CAESIM

Water Jet Simulations

Validation & Verification Report

CONTENTS

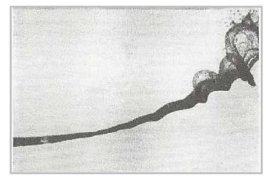
SECTION **PAGE**

1.0 Introduction

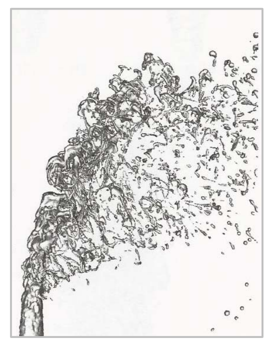
The purpose of the Verification and Validation tasking is to ensure that the CAESIM software performs its intended function for high speed water jet flows, including gaseous cross flow conditions. Task VI consists of benchmarking the CAESIM software against published experimental/numerical research.

This report documents five V&V CFD models developed, and presents the simulation results compared to published research. The following experimental and numerical research will be compared to simulation results produced by CAESIM.

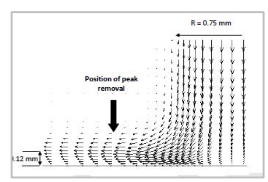
1. “Breakup of Round Non-turbulent Liquid Jets in Gaeseous Crossflow”, K. A. Sallam, AIAA Journal Vol.42, No. 12, Dec 2004.



2. “Detailed simulations of primary breakup of turbulent liquid jets in crossflow”, M. G. Pai et al, Center for Turbulence Research Annual Research Briefs, 2008.



3. “Material Removal in Magneto Rheological Finishing of Optics.” QED Technologies International.



4. “An Experimental and Numerical Study of a Water jet Cleaning Process”. Mechanical & Materials Engineering Dept, University of Windsor, Windsor, Ontario Canada.

2.0 Verification & Validation - Cross Flow Cases

a. Case # 1 – “Breakup of Round Non-turbulent Liquid Jets in Gaseous Cross flow”

Research Publication Abstract

“An experimental investigation of the primary breakup of round non-turbulent liquid jets in gaseous cross flow is described. Pulsed shadowgraph and holograph observations were made to determine the following breakup properties: primary breakup regimes, conditions required for the onset of ligament and drop formation, ligament and drop sizes along the liquid surface, drop velocities after breakup, rates of liquid breakup along the liquid surface, conditions required for the breakup of the liquid column as a whole, and liquid column trajectories…”

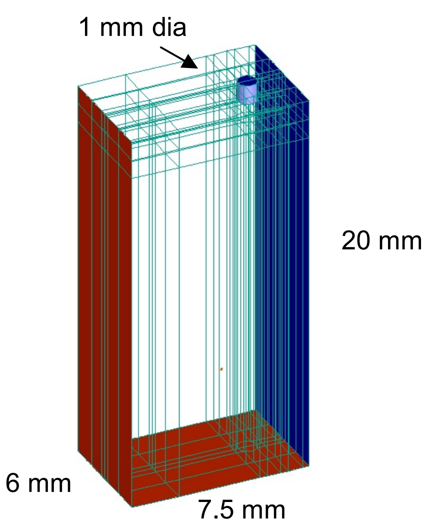
Comparison/Validation Case Definition

The main objective for this validation case is to produce CFD simulation results that compare well to a single case presented in the publication, focusing on liquid column characteristics (i.e., length and trajectory). The case selected from the publication is depicted in Figure 1c for a Weber number equal to 8 which shows the “bag breakup” behavior.

