

Article

Analysis of Water Flow through the Active Parts of an Abrasive Water Jet Machine: A Combined Analytical and CFD Approach

Iulian Pătrînac ¹, Razvan George Ripeanu ¹, Catalin Popescu ^{2,*} and Maria Tănase ^{1,*}

¹ Department of Mechanical Engineering, Petroleum-Gas University of Ploiesti, B-dul Bucuresti, No. 39, 100680 Ploiesti, Romania; ipatirnac@gmail.com (I.P.); rrapeanu@upg-ploiesti.ro (R.G.R.)

² Business Administration Department, Petroleum-Gas University of Ploiesti, 100680 Ploiesti, Romania

* Correspondence: cpopescu@upg-ploiesti.ro (C.P.); maria.tanase@upg-ploiesti.ro (M.T.)

Abstract: This study has the main objective of the analysis of water flow through the active parts (cutting head CH) of an abrasive water jet (AWJ) machine, model YCWJ-380-1520, performed on a high-pressure nozzle (HPN) and mixing tube (MT). The flow is analyzed through the ruby orifice with a diameter of 0.25 mm by assimilating it with a circular pipe. Taking into account the fact that the average flow velocity through the ruby orifice is about 622 m/s, the value of 155,500 according to the Reynolds criterion was obtained. Regarding the turbulent flow regime, the flow section is divided into four zones; for each of them, the limits of flow layers and the maximum values of water velocities were determined. In the second part of this work, a 2D analysis of the flow through the CH assembly was carried out. Since the abrasive inlet tube (AT) also appears in the CH componence, two situations were analyzed in this study, namely, the case when the inlet through AT is restricted and the case when the AT is free. For each case, three values of flow diameters were considered, both for HPN and MT. The water flow characteristics were established and comparisons between theoretical models and CFD simulation were performed.

Keywords: abrasive water jet machine; water flow; analytical method; numerical simulation



Citation: Pătrînac, I.; Ripeanu, R.G.; Popescu, C.; Tănase, M. Analysis of Water Flow through the Active Parts of an Abrasive Water Jet Machine: A Combined Analytical and CFD Approach. *Processes* **2024**, *12*, 1551. <https://doi.org/10.3390/pr12081551>

Academic Editor: Ireneusz Zbicinski

Received: 3 July 2024

Revised: 19 July 2024

Accepted: 23 July 2024

Published: 25 July 2024



Copyright: © 2024 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

Abrasive water jet (AWJ) machining is a non-conventional cutting process that uses a high-velocity stream of water mixed with abrasive particles to cut a wide range of materials. Abrasive water jet (AWJ) machining emerged as a highly versatile and efficient method for processing a variety of materials due to its ability to cut without inducing thermal damage, its high applicability, and its flexibility [1–4]. This non-traditional machining technique uses a high-velocity jet of water mixed with abrasive particles to erode material, making it suitable for complex and delicate operations across multiple industries [5]. In addition to AWJ, other advanced manufacturing methods, such as carbon dioxide (CO₂) laser cutting, offer significant benefits. CO₂ laser cutting, a non-contact, thermal-based method, provides high precision and excellent operational control, making it particularly suitable for processing thermoplastics such as polyethylene [6]. Previous studies identified numerous advantages associated with advanced manufacturing technologies, including increased market share and profitability, enhanced flexibility, improved delivery times, reduced costs, higher quality products, greater productivity and efficiency, and significant competitive advantages [7]. In comparison, AWJ machining stands out for its ability to cut without thermal damage, providing a complementary solution to other advanced manufacturing techniques by offering unique benefits in specific applications.

The increasing application of AWJ machining necessitates a comprehensive understanding of the factors affecting its performance, including the characteristics of the water jet, the properties of the abrasive materials, and the interaction between these elements. The efficiency and performance of AWJ machines depend significantly on the dynamics

of water flow through critical components, such as the cutting head (CH), high-pressure nozzle (HPN), and mixing tube (MT) [4]. Understanding the water flow characteristics through these components is vital for optimizing the machining process and improving the quality of cuts. The study of turbulent water flow through these orifices provides insights into the complex interactions between fluid mechanics and material removal. Previous research focused on both experimental and numerical methods to analyze these flows, often employing computational fluid dynamics (CFD) simulations [8–15] to complement analytical approaches [16–18].

The modeling and simulation of abrasive water jets are critical for optimizing performance and extending the lifespan of machine components. Computational fluid dynamics (CFD) simulations are widely utilized to analyze the characteristics and behavior of water jets under various conditions. Liu et al. [10] used CFD simulations to model ultrahigh velocity water jets, examining the jet dynamics downstream from a fine nozzle and providing insights into the kerf formation process in AWJ cutting. Similarly, Pătrînac et al. [9] conducted CFD simulations to study the flow through the cutting head of a water jet machine, focusing on the erosion wear of components due to high-velocity impacts.

The impact of system and geometric parameters on nozzle is also a significant area of research. Nanduri et al. [19] investigated how variables such as nozzle length, inlet angle, diameter, and water pressure influence the wear rate of nozzles. Their experimental procedures helped develop an empirical model correlating these parameters with nozzle wear, providing a basis for optimizing nozzle design. Further experimental investigations by Yu et al. [20] explored the effects of abrasive properties, including material, size, and flow rate, on the cutting performance of AWJs. Their study highlighted how variations in these parameters impact cutting depth, kerf width, and surface roughness, which are critical indicators of cutting quality. Additionally, Shao et al. [8] employed CFD simulations and experimental methods to analyze the particle flow inside nozzles and the resulting erosion damage mechanisms, identifying factors such as pump pressure and particle size as significant influences on erosion rate. Innovations in nozzle design were driven by the need to improve the efficiency and lifespan of AWJ systems. Li et al. [21] utilized hydrodynamic technology and multi-parameter optimization to enhance the geometry of suspension abrasive water jet nozzles, achieving better performance and reduced wear. Wang et al. [22] provided a comprehensive review of the principles, characteristics, and future directions of AWJ machining technology, emphasizing the importance of process optimization and the development of new technologies to advance the field. In conclusion, the literature highlights the extensive research efforts focused on understanding and improving the performance of abrasive water jets. From CFD simulations to experimental investigations, these studies provide valuable insights into the factors influencing jet behavior, nozzle wear, and overall cutting performance. Future research should continue to explore advanced modeling techniques and innovative designs to further enhance the capabilities and applications of AWJ machining.

This paper aims to build on these foundational studies by conducting a comprehensive analysis of water flow through the active parts (Figure 1) of an AWJ machine model YCWJ-380-1520. By integrating analytical methods with CFD simulations, this research seeks to deepen the understanding of flow dynamics and erosion characteristics, ultimately contributing to the optimization and longevity of AWJ nozzles.

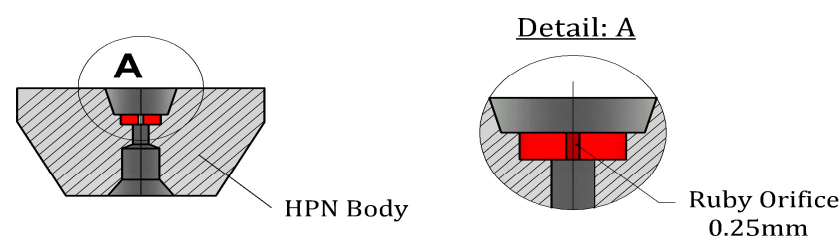


Figure 1. Schematic of HPN.

For the calculation of the hydraulic parameters of the ruby nozzle, the assumption is made of its assimilation with a pipe through which liquid circulates. As a first hydraulic parameter, it is necessary to know the average water velocity in the cross section of the ruby nozzle of diameter d_a . The expression of this velocity is given by relation (1):

$$v_m = \frac{4 \cdot Q_a}{\pi \cdot d_a^2}, \quad (1)$$

where v_m —average velocity of water [m/s]; Q_a —represents the water flow, [m³/s]; and d_a —the diameter of the HPN ruby hole, [m]. In this case, the measured water flow generated by the water jet machine is 3.05×10^{-5} m³/s.

In any section through which fluid flows, the flow regime is characterized by the Reynolds number, and in the analyzed case it is given by the expression (2):

$$Re = \frac{\rho_a \cdot v_m \cdot d_a}{\mu_a}, \quad (2)$$

where ρ_a —is water density [kg/m³]; μ_a —is water dynamic viscosity, [Pa·s]. In this case, based on relation (2), the Reynolds number resulted as 155,500.

2. Methods

2.1. Study of Water Flow through Ruby Nozzle Orifice Using Analytical Method

Knowing the value of the Reynolds number as well as the value of the average velocity through the water nozzle, a study can be made regarding the turbulent motion through it. If the ruby nozzle orifice is assimilated to a circular pipe, it is observed that the value of the Reynolds number resulting from the turbulent motion calculation is fully developed. It is assumed that the water enters the nozzle with the constant speed v_m .

