNS solver rather than relying on turbulence modeling in their CFD analysis. Directly solving the Navier-Stokes equations yields the most reliable and accurate results, as it captures all relevant spatial turbulences down to the Kolmogorov scale [1]. This full resolution of turbulence is methodologically essential for investigating the phenomenological effects present in low-pressure cold spray (LPCS) applications. The LPCS system is characterized by alternating swirling and irregular fluctuations that are inherently phenomenological. These features undermine the accuracy of time-averaged results produced by RANS solvers. Experimental observations further support this, revealing the emergence of self-sustained fluid instabilities, wake development, and intermittent vorticity [2]. Figure 1 illustrates the enhanced fidelity that DNS simulations offer compared to RANS.

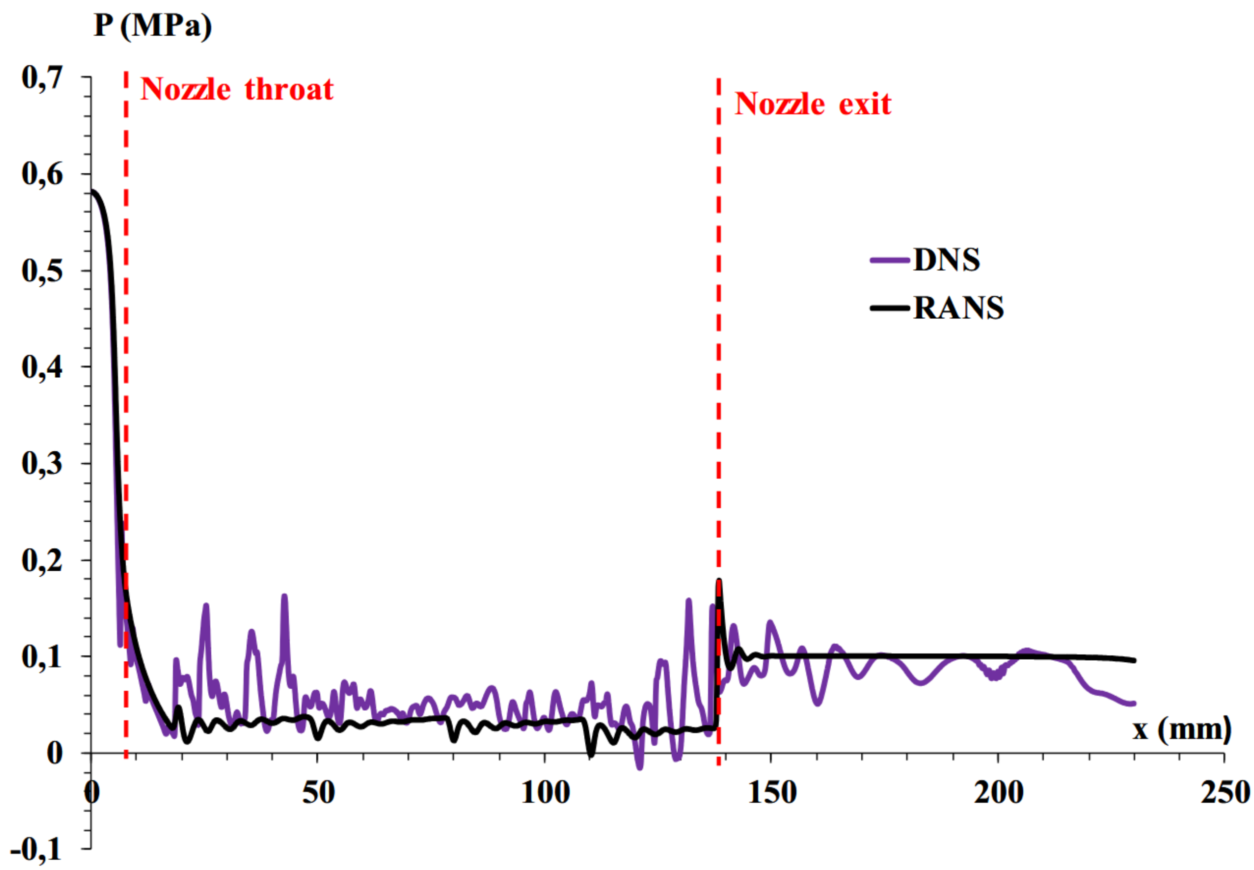

Figure 1: Comparison of pressure variation in the x-direction between DNS and RANS simulation [2].