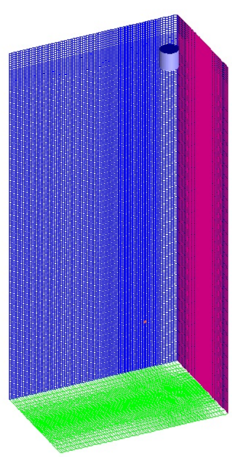
CFD Model Definition

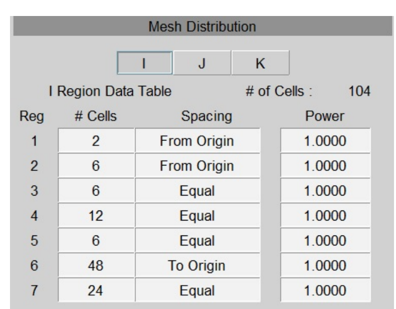
Water jet in fluid
Water jet diameter: 1.0 mm
Inlet velocity (water) 26.24 m/s (Reynolds Number = 3e+04)
Cross flow velocity (air) 22.12 m/s (Weber Number = 8)

CFD Model Geometry / Mesh



Mesh: 104 x 220 x 48 ~ 1.1 M cells




Mesh Distribution

I J K

I Region Data Table # of Cells : 104

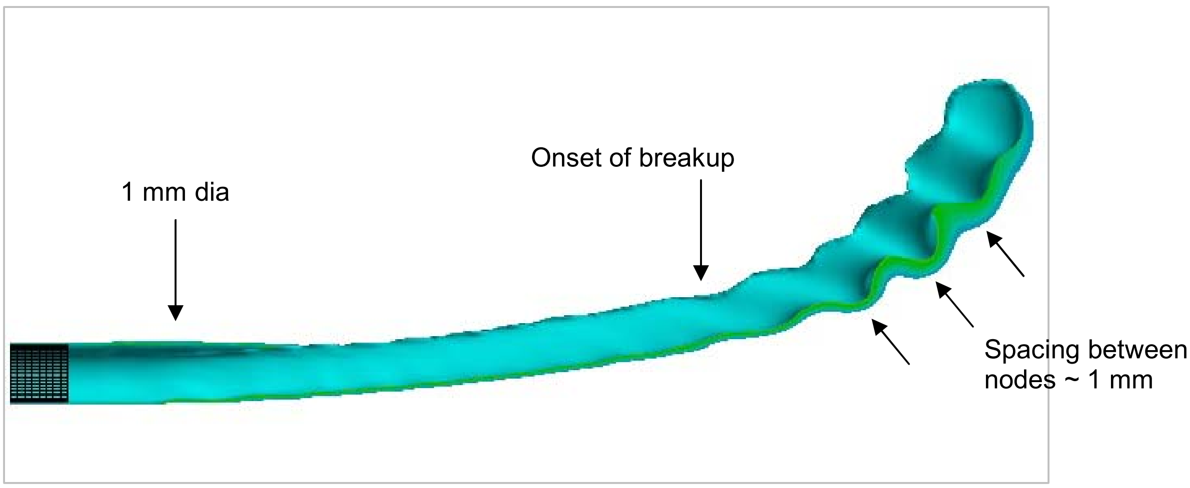
Reg	# Cells	Spacing	Power
1	2	From Origin	1.0000
2	6	From Origin	1.0000
3	6	Equal	1.0000
4	12	Equal	1.0000
5	6	Equal	1.0000
6	48	To Origin	1.0000
7	24	Equal	1.0000

Experiment Results

The primary experimental result presented in the publication was directly from pulsed shadowgraph and holograph “observations”. Presented measured data related to jet flow (etc) is very limited.

One stated observed measurement related to the bag breakup regime at a Weber number of 8, is related “to the spacing between nodes” (refer to page 2532, paragraph 2). The publication states that “In this regime, cross flow Weber numbers have reached values where the spacing between the nodes is comparable to the liquid jet diameter”.

This can be directly seen from the CFD simulation (refer to the figure below). This is also shown in Figure 3 as a ratio of liquid surface wavelengths as a function of Weber number, page 2533.