In the study of the turbulent movement through a pipe of circular section, the friction that occurs between the fluid and the pipe wall is taken into account, introducing the notion of friction velocity given by relation (3) [16,23].

$$v_f = \sqrt{\frac{\tau_p}{\rho_a}} \quad (3)$$

where τ_p —is the tangential unit stress [Pa]; ρ_a —is water density [kg/m³].

At high flow velocities, the average value of the tangential unit stress can be determined [24]:

$$\tau_p = 0.5 \cdot \rho_m \cdot f_f \cdot v_m^2. \quad (4)$$

According to [24], the friction coefficient f_f is calculated as:

$$f_f = 0.001375 \left[1 + \left(2 \cdot 10^4 \cdot \frac{R_a}{d_a} + \frac{\mu_a}{\rho_a \cdot d_a \cdot v_m} \cdot 10^6 \right)^{1/3} \right] \quad (5)$$

where ρ_m —represents the density of the fluid mixture [kg/m³]; v_m —the average velocity of the water through the nozzle [m/s]; and μ_a —dynamic viscosity of water [kg/m·s], R_a —represents the average ruby roughness [μm].

Considering the roughness of the ruby between 0.012 and 0.025 μm [25], and only water flows through the hole, substituting in relation (5), the value of the friction coefficient is obtained:

$$f_f = 0.001375 \left[1 + \left(2 \cdot 10^4 \frac{0.012 \cdot 10^{-6}}{0.25 \cdot 10^{-3}} + \frac{1.008 \cdot 10^6}{998.2 \cdot 0.25 \cdot 10^{-3} \cdot 622.35} \right)^{1/3} \right] = 3.932 \cdot 10^{-3}$$

substituting the value of the friction coefficient in relation (2), the tangential unitary effort is obtained in the analyzed case.

$$\tau_p = 0.5 \cdot 998.2 \cdot 3.932 \cdot 10^{-3} \cdot 622.35^2 = 7.6 \cdot 10^5 \text{ Pa}$$

The value of the friction velocity in the case of turbulent flow through the ruby orifice of the water nozzle will be:

$$v_f = \sqrt{\frac{7.6 \cdot 10^5}{998.2}} = 27.59 \frac{\text{m}}{\text{s}}.$$

Knowing the value of the friction speed, a reference length given by relation (6) [23] can be defined and used to express the speed distribution laws, as well as the size of the flow zone intervals.

$$y_f = \frac{\mu_a}{\sqrt{\rho_a \cdot \tau_p}} = 3.653 \cdot 10^{-8} \text{ m} \quad (6)$$

As in the case of pipes, in the ruby nozzle where the fluid movement is turbulent, several specific zones are formed, as shown in Figure 2. These areas refer to the variation in the average velocity with respect to the distance from the nozzle wall. It is specified that the notation v represents the local average speed.

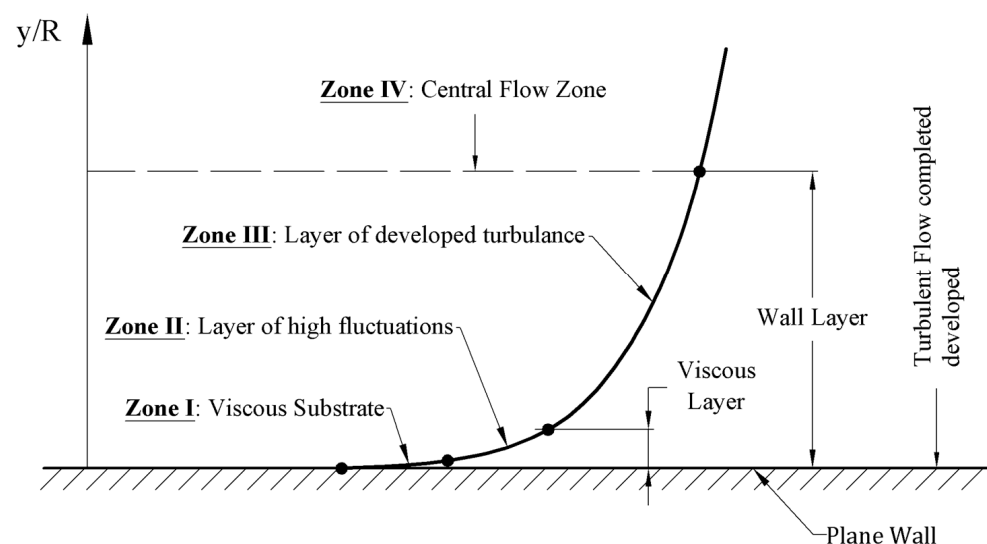


Figure 2. Turbulent motion past a plane wall.

The first area appears in the immediate vicinity of the wall; this area is called viscous substrate. According to [23], the viscous substrate has a thickness (y) given by relation (7). This layer has a very small thickness but plays a very important role in the flow characteristics involving a high velocity gradient. In this substrate the turbulent unit effort is negligible.

$$0 \leq \frac{y}{y_f} \leq 5 \quad (7)$$

In the viscous substrate, the average velocity distribution law has the expression given by relation (6) [23], where a linear distribution can be observed.

$$v = \frac{\tau_p}{\mu_a} (r_a - r) = \frac{\tau_p}{\mu_a} y \quad (8)$$

where r_a —inner radius of the nozzle [m]; r —the radial distance measured from the axis of the nozzle to the wall [m]; and y —the thickness of the viscous layer [m].

The second zone is where the viscous and turbulent tangential unit stresses are of comparable magnitude. This zone is called the intermediate layer (buffer), or zone of intense fluctuations. The limits of this zone are given by the relation (9) [23]:

$$5 \leq \frac{y}{y_f} \leq 30. \quad (9)$$

It is observed that in the intermediate layer, the average velocity distribution law is obtained by integrating the expression [23] (10).

$$\frac{dv}{dy} = \frac{\tau_p}{\mu_a} \cdot \frac{1}{1 + \frac{\varepsilon_m}{\nu_a}} \quad (10)$$

In the Formula (10), ε_m is the apparent or turbulent viscosity, and ν_a represents the kinematic viscosity of water [m²/s]. If the notations are introduced, according to [17,23], it results as:

$$v^+ = \frac{v}{v_f} \text{ and } y^+ = \frac{v_f \cdot y}{\nu_a},$$

and for the integration, one of the laws proposed for the variation in the turbulent viscosity in the intermediate layer must be considered. As for laws, reference is made to the one proposed by Deissler (11) [23], as well as to the one proposed by van Driest (12) [18,23]:

$$\frac{\varepsilon_m}{\nu_a} = n^2 \cdot y^+ \cdot v^+ \left(1 - e^{-n^2 y^+ v^+} \right). \quad (11)$$

The coefficient n has the value 0.124, valid for $y^+ < 26$, according to [23].

$$\frac{\varepsilon_m}{\nu_a} = K^2 \cdot (y^+)^2 \cdot v^+ \left(1 - e^{\frac{y^+}{y_s^+}} \right)^2 \frac{dv^+}{dy^+} \quad (12)$$

In Formula (12), K is called Karman's constant and has a value around 0.4 [23], and $y_s^+ = 26$ [23].

The first and second zones occupy only a small part of the nozzle radius, but have an important influence on the movement as a whole. The thickness of these two layers depends on the value of the Reynolds number. The first two areas form the viscous layer; this layer is formed only when the roughness of the wall is small in relation to its thickness, i.e., it is necessary that the micro-asperities of the wall do not penetrate this layer. From Table 1, the validation of this fact can be seen, and that the determination of the average velocity in the intermediate layer using the methodology described by relations (10), (11), and (12) is very complicated.

Table 1. Characteristics of turbulent flow through the water nozzle.

Layer Type	Layer Limit Range [mm]		Maximum Speed [m/s]
	Lower	Upper	
Viscous substrate	0	1.812×10^{-4}	136.61
The intermediate layer	1.812×10^{-4}	1.087×10^{-3}	-
The fully turbulent layer	1.087×10^{-3}	1.812×10^{-2}	562.33
The turbulent core	1.812×10^{-2}	0.125	636.43

The third zone, called the fully turbulent layer, presents sufficiently developed turbulence for the effect of turbulent unit stress to be predominant. In this area, the nozzle wall still influences the flow. The boundaries of the fully turbulent layer are given by relation (13) [23].

$$30 \leq \frac{y}{y_f} \leq 500 \quad (13)$$

The average velocity distribution law in the turbulent layer is given by relation (14) [26], where a logarithmic law is observed.

$$\frac{v}{v_f} = A \cdot \ln\left(\frac{y}{y_f}\right) + B \quad (14)$$

In (14), A and B are experimental constants and their values show some scatter. From [23,27], it can be seen that the values of these constants are $A = 2.5 (\pm 15\%)$ and $B = 5 (\pm 25\%)$.