The flow of LPCS feels compressibility effects due to its high-speed nature. As such, the equations solved for by the DNS Eulerian solver are the standard equations for mass, momentum, and energy of a 2D compressible system. The system of equations is closed by the ideal gas law, and the dynamic viscosity is determined by Sutherland's law due to its temperature dependence [3]. In the following equations, the time derivative is the advective derivative formulation. ρ (kg/m^3), p (Pa), T (K) and V (m/s) are the gas density, pressure, temperature and velocity. C_p (J/(K·kg)) is the calorific capacity, μ (kg/(m·s)) is the dynamic viscosity, λ is the thermal conductivity (W/(m·K)), and the term $\mu\tau' : \nabla\vec{V}$ represents viscous dissipation [2].

$$\frac{D\rho}{Dt} + \rho \cdot div(\vec{V}) = 0 \quad \text{Mass Conservation} \tag{1}$$

$$\rho\frac{D\vec{V}}{Dt} = -\vec{\nabla}p + \mu\vec{\nabla\cdot}\underline{\underline{\tau'}} \quad \text{Momentum conservation} \tag{2}$$

$$\rho C_p\frac{DT}{Dt} = \nabla \cdot \left(\lambda\vec{\nabla}T\right) + \mu\underline{\underline{\tau'}} : \nabla\vec{V} \quad \text{Energy Conservation} \tag{3}$$

$$\underline{\underline{\tau'}} = \mu(\nabla\vec{V} + \nabla\vec{V}^T - \frac{3}{2}\nabla\cdot\vec{V}\underline{\underline{I}}) \quad \text{Viscous Stress Tensor} \tag{4}$$

$$\mu = \mu_{\text{ref}}\left(\frac{T}{T_{\text{ref}}}\right)^{3/2}\frac{T_{\text{ref}} + S}{T + S} \quad \text{Sutherland's Law} \tag{5}$$

Because these equations describe a compressible system, greater computational rigor is required compared to traditional incompressible formulations. This demand becomes especially evident when evaluating the system's nonlinear fluxes, which are themselves functions of the conservative variables: mass, momentum, and energy. The coupling of these variables introduces inherent complexity and necessitates increased computational effort [4]. Moreover, computational complexity scales substantially with the Reynolds number. In DNS solvers, the number of grid points required for two-dimensional geometries scales approximately as $\sim Re^{4/3}$ [5]. Given that LPCS applications operate at speeds well above Mach 1, they exhibit extremely high Reynolds numbers—resulting in substantial computational demands for DNS simulations. To ensure accurate resolution of flow disturbances down to the Kolmogorov scale, both the computational grid and time step must be sufficiently refined. In this case, a mesh size of 70 microns and a time step of 1 µs were used. Despite the magnitude of the simulation, such resolution was achievable due to the use of a small and simplified simulation geometry, as illustrated in Figure 2. Additionally, Raoelison *et al.* employed a computing cluster with specifications detailed in Table 1 to carry out the simulation [2].

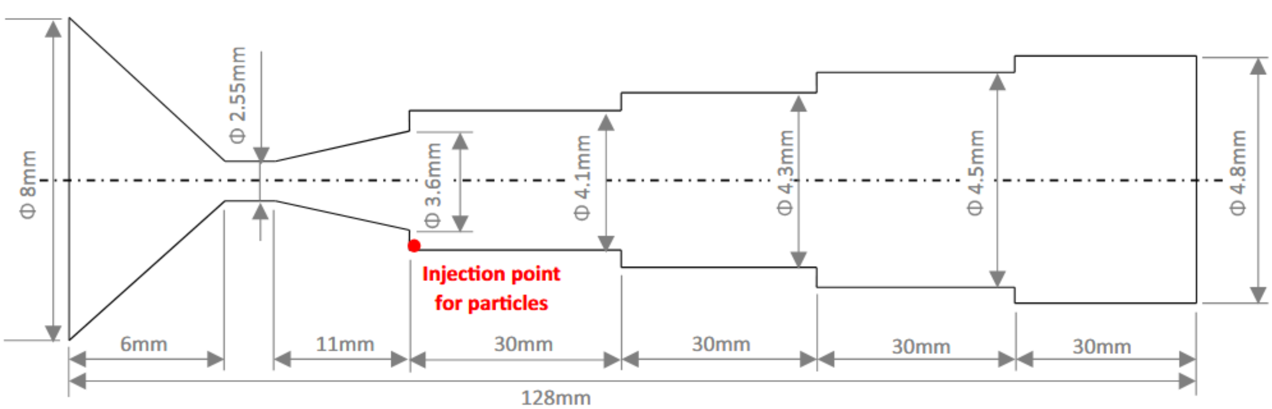

Figure 2: Specification of the nozzle shape and dimension with the radial powder injection location [2].

