CDF STUDY OF BIPHASIC FLUID IN CONVERGENT-DIVERGENT DE LAVAL NOZZLE

Maricruz Hernández-Hernández[1], Victor Hugo Mercado-Lemus[1], Hugo Arcos Gutiérrez[2], Isaías Garduño-Olvera[2], Adriana Del Carmen Gallegos Melgar[1], Jan Mayen[2], Raul Pérez Bustamante[1].

[1]CONACYT, Corporación Mexicana de Investigación en Materiales, Saltillo, Coahuila, C.P. 25290, México.

[2]CONACYT, CIATEQ, Unidad San Luis Potosí, Eje 126 No. 225, Zona Industrial, San Luis Potosí C.P. 78395, México.

Abstract

The fluid behavior in a convergent-divergent nozzle employed in Cold spray process is numerically analyzed. Cold spray, also called Cold gas dynamic spray, has a high potential, both for the generation of coatings and for the additive manufacturing technique. One of the main components of this equipment is ***the spray gun***, its configuration is highly important in the control of the final characteristics of the coating, and in the efficiency of the process. Gas dynamics are responsible for delivering a power at a desired velocity and temperature. A high-pressure gas flows into a de Laval nozzle with the ability to accelerate compressible fluids at supersonic speeds, largely determined by the nozzle configuration. The transporter gas operates in an adiabatic, reversible regimen, and calorically perfect - mean the gas behavior is governed by isentropic flow relation. In this work, the gas dynamic behavior in two different nozzle geometry is numerically analyzed using OpenFOAM under compressible conditions.

Introduction

In the Laval nozzle, dissipative effects like viscosity and heat transfer occur mainly in thin boundary layers near the nozzle walls. This means that a large part of the gas operates in an adiabatic, reversible regimen. Furthermore, temperatures and pressures are low enough to ignore intermolecular forces and to consider the carrier gas as calorically perfect: the specific heats are approximated as constant. The combination of these three approximations – adiabatic, reversible, and calorically perfect means the gas behavior is governed by isentropic flow relation.

For supersonic nozzle flow, OpenFOAM compressible flow Navier-Stokes (NS) solvers are of interest; *rhoCentralFoam* is a density-based, transient solver that merges PISO and SIMPLE algorithms; and it is a KT/KNP solver that uses the central-upwind schemes proposed by Kurganov and Tadmor. This solver has the ability to handle supersonic compressible flows and it has proved its capabilities in the viscous supersonic flow regime.

Gas Flow in a de Laval nozzle

Gas dynamics behavior in Cold spray gun can be described by these relationships. it is possible to observe that the variables along the axial nozzle length are possible to calculate with the value of the input variables. The principal feature of the isentropic relations is that gas velocity increases at the expense of pressure and temperature. These isentropic relations will be used to test the validity of the simulation results in the compressible regime. At $M = 1$, we can define a sonic temperature T^*, a sonic pressure $p*$, a sonic density ρ^*, and a sonic area A^*.

$$\frac{A}{A^*} = \frac{1}{M}\left[\frac{2}{\gamma+1}\left(1+\frac{\gamma-1}{2}M^2\right)\right]^{\frac{\gamma+1}{[2(\gamma-1)]}}$$

$$\frac{P}{P^*} = \left(1+\frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}$$

$$\frac{T}{T^*} = 1+\frac{\gamma-1}{2}M^2$$

$$\frac{\rho}{\rho^*} = \left(1+\frac{\gamma-1}{2}M^2\right)^{\frac{1}{\gamma-1}}$$