The fourth zone, which is called the turbulent core of the motion, presents a deviation from the logarithmic law. From [23] it is found that the point where this deviation occurs is approximately 15% of the nozzle radius. The average velocity distribution law in this area is given by relation (15), also known as the power law of the velocity profile [26].

$$\frac{v}{v_{max}} = \left(\frac{y}{r_a}\right)^{1/n} \quad (15)$$

In relation (15), v_{max} represents the maximum velocity, this being present in the axis of the nozzle, and n is a constant whose value depends on the Reynolds number. The value of n increases with increasing R_e number.

The power law variation in the velocity profiles is shown in Figure 3, for different values of the constant n , together with the velocity profile for a fully developed laminar flow.

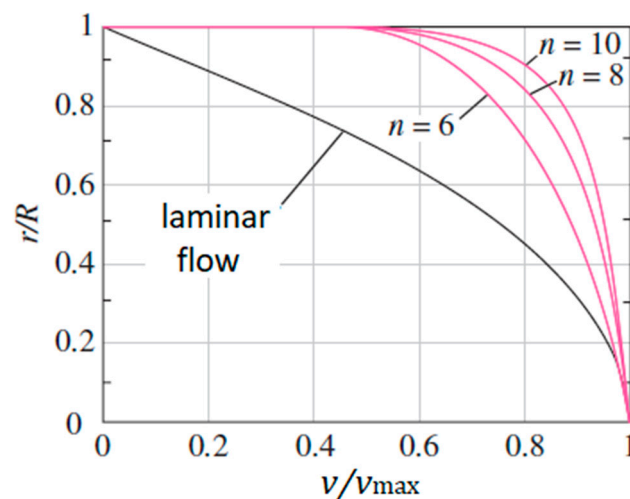


Figure 3. Polinomial curves of velocity distribution.

The thicknesses of the flow layers in the turbulent regime, as well as the values of the velocities in these intervals obtained analytically, in the case of the water nozzle with a diameter of 0.25 mm used in the water jet cutting machine type WUXI YCWJ-380-1520, are presented in Table 1.

It is observed that the lower boundary of the turbulent core of the motion is at a distance of 0.01812 mm from the ruby hole wall. If we consider the fact that the fourth flow zone starts at about 15% of the nozzle radius (measured from the wall), then this layer would start at a distance of 0.01875 mm from the nozzle ruby orifice wall. The difference between these two values is 6.3×10^{-4} mm, this difference being comparable to the reference length y_f .

The diagram in Figure 4 shows the variation in the average speed through the water nozzle hole depending on the thickness of each layer, obtained both theoretically and experimentally. On the experimental curve, it can be seen that the maximum speed in the intermediate layer was established through tests, so that this curve is as close as possible to the real situation. The experimental curve has an acceptable shape ($R^2 = 0.9277$) for values of the speed in the intermediate layer between 445 m/s and 465 m/s.

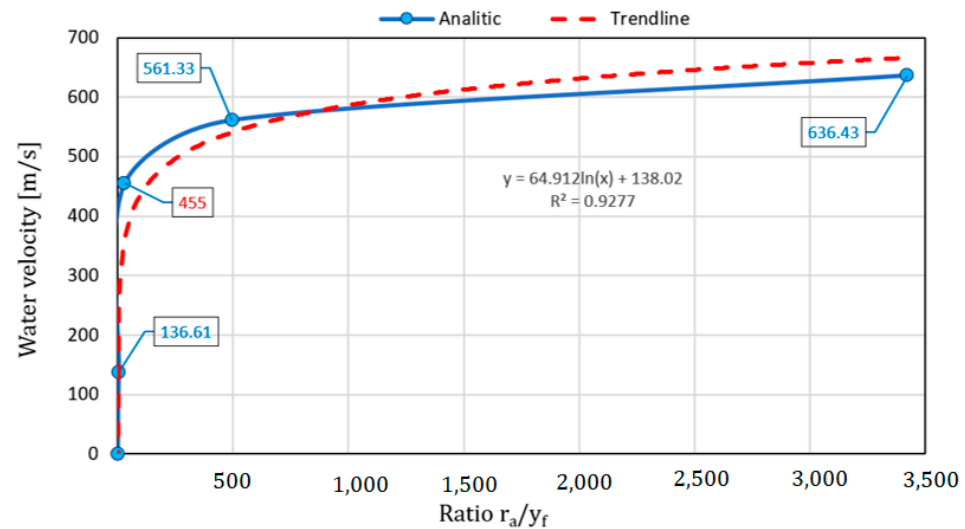


Figure 4. The law of variation in the velocity of water through the ruby nozzle.

The same can be said about the curve obtained theoretically, whose equation is represented on the diagram. If we divide this equation by the friction velocity (v_f), we obtain the velocity distribution law in the ruby hole section, given by relation (16).

$$\frac{v}{v_f} = 2.352 \cdot \ln(x) + 5.002, \quad (16)$$

the constants in the equation can be identified as: $A = 2.352$ and $B = 5.002$. It is observed that the difference between the values of these constants obtained for this case and their theoretical values from [23,27] are included in the deviation field of $\pm 15\%$ and $\pm 25\%$, respectively.

According to the experiments made by Nikuradse, a pipe can be characterized according to the ratio between the equivalent roughness (k_s) and the reference length (y_f), as [23]:

- Smooth hydraulic pipe, for: $(k_s/y_f) < 4$;
- Mixed pipe, for: $4 < (k_s/y_f) < 60$;
- Completely rough pipe, for: $(k_s/y_f) > 60$.

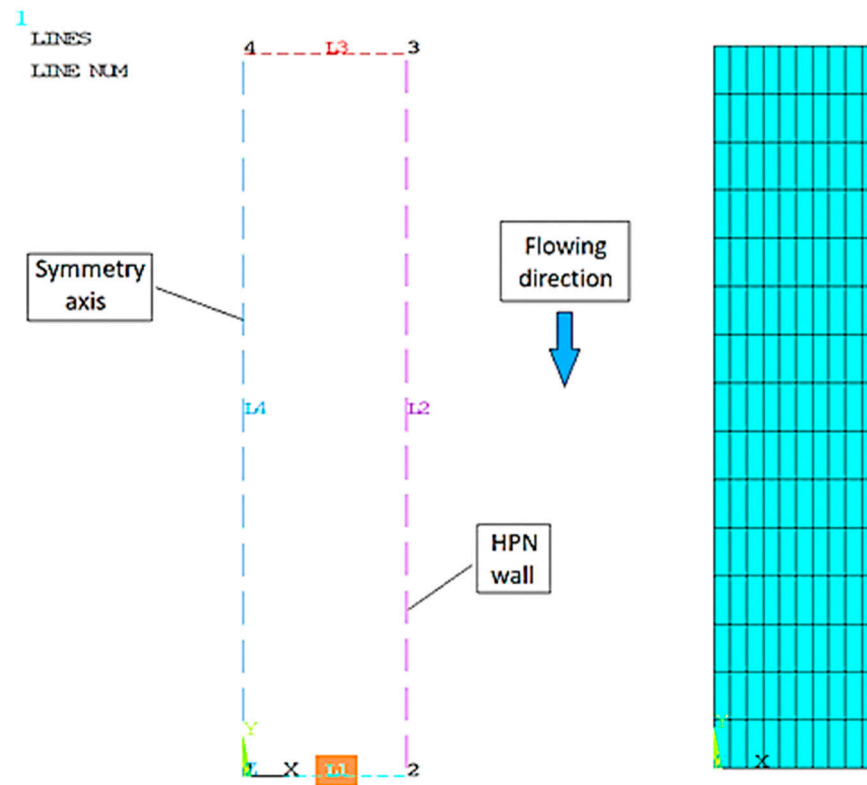
Taking into account this characterization, as well as the fact that the roughness of the ruby hole wall is between $0.012 \mu\text{m}$ and $0.025 \mu\text{m}$, it is observed that the value of this ratio is between 0.331 and 0.689, respectively, which means that the turbulent movement through the nozzle is not influenced by the degree of roughness. From this observation we can also draw the conclusion that Nikuradse's relation for calculating the coefficient of hydraulic resistance λ was adopted correctly.

2.2. Study of Water Flow through the Ruby Nozzle Orifice Using the CFD Method

The study of the flow through the ruby orifice in the framework of the finite element analysis is conducted using Ansys FLOTRAN-CFD APDL 2010, and the analysis element is FLUID141. The geometric model of the ruby nozzle used in this program is presented in Figure 5, and the geometric characteristics of the model are presented in Table 2. Since the water nozzle is a revolution part, the geometric pattern was adopted symmetrically along the Y-axis. As can be seen, the pattern was divided into quadrilateral-type elements, resulting in a total of 150 elements.

Table 2. CFD geometric characteristics of the ruby nozzle.

Line No.	L1	L2	L3	L4
Line name	Water outlet	The wall of the nozzle	Water inlet	Axis of symmetry
Length [mm]	0.125	0.55	0.125	0.55
No. divisions	10	15	10	15

**Figure 5.** HPN ruby orifice geometry in CFD analysis (1, 2, 3 and 4 are keypoints used to define the lines L1, L2, L3 and L4).