Table 1: Computational details with performance allocated by the Cluster Lumière during simulation [2]

Computation performance			**Space-time discretization**			**Computation duration and storage**		
RAM	Core	Speed	Nodes	Δt	Total time	CPU time	Backup time	Storage
144 GB	96	66.33 Tflops	823,901	1 μs	0.05 s	36 Hrs 14 min	every 1 ms	24.8 GB

2.2 Shadowgraphy

To validate the results from the DNS simulation, high-speed shadowgraphy was employed—a process used to visualize the flow fields and kinematics of a system. Shadowgraphy, in particular, relies on the refractive index variations of a transparent medium to reveal flow structures [6]. The primary mechanism underlying shadowgraph visualization is the curvature of light caused by disturbances within the flow. The refractive index varies spatially when light passes through regions of inhomogeneous density, and these variations deflect light rays, altering their intensity (I) as observed on a screen located a distance L away. The change in intensity is proportional to the spatial rate of light deflection [6]. This relationship is expressed mathematically in equation (6), which links the normalized change in intensity to the two-dimensional Laplacian of the refractive index n [7]:

$$\frac{I(x,y) - I_0}{I_0} \propto -L\left(\frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2}\right)\int_0^{\Delta z} n(x,y,z)\,dz \tag{6}$$

Additionally, through the Gladstone–Dale relation, the refractive index of a gas is directly related to its density. Consequently, shadowgraphs are sensitive to the second derivative of density. This sensitivity makes shadowgraphy particularly well-suited for analyzing supersonic flows, where phenomena such as shock waves produce large density gradients that are readily visible in shadowgraph images [8].

Raoelison *et al.* employ a specialized high-speed shadowgraphy system called SprayCam, focused on the region of flow just outside the nozzle exit. The ability to freeze motion and eliminate blur is particularly critical at the high velocities characteristic of LPCS systems. Image clarity is enhanced through the use of a pulsed laser with a 40 ns pulse length, which reduces motion blur to approximately 20 µm or less. A high-speed CCD sensor containing 1024 pixels and a spatial resolution of 16 µm/pixel was used to collect the image data. The copper particles typical of LPCS applications generate distinct shadows that are captured by the CCD sensor and converted into voltage signals, which are then reconstructed into successive frames to accurately represent the flow behavior [2, 9].

3: Critical Review of Article

The subject of Raoelison et al.'s article is the effect of phenomenological turbulence on cold spray processes. This focus inherently excludes other simulation methods, as both RANS and LES rely on turbulence models to approximate small-scale flow behavior, which introduces error in all cases [1]. In particular, several aspects of RANS analysis conflict with the research goals—most notably, its time-averaging of results and dependence on turbulence models that do not fully align with physical reality. These issues are present to a lesser extent in LES, where only small-scale flow characteristics are modeled. However, LES remains impractical for Raoelison et al.'s investigation, since even minor flow disturbances can cascade and generate large-scale turbulence [2]. Consequently, DNS was the only suitable simulation method for their application. This choice is typical of studies that aim to establish a framework for future research, which is one of the primary goals of this work. Nevertheless, the LPCS simulation still incorporates certain simplifications that limit its ability to represent real-world systems.

The most notable simplification is dimensional. The simulation was conducted on a two-dimensional geometry, which is not representative of actual cold spray systems [2], especially because turbulence is inherently a three-dimensional phenomenon. This simplification was made to reduce computational complexity; however, given the computational resources Raoelison et al. had access to and their stated goal of creating a comprehensive framework for LPCS turbulence studies, a more computationally intensive approach would have been justified. Given access to a 96-core cluster, a limited 3D wedge or axisymmetric simulation may have been feasible. Compared to other DNS studies of similar systems, the computational complexity of Raoelison et al.'s simulation appears modest, particularly when considering the resources reportedly available to the authors.

DNS simulations have been conducted on three-dimensional geometries for Mach numbers far exceeding that seen in the LPCS system of Raoelison *et al.*'s research. One such example is the study of hypersonic shock wave/turbulent boundary layer interactions (STBLI) on a three-dimensional compression ramp at Mach 6 [10]. While the geometry in this example is slightly simpler than that of the De Laval nozzle studied by Raoelison *et al.*, the Mach number far exceeds that of the flow within the De Laval nozzle. This is not to say that the geometry in the Zhang *et al.* study is trivial; it contains a ramp with both an angle of 34° from the flat horizontal axis and a sweep angle of 45° inward. Additionally, because a higher Reynolds number consequently reduces the Kolmogorov scale, the mesh size is smaller as a result: Zhang *et al.* report a minimum wall-normal resolution of 0.008 mm, or 8 μm, nearly a factor of ten smaller than the 70 μm resolution used in the LPCS study. Furthermore, because the geometry is three-dimensional, the number of cells is several orders of magnitude higher, as the ramp geometry contains $5.51 \cdot 10^9$ cells, compared to the nozzle geometry's approximately 823,901 cells.

