Journal of Thermal Spray and Engineering, 1(1): 24-31
ISSN (Online): xxxx-xxxx; 10.jtse/xxxx-xxxx/1-1.5

Analysis of Jet Characteristics Among Various Cold Spray Nozzles

Sai Rajkumar Vadla · Jeffrey Doom

South Dakota State University, Brookings, SD 57007, USA.

ABSTRACT

This research is conducted mainly to analyze the jet characteristics of various cold spray nozzles. This study presents the theoretical and practical aspects of Cold Spray process modeling, discusses multiple numerical analysis research areas, and determines the significant parameters to be considered while developing a custom cold spray setup and exhibits analysis-based correlations. The simulations were performed on some meshes of different density using the SST turbulent model in Star CCM+ solver. For the first time, in this work, the jet characteristics inside a step drilled nozzle was presented; Furthermore, shock diamond formation was found inside the divergent section of step drilled nozzle which strongly influence the flow regime with sharp fluctuations. The comprehensive comparison between step drilled nozzle, conical nozzle and curved nozzle indicates that curved nozzle results in slightly higher nozzle exit velocity. However, results have suggested that the curved nozzle can achieve much higher velocities by optimizing the nozzle length.

ARTICLE HISTORY
Received 07-10-2018
Revised 15-10-2018
Accepted 17-10-2018
Published 23-10-2018

KEYWORDS
Cold spray
Spray Nozzles
Computational fluid dynamics (CFD)

Introduction

The utilization of computational fluid dynamics (CFD) as a research study tool in the aerodynamics and turbomachinery industry reinforces efficiency in the design of aircraft or for understanding the flow through pipes. CFD offer tools to model different geometries and perform a more extensive study of the flow phenomena. This gives the opportunity to model a variety of geometries and analyze their behavior under a variety of operating conditions [1]. A similar approach can be implemented in coating technologies. Coating technologies play an essential role in the manufacturing industry. Their ability to form layers of specific materials onto engineering components to enhance mechanical and physical properties has numerous applications [2]. The applications include corrosion protection, repair, and thermal protection. In recent years, CFD simulations are increasingly used in Cold Spray Technology, which is a relatively new and novel coating technology used to manufacture coatings in the solid state fully preserving feedstock material properties [3]. Cold spray refers to a direct material deposition process, the kinetic energy of the particles sprayed at very high velocities leads to the bonding of the particles to the substrate. In this process, small powdered particles in a range of 5 to 100 μm [4] are accelerated to a high velocity in the range of 300 to 1200 m/s [4] in a supersonic jet of heated gas and then impinged onto a substrate surface in solid state without significant fusion, undergoing intensive plastic deformation. Cold spray produces coatings based on a wide selection of materials with superior characteristics. Before cold spray technology, it was necessary to either dissolve the metal in a chemical bath, melt them or vaporize them. Now, through cold spray technology, metals can be deposited rapidly in the solid state, and thus, drawbacks connected with melting, such as oxidation and undesirable phase transformations can be avoided. Cold spraying is recognized as a promising spray coating technique capable of producing thick metal and in some cases metal-ceramic coatings on metal or ceramic substrates at relatively low temperatures. Cold Spray has moved from a scientific curiosity to an established and integrated manufacturing process in a short span of 30 years [5]. Special material powders with specific characteristics, required for cold spray processing are readily available. Today, Cold Spray systems are being utilized in a wide range of applications including performance-enhancing coatings, protective layers, manufacturing new components and repairing fabricated parts [2, 5]. The number of industries adapting to cold spray technology is increasing, from aerospace to automobile to turbine and defense to power to sputter targets.

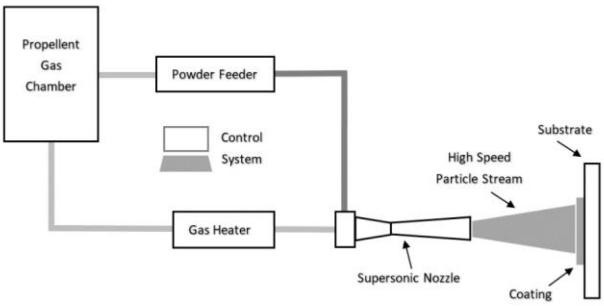

Figure 1: Schematic of Cold Spray Process

The principle of cold spray is based on the metal particle deformation behavior during high-velocity impact with a solid substrate (Figure 1). In the process, the propellant gas is accelerated to supersonic velocity in a convergent-divergent (de 'Laval) nozzle. The coating powdered material is injected into the gas stream, stimulated by the propellant gas in the nozzle and propelled towards the substrate to be coated. If impact velocity of the particle exceeds a specific critical value, the impact energy from the particle provokes an intense plastic deformation of the particle. Upon impact, the particle breaks thin film on the substrate which helps to establish intimate, clean contact between the particle and substrate which leads to the creation of intense bonding. Hence, a dense and solid adhesive coating on the substrate surface is formed [6].

Corresponding Author: Jeffrey Doom, Tel: 6056886703
Email: jeffrey.doom@sdstate.edu

The main components of Cold Spray system include:

- Powder feeder
- Propellant Gas
- Gas heater to preheat the gas
- Supersonic nozzle
- Spraying chamber with a motion system
- Method for controlling spray parameters

The equipment used for cold spray has been continuously developed to achieve optimum impact conditions for a large variety of materials. Materials that have low melting temperatures can be successfully deposited by moderate conditions, using the less expensive low pressure/temperature equipment. There is a necessity of having more powerful hardware for providing higher pressures and temperatures for processing high strength materials.