The element type FLUID141 is used in the 2D calculation of the velocity and pressure distribution for a single phase of a Newtonian fluid. In the flow analysis with this type of element, quadrilateral or triangular elements can be used. In the analyzed case, an axial-symmetric type element with 4 nodes per element is used.

The equations used by Ansys Flotran-CFD in the case of quadrilateral elements with four nodes per element are given by the relations (17a), (17b), and (17c), to establish the velocities along the three directions. The pressure is given by solving the Equation (16), and in the case of the turbulent kinetic energy, it is given by the relation (19) [28].

$$v_x = \frac{1}{4}v_{x_I}(1-s)(1-t) + v_{x_J}(1+s)(1-t) + v_{x_K}(1+s)(1+t) + v_{x_L}(1-s)(1+t) \quad (17a)$$

$$v_y = \frac{1}{4}v_{y_I}(1-s)(1-t) + v_{y_J}(1+s)(1-t) + v_{y_K}(1+s)(1+t) + v_{y_L}(1-s)(1+t) \quad (17b)$$

$$v_z = \frac{1}{4}v_{z_I}(1-s)(1-t) + v_{z_J}(1+s)(1-t) + v_{z_K}(1+s)(1+t) + v_{z_L}(1-s)(1+t) \quad (17c)$$

where v_x , v_y , v_z , represents the velocities along the three directions of travel.

$$p = \frac{1}{4}p_I(1-s)(1-t) + p_J(1+s)(1-t) + p_K(1+s)(1+t) + p_L(1-s)(1+t) \quad (18)$$

where p_I, p_J, p_K, p_L , represent the pressures in nodes I, J, K, L .

$$E^K = \frac{1}{4}E_I^K(1-s)(1-t) + E_J^K(1+s)(1-t) + E_K^K(1+s)(1+t) + E_L^K(1-s)(1+t) \quad (19)$$

where E^K —represents the turbulent kinetic energy for the analyzed element.

For FLOTTRAN CFD elements, velocities are obtained based on the principle of conservation of momentum, and pressure is obtained based on the principle of conservation of mass. The system matrix derived from the finite element discretization of the equation governing the motion for each individual degree of freedom is solved separately. The flow problem analyzed with this type of element is nonlinear, and the equations governing the motion are tied together. The sequential solution for all equations, combined with some properties such as pressure or temperature, constitutes a global iteration. The number of global iterations required to achieve a solution can vary considerably, depending on the size of the problem in question.

Table 3 presents data related to the modeling of the flow through the ruby nozzle orifice. All loads as well as bond conditions have constant values and are applied to the lines of the model. The CFD analysis is turbulent according to the R_e number.

Table 3. Ruby nozzle flow modeling data.

Model Type:	2D—Axisymmetric	
Element type:	FLUID 141	
The fluid material:	water	
Density:	$\rho_a = 998.2 \text{ kg/m}^3$	
Dynamic viscosity:	$\mu_a = 1.008 \text{ Pa}\cdot\text{s}$	
Boundary conditions	DOF input speed (L3):	$V_Y = 622.35 \text{ m/s}$
	Wall velocity DOF (L2):	$V_X = V_Y = 0$
	DOF outlet pressure (L1):	0
	Axis of symmetry DOF (L4):	$V_X = 0$
Environment conditions:	Temperature:	293 K
	Pressure:	101,350 Pa

3. Results and Discussion

3.1. CFD Analysis of the Flow through the Ruby Nozzle

The results obtained from the CFD analysis of the flow through the ruby nozzle using the Ansys Flotran-CFD program are presented below. Thus, in the diagram in Figure 6, the outline of the pressure drops on the ruby nozzle is represented. It can be seen from the pressure map that the maximum value indicated is 12.4 MPa and occurs at the entrance of the fluid to the nozzle, i.e., in the area adjacent to the nozzle wall.

As can be seen, the pressure at the inlet of the water nozzle is not constant. The graph in Figure 7 shows a parabolic variation of the inlet pressure at a distance of 0.1125 mm from the symmetry axis with a pressure difference of $24.708 \times 10^5 \text{ Pa}$, and on the interval from 0.1125 mm to the wall, an almost linear distribution can be observed with a pressure difference of $3.11 \times 10^5 \text{ Pa}$. From this graph it can be concluded that a variation in pressure can implicitly lead to a variation in velocity, so where the pressure is low, a high velocity of the fluid is obtained (the velocity in the axis of symmetry), and where the pressure has high values, the velocity tends to zero (the case of the velocity at the wall). The pressure study is useful in calculating the hydraulic losses imposed by the water nozzle, as well as in determining the tangential stress on the nozzle wall.

In the diagram from Figure 8, the pressure drop along the axis of the nozzle is graphically represented. An almost linear pressure distribution along the ruby orifice can be observed, and the axial pressure drop value is $96.491 \times 10^5 \text{ Pa}$ (9.649 MPa). In the diagram from Figure 9, the pressure drop along the ruby wall is graphically represented. As can be seen, in this case too there is an almost linear dependence of the pressure along the nozzle wall, the value of the pressure drop in this area being $124.31 \times 10^5 \text{ Pa}$ (12.431 MPa).

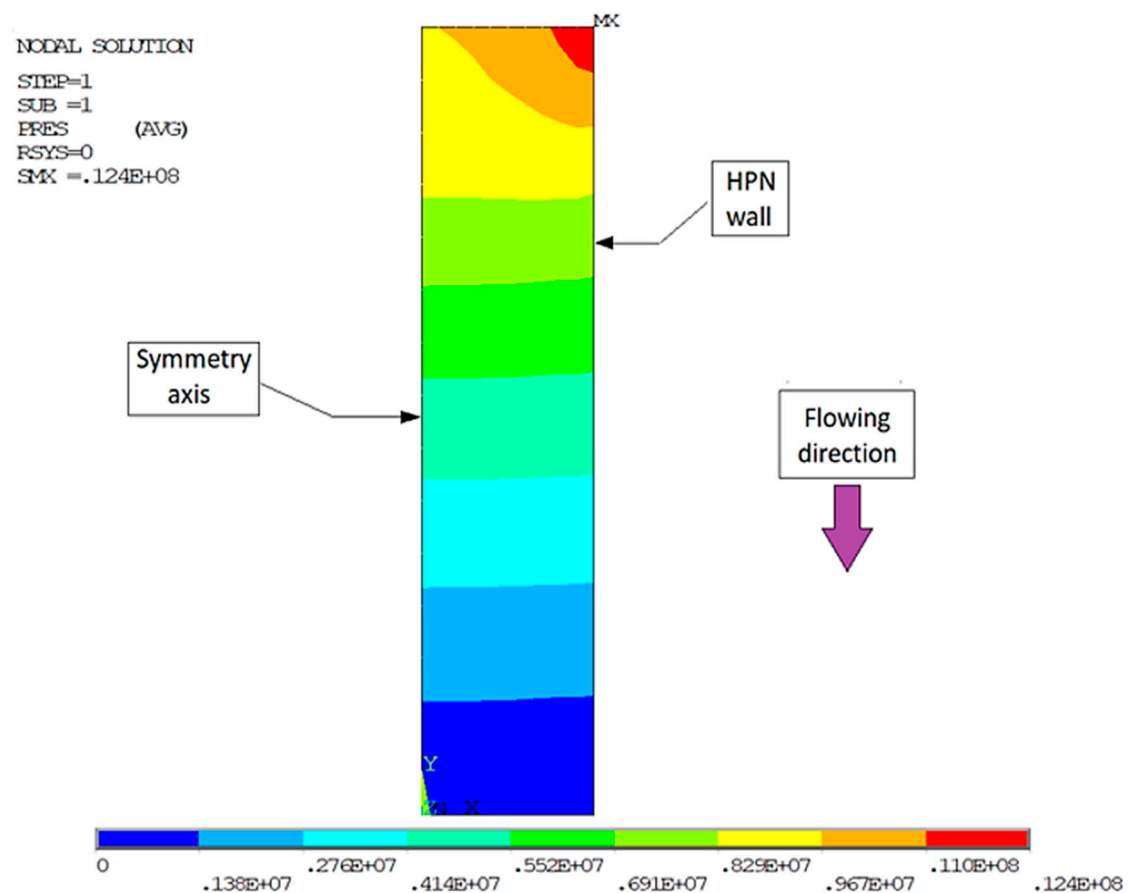


Figure 6. Pressure contour through the 0.25 mm ruby HPN (units in Pa).