Table 2: Comparative Computational Specifications for High-Speed DNS Studies

Parameter	**Cold Spray DNS [2]**	**Hypersonic STBLI DNS [10]**
Simulation Dimensionality	2D (Bi-dimensional geometry) [2]	3D (Swept Compression Ramp) [10]
Flow Regime / Mach No.	Supersonic Cold Spray ($M > 1$) [2]	Hypersonic STBLI ($M_\infty = 6$) [10]
Grid Resolution	70 μm mesh size in supersonic region [2]	5.51×10^9 points (5.51 billion) [10]
Time Step (Δt)	1×10^{-6} s (1 μs) [2]	0.006 (dimensionless) [10]
Total Physical Time	0.05 s [2]	Not explicitly provided [10]
Computational System	CPU cluster (96 cores, 66.33 Tflops) [2]	GPU-accelerated solver (OpenCFD-SCU) [10]
Total CPU/Wall Time	36 hrs 14 min (CPU time) [2]	Not explicitly provided; $\sim$ 98.7% scaling on 24,576 GPUs [10]

Raoelison *et al.* also make no note of mesh adaptation near walls or of any mesh-convergence study, leaving a critical aspect of result validation unaddressed. Without mesh convergence, it is impossible to determine whether the mesh fully resolves all relevant flow features and therefore yields an accurate depiction of the flow. Although DNS carries extreme computational cost, mesh-convergence testing remains essential for validating results and has been performed in studies with far larger cell counts and greater computational complexity [10]. While the experimental setup provides a basis for comparison, all quantifiable results in Raoelison *et al.* come from the DNS. Thus, to fully interpret and justify the study's findings, a mesh-convergence analysis should be employed.

Another report, studying the properties of turbulent boundary layers (TBLs) under high-speed compressible conditions, shows a similar adoption of high-fidelity three-dimensional DNS [11]. Simulations were run at Mach numbers of 2.3 and 4.9, once again scaling the minimum grid requirement as a result. Because of this, and the high-fidelity nature of the simulation, a mesh containing $3.3 \cdot 10^{10}$ cells was required. This was driven by a non-negotiable research requirement that the wall-bounded geometry must satisfy a resolution criterion of $\max(\Delta x_1^+, \Delta x_2^+, \Delta x_3^+) \leq 1$ for the inner layer across all locations and orientations [11].

While the scientific goals of these papers do not relate directly to LPCS, they demonstrate that high-fidelity, three-dimensional direct numerical simulations are producible at scales far exceeding that of Raoelison *et al.*'s simulation. The decision to reduce computational time and complexity has merit, but in applications with little prior research, ample computation power, and a three-dimensional experimental model for validation, a three-dimensional simulation should be considered.

An additional, albeit small, critique is the inconsistency in nozzle dimension as shown in Figures 2 and 3 of this paper. The nozzle dimensions are taken from Raoelison *et al.*'s work. It can be seen that the dimensions of the nozzle are not accurate. The entire nozzle is stated to be 128 mm in length, while the individual components sum to more than that. The bounding length of the nozzle should be changed to 138 mm, which is consistent with the labeled nozzle exit lines in Figures 1 and 4. Inconsistencies such as this should not appear in peer-reviewed, published articles.

4: Impact of Article

Raoelison *et al.*'s study of the phenomenological flow in low-pressure cold spray (LPCS) systems was conducted to better understand how supersonic flow affects particle behavior, a notion supported by their claim that particle flow is influenced by even the smallest flow perturbations [2]. Because of this, the paper has been used heavily in studies examining various aspects of metal particle behavior in LPCS systems. One such example is a study of particle temperature and substrate adhesion in cold spray systems by Nastic *et al.* [12].

Raoelison *et al.*'s article in particular provides a crucial framework for understanding how properties of the gas and particle-feeding system impact substrate adhesion through the development of flow characteristics such as oblique shocks and Mach disks. Consequently, these flow properties significantly alter the thermal history of the particle and, in turn, its adhesion behavior [12]. Their work continues to be viewed positively, as its physically accurate DNS simulations and comparison to experimental results are widely lauded.

