Computational Fluid Dynamic Analysis of Water Flow Through Nozzle of a Gravimetric Measuring Method System

Aguilar Corona Miguel[1], Nuñez Martín C. Alexander[2] Mendez Miguel Luis Rolando[3]

1 Posgrado CIATEQ A.C, Querétaro, Qro.
2 Tecnológico Nacional de México, Campus Querétaro, Qro.
3 CIATEQ, Querétaro, Qro.

DOI: https://doi.org/10.56293/IJASR.2024.6017

IJASR 2024
VOLUME 7
ISSUE 4 JULY – AUGUST **ISSN: 2581-7876**

Abstract: The gravimetric measuring method incurs errors and uncertainties in calibration, originating from various sources, with the diverter valve being one of the most significant. For optimal operation, the output flow from the diverter valve should form a jet in a thin, rectangular sheet form to ensure a low uncertainty value. The current work explores different scenarios in the configuration of an upstream diverter valve installation to determine its effect on velocity profile and flow path line at the nozzle outlet. The analysis is conducted using CFD Fluent edition 2023, varying operational conditions and nozzle geometry. In the results section, a pressure drop along the nozzle at different velocities as well as uniformity in velocities vectors and pressure magnitude at outlet nozzle, are presented.

Keywords: Nozzle, CFD, Fluids, Gravimetric measuring method, diverter valve

1. Introduction

In flow measurement field, to ensure the measured magnitude is being correctly determined is the main goal. one way in order to achieve a certain level of accuracy, a calibration of the flowmeter device is required. The calibration process is carried out using equipment with higher metrological hierarchy, called standard. The calibration by gravimetric method complies with requirements provided by standard ISO 4185. The system consists of a diverter valve, a time measuring device and a weighing system. Each element of the system induces an error in flow measurement. The present work is focused on the error induced by the diverter valve, the most significant element in the uncertainty budget [1]. The performance of the diverter valve is affected by its outlet nozzle, located upstream. The nozzle has the function of conditioning the liquid's stream to form a rectangular jet of water. Water jet must be thin and stable in order to avoid splashing during supply liquid to measurement tank and during diversion of fluid. The flow of fluid its affected by piping configuration as length of straight pipe, direction changes, valves and fittings that can induces swirls or air pockets. A CFD Analysis is carried out by parameterizing variables as geometry nozzle and two configurations of installation: elbows in same plane and elbows in perpendicular plane. The goal of analysis is to determine the pressure drop through line, uniformity of velocity vectors at the outlet nozzle and streamline to identify a swirl zone in order to propose a more efficient design of nozzle for a prototype.