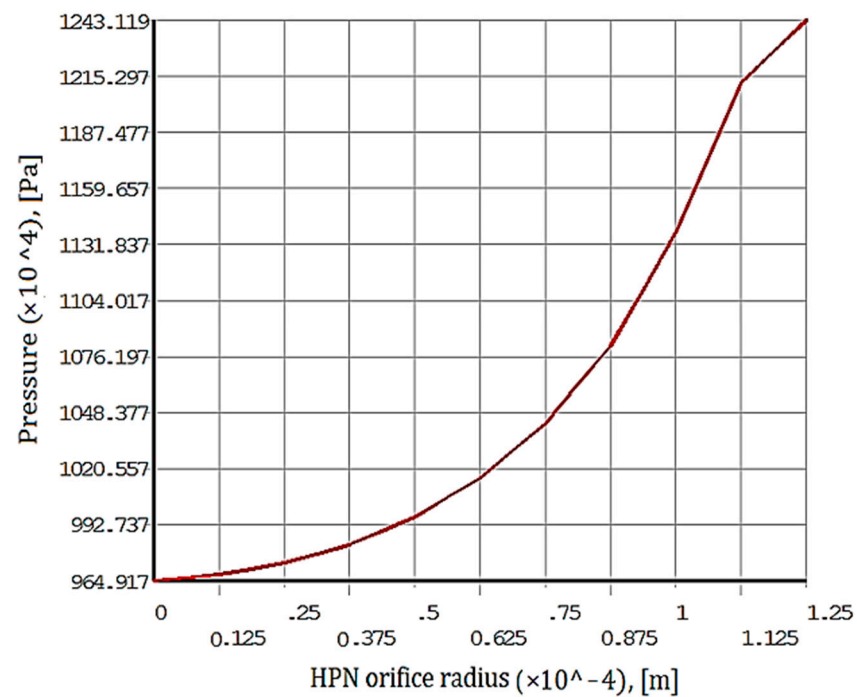


Figure 7. Water pressure distribution at the entrance to the HPN.

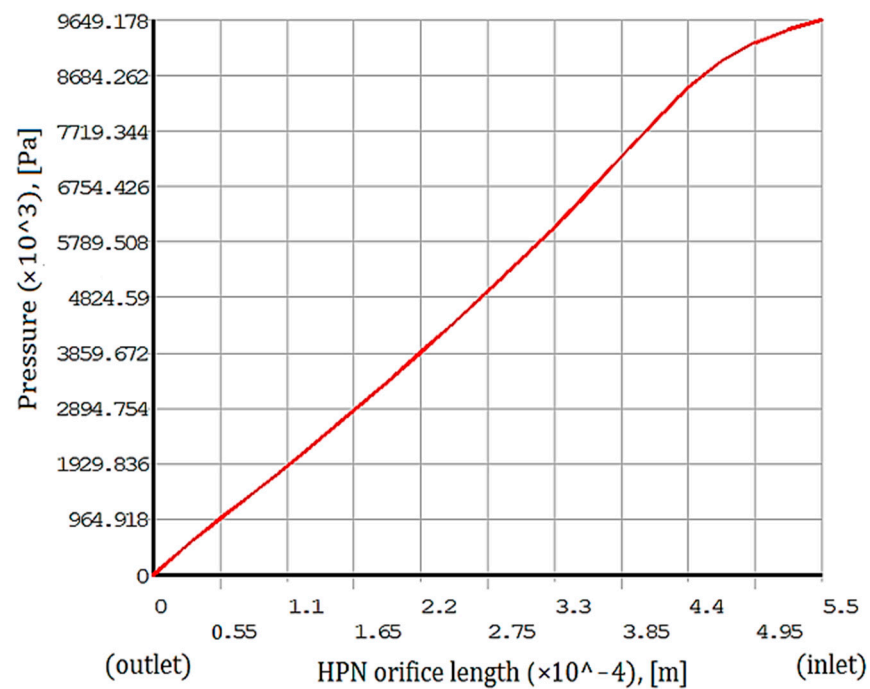


Figure 8. Pressure distribution in the axis of HPN.

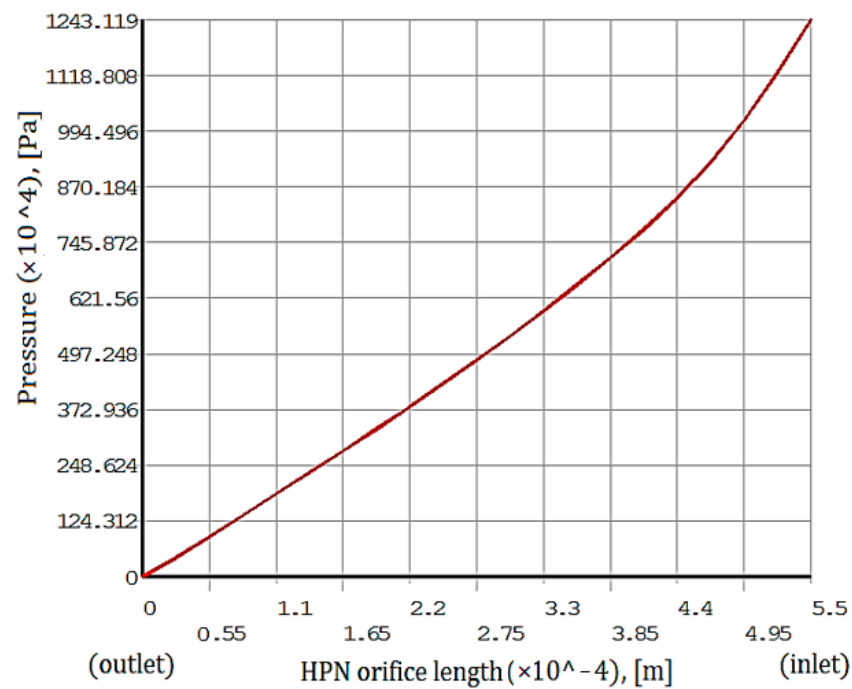


Figure 9. Pressure distribution at the HPN wall.

In the diagram presented in Figure 10, the contour of the velocities according to the flow direction is represented, as well as their graphic representation in the form of vectors, resulting from the Ansys Flotran-CFD analysis. From the velocity map, it can be seen that the maximum speeds have values in the range of 571.1 m/s to 642.5 m/s, a range that corresponds to the turbulent core of the movement. The minimum velocities are recorded towards the nozzle wall, showing an almost uniform flow in the wall layer. This wall layer creates turbulent conditions of motion.

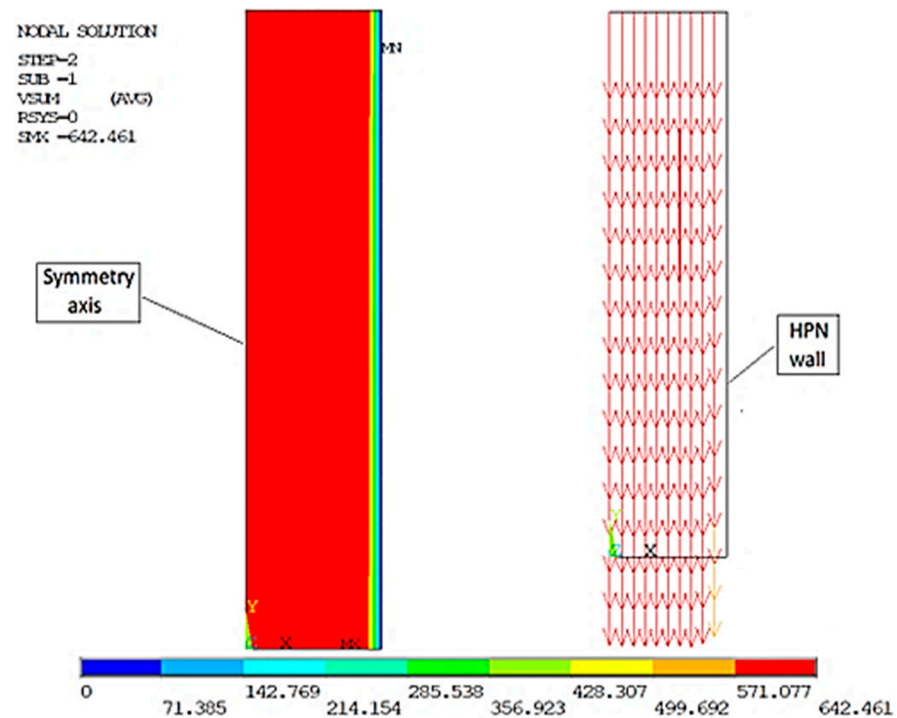


Figure 10. The contour of the velocities through the 0.25 mm ruby of HPN (units in m/s).

Figure 11 shows the variation in the water velocity at the exit from the ruby hole obtained by the finite element analysis.

