Cold Spray

Simulation results – 1

Fraunhofer Institute for Production Systems and Design Technology

Dinesh Poluru Baskar

Hochschule Furtwangen University

June 5, 2021

Contents

1 Introduction

This report postulates the simulation results which has been carried out in ANSYS FLUENT for the desired geometry. The geometry has not been changed till now and the simulation is run with some boundary conditions which are discussed in the report below. The simulation is running in two possible ways namely:

- Simulation with a carrier gas and without deposition particles.

- Simulation with a carrier gas and deposition particles.

2 Simulation

ANSYS 2020 R2 version of the software is used for the simulation and the geometry is unchanged for the first set of simulations. The reason the geometry is unchanged because to study the flow properties of fluid inside the nozzle. Two models have been used in the simulation namely Eulerian and DDPM (Dense Discrete Phase model). The energy equation is used for the simulation with carrier gas without deposition materials.

2.1 Eulerian Model

Multiphase models have different types of Eulerian models which can be used for the simulation. Since our application is tracking the particles as well as the flow which promotes that particle, the Eulerian model is best suited for our application. The VOF (Volume of Fluid) and the Mixture model simulate the multiphase model has a dispersed-continua model which is that the particles are dispersed in the fluid in which the tracking of particles is difficult or impossible.

2.2 Dense Discrete Phase Model

This model helps to solve the second phase of spherical particles (droplets or bubbles) dispersed in the continuous phase. FLUENT computes the trajectory of these discrete phase entities, as well as heat and mass transfer to/from them. The initial velocity, diameter, and mass flow rate can choose for different materials from the DPM model. [1]

3 Procedure

The simulation procedure from creating the geometry to the results is explained below. The simulation is run for 2000 iterations in which some solutions have been converged before reaching the maximum value and some results show errors that should be rectified in the upcoming simulations results.

3.1 Geometry

The geometry is created using Spaceclaim modeling available in the ANSYS and the model is created with the base dimension which was provided for the simulation purpose. In the upcoming days, the model can be changed according to the simulation results to match the application.
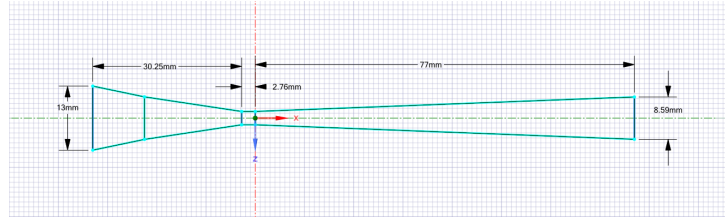
3.2 Mesh

The created geometry is split into 4 faces by using the FaceSplit command and the edge sizing is chosen for all the edges in the geometry for the meshing. To make an ordered mesh face mesh is chosen for all the faces in the geometry.
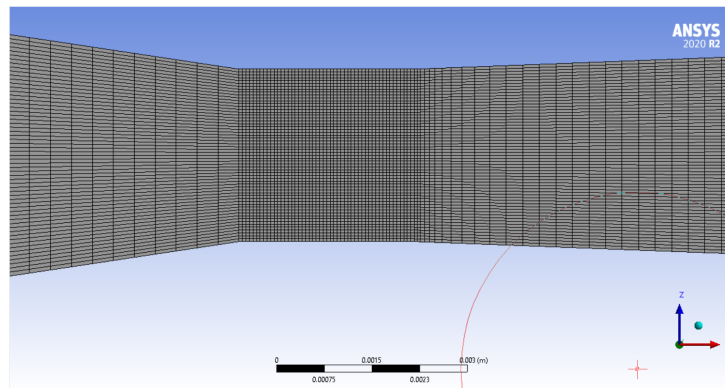
4 Setup

The setup section can be divided into two namely simulations with a carrier gas, simulation with a carrier gas, and particles. Both the simulation result shows not even close to expected results as there are some noises and errors which has to be taken care.

4.1 Simulation with carrier gas

The carrier gas used in Nitrogen with the provided initial condition. Inlet pressure is about 50bar and inlet temperature is about 750-degree. In this model, the energy equation is used for the simulation with inviscid flow conditions. The inviscid flow is used because the flow is compressible and the density-based flow is checked in the general setting of the setup.

As the inlet condition for the flow is set the materials have been chosen from the materials tab from the fluent database. The density and the specific

heat of the nitrogen material have been set constant because the study of flow must provide a better result with the constant values provided by the FLUENT solver.