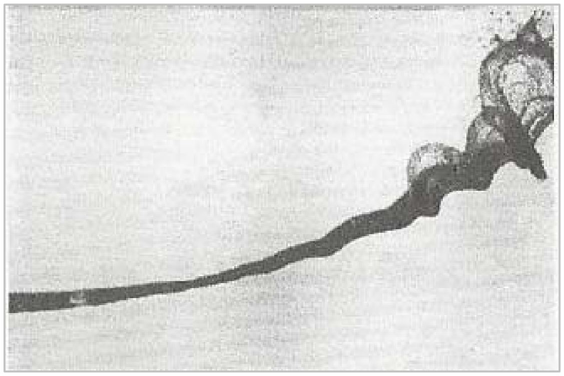
Another stated measurement is related to liquid surface velocities (refer to page 2534, paragraph 3 and Figure 4). Note that reliable measurements were only obtainable for values of Vs/Vj in the range of Y/Yb = 0.1 – 0.7 (due to the surface becoming difficult to define near the liquid jet breakup condition). At a value of Y/Yb=0.2, the Vs/Vj value is ~unity (which the CFD simulation reproduces).

A final comparison parameter involves the deformation at onset of primary breakup as a function of Weber number (refer to Figure 5, page 2534). For a Weber number of 8, the ratio of initial jet diameter to jet diameter at the onset point, Dj/Di, is approximately equal to 1.1 (see figure above for reference). The CFD simulation reproduces this flow characteristic.

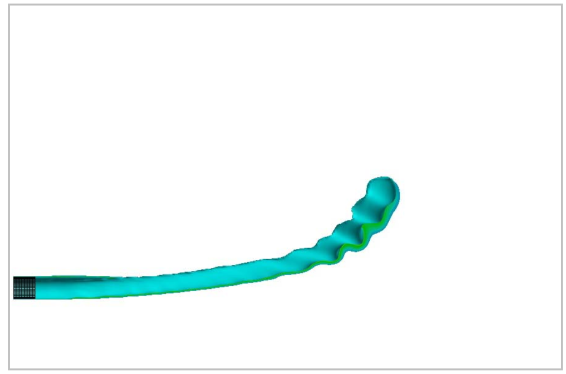
Note that a large part of the publication deals with droplet formation and related characteristics. The CFD simulation did not model/simulate this phenomena, thus comparisons are not presented.

Experiment and Simulation Result Visual Comparison

Experiment



CFD Simulation

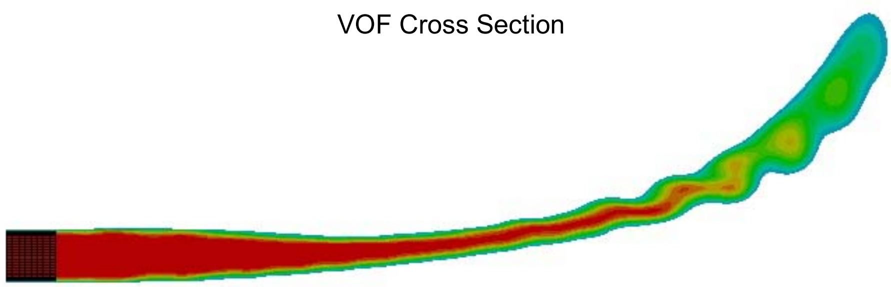


~ 0.002 seconds

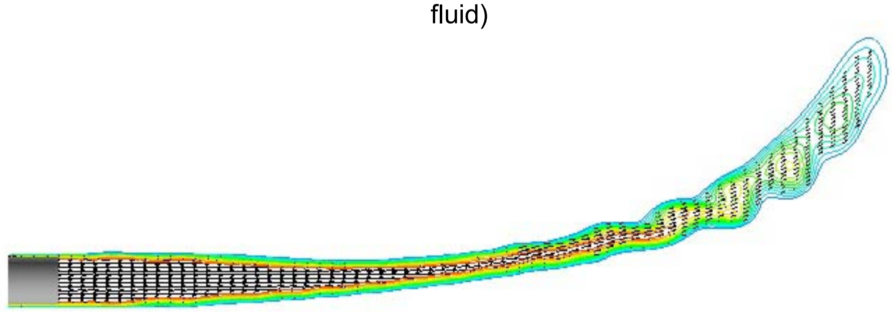
The CFD simulation result shows that the water jet follows a similar trajectory, and that even with a fairly coarse mesh used in the CFD model, the “bag breakup” behavior is captured.