A Cold Spray System can be designed in fixed, manual, portable or robotic systems. There are majorly two main categories of cold spray systems, 1. Low-Pressure Cold Spray System, 2. High-Pressure Cold Spray System [7].

In the Low-Pressure cold spray system (LPCS), the powder is injected in the diverging section of the nozzle where the gas is expanded [4, 7]. Atmospheric pressure air supplied to transport powder from the feeder. Hence, the LPCS does not require a pressurized feeder. LPCS is typically smaller and often found in portable systems. The range of particle velocities that can be achieved through this system usually ranges between 300 to 600 m/s [4, 8]. They are used for the application of lighter materials and are generally available with air or nitrogen as a propellant gas at pressures on the order of 0.5 to 1.0 MPa [5, 8].

In High-Pressure Cold Spray (HPCS), small particles can generate relatively higher particle velocities compared to LPCS ranging from 800 to 1400 m/s [5]. Lower density gasses like helium or nitrogen are usually preferred for this system. The gases are pressurized high, typically high in the range of 1 to 5 MPa [5, 8] through the converging-diverging nozzle to achieve high particle impact velocities. For HPCS, high-pressure powder feeder running at a higher pressure than main gas stream is required to avoid powder backflow. High-pressure powder feeders are usually very expensive and large. Nozzle clogging is the other major problem with HPCS. When the particle velocity and temperature are increased, it can get worse. To overcome this problem, either a larger average particle diameter or higher yield strength second particle population should be mixed with the first particle population [4]. Due to particle erosion, severe wear will occur at nozzle throat which affects the operation of the nozzle and leads to more considerable variations in deposit quantity. It gets worse when particles of harder material are sprayed. On the other hand, the LPCS has simpler equipment. However, LPCS can only achieve relatively fewer particle velocities compared to HPCS as the exit Mach number, and inlet pressure is low, usually below 3 and 1MPa [8], respectively. Pressure can no longer be able to supply powders into the supersonic nozzle.

Cold spray offers many technical benefits when compared with other coating processes [2-3]. Because cold spray does not use a high-temperature heat source, such as a flame or plasma, to melt the feed material, it does not deposit large amounts of heat into a coated part does it degrade thermally sensitive coating materials through oxidation or other in-flight chemical reactions. For this reason, cold spray seems very attractive for depositing oxygen-sensitive materials, such as copper or titanium.

Thermally sprayed coatings are usually evaluated by considering porosity, adhesivity to the substrate, and oxide contents in the layer. Presently cold spray enables the obtainment of coatings characterized by very low porosity, oxide content and high adhesion. Also, the lack of thermal stress in the layer and substrate increases the spectrum of cold spray method application in comparison with other thermal spraying methods. Cold spray can be used for applying coatings to most engineering materials. Cold spray applications are vast and primarily includes corrosion resistant coatings, composites coatings, fretting fatigue resistant coatings, high-temperature protection coatings, oxidation protection coatings, self-lubrication coatings, and wear-resistant coatings. Repairs and restoration in some of the industries including aerospace, agriculture, automotive, and tooling [3].

To meet strong bonding, setting up the process, and powder parameters are vital. The following are the significant parameters (but not limited to) to be monitored in a typical Cold Spray system:

- Nozzle design (Inlet diameter, throat diameter, exit diameter, convergent length, divergent length, and profile shape)
- Propellant Gas (Nature, temperature, and pressure)
- Particle (Nature, critical velocity, impact velocity, temperature, and size)
- The substrate (Nature, standoff distance, and substrate temperature)
- Deposition Efficiency

To ensure optimized and successful cold spraying, the influence of various process and powder parameters on the critical velocity, deposition efficiency should be well understood. Application of modeling and simulation in cold spray research is a feasible and robust way to reach this goal. Works related to modeling of the cold spray powder spraying process cover two primary research areas, i.e., modeling of powder particle deformation and modeling of powder particle velocity. Other research areas include modeling of substrate heating and building up residual stress in cold spraying [9].

The numerical simulation of gas/powder velocity enables the determination of the gas state parameters inside the nozzle and once the gas has left it as well as the distribution of powder particle velocity and temperature. Particle velocity is an important factor that determines whether particles can adhere to the substrate surface. It is known that particle in flight velocity is highly dependent on the character of the gas flow field inside and outside the nozzle because the powders are only dragged and accelerated by the compressed driving gas during the process. As for the gas flow field, it is influenced by several factors, including operating parameters, nozzle geometry, and standoff distance. Therefore, to achieve a high particle impact velocity, a large body of works has been carried out to optimize the parameters of nozzle geometry and operating parameters, and the following flow field. Among these studies, Computational fluid dynamics (CFD) technique was always adopted to predict the gas flow field and particle velocity due to its lower cost and time consuming compared to experimental implementations [8, 10-15].

Literature survey

Shuo Yin [16] conducted a numerical investigation to study the effect of nozzle cross-section shape on gas flow and particle acceleration in cold spraying. He presented a comprehensive comparison between rectangular nozzles and elliptical nozzles. Based on his calculation results, he concluded that the nozzle cross-section shape could affect the gas flow field, particle acceleration and thus particle impact velocity. His works showed that rectangular nozzles, result in slightly lower mean particle impact velocity than elliptical nozzles. However, for rectangular nozzles, more particles may achieve relatively high velocity due to the larger sectional area of their potential core.