Overall, these contributions enable a shift in focus from the underlying mechanics of cold spray systems to the physical performance of the systems, specifically the material characteristics of cold spray deposits.

This shift centers on the idea that thermal history can compensate for inherent flow instability and imperfections [12]. Because Raoelison *et al.*'s work highlights the uncontrollable flow complexity caused by the particle-feeding mechanism—and its negative effect on cold spray adhesion—a new research question emerged: can altering the thermal history of the particle increase adhesion despite these flow disturbances? It was ultimately found that at high temperatures, particle adhesion becomes more reliant on temperature than on in-situ peening, enabling further advancement of cold spray systems through material characteristics.

An exploration of other particle characteristics is seen in a study of particle dispersion resulting from flow turbulence by Mahato *et al.*, providing a shift in focus from phenomenological turbulence to its quantifiable effects on particles of different sizes [13]. Mahato *et al.* both adopt and critique certain aspects of the DNS study employed by Raoelison *et al.*. Primarily, they express appreciation for DNS simulations for their ability to capture unstable flow characteristics that are absent in RANS analyses, but also note their disapproval of the two-dimensional geometry used in Raoelison *et al.*. To address the computational challenges of a full 3D DNS, Mahato *et al.* employ LES to achieve a balance between high fidelity and reduced computational cost.

Through the work of Raoelison *et al.*, it was shown that simplified turbulence modeling can never fully represent the supersonic flow of cold spray systems, enabling higher-fidelity analyses of how such flows affect particle dispersion. In particular, Mahato *et al.* demonstrate that particles smaller than 25 µm in diameter are more susceptible to turbulent flow disturbances due to their lower Stokes number, defined as the ratio of the particle time scale to the flow time scale. Additionally, they enhance the fidelity of their analysis by modeling drag coefficients for dendrite-shaped particles [13]. These improved understandings of particle behavior rely heavily on the framework established by Raoelison *et al.*, particularly their detailed description of turbulence across all relevant flow scales.

Raoelison *et al.* have leveraged their earlier research and DNS simulations to expand their focus to particle behavior, particularly the effect that turbulence has on the flight of particles composed of different materials [14]. This work builds directly on the DNS results and the understanding that flow in a cold spray system begins as a concentrated jet and progressively disperses with increasing distance from the nozzle [2].

This foundation allows Raoelison *et al.* to pursue a more focused line of inquiry, studying the influence of particle material on dispersion behavior and converting the qualitative dispersion characteristics identified in the DNS study into quantifiable metrics. Through this approach, Raoelison was able to quantify that particle weight, expressed as $\rho_P D_p$ (particle density multiplied by particle diameter), is inversely related to dispersion in a nonlinear fashion. These findings have direct applications to physical cold spray systems, as the optimized standoff distance—defined as the distance at which adhesion is maximized—can be more accurately identified and utilized for the materials studied.

5: Future Work

A key element of Raoelison *et al.*'s work is the negative effect of phenomenological turbulence on the entire cold spray system. One of the most striking results of their study is the explanation for why cold spray nozzles tend to wear on the top surface rather than the bottom. These findings—particularly the localized heating and cooling near the top wall—can be leveraged to improve the efficiency, reliability, and broader adoption of cold spray technology. This topic remains an active area of research for many groups, including the Ozdemir group at Northeastern University [15]. O. C. Ozdemir and his team specifically highlight the influence of flow characteristics, such as gas pressure and temperature, on nozzle reliability [16]. However, while Ozdemir *et al.* recognize the importance of these characteristics in cold spray's industrial applications, they have not yet produced a study focused on reducing negative flow properties or mitigating wear at its source. Instead, their work explores in situ detection methods for wear and clogging, such as acoustic monitoring. As a whole, Ozdemir's work is representative of the current academic landscape and underscores a clear gap in research aimed at reducing the root causes of nozzle wear.

To effectively mitigate cyclical heating in cold spray nozzles, the primary source of turbulence must be considered—namely, the influx of metallic particles and the complex flow structures associated with them. With this focus, reducing flow asymmetry and its downstream effects may be achievable in physical cold spray systems, thereby enhancing industrial relevance.