Additional CFD Solution Result Presentation

VOF Cross Section



Velocity Vectors (within fluid)



b. Case # 2 - "Detailed simulations of primary breakup of turbulent liquid jets in cross flow"

Research Publication Abstract

"In this study, we employ a recently proposed spectrally refined interface (SRI) tracking method (Desjardins & Pitsch 2008) to study liquid jets in cross flow through detailed numerical simulations. … From the non-dimensional parametric space, the effect of two dimensionless groups, namely the Weber numbers corresponding to the liquid jet and cross flow, which dictate the likelihood of breakup of the liquid jet are investigated."

Comparison/Validation Case Definition

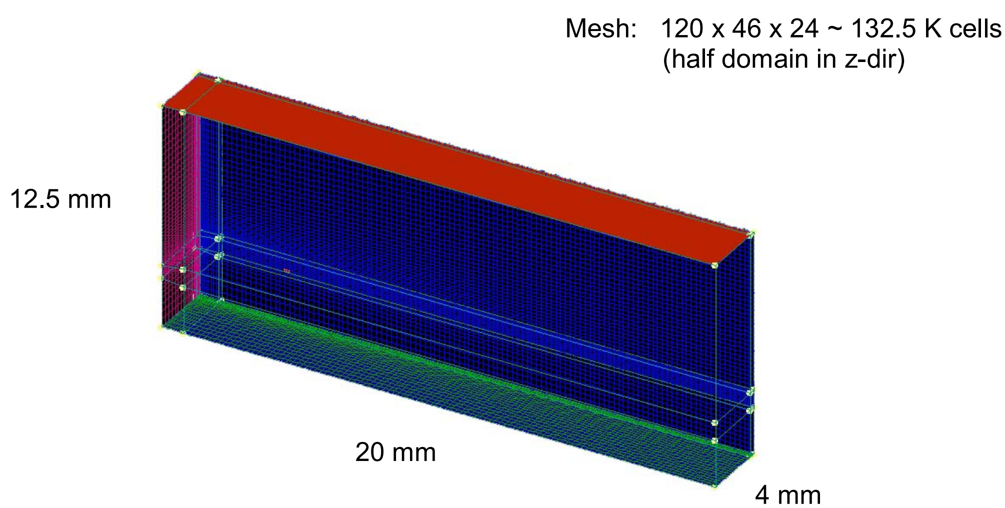
The main objective for this validation case is to produce CFD simulation results that qualitatively compare to the cases presented in the publication, focusing on liquid column characteristics (i.e., length and trajectory). The case selected from the publication for comparison is depicted in Figure 2c of the publication.

CFD Model Definition

Water jet in air

Water jet diameter:	1.0 mm
Inlet velocity (water)	33.6 m/s
Cross flow velocity (air)	24.5 m/s

CFD Model Geometry / Mesh

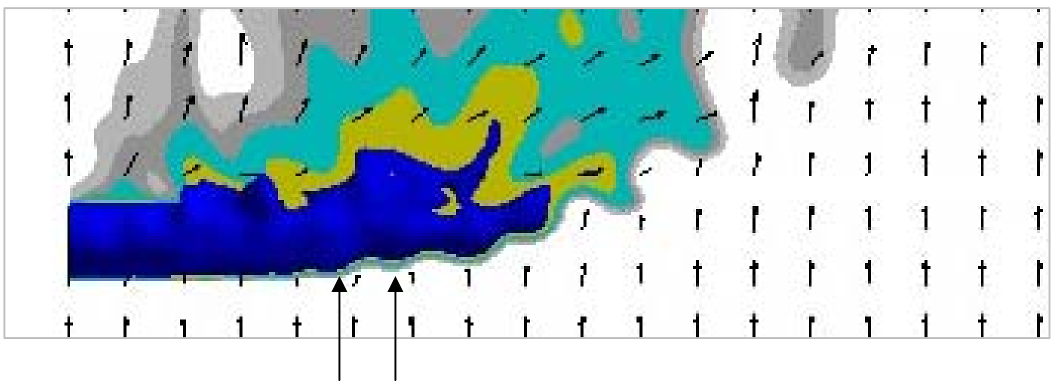


Publication Simulation Results

The results presented in the publication are simulation based only (no experimentation results presented).