Masahiro Fukumoto [17] has carried out optimization in nozzle design by numerical simulation to improve the deposition efficiency of copper fine particles in cold spray process onto a metallic substrate. His team developed a special nozzle to reduce the bow shock effect on the substrate surface. By using his newly designed nozzle, he could able to decrease the pressure level on the substrate effectively when compared to the conventional nozzle. His research achieved a remarkable improvement in deposition efficiency, almost eight times higher than the traditional nozzle.

Wen-Ya Li [18] designed a convergent barrel cold spray nozzle through numerical simulation. He found that the main factors influencing are particle velocity and temperature including the length and diameter of the barrel section, nature of the accelerating gas, pressure, and temperature of the accelerating gas, and the particle size. His analysis showed that under constant gas pressure, particles could achieve high temperature but relatively low velocity when the convergent-barrel nozzle is used compared to a convergent-divergent nozzle. On his other research, he showed that the gas conditions, particle size, and the divergent section length of the nozzle influence the optimal expansion ratio.

Zheng-Dong, Zhou [19] compared the Laval orifice and straight orifice nozzles. In their work, they calculated gas flow field inside and outside of the nozzles. His numerical simulations show that the flow generated by the Laval nozzle had a higher exit velocity in comparison with that of the straight nozzle. He found that, in the Laval nozzle, the gas is compressed, and the pressure is reduced at the throat part, whereas, at the accelerative part, the gas is almost entirely expanded, and the velocity is driven close to the maximum.

Notwithstanding these critical findings, several vital problems still need to be further studied. In actual application, many industries use step drilled nozzle for its cheaper design and manufacturing cost. There are no research studies performed on determining the effects of step drilled nozzle numerically. In this study, as a first approximation, a numerical investigation is conducted on a step drilled nozzle design and a couple of other models (conical and curved shaped divergent sections) with the same operating parameters except the divergent cross-section shape. The main aim is to clarify the effect of nozzle cross-section shape on gas flow, numerically expose the losses of step-nozzle design and determine preferable nozzle shape.

Experimental

Problem Statement

The supersonic jet characteristics of a step drilled nozzle are compared with a conical and curved nozzle through numerical study using Star-CCM+, a commercial CFD software [20].

Geometry and CAD model generation

All the geometrical parameters except the divergent section profile are kept constant among three different nozzle designs to ensure consistency. The nozzle geometries are illustrated in Fig. 2.

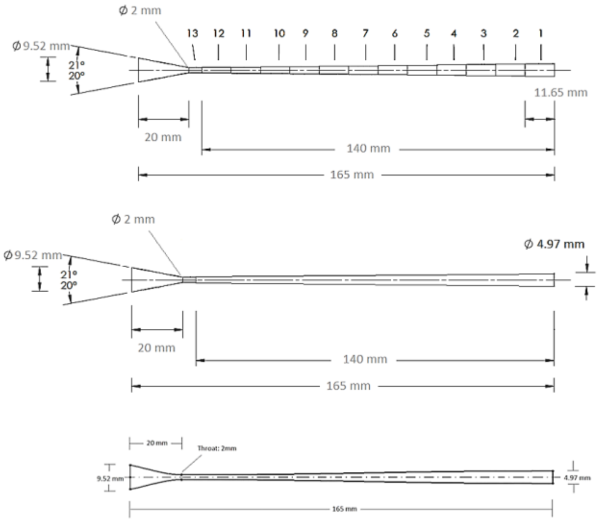

Figure 2: Dimensions of Step drilled, conical and curved nozzles respectively

Table 1: Summary of geometrical dimensions of various nozzles

	Nozzle 1	Nozzle 2	Nozzle 3
Inlet Diameter (mm)	9.52	9.52	9.52
Outlet Diameter (mm)	4.97	4.97	4.97
Throat (mm)	2	2	2
Convergent Length (mm)	20	20	20
Divergent Length (mm)	145	145	145
Divergent Profile (mm)	Stepped	Conical	Curved
Stand Off Distance (mm)	20	20	20

Table 2: Step dimensions used in Step drilled nozzle

	Diameter (mm)	Step length (mm)
Step 1	4.97	11.65
Step 2	4.69	11.65
Step 3	4.49	11.65
Step 4	4.21	11.65
Step 5	3.98	11.65
Step 6	3.73	11.65
Step 7	3.45	11.65
Step 8	3.25	11.65
Step 9	2.94	11.65
Step 10	2.69	11.65
Step 11	2.43	11.65
Step 12	2.18	11.65
Step 13	2	Remainder

Table 1 summarizes the design parameters among various nozzles used in the study and Table 2 shows a summary of step dimensions used in step drilled nozzle. CAD models are generated using StarCCM+ [20], as shown in Fig. 3.