4.2 Result for the carrier gas

The simulation is run for 2000 iterations. A pressure plot has been added to the simulation graphs. The initialization is computed from the inlet and the pressure inlet is set to the 50bar.
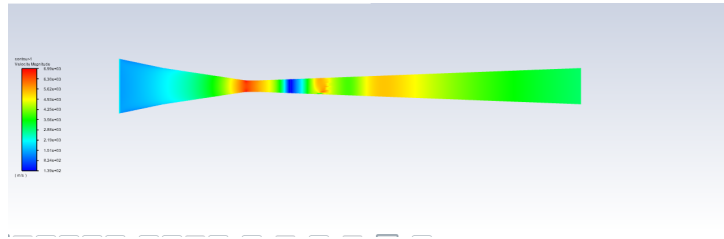
This picture represents the velocity magnitude for the nitrogen material. Due to some errors in the calculation, the solution is blown up and the graph shows an error near the throat. This can be viewed better with the line graph.
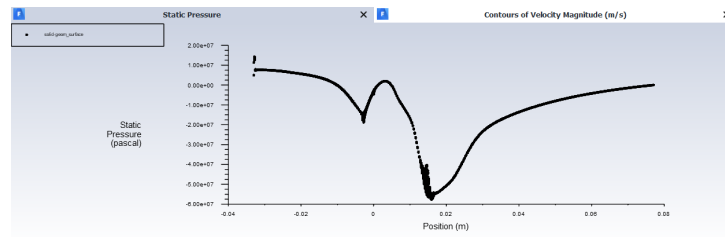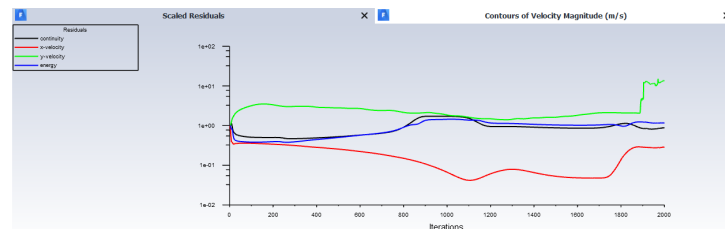

This graph represents the static pressure inside the nozzle. From the graph, it is clearly shown that there are errors in the simulation that was run, and in the upcoming simulation, these errors should be sorted out.


```
Calculation complete.

            Mass Flow Rate                 (kg/s)
    -----------------------------    ----------------------
                         inlet           -36.822194
    -----------------------------    ----------------------
                           Net           -36.822194

            Mass Flow Rate                 (kg/s)
    -----------------------------    ----------------------
                        outlet           -29.583366
    -----------------------------    ----------------------
```

And the mass flow rate which was recorded shows that the flow is reversed in both cases which does not make any sense in this application.

4.3   Simulation with deposition material

The carrier gas used is Nitrogen with the same initial condition of the pressure of temperature. As we discussed DPM model is a particle simulator where we can track the individual particles in the flow. The model is set to Eulerian and the DPM model is turned on with a maximum number of steps of 500 is set. This value is the default value that is available for copper particles in the DPM model tab. Before setting the DPM model, injection particles should be set up in the materials tab.

The diameter distribution is set to linear because we are dealing with particles with the same diameter. If we are dealing with different diameters, we can choose the rosin-rammler diameter distribution. The starting and ending point of distribution is given on the axis of the nozzle. These values which I have shown are from my point whereas the values are not mentioned so I have chosen some values which could suit the simulation as I already run some simulations regarding these parameters. There is no change in the boundary and initial condition as we used the same in the last simulation. The iteration steps have been set to 2000 and the results are recorded.

4.4   Result for deposition material

The simulation result which I have shown here is pressure gradient in which the results show some errors.
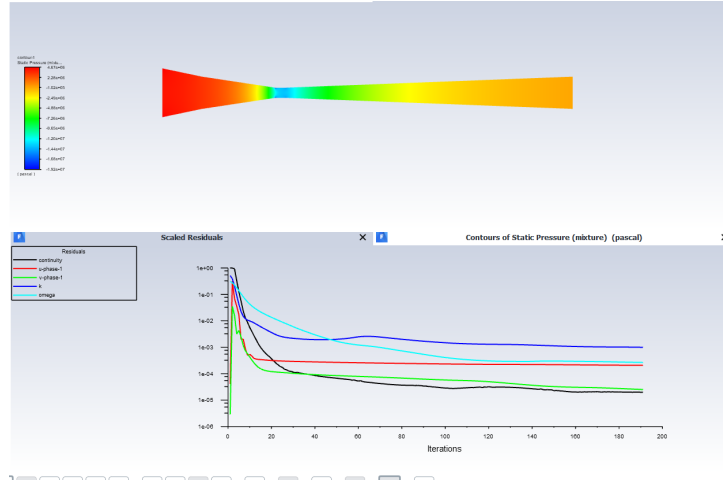
As we can see from the residual graph it shows some irregularities in the velocity phase from the inlet of the nozzle.

5   Conclusion

This report gives a view about the simulation results and the errors which in the upcoming process improvement in the result should be done for better efficiency. The problem with the simulation results could be the mesh or the setup cases used in the simulation and the boundary conditions. These problems should be rectified for the next simulation.

References

[1]   UDF Manual. "ANSYS FLUENT 12.0". In: *Theory Guide. Canonsburg, PA* (2009).