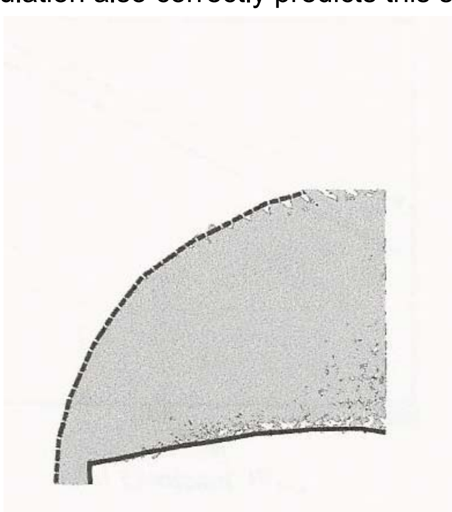
Similar to the previous V&V case, one flow characteristic reported in the publication was related to “liquid surface disturbances” (refer to page 462, section 6.2.3). The publication presents the characteristic length scale of the KH-like instability measured along the liquid jet trajectory (i.e., the spacing between nodes). Figure 6 on page 463 shows the predicted wavelength of the liquid surface disturbances. The value is ~ 0.5 mm for Weber numbers ranging from 100-300.

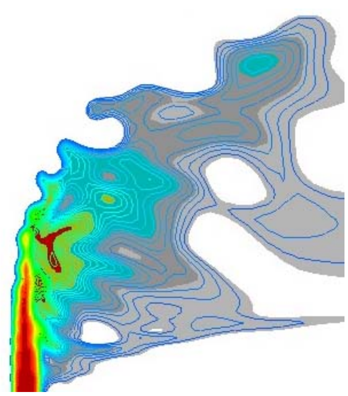
The following CFD solution image depicts a comparable value for the wavelength.



Spacing between nodes ~ 0.5 mm

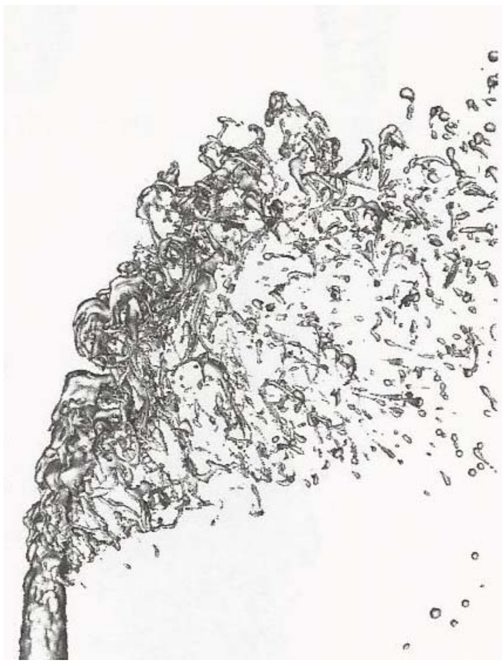
The publication presents an inference of the jet trajectory (dashed line) and the lowermost extent (solid line) based on several superimposed spray images (see figure to the left below). The CFD simulation also correctly predicts this spray pattern (figure below on the right).