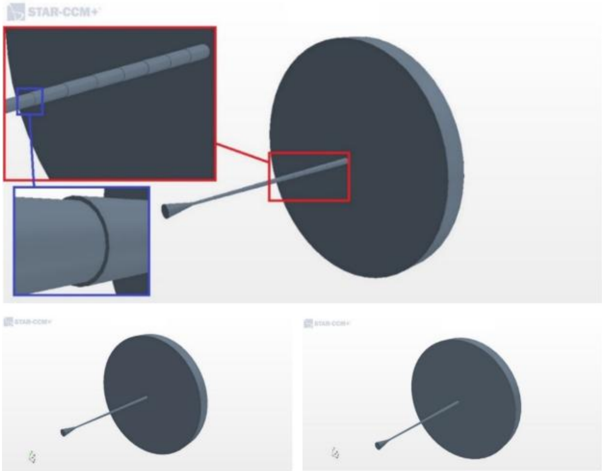

Figure 3: CAD models of Step drilled (top), conical (bottom-left) and curved (bottom-right) nozzles respectively

Discretization

There are numerous methods of discretization, which can broadly have classified into a mesh (grid) methods and mesh-free methods, but the mesh methods are more widely used. In the Meshing process, the region of interest is divided into smaller sub-regions. These smaller regions may be of different shapes like triangles, rectangles in case of 2D geometry, hexahedrons, tetrahedrons in case of 3D geometry.

Meshing models used for the current study

- Extruder
- Advanced Layer Mesher
- Surface Wrapper
- Surface Remesher

Table 3: Summary of generated cells and faces

	Nozzle 1 (Step Drilled)	Nozzle 2 (Conical)	Nozzle 3 (Curved)
Number of Cells	3033421	1686663	1657432
Number of Faces	7668366	7778737	7610608

Governing Equations

The governing equations used are the Navier-Stokes equations:

- Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_j) = 0$$

- Momentum Equation:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \left[\frac{\partial \tau_{ij}}{\partial x_j}\right]$$

$$\tau_{ij} = \mu\left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right]$$

- Energy Equation:

$$\frac{\partial \rho E}{\partial t} + \frac{\partial}{\partial x_j}(\rho E + p) = \frac{\partial \tau_{ij}}{\partial x_j}u_i + \frac{\partial}{\partial x_j}\left(k\frac{\partial T}{\partial x_j}\right)$$

For this simulation, we will consider steady-state supersonic turbulent flow with heat transfer during the process. Fluid motion equations are generally very complex and require computational ways to solve. The computational methods of discretization are the finite difference methods, finite element methods, and finite volume methods. The method used is finite-volume methods (FVM). The equations are solved with an algebraic multi-grid solver. The numerical algorithm is discussed in detail by Siemens PLM software [20]. In this study, CFD simulations were performed using STAR-CCM+ to predict the gas flow field.

Selected physics models for the current study

- Steady State
- Ideal Gas
- Reynolds Averaged Navier Stokes
- Coupled Flow
- Turbulent Flow
- SST K – Omega

Table 4: Summary of various boundary conditions used in the study

	Nozzle 1 (Step Drilled)	Nozzle 2 (Conical)	Nozzle 3 (Curved)
Stagnation Inlet	Nozzle Inlet	Nozzle Inlet	Nozzle Inlet
Pressure Outlet	Radial Outlet	Radial Outlet	Radial Outlet
Symmetry Plane	Top1, Top2	Top1, Top2	Top1, Top2
Wall	Convergent, Divergent	Convergent, Divergent	Convergent, Divergent

Summary of the boundary conditions set for various model is listed in Table 4. For all nozzles in the current research, Helium is set as carrier gas whose pressure and temperature are set at 3.103 MPa and 523 K simultaneously.

Results and Discussion

Visualization

CFD simulations of all the nozzle models specified in the previous sections were run on the high-performance computer at South Dakota State University. Figure 4-6 are the contour of plots pressure, Mach number, Temperature and velocity magnitude for the drilled, conical, and curved nozzle. A MATLAB [21] code is written to visualize the scalar fields along the nozzle axis among various nozzles using a line graph.

Summary of the results

To best display the effects of the nozzle shape, a typical operating condition was chosen: 3.2 bar, 523 K. Figure 7 shows center-line plots of Pressure, Mach number, Temperature, and Velocity magnitude for various nozzles used in the study. The absolute pressure plots show that the step drilled nozzle exhibits 40.5% higher than conical and 9.64% higher than the curved nozzle, which means, the