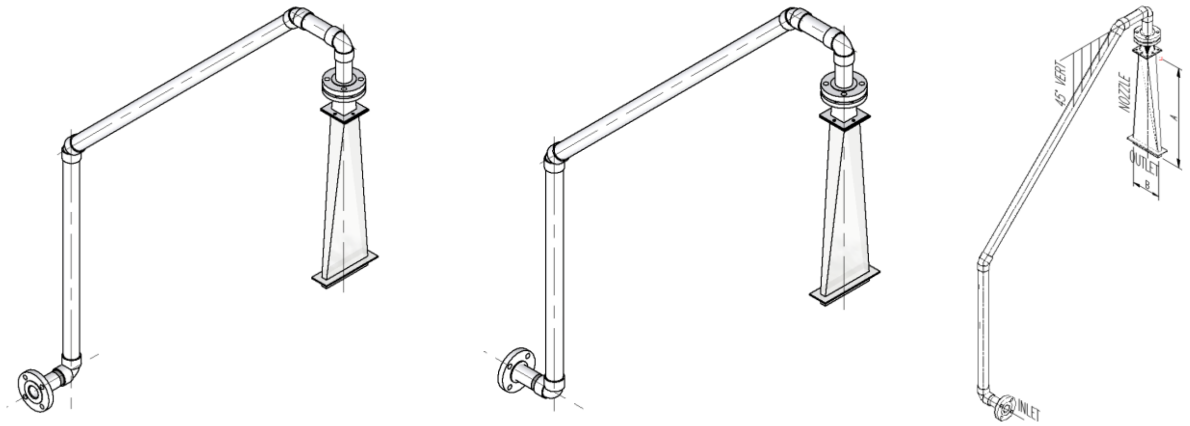
Method description

The method used is divided into four essential stages which will provide the guidelines for the development of the research carried out in order to meet the goal. The stages are described below.
Stage 1 Define the conceptual installations; propose the possible configurations to evaluate and define the best choice to manufacture, the considered model is 2^k with eight installation scenarios
Stage 2 Defining the model; choose the mathematical formulation that best describe the phenomena of fluid in piping according to operational condition.
Stage 3 CFD simulation; to execute simulation using CFD software teaching version, following steps preprocessing, solver and post processing. Interest area it's the outlet nozzle.
Stage 4 Results analysis: according to results, choose the configuration to consider in manufacturing, the criteria it's the velocities profile at outlet nozzle.

Methodology

Installation description and considerations

The arrangement is confirmed by hydraulic piping and fittings in 1 ½" size of diameter, material PVC, schedule 40, the test fluid considered is pure water, operation conditions max flow 200 kg/min and 20 °C. The interest region in the analysis is the output of nozzle, it should be clarified that cross-sectional area it is the same at inlet than outlet nozzle to any case, the geometry nozzle is parameterized modifying length dimension and two scenarios are considered to analyze: elbows y same plane and elbows in perpendicular plane. Figure 1 shows the three different scenarios.



Long nozzle, elbows in perpendicular plane

Long nozzle, elbows in same plane

Varying length of nozzle

Figure 1. scenarios to analysis

Reynolds number based on the pipe diameter (40 mm) and average velocity (1 m/s), at 20°C will be 40 000 which is higher than 4000. At this Reynolds number, the flow is usually completely turbulent.

Dimensions of nozzle are indicated in table 1, it shown two lengths of nozzle.

Table 1. nozzle dimensions

Parameter	Dimension m	Dimension m
Length (A)	0,60	0,30
Width (B)	0,15	0,15
Thick (C)	0,01	0,01

Mathematical formulation: The fluid considered in the analysis presented in this work is pure water, considered as Newtonian fluid and turbulent flow regime, the shear stress is linearly proportional to shear strain ratio [2]

The equations that define the model of flow are the equation of conservation of mass or continuity equation and the second Newton´s law (Navier – Stokes equation)

$$\rho = \frac{D\vec{V}}{Dt} = -\vec{\nabla}P + \rho\vec{g} + \mu\nabla^2\vec{V} \quad (1)$$

Where:

r = density of the fluid
m = viscosity
g = gravity
P = pressure
V = velocity

Continuity equation of incompressible flow.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad (2)$$

Components (x, y, z) Navier – Stokes Equation

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial P}{\partial x} + \rho g_x + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad (3)$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial P}{\partial x} + \rho g_y + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \qquad (4)$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial P}{\partial x} + \rho g_z + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \qquad (5)$$

Model

Due characteristics installations and test liquid, the model used in CFD analysis is the k- ε turbulence model. k- ε turbulence model is by far the most frequently used model in industrial applications due to its robustness and its low computational cost [3]. Although, its performance is poor in cases of strong adverse pressure gradients. The model consists of solving two additional equations: for the transport of turbulent kinetic energy k (which determines the energy in the turbulence) and for the dissipation rate of turbulent kinetic energy ε (which determines the scale of the turbulence). uses the hypothesis of a convection and diffusion gradient to relate Reynolds stresses to mean velocity gradients and turbulent viscosity. Used to high Reynolds number flows, either in the incompressible or compressible regime with Ma < 0.3. The flow near the walls is modeled using wall functions, therefore it is not simulated. It can be applied in stationary or transient studies.