The present study will focus on numerically validating the effect of controlled injected flows within cold spray nozzles. The model geometry will follow that of the De Laval nozzle used in the two-dimensional DNS of Raoelison *et al.*, but revolved about its central axis to form a three-dimensional geometry. Two additional flow introduction points will be placed downstream in the nozzle, where controlled injections of propellant gas (N_2) will be introduced with the goal of restructuring the flow and reducing cyclical asymmetries near the top wall. The secondary flow will be introduced along the horizontal axis of the De Laval nozzle. With the addition of a third dimension, computational complexity increases. To manage

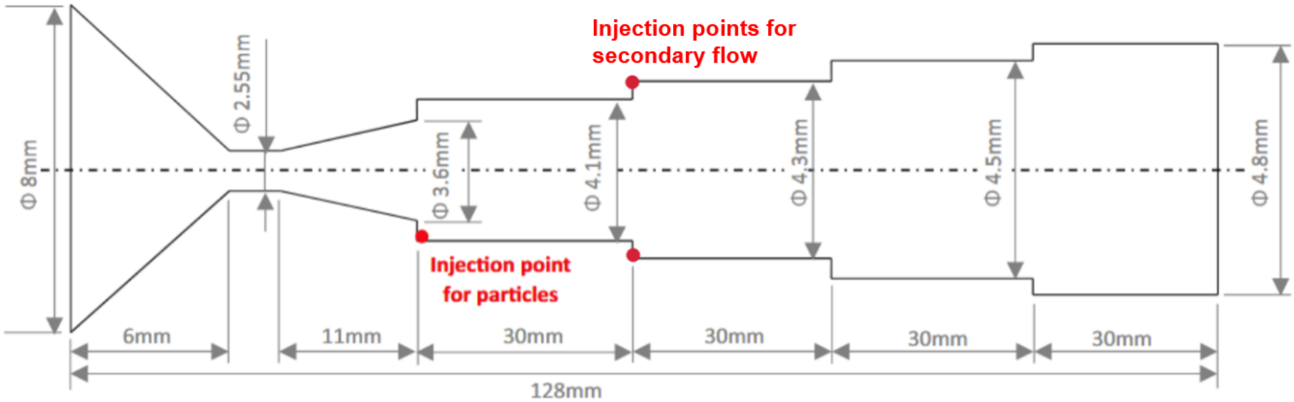

Figure 3: Cross-section of updated simulation geometry, including flow injection points [2].

this complexity, an LES approach will be used instead of DNS. LES serves as an intermediate method, balancing the fidelity of DNS with the computational efficiency of RANS, and has demonstrated reliable performance in cold spray systems with similar geometries, with a dynamic kinetic energy subgrid-scale (SGS) model used as the turbulence model. [13]. While the choice of using LES as opposed to DNS fundamentally differs from the provided critique of Raoelison *et al.*'s work, it stands as a compromise made from using a three-dimensional nozzle. In addition to using a three-dimensional geometry, the study consists of many runs. This adds another layer of complexity that Raoelison et al. did not have to manage. The mesh and time scales will be selected to reflect those of Raoelison *et al.*'s study, enabling resolution of relevant flow characteristics and allowing for direct comparison to DNS results. Being able to compare to

DNS results of heating and pressure near the top wall as a benchmark provides a clear means of evaluating improvements in nozzle wear. A mesh convergence will also be employed to see if the grid resolution employed is sufficient. The results of the grid-convergence tests cannot be directly related to Raoelison *et al.*'s work, due to the differences spurred from altering simulation method and geometry.

The study will consist of two distinct runs: one with fluctuating pressure and one with fluctuating velocity. Figure 1 provides a pressure profile along the nozzle geometry. Around 50 mm downstream of the nozzle entrance—where the secondary jets are introduced—the jet pressure varies around an average value of 0.5 MPa. This value will serve as the benchmark for the pressure trials. Similarly, Figure 4 shows velocity as a function of nozzle length. At the same location, 50 mm downstream, the jet velocity is approximately 700 m/s, which will be used as the benchmark for the velocity study. The pressure study