Contents lists available at http://www.inscience.in/JTSE.html

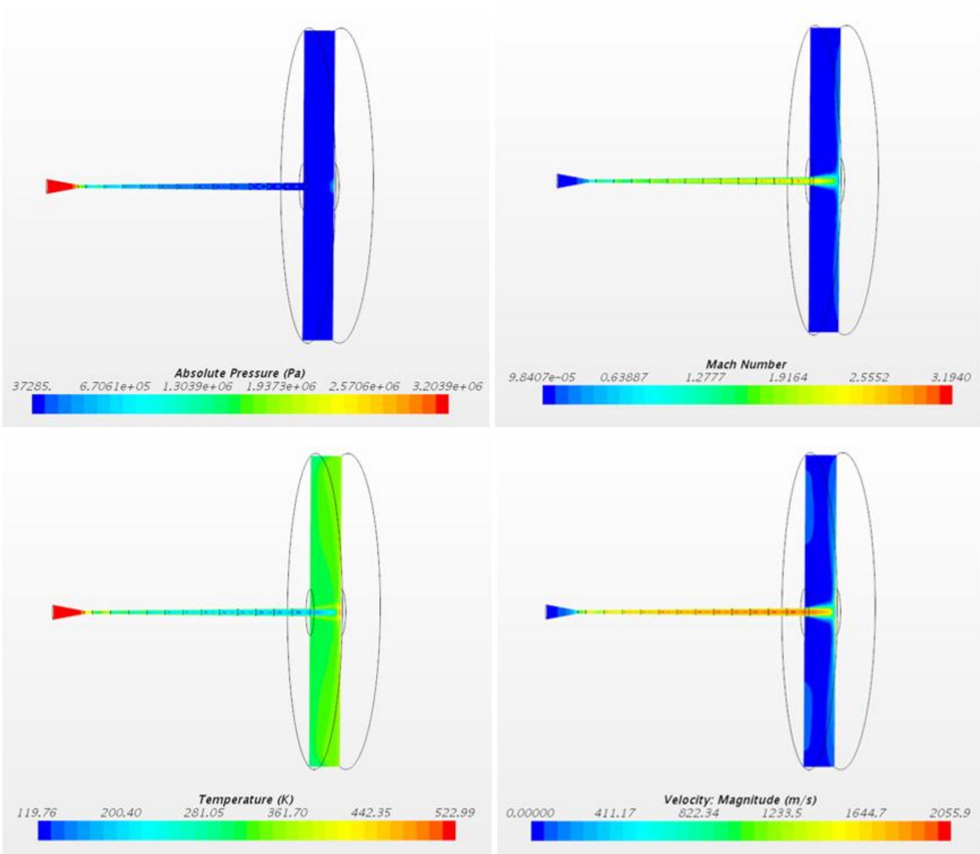

Figure 4: Various scalar scenes at the cross-section of Step Drilled nozzle (Nozzle 1)

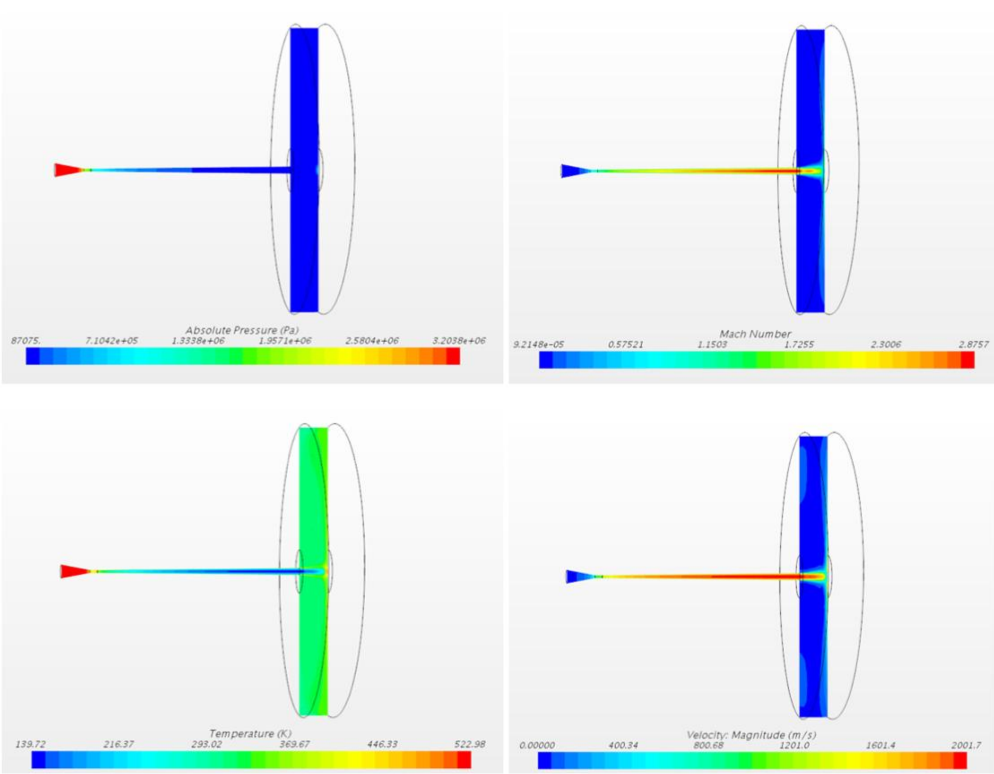

Figure 5: Various scalar scenes at the cross-section of the Conical nozzle (Nozzle 2)