Computational fluid analysis consists of three stages, preprocessing, solver and post processing, described as next

- Goal of the analysis, predict behavior of velocity vectors at nozzle outlet and determine drop pressure in the line.
- Identify the domain to model; The area of interest of the system to be analyzed is the nozzle outlet, must be pay attention to the geometry, preferably avoiding geometries or configuration that compromise the quality of the mesh.
- Create the robust domain model; generate the solid with parametric dimensions for design optimization.
- Meshing, design and create the mesh; refine meshing in areas of interest, the meshing consists of a large number of cells or finite volumes for the phenomenon of study. A CFD type mesh is suitable in this case.
- Setup the solver; For the analysis of the input line, it is considered to use a viscous model of the k-epsilon (2 eq) type of the type realizable without functions for standard treatment on the wall, in addition the multiphase model of the volume of fluid VOF type is used since it its necessary ensure if there are discontinuities due to empty spaces like bubbles or air pockets in the fluid.
- Solver
- Analysis of results
- Finally consider if its necessary review the model or if results are different to expected.

The simulation is carried out in the Ansys Fluent 2023 R2 [4] software in a three-dimensional model. Considerations for the analysis are listed below in table 2.

Table 2. input data

Setup	
General	
Solver Type	Pressure type
Velocity formulation	Absolute

Time	Steady
Gravity	-9,81 m/s² Y direction
Models	
Viscous	k-epsilon (2eqn), Realizable Near-wall treatment; standard wall functions Model constants; 1,9 C2-Epsilon, 1 TKE Prandtl & 1,2 TKE Prandtl.
Multiphase	Models multiphase - volume of fluid Homogeneous model; volume of fluid Number of Eulerian phases 2, Volume fraction parameters; formulation implicit Phases; primary phase 1 air, secondary phase 2 water-liquid Force setup; Surface tension coefficient constant 0,073 VOF immiscible fluid
Materials	
Fluid	Air and water-liquid (fluent data base)
Boundary conditions	
Velocity inlet	Min 1 m/s max 2,5 m/s Turbulent intensity 2,5 % Hydraulic diameter 0,041 m Volume fraction 1 to phase-2
Outlet	Mixture Turbulence; specification method intensity and viscosity ratio 5% backflow turbulent intensity 10 backflow turbulent viscosity ratio Outlet phase-2 Volume fraction specification method; backflow volume fraction 0 Backflow volume fraction
Solution	
Methods	Scheme SIMPLEC Spatial discretization; least squares cell based Pressure PRESTO Momentum; second order upwind Volume fraction compressive Turbulent kinetic energy; second order upwind Turbulent dissipation rate; second order upwind
Initialization	Standard, compute from inlet Patch, zones to patch solid
Run calculation	750 number iterations
Results	
Contours	Contours of phases, volume fraction, phase-2, surface plane-n (outlet nozzle)
Vector	Vectors of velocity, phase mixture, velocity magnitude, surface plane-n
Pathlines	Turbulence, phase mixture

Scenarios

1. Elbow in same plane Nozzle length 600 mm, speed 1 m/s at inlet piping
2. Elbow in same plane Nozzle length 600 mm, speed 2,5 m/s at inlet piping
3. Elbow in same plane Nozzle length 300 mm, speed 1 m/s at inlet piping
4. Elbow in same plane Nozzle length 300 mm, speed 2,5 m/s at inlet piping
5. Elbow in perpendicular plane Nozzle length 600 mm, speed 1 m/s at inlet piping
6. Elbow in perpendicular plane Nozzle length 600 mm, speed 2,5 m/s at inlet piping
7. Elbow in perpendicular plane Nozzle length 300 mm, speed 1 m/s at inlet piping
8. Elbow in perpendicular plane Nozzle length 300 mm, speed 2,5 m/s at inlet piping

The meshing of the model is carried out according to the criteria that allow us to minimize the error in the results; Aspect ratio (max 20, recommended 10), skewness (max 0,85 recommended 0,25) and orthogonal quality (0 ideal and 90 worst).