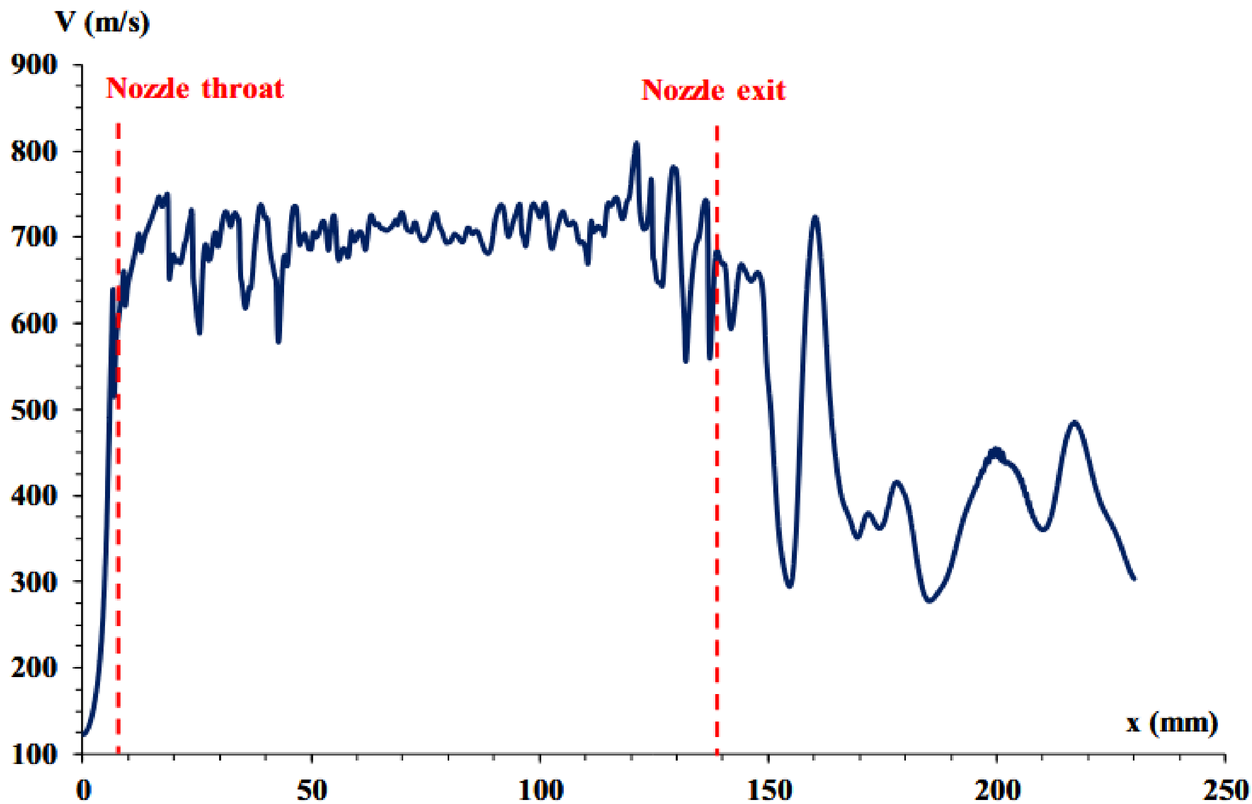

Figure 4: Plot of velocity from the DNS result as a function of distance from the nozzle start. [2]

will maintain a secondary inlet velocity of 700 m/s and vary pressure from 0.40 MPa to 0.60 MPa in increments of 0.02 MPa. Likewise, the velocity study will maintain a secondary inlet pressure of 0.50 MPa and vary velocity from 650 m/s to 750 m/s in increments of 10 m/s. The runs are summarized in Table 3. It should be noted that introducing supersonic flow increases mechanical complexity significantly; if results from this study prove favorable, future work could explore methods for coupling the secondary flow with the primary flow to reduce operational cost and system complexity.
Being able to understand and control flow asymmetries in cold spray nozzles has the potential to dramatically improve cold spray performance, as performance characteristics, such as deposition efficiency, will dramatically improve. Waste in deposition is the main cost driver in cold spray systems, as it leads

Table 3: Summary of Simulation Runs for Pressure and Velocity Studies

Study Type	**Fixed Parameter**	**Varied Parameter (Range)**	**Step Size**
Pressure Study	Secondary inlet velocity: 700 m/s	Pressure: 0.40–0.60 MPa	0.02 MPa
Velocity Study	Secondary inlet pressure: 0.50 MPa	Velocity: 650–750 m/s	10 m/s

to unused particles and increased operational costs, such as labor [17]. By introducing controlled secondary flow to stabilize the internal jet, this work could reduce thermal fatigue and flow-driven erosion, in turn extending nozzle lifetime, reducing downtime, and increasing the economic viability of cold spray for industrial adoption [17].

References

[1] Fidelis. "Turbulence Modeling Techniques In CFD – DNS vs LES vs RANS." Fidelis Consulting. Accessed November 11, 2025, URL `https://www.fidelisfea.com/post/turbulence-modeling-techniques-in-cfd-dns-vs-les-vs-rans`.