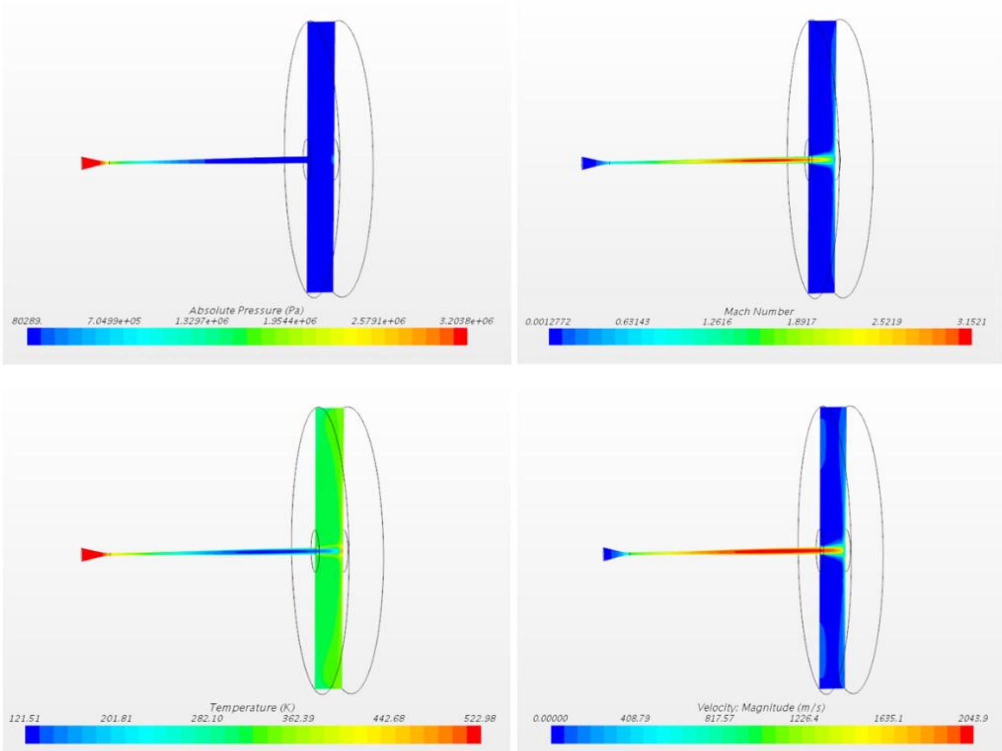

Figure 6: Various scalar scenes at the cross-section of the Curvednozzle (Nozzle 3)

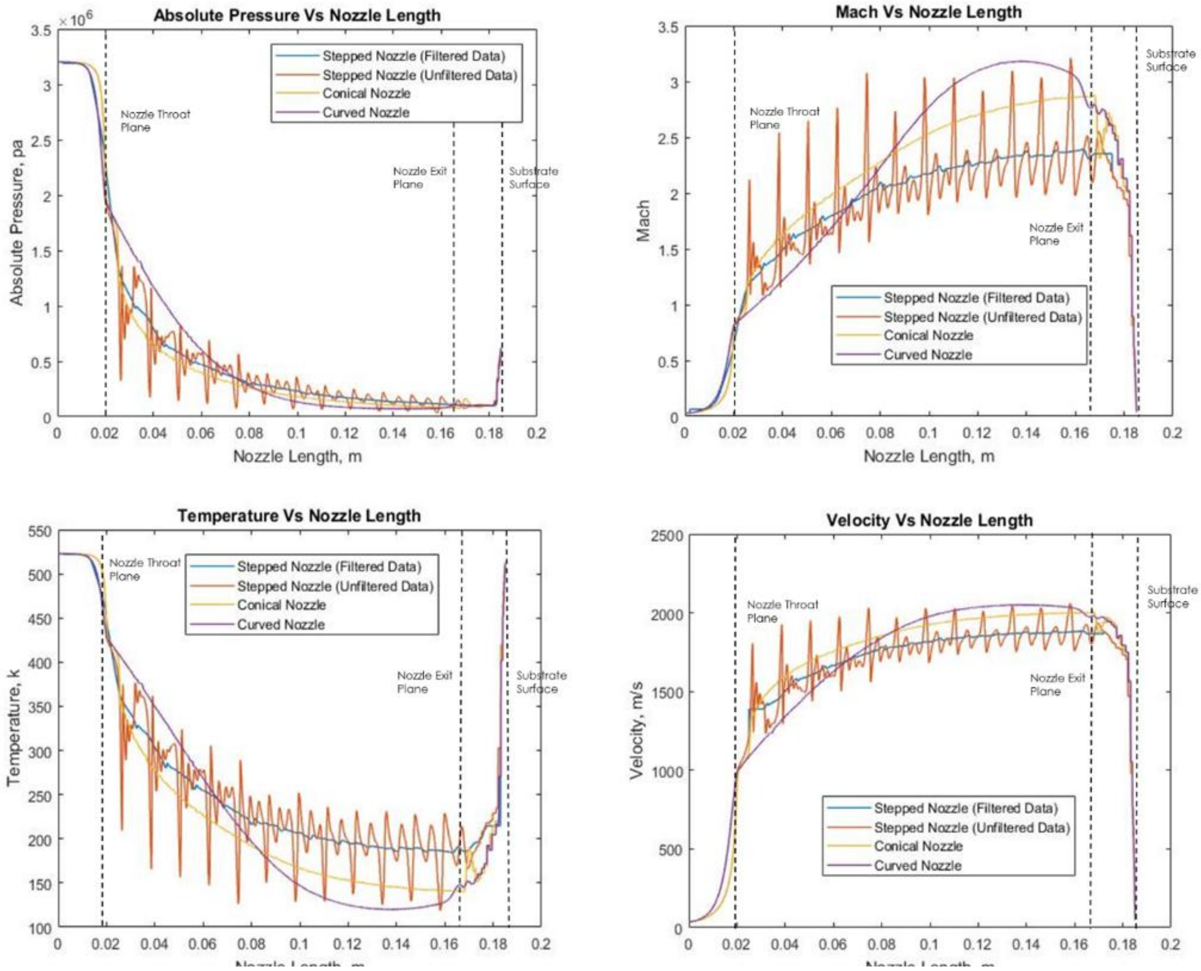

Figure 7: Scalar field plots at the nozzle axis of Step drilled (unfiltered, filtered), Conical and Curved nozzles

conversion of pressure to kinetic energy is lowest among other two nozzles. Density among three nozzles does not differ much, but step drilled nozzle achieved 13.79% and 12.28% lesser density than curved and conical nozzle respectively. The temperature at the nozzle exit is highest in case of step drilled nozzle (34.94% higher than conical, 30.57% higher than curved). Turbulent Kinetic Energy at the nozzle exit in case of step drilled nozzle is 2.59 times higher than conical and 4.82 times higher than the curved nozzle. Whereas the velocity is concerned, step drilled nozzle obtained the least compared to the other two nozzles (6% lesser than conical, 8% lower than curved nozzle).
Considerable losses happen when shock formation occur inside the nozzle. From velocity scalar field image, one can see the shock formation inside the step drilled nozzle. The presence of sharp cornered steps mainly caused the shock formation inside the nozzle. On the other hand, the curved nozzle showed the best performance since its design supports better gas expansion. Summary of scalar field values at the nozzle exit plane is shown in Table 5.