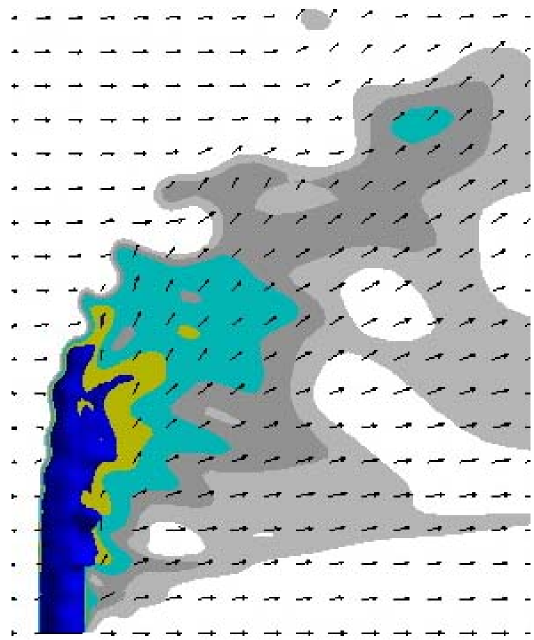


Simulation Result and Comparison

Experiment



CFD Simulation



Blue iso-surface @ VOF=0.6

The CFD simulation result shows a similar flow path trajectory and spread of water. The CAESIM simulation used a very coarse mesh compared to the numerical simulations produced/documented in the publication.

3.0 Verification & Validation – Other Cases

a. Case # 3 - "Material Removal in Magneto Rheological Finishing of Optics."

Research Publication Abstract

"A concept of material removal based on the principle of conservation of particles momentum in a binary suspension is applied to analyze material removal in Magnetorheological Finishing (MRF) and Magnetorheological Jet processes widely used in precision optics fabrication. According to this concept, a load for surface indentation by abrasive particles, which fluctuate (due to collision) in the shear flow of concentrated suspension. The model is in good qualitative agreement with experimental results."

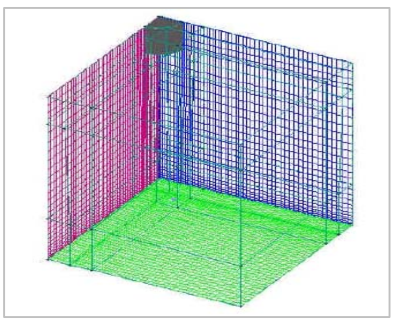
Comparison/Validation Case Definition

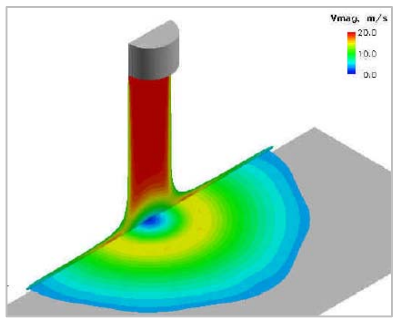
The main objective for this validation case is to produce CFD simulation results that compare well to a single case presented in the publication. The unique aspect of this case is that the liquid jet impacts a solid surface. The validation is focused on determining a “removal rate”, based on the resulting shear stresses produced by the impacting liquid jet flow.

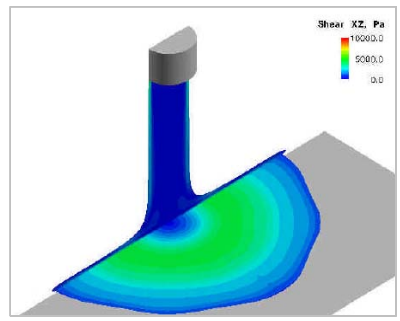
CFD Model Definition

Water jet in air
Water jet diameter: 1.0 mm
Inlet velocity (water) 20 m/s

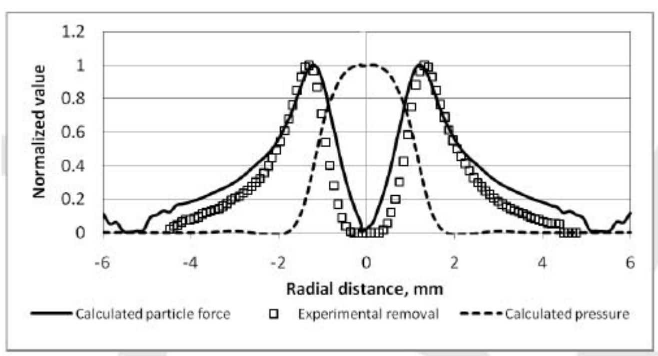
CFD Model Geometry / Mesh







Simulation Result compared to Experiment



b. Case # 4 - "An Experimental and Numerical Study of a Water jet Cleaning Process".