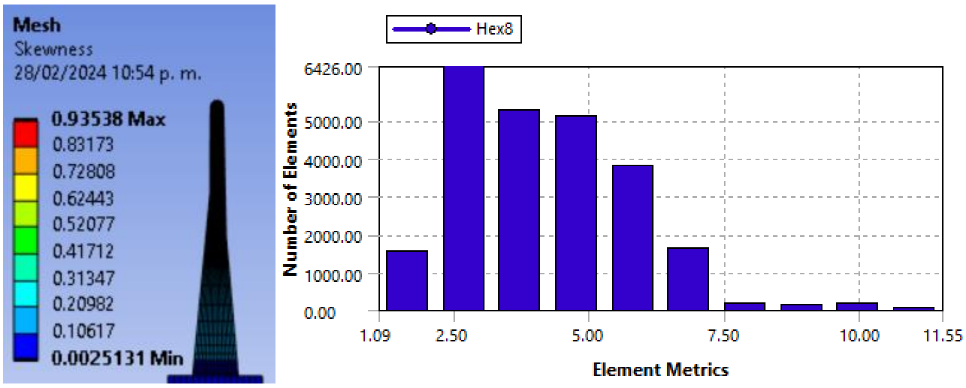


Figure 2. Mesh quality

In Figure 3 the displacement of the air phase (phase-1) in the pipe, in order to verify absence of bubbles or air pockets are shown

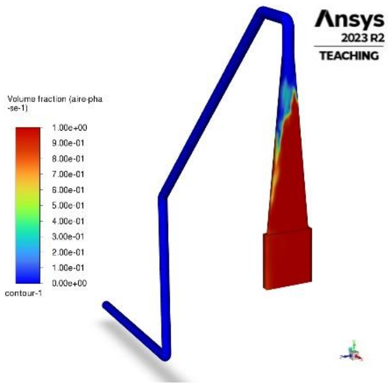
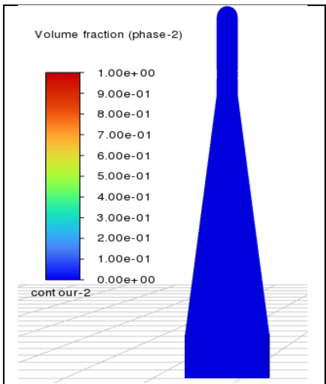
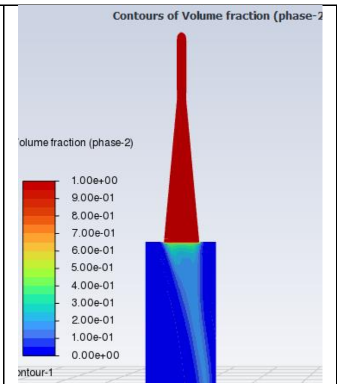
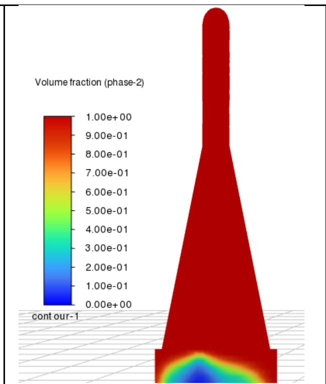
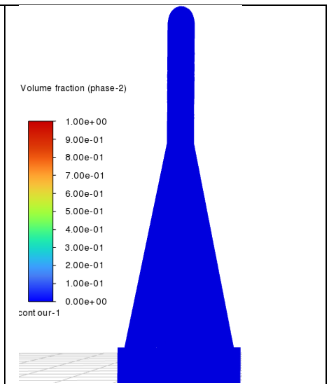
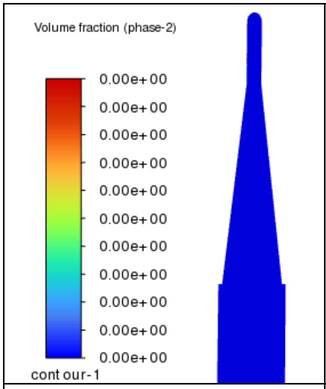