Table 5: Summary of various scalar field values at the nozzle exit

Scalar Field	Unfiltered stepped	Filtered stepped	Conical	Curved
Absolute Pressure, Pa	83969	113526	80767	103542
Density, kg/m^3	0.24	0.28	0.28	0.34
Mach	2.51	2.32	2.87	2.79
Pressure, Pa	-17356	12201	-20558	2217
Temperature, K	170	190	141	146
Turbulent Kinetic Energy, J/kg	5366	5577	1580	957
Total Pressure, Pa	1327596	1336202	2081172	2441148
Velocity, m/s	1926	1869	2001	1981

Table 6: Comparison of the process variables used in the current study with the validation case

	Current Study	Validation Case
Gas Inlet Pressure, MPa	3.2	3
Gas Inlet Temperature, C	250	800
Gas Type	Helium	Nitrogen
Nozzle Throat Diameter, mm	2	2.7
Stand Off Distance, mm	20	35
Convergent Section Length, mm	20	51.2
Divergent Section Length, mm	145	70.3
Divergent Section Profile	Conical	Conical

Verification

Verification of simulations is performed by comparing the results of Muhammad Faizan Ur Rab's simulation [22] to the results obtained from the conical nozzle simulation in the current study. Figure 8 shows the computed velocity profile concerning the nozzle axis for cold spray supersonic jet at 800C, 3 MPa (Verification Case) and at 250 C, 3.2 MPa (Current Study) respectively. As it can be seen from the velocity profiles of the validation case and current study, the velocity along the axis on both nozzles exhibits striking similarity in the trend. Both the nozzles witness significant acceleration in velocity when the gas enters to diverging section, a gradual increase in the velocity till the gas exits the nozzle, negative peak during the travel in the stand-off distance and comes down to zero when it the gas impacts the surface of the substrate. Since the process variables vary in both cases, the velocity magnitude varies but not the trend. Table 6 shows the differences in the process variables. Since there are not much research articles published on a step drilled nozzle used for cold spray, further investigation regarding experimentation and modeling will be required in future work for more exact comparison and verification/validation. Therefore, it is sensible to conclude that the simulation results obtained for the current study are reasonable.

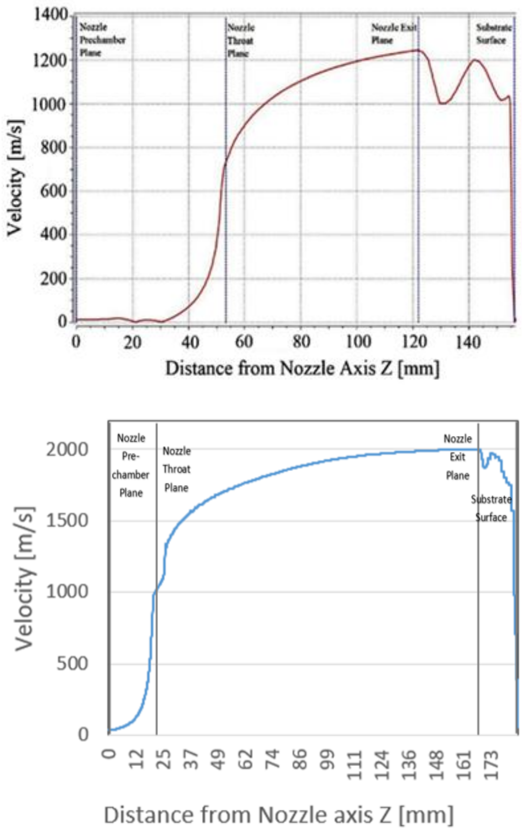

Figure 8: Comparison of velocity plot at the center line between the validation case and the current study

Research Contribution

For the first time, this research study contributes the numerical analysis performed on the step drilled nozzle. In this study, the effect of the internal nozzle steps on the flow characteristics of the cold spray process is systematically investigated through numerical analysis. The results are compared with the conical and curved nozzle designs. CFD approach is employed to achieve this objective by solving 3D full Navier-Stokes equations. Based on the numerical results obtained from the simulation, it is found that the shock formation inside the stepped drilled design significantly influences the flow regime and the gas acceleration.