Approach

The open-source software **OpenFOAM** was used in order to solve the gas dynamics into the de Laval nozzle used in the Cold spray process by using the solver rhoCentralFoam. The fluid parameters tracked throughout the simulation are temperature, velocity, density, and pressure. Because this simulation is run under non-isothermal conditions, a thermophysicalProperties file is required. The thermodynamic qualities of the fluid are set such to approximate the fluid like air. The flow is assumed to be laminar, which is set in the turbulenceProperties file. In these simulations, a maximum Courant number of 0.1 was chosen to ensure stability and the time step set up was $1x10^{-8}$ s. The simulation was run for 0.07 seconds. The nozzle itself is adiabatic, assuming negligible heat flux between the flow and the nozzle, and a no slip condition is imposed at the wall assuming continuum flow. The gas is assumed to satisfy the ideal gas relation. The gas is also assumed to have a constant ratio of specific heats ($\gamma = 1.4$). The geometry is an axisymmetric converging-diverging duct. Figure1 shows the nozzle domain and boundaries. In this study, two expansion ratios are considered for nozzles, 2.96, and 4. The final computational domains were designed with the OpenFOAM meshing tool *blockMesh*. The same inlet conditions were used for both nozzles. At the inlet, total temperature and total pressure are specified. The total inlet temperature is set to 750 K, and the inlet pressure of 35 bar. Conditions of zeroGradient for pressure and temperature are imposed at the outlet.

Results

	Nozzle 1	Nozzle 2
Inlet Diameter (mm)	13	10
Outlet Diameter (mm)	4	6
Throat (mm)	1.35	1.5
Convergent Length (mm)	10	10
Divergent Length (mm)	100	140

Table : Summary of Nozzle Dimensions

Figure 1:Geometry domain and boundary conditions of nozzle.

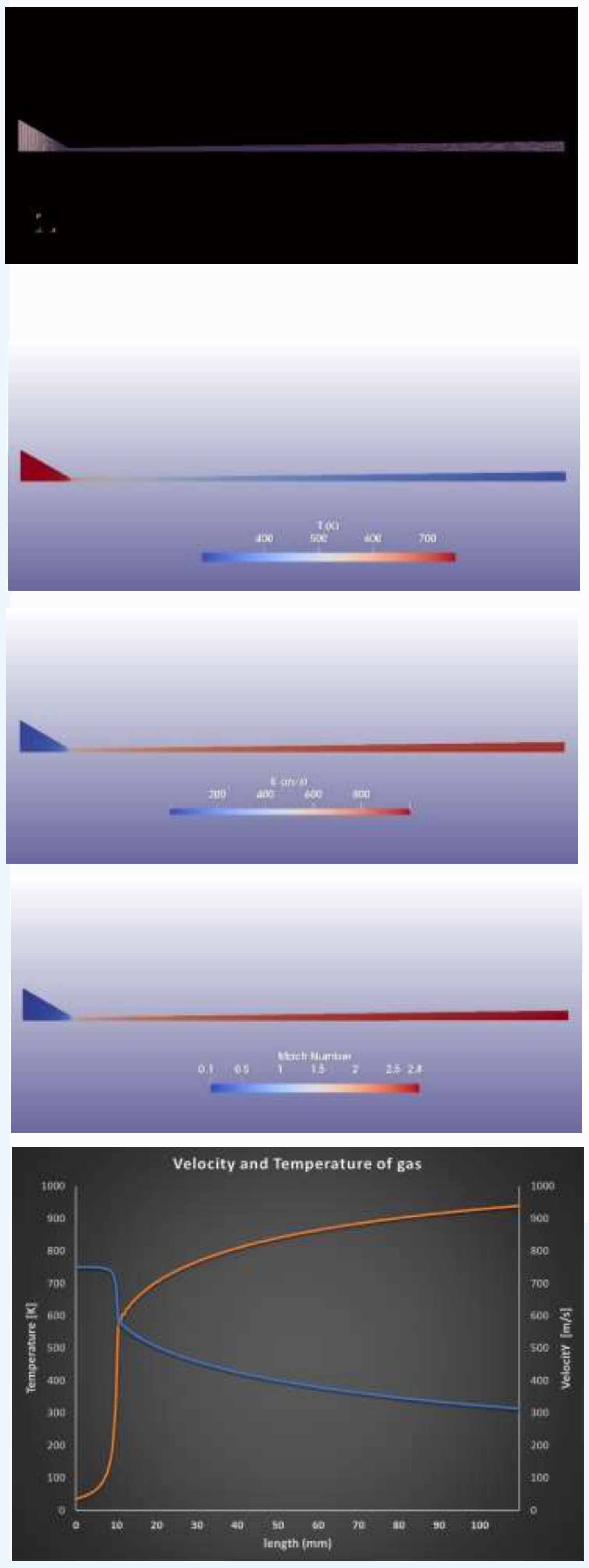

Figure 2:Geometry, T, U, number Mach and graph for nozzle1.