Figure 3. inlet fluid through nozzle

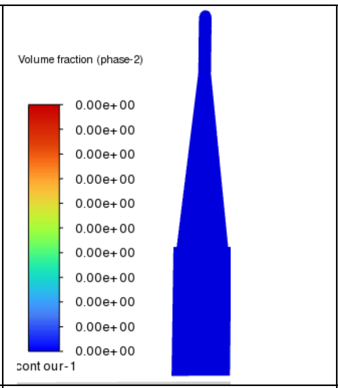
Results

The interest area is the outlet nozzle, thorough a plane in sheet liquid we can see the contours, vectors and streamline; to determine the best configuration, results are shown in figures below.

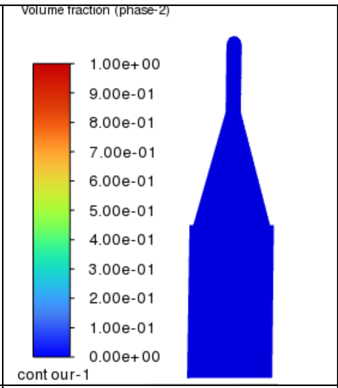
			
Contours Scenario 1	Contours Scenario 2	Contours Scenario 3	Contours Scenario 4



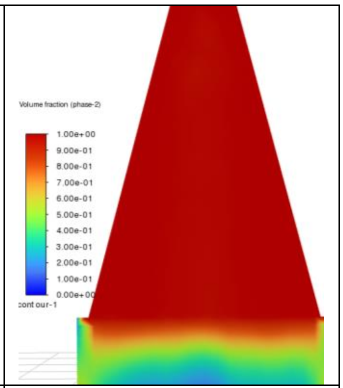
Contours Scenario 5



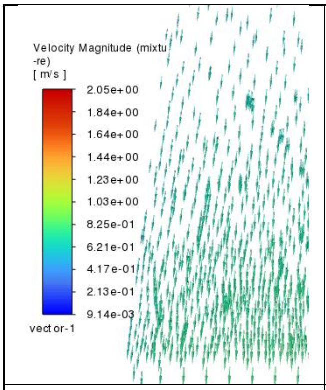
Contours Scenario 6



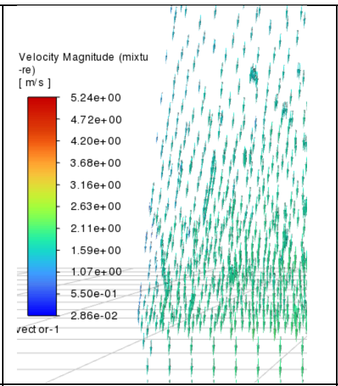
Contours Scenario 7



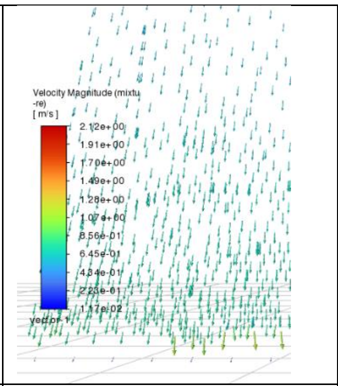
Contours Scenario 8



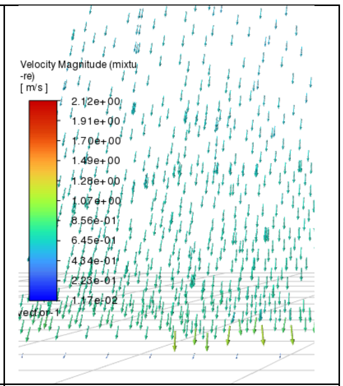
Vectors Scenario 1



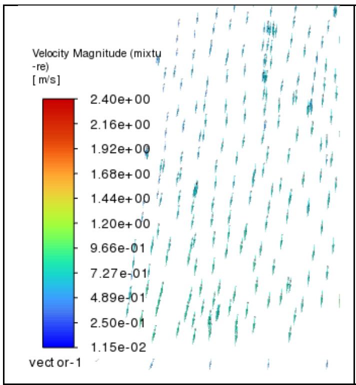
Vectors Scenario 2



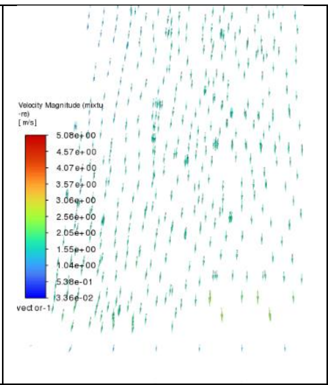
Vectors Scenario 3



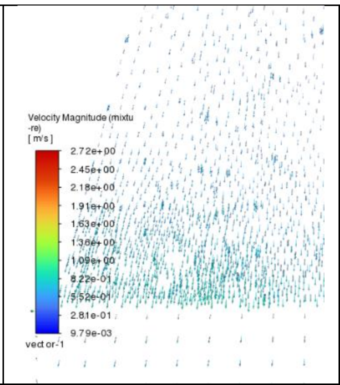
Vectors Scenario 4



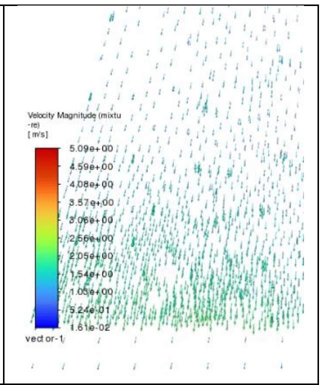
Vectors Scenario 5



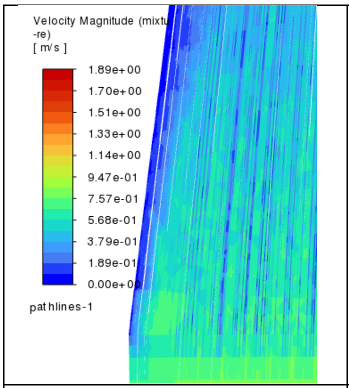
Vectors Scenario 6



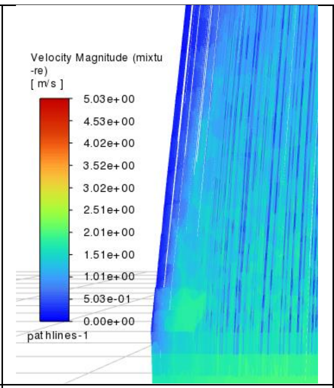