Conclusions

The conclusion of the research study provides with some suggestions for future work in this area. Comparison of step drilled nozzle, conical, and curved designs were performed by numerical simulation using the typical operating parameters. The results obtained are summarized as follows:

- Presence of shock formation inside the divergent section of the step drilled nozzle creating intense

Contents lists available at http://www.inscience.in/JTSE.html

fluctuations regarding pressure, temperature, velocity and other scalar fields presented in the study

- Step drilled nozzle attained the lowest jet velocity in the divergent section as well as at the substrate surface
- Curved nozzle outperformed the other two nozzles because the divergent section allows significant gas expansion and its design itself will enable a smooth transition from convergent to divergent avoiding the presence of sharp corners
- This research has been mainly focused on analyzing the jet characteristics inside the various nozzle, leaving the study of particle behavior in the nozzle outside the scope of the study. The ideas that could be tested include injecting particles inside multiple nozzles and study their behavior, performing Large Eddy simulation for more detailed analysis, designing De-Laval nozzle using the equations mentioned in the current study, and performing research study on the shock formation inside the nozzle.

References

1. S. Grigoriev, A. Okunkova, A. Sova, P. Bertrand, I. Smurov, "Cold spraying: From process fundamentals towards advanced applications." Surface & Coating Technology, 2014, 256: 77-84.
2. V. Champagne, D. Helfritch, "The unique abilities of cold spray deposition.", International Materials Reviews, 2016, 61(7), 437-455.
3. Hamid Assadi, Frank Gärtner, Thorsten Stoltenhoff, Heinrich Kreye, "Bonding mechanism in cold gas spraying." Acta Materialia, 2003, 51(15): 4379-4394.
4. M. Meyer and R. Lupoi, "An analysis of the particulate flow in cold spray nozzles." Mechanical Sciences, 2015, 6(2): 127-136.
5. A. Moridi, S. M. Hassani-Gangaraj, M. Guagliano, M. Dao, "Cold spray coating: a review of material systems and future perspectives." Surface Engineering, 2014, 30(6): 369-395.
6. T. Stoltenhoff, H. Kreye, H.J. Richter "An analysis of cold spray process and its coatings." J. Thermal Spray Technology, 2002, 11(4), 542-550.
7. A.I. Kashirin, O.F. Klyuev, and V. Buzdygar: "Apparatus for gas-dynamic coating", US Patent 6402050, 2002.
8. Shuo Yin, Meng Zhang, Zhiwei Guo, Hanlin Liao, Xiaofang Wang, "Numerical investigations on the effect of total pressure and nozzle divergent length on the flow character and particle impact velocity in cold spraying." Surface and Coatings Technology, 2013, 232: 290-297.
9. A. Arabgol, H. Assadi, T. Schmidt, F. Gartner, and T. Klassen, "Analysis of Thermal History and Residual Stress in Cold-Sprayed Coatings", J. Thermal Spray Technology, 2014, 23 (1-2), 84-90.
10. V. Varadaraajan, P. Mohanty, "Design and optimization of rectangular cold spray nozzle: Radial injection angle, expansion ratio, and traverse speed." Surface and Coatings Technology 316: 246-254.
11. S. Yin, X. Suo, H. Liao, Z. Guo, X. Wang, "Significant influence of carrier gas temperature during the cold spray process." Surface Engineering, 2014, 30(6): 443-450.
12. M. Karimi, A. Fartaj, G. Rankin, D. Vanderzwet, W. Birtch, J. Villafuerte, "Numerical Simulation of the Cold Gas-Dynamic Spray Process." Journal of Thermal Spray Technology, 2006, 15(4): 518-523.
13. R. Lupoi and O'Neill, "Powder stream characteristics in cold spray nozzles." Surface & Coatings Technology, 2011, 206(6), 1069-1076.
14. X. K. Suo, T. K. Liu, W.Y. Li, Q. L. Suo, M. P. Planche, H. L. Liao, "Numerical study on the effect of nozzle dimension on particle distribution in cold spraying." Surface and Coatings Technology, 2013, 220: 107-111.
15. M Grujicic, W S DeRosset, D Helfrich "Flow analysis and nozzle-shape optimization for the cold-gas dynamic-spray process.", Journal of Engineering Manufacture, 2003, 217(11), 1603-1613.
16. Shuo Yin, Xiao-fang Wang, Wen-ya Li, "Computational analysis of the effect of nozzle cross-section shape on gas flow and particle acceleration in cold spraying." Surface & Coatings Technology, 2011, 205(8), 2970-2977.
17. Masahiro Fukumoto, Hiroki Terada, Masahiro Mashiko, Kazunori Sato, Motohiro Yamada and Eiji Yamaguchi "Deposition of Copper Fine particle by the cold spray process.", Material Transactions, 2009, 50(6), 1482-1488.
18. Wen-Ya Li, Hanlin Liao, Hong-Tao Wang, Chang-Jiu Li, Ga Zhang, C. Coddet, "Optimal design of a convergent-barrel cold spray nozzle by numerical method." Applied Surface Science, 2005, 253(2): 708-713.
19. Zheng-Dong, Zhang Guo-Qing, Zhou, Zhou LI, Zhang Yong, Xu Wen-Yong, "Simulation of Gas Flow Field in Laval Nozzle and Straight Nozzle for Powder Metallurgy and Spray Forming," Journal of Iron and Steel Research, 2008, 15(6), 44-47.
20. https://mdx.plm.automation.siemens.com/star-ccm-plus
21. https://www.mathworks.com/products/matlab.html
22. Muhammad Faizan Ur Rab, Saden Zahiri, Syed H. Masood, Mahnaz Jahedi and Romesh Nagarajah "Development of 3D Multicomponent Model for Cold Spray Process Using Nitrogen and Air.", Coatings, 2015, 5(4), 688-708.