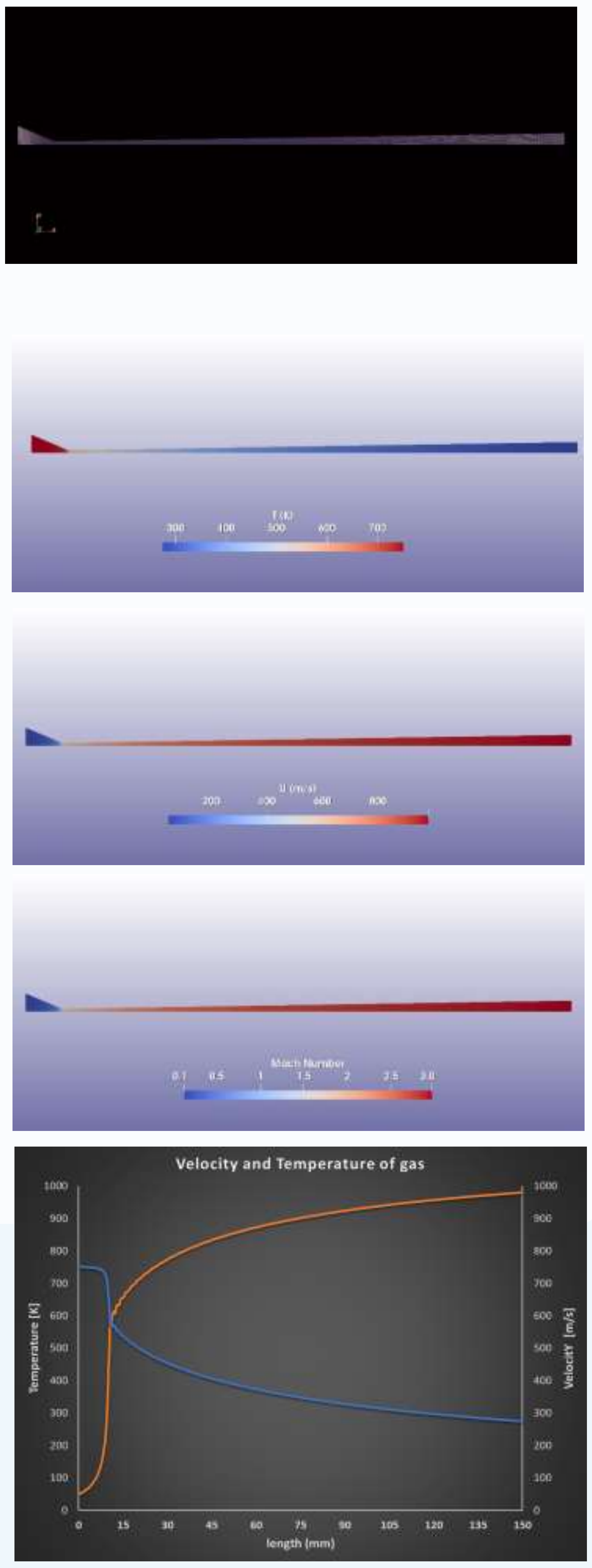

Figure 3: Geometry, T, U, number Mach and graph for nozzle2.

Conclusion

The results presented demonstrate, that OpenFOAM along with rhoCentralForam are powerful tools to analyze de Laval nozzles. The simulation results obtained for different nozzle geometries show a good correlation to theory. From the figures, it is possible to conclude that the largest converging area the exit velocity of the gas increases.

The numerical simulation tools are valuable to identify governing parameters in the de Laval nozzle and can help to develop a deeper understanding of that technology.

Acknowledgements

This work has been funded by CONACYT CB project A1-S- 23339 "Estudio del efecto de la aglomeración de partículas en una tobera convergente-divergente tipo Laval mediante CFD".

Acknowledgments: The authors would like to acknowledge the support from CONACYT through the program Cátedras CONACYT by Project grant number 674 and 850.

For any question, please contact:

maricruz.hdz@comimsa.com