Vectors Scenario 7



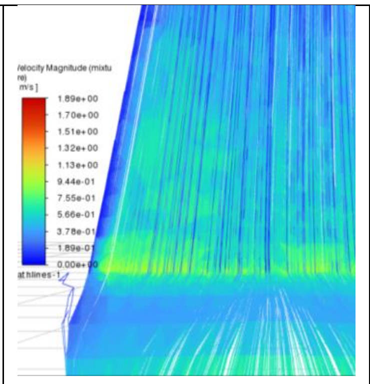
Vectors Scenario 8



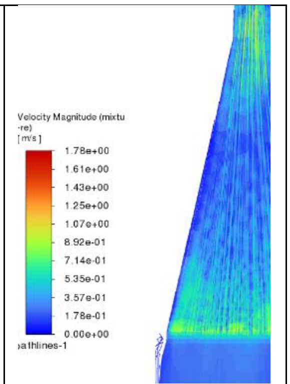
Streamlines Scenario 1



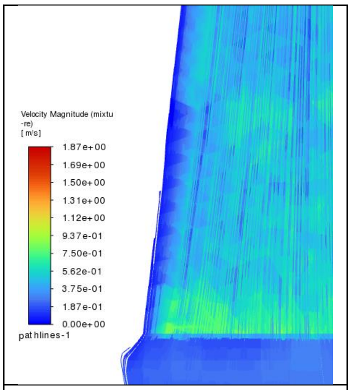
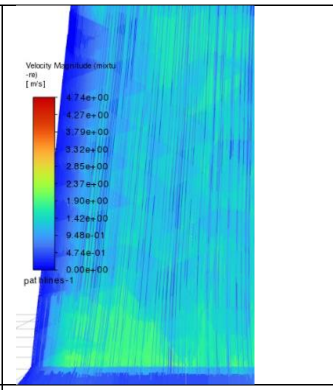
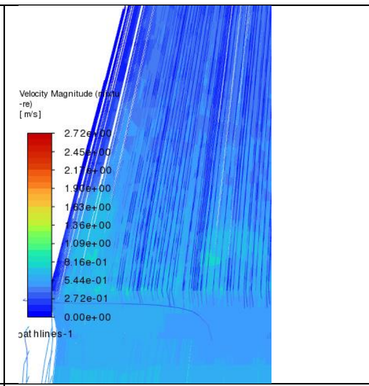
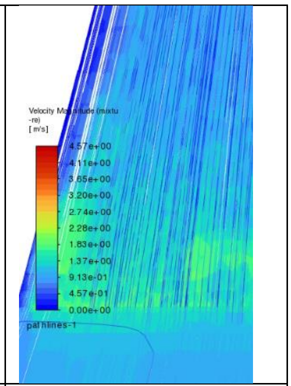
Streamlines Scenario 2



Streamlines Scenario 3



Streamlines Scenario 4

			
Streamlines Scenario 5	Streamlines Scenario 6	Streamlines Scenario 7	Streamlines Scenario 8

Inlet pressure, outlet pressure and drop pressure through nozzle are shown in table 3

Table 3. pressures

Scenario	Inlet pressure Pa	Outlet pressure Pa	Drop pressure Pa
1	6243.55	998.06	5245.49
2	5917.19	1076.77	4840.42
3	2996.31	373.94	2622.37
4	3389.35	2140.63	1248.72
5	9292.57	4060.07	5232.50
6	9575.47	4800.34	4775.13
7	3002.50	170.87	2831.63
8	7152.54	4704.22	2448.32

A criteria to consider in design of nozzle it's the drop pressure, according to ISO 4185 [5] The pressure drop across the nozzle slot should not exceed about 20 000 Pa to avoid splashing. From table 3, all scenarios meet the criteria.

Conclusion and discussion

When a fluid pass around a bend the friction resistance of the pipe walls and the action of centrifugal force combine to produce rotation and to induce swirls in flow, however, due lengths of the installation, the profile its fully developed at nozzle outlet, therefore, we choose the scenario with long nozzle and elbows in same plane has a better performance in the velocities vector as presented and will reproduce in prototype in order to verify results in CFD.

References

1. Jaiswal, Shiv & Yadav, Sanjay & Bandyopadhyay, Ashis & Agarwal, Ravinder. (2012). Global Water Flow Measurement and Calibration Facilities: Review of Methods and Instrumentations. Mapan-Journal of Metrology Society of India. 27. 63-76.
2. Yunes A. Çengel John M. *Mecánica de fluidos; fundamentos y aplicaciones*, Cimbala, 4ta ed.
3. David C. Wilcox, *Turbulence Modeling for CFD*, DWC Industries, third edition
4. John E. Matsson Ph.D., P.E *An introduction to ANSYS Fluent 2023*, SDC Publications
5. ISO 4185 Measurement of liquid flow in closed conduits - Weighing method