Research Publication Abstract

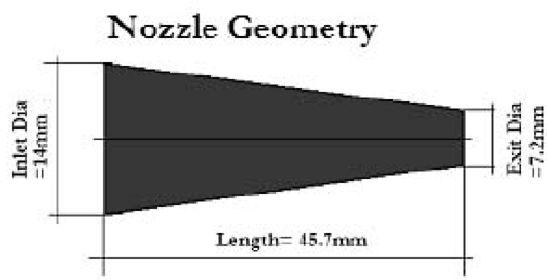
"In this paper, we have experimentally, numerically and theoretically investigated the water jet cleaning process. Very high speed water jets (~80-200 m/s) are typically used in such cleaning operations. These jets diffuse in the surrounding atmosphere by the process of air entrainment and this contributes to the spreading of the jet and subsequent decay of pressure". "Numerical simulations are performed to capture the process of air entrainment in the jet and the subsequent pressure characteristics. The simulation results are found to correctly predict the experimental data."

Comparison/Validation Case Definition

The main objective for this validation case is to produce CFD simulation results that compare well to the case presented in the publication, focusing on nozzle flow characteristics (i.e., pressure and velocity). The case selected from the publication is depicted in Figure 11 for "Test Case 2".

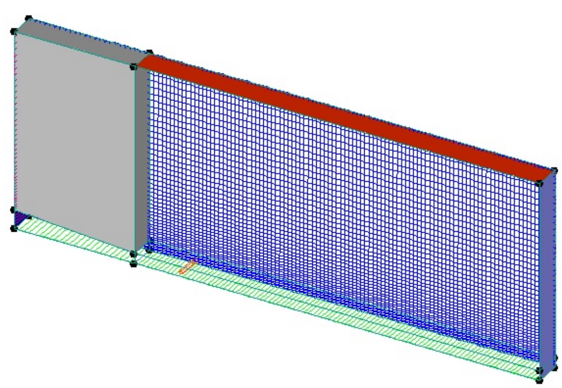
CFD Model Definition

Water jet in air
Nozzle configuration:



Inlet velocity (water)	46 m/s
Inlet kinetic energy	31.74 m2/s2
Inlet dissipation	4.197e4 m2/s3

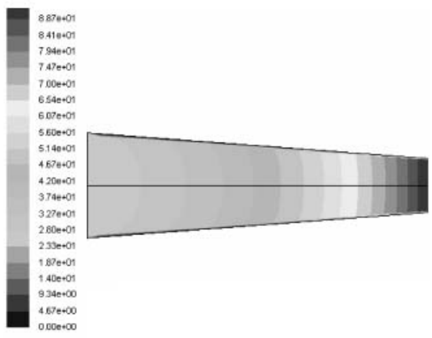
CFD Model Geometry / Mesh

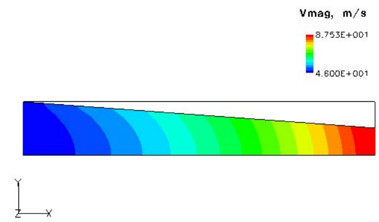


Simulation Result and Comparison

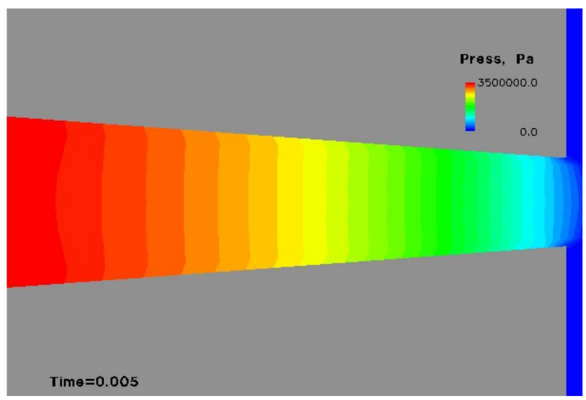
Paper Result (numerical) CAESIM Simulation

Nozzle Velocity





Nozzle Pressure (CFD2000)



The pressure obtained is comparable to what is reported by Guha &al (>3 Mpa).