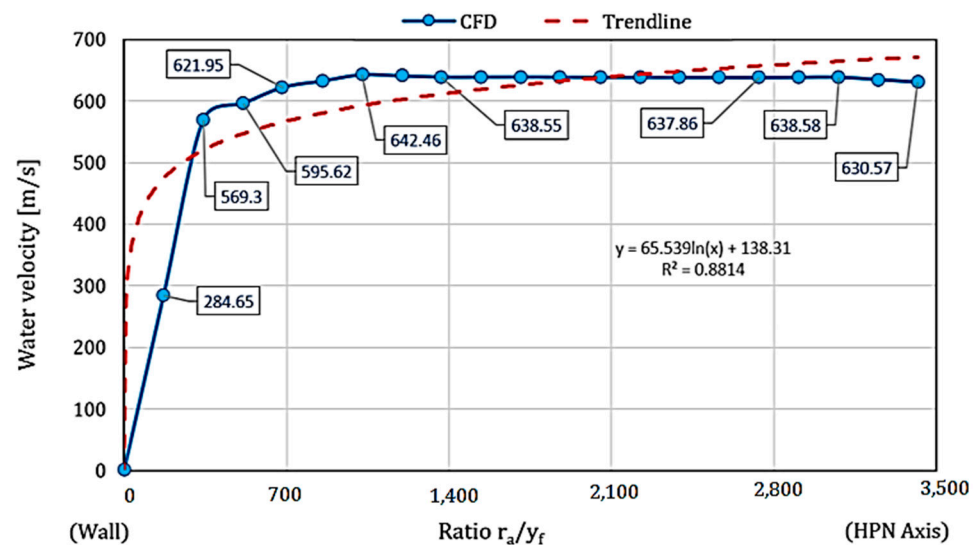


Figure 11. Variation curves of water velocity at the exit from ruby nozzle using Ansys Flotran-CFD.

On the CFD curve (generated in Ansys), a maximum velocity value of 642.46 m/s is observed at a distance of 0.038 mm from the nozzle wall, after which the velocity tends to decrease with a very small slope towards the value of 637.86 m/s obtained at a distance of 0.1 mm from the nozzle wall. In the range of 0.038 mm to 0.125 mm, the CFD graph shows some relatively small fluctuations of the speed, so that in the axis of symmetry of the nozzle, the water speed reaches the value of 630.57 m/s. Moreover, on this diagram, an almost linear dependence of the speed can be observed up to the value of 569.3 m/s at a distance of 0.013 mm from the nozzle wall. By comparing with the analytical diagram from Figure 3, a velocity value close to this (578.91 m/s) is observed at a distance of 0.018 mm from the nozzle wall, i.e., at the end of the fully turbulent layer.

The theoretical curve in this case was established through experimental tests so that it has an allure as close as possible to the experimental one. As can be seen from the diagram, the theoretical speed distribution law is a logarithmic one, whose expression is given by the relation (18). If we consider the same assumptions as in the case of the analytical study, i.e., the friction velocity being 27.59 m/s and dividing the equation by this value, we obtain the velocity distribution law in the cross section of the ruby, for the case of the CFD analysis.

$$\frac{v}{v_f} = 2.375 \cdot \ln(x) + 5.013 \quad (20)$$

As in the previous case, by analogy with relation (14), the constants A and B can be identified as: $A = 2.375$ and $B = 5.013$. It is observed that the values of these constants obtained by CFD analysis and those given by the theoretical law from the analytical study are comparable. According to the relation (20), the average speed at the end of the intermediate zone corresponding to thickness of 0.018 mm is 544.65 m/s.

3.2. 2D Analysis of the Flow through the Cutting Head of the AWJ Machine

In this part of the study, a 2D analysis of the water flow through the active elements of the cutting head from the composition of the AWJ machine from the equipment of the IMN department of UPG Ploiești is made. For this analysis, three ranges of sizes of the water nozzle hole and the mixing tube were considered, as presented in Table 4. It was also considered that the machine discharges the same flow of water and the average velocity of the water through the ruby orifice varies with the flow section.

Table 4. The dimensions of the holes of the cutting head.

The Diameter of the Ruby Hole, d_a [mm]	The Diameter of the Mixing Tube, d_t [mm]	Average Speed through the Ruby Hole, v_m [m/s]
0.20	0.76	972.40
0.25	0.90	622.35
0.30	1.02	432.18

A 2D analysis of the flow through the active elements of the cutting head was conducted using Flotran-CFD mode in Ansys. The geometric model from Figure 12 was adopted in the absence of the abrasive tube and in the assumption that the water flow is symmetrical along the axis of symmetry of the cutting head assembly (line L21).

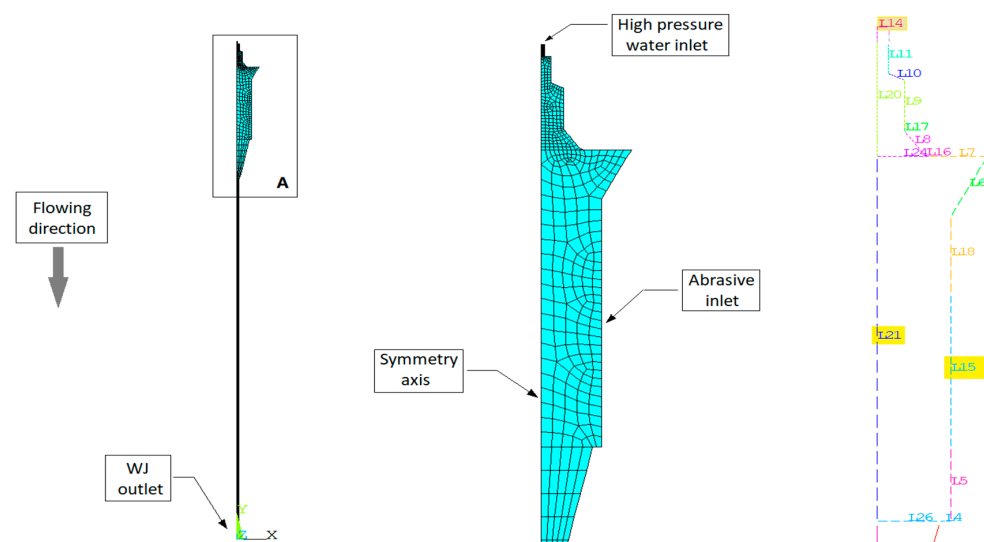


Figure 12. CFD geometric model of the cutting head.

The hydrodynamic parameters used for the purpose of the CFD simulation in this case are presented in Table 5. This analysis was made considering that only one fluid medium (water) circulates through the cutting head, and the flow is achieved without heat exchange processes. Water enters the system through line L14 of the geometric model.

Table 5. Data relating to the modeling of the flow through the cutting head.

Model Type:	2D—Axisymmetric
Element type:	FLUID 141
The fluid material:	water
Density:	$\rho = 998.2 \text{ kg/m}^3$
Dynamic viscosity:	$\mu_a = 1.008 \text{ Pa}\cdot\text{s}$
Boundary conditions	DOF input speed: VY , according to Table 4 DOF wall velocity: $VX = VY = 0$ DOF outlet pressure: 0 Axis of symmetry DOF: $VX = 0$ Abrasive input speed: $VXY = -0.547 \text{ m/s}$
Environment conditions:	Temperature: 293 K Pressure: 101,350 Pa

The graphic representation of the water velocity at the exit from the ruby hole for the three sizes adopted are presented in Figures 13–15. Given the fact that the same flow of water flows through the three holes of the nozzles, it is normal for the velocities at the exit from them to have a decreasing tendency depending on the increase in the flow section. Due to the turbulent nature of the flow, a logarithmic distribution of velocities is observed in the flow section.

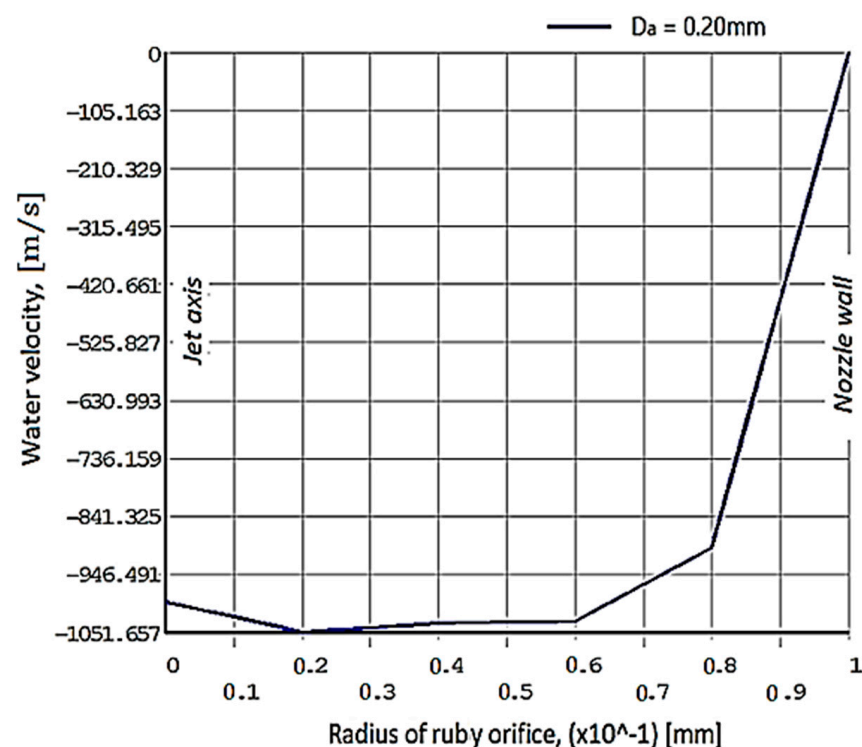


Figure 13. Water velocity at the exit of the 0.2 mm nozzle.

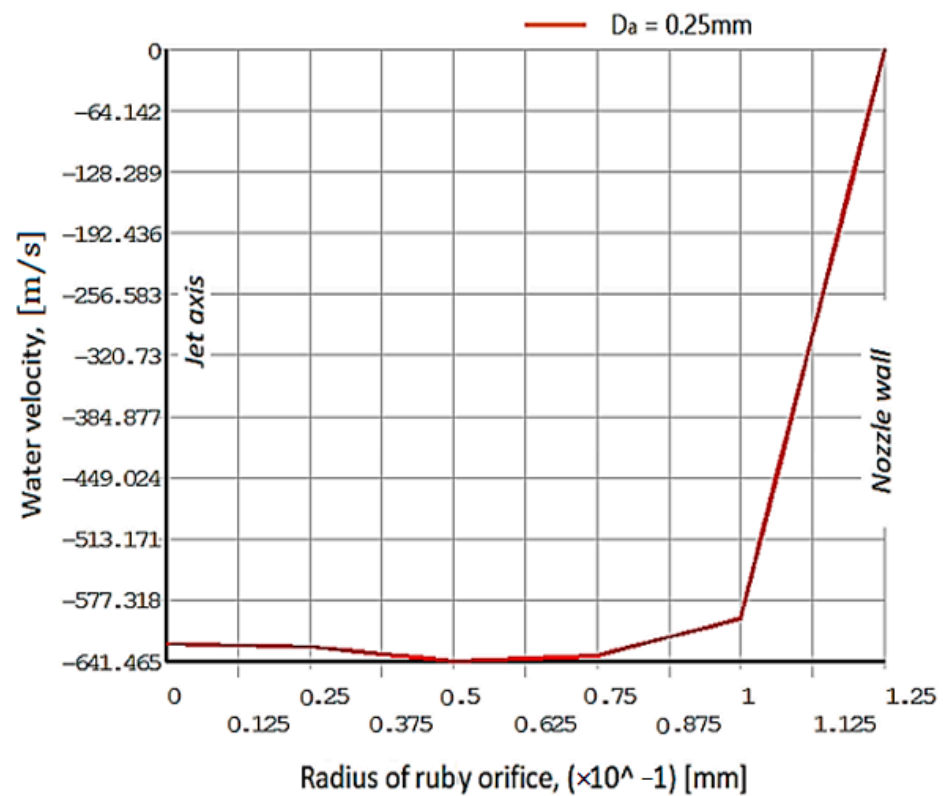


Figure 14. Water velocity at the exit of the 0.25 mm nozzle.

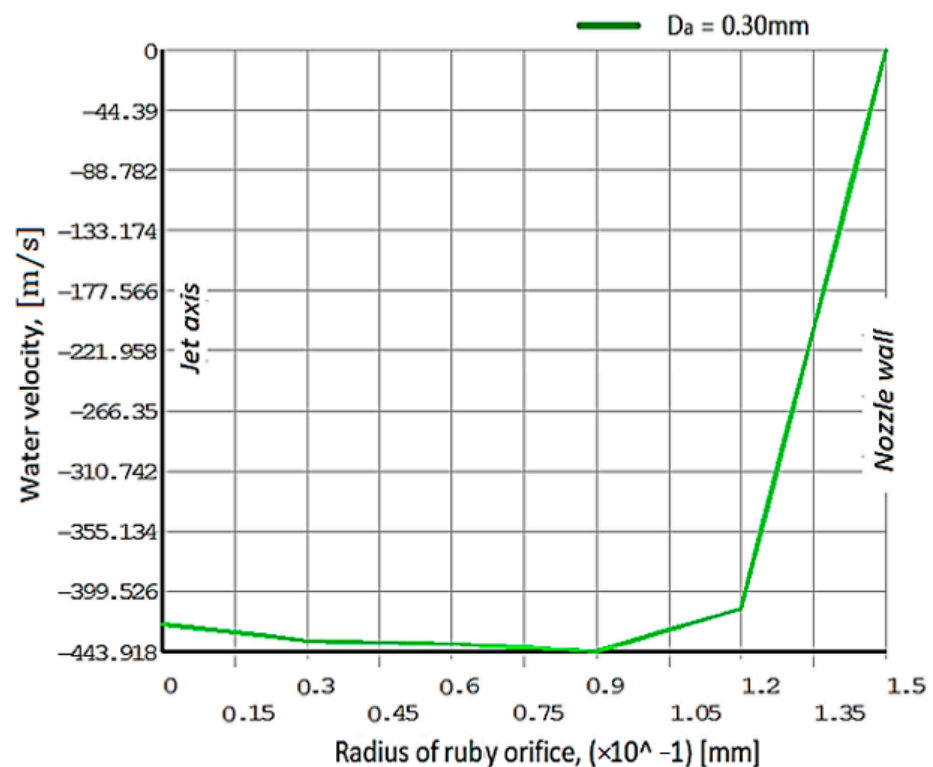


Figure 15. Water velocity at the exit of the 0.30 mm nozzle.

Figure 16 shows the characteristic of normal and shear stresses for a plane jet [29]. Compared to the flow results obtained through a circular duct, a certain qualitative similarity can be observed. It can also be seen that the peaks of the Reynolds stress profile in the case of flow through the pipe are quite sharp and closed towards its wall areas. It

follows that the turbulence produced during the flow occurs near the wall of the pipe (in the present case, at the wall of the ruby orifice). The peaks of the turbulent stress profile developed in a jet are much more developed in the vicinity of the axis of symmetry and less in its center. This leads to the diffuse nature of a plane jet and the entrainment of the surrounding medium (fluids or solid particles) towards its interior.

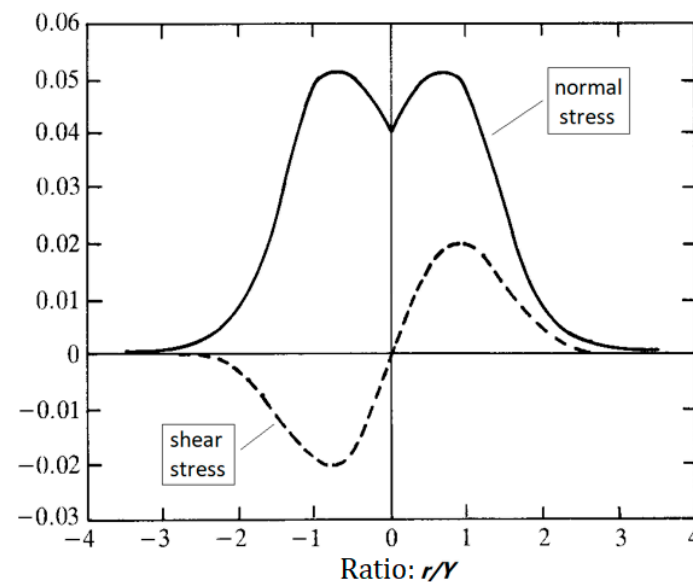


Figure 16. Variation in Reynolds stresses for a free jet.

From the presented results, it can be seen that the water velocity is lower in the axis of symmetry of the jet compared to the rest of the flow interval, showing a peak inside the flow interval, in the fully developed turbulent flow area (the III-rd zone of flow). The values of the velocities in the area adjacent to the ruby nozzle for the three dimensions analyzed are presented in Table 6. This graphical feature influences the diffuse nature of the jet flow, resulting in the variation in normal and shear Reynolds stresses in the jet flow section.

Table 6. Characteristic velocities through the water nozzles.

Hole diameter, d_a [mm]	0.20	0.25	0.30
The water velocity in the axis is symmetrical, v_{ax} [m/s]	996.35	623.38	423.64
Top speed, v_{pk} [m/s]	1051.7	641.46	443.92
Distance from axis of v_{pk} , [mm]	0.020	0.050	0.090
Maximum water speed, [m/s]	1108	726.206	563.308

Modeling of the flow through the three adopted dimensions of the mixing tube was conducted in two ways: with the blocked entrance of the abrasive material access tube and with the free entrance through the abrasive tube.

3.2.1. Case I—Blocked Inlet through the Abrasive Tube

In this analysis, the degrees of freedom (DOF) were blocked along the L15 line of the geometric model, this line being the access gate of the abrasive material in the mixing chamber. Figure 17 shows the speed contour in the case of flow simulation using the pressure nozzle with a diameter of 0.25 mm and the mixing tube of 0.76 mm. For the other cases there is a proportional variation in velocities due to similar flow conditions. From the figure below, it can be seen that the speed values along the MT length (detail A) are in the range of 0–80 m/s.

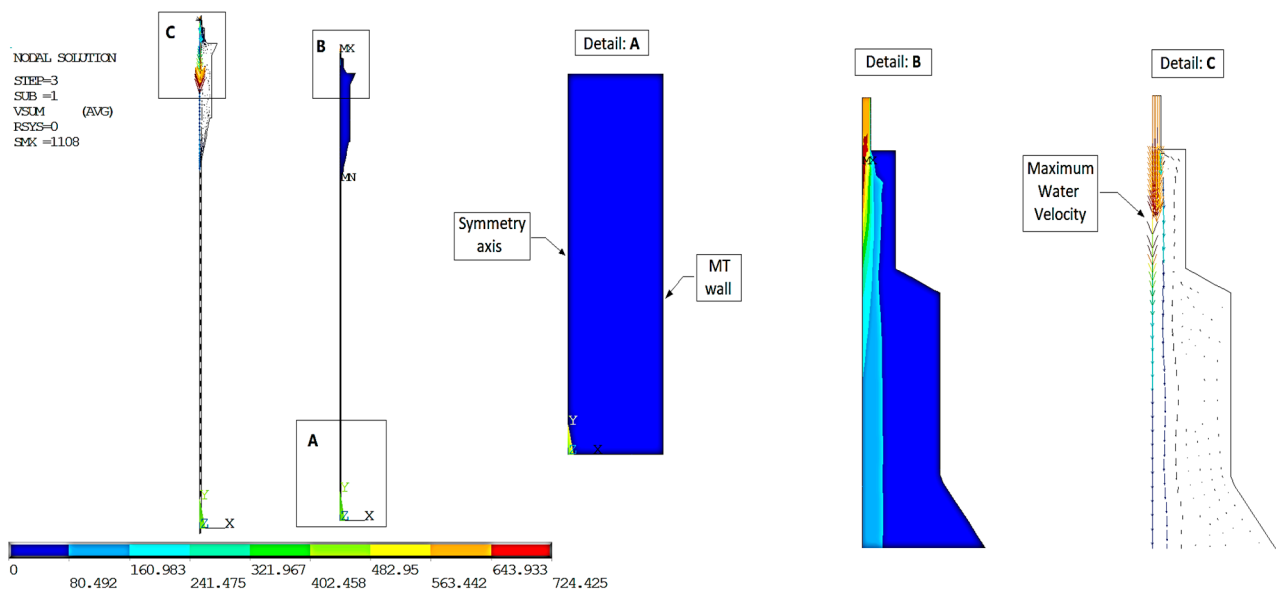


Figure 17. Velocity distribution in case of TA locked.

In Figure 18, the variation in the water speed through the flow hole of the MT in the blocked TA situation is represented graphically for the three considered values of the diameters. Due to the fact that the same water flow flows through the considered model, a decreasing dependence of the flow rates is observed as the diameter of the MT orifice increases. A similar shape of the velocity profile is also observed in the work [30], where the flow of the liquid phase is found in the laminar regime, having the Re number between 303 and 1532.

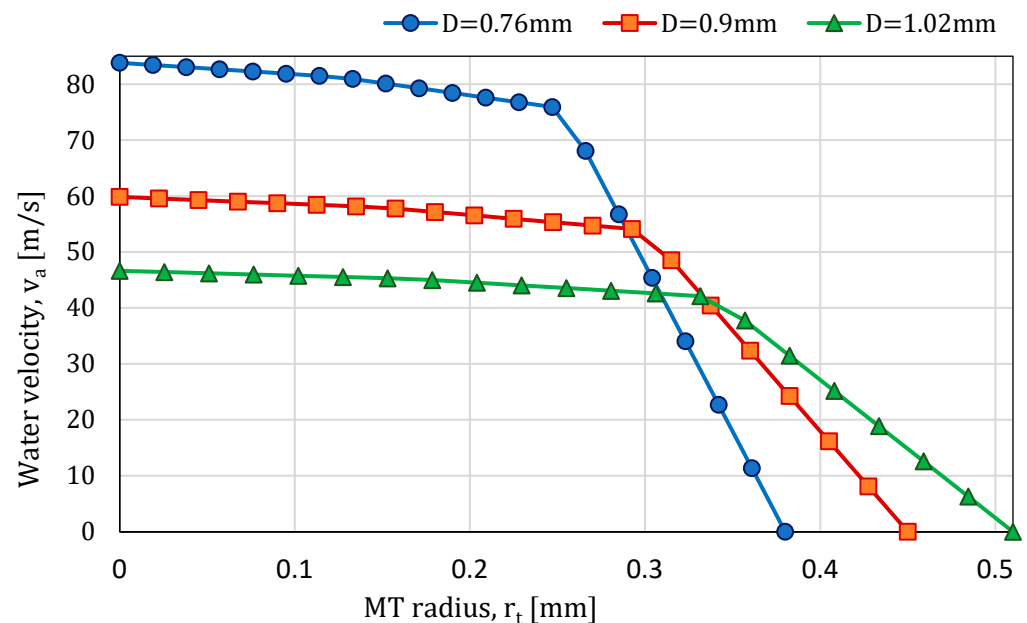


Figure 18. Variation in water velocity through MT in case of blocked TA.

The numerical values of the flow velocities at the exit from the MT in the case of blocked TA, for the three adopted sizes of the diameters (d_t), are presented in Table 7. The average theoretical values of the velocities were obtained using the flow continuity law, and the calculation error was obtained using the relation (21).

$$\varepsilon_r = \frac{v_{th} - v_{CFD}}{v_{th}} \cdot 100 \quad (21)$$

where ε_r —represents the error of the obtained values; v_{th} —represents the theoretical average speed of the water; and v_{CFD} —water velocity obtained by CFD simulation.

Table 7. Flow characteristics in case of blocked TA.

MT Diameter, d_t [mm]	0.76	0.90	1.02
Wall layer thickness, [mm]	0.133	0.158	0.179
Maximum velocity, v_{max} [m/s]	83.824	59.847	46.636
Average velocity CFD, v_{CFD} [m/s]	65.041	46.398	36.134
Average theoretical velocity v_{th} [m/s]	67.343	48.021	37.38
Velocity error, ε_r [%]	3.41	3.37	3.33
Reynolds number, Re	3015.9	2546.78	2246.76

The maximum flow velocity was obtained in the axis of symmetry for all three values of the diameters, and the thickness of the wall layer was determined according to the point where the velocity graph changes its slope.

From Table 7, it can be seen that only in the case of the tube with a diameter of 0.76 mm the flow is turbulent, and in the other two cases it is transient. However, the graphical form of the distribution of water velocities at the exit of the tube openings is of a turbulent type, according to the CFD simulation performed.

3.2.2. Case II—Free Entry through the Abrasive Tube

In this analysis, it was considered that the fluid enters both through line L14 of the geometric model, and through the space intended for the abrasive material to enter the mixing chamber (L15). It was taken into account that the fluid from the abrasive medium enters the mixing chamber with the same flow rate, regardless of the sizes of the nozzles adopted. Since the construction of the cutting head is asymmetric, it was hypothesized that the fluid from the abrasive medium flows into the mixing chamber through a lateral surface generated by line L15, according to the relation:

$$A_{L,abr} = \pi \cdot D_{MC} \cdot h_{L15} = \pi \cdot 0.005 \cdot 5.8 \cdot 10^{-3} = 91.12 \cdot 10^{-6} \text{ m}^2 \quad (22)$$

where $A_{L,abr}$ —the lateral flow area of the abrasive through the MC wall; DMC—diameter of the mixing chamber; and h_{L15} —the height of the L15 generator in the geometric model.

Knowing the flow rate of the fluid through the TA, from the law of continuity, the average velocity of the fluid entering through the TA is determined, with the relation (23):

$$v_{m,abr} = \frac{Q_g}{A_{L,abr}} = \frac{4.991 \cdot 10^{-5}}{91.12 \cdot 10^{-6}} = 0.547 \frac{\text{m}}{\text{s}} \quad (23)$$

where $v_{m,abr}$ —represents the average flow speed of the abrasive in the case of free TA; Q_g —fluid flow through TA.

Knowing that the direction of fluid entry from TA to MC is at an angle of approximately 30 degrees to the horizontal, it follows that the value of the horizontal component of the velocity is 0.471 m/s, and the vertical component is 0.277 m/s.

Figure 19 shows the speed contour obtained by CFD analysis in the case of free TA. Unlike the previous case, the velocity values along the mixing tube can reach up to 240 m/s. It follows that the cutting head assembly acts as an amplifier depending on the input velocity of the abrasive medium.

The graphic form of the distribution of water velocities through the adopted openings of the MT is similar to those analyzed in the blocked TA case, except for their values, as shown in Figure 20. This is due to the fact that fluid flows through the TA at a certain rate, and this leads to the amplification of the values of the velocities in the mixing tube.