[2] Raoelison, R.N., Koithara, L.L, Costil, S. and Langlade, C. "Turbulences of the supersonic gas flow during cold spraying and their negative effects: A DNS CFD analysis coupled with experimental observation and laser impulse high-speed shadowgraphs of the particles in-flight flow." *International Journal of Heat and Mass Transfer* Vol. 147 (2020). DOI 10.1016/j.ijheatmasstransfer.2019.118894.

[3] Online, CFD. "Sutherland's Law." CFD Online. Accessed November 11, 2025, URL `https://www.cfd-online.com/Wiki/Sutherland%27s_law`.

[4] Gerritsen, Margot and Olsson, Pelle. "A High Order Finite Difference Scheme with Sharp Shock Resolution for the Euler Equations." Technical Report No. RIACS-TR-96-01. Research Institute for Advanced Computer Science (RIACS) and NASA Ames Research Center. 1996. NASA-CR-199468.

[5] Coleman, Gary N. and Sandberg, Richard D. "A primer on direct numerical simulation of turbulence - methods, procedures and guidelines." Technical Report No. AFM 09/01a. University of Southampton. 2010. URL `https://eprints.soton.ac.uk/66182/1/A_primer_on_DNS.pdf`. School of Engineering Sciences Aerospace Engineering AFM Reports.

[6] Settles, Gary S and Hargather, Michael J. "A review of recent developments in schlieren and shadowgraph techniques." *Measurement Science and Technology* Vol. 28 (2022). DOI 10.1088/1361-6501/aa5748.

[7] Gopal, Amrutha, Minardi, Stefano and Tatarakis, Michael. "Quantitative two-dimensional shadowgraphic method for high-sensitivity density measurement of under-critical laser plasmas." *Optics Letters* Vol. 32 (2007). DOI 10.1364/OL.32.001238.

[8] Dingfield, Allie and Viganò, Dr. Davide. "Quantitative Shadowgraph for Turbulence Measurements in Supersonic Flows." Technical report no. Department of Mechanical and Aerospace Engineering, Missouri ST. 2025.

[9] Stevenin, C., Tomas, S., Vallet, A., Amielh, M. and Anselmet, F. "Shadowgraphy investigations of high speed water jet atomization into still air." *International Conference on Liquid Atomization and Spray Systems* Vol. 12 (2012).

[10] Zhang, Ji, Guo, Tongbiao, Dang, Guanlin and Xin-Liang, Li. "Direct numerical simulation of shock wave/turbulent boundary layer interaction in a swept compression ramp at Mach 6." *Physics of Fluids* Vol. 34 (2016). DOI 10.1063/5.0118578.

[11] Poggie, Jonathan. "Compressible Turbulent Boundary Layer Simulations: Resolution Effects and Turbulence Modeling." Technical report no. American Institute of Aeronautics and Astronautics. 2015.

[12] Nastic, A., Jodoin, B., Poirier, D. and Legoux, J.-G. "Particle temperature effect in cold spray: A study of soft particle deposition on hard substrate." *Surface Coatings and Technology* Vol. 406 (2021). DOI 10.1016/j.surfcoat.2020.126735.

[13] Mahato, Bikash, Yoder, Jay, Simmons, Gloyd, Huft, Nathan, Nault, Isaac and Lucon, Peter. "Particle dynamics in low-pressure cold spray additive manufacturing — A numerical and experimental study." *Additive Manufacturing* Vol. 10 (2025). DOI 10.1016/j.addma.2025.104937.

[14] Koithara, Libin Lalu, Raoelison, Rija Nirina and Costil, Sophie. "Flow phenomenon of micron-sized

particles during cold spray additive manufacturing: High-speed optic observation and characterization." *Advanced Powder Technology* Vol. 31 (2020). DOI 10.1016/j.apt.2019.12.037.

[15] Group, Ozdemir Research. Northeastern Univeristy. Accessed November 16, 2025, URL `https://www.ozdemirresearchgroup.com/projects`.

[16] Ozdemir, O.C., Conahan, J. and Muftu, S. "Particle Velocimetry, CFD, and the Role of Particle Sphericity in Cold Spray." *Coatings* Vol. 10 (2020). DOI 10.3390/coatings10121254.

[17] Guo, D., Kazasidis, M., Hawkins, A., Fan, N., Leclerc, Z., MacDonald, D., Nastic, A., Nikbakht, R., Ortiz-Fernandez, R., Rahmati, S., Razavipour, M., Richer, P., Yin, S., Lupoi, R. and Jodoin, B. "Cold Spray: Over 30 Years of Development Toward a Hot Future." *Journal of Thermal Spray Technology* Vol. 31 (2022). DOI 10.1007/s11666-022-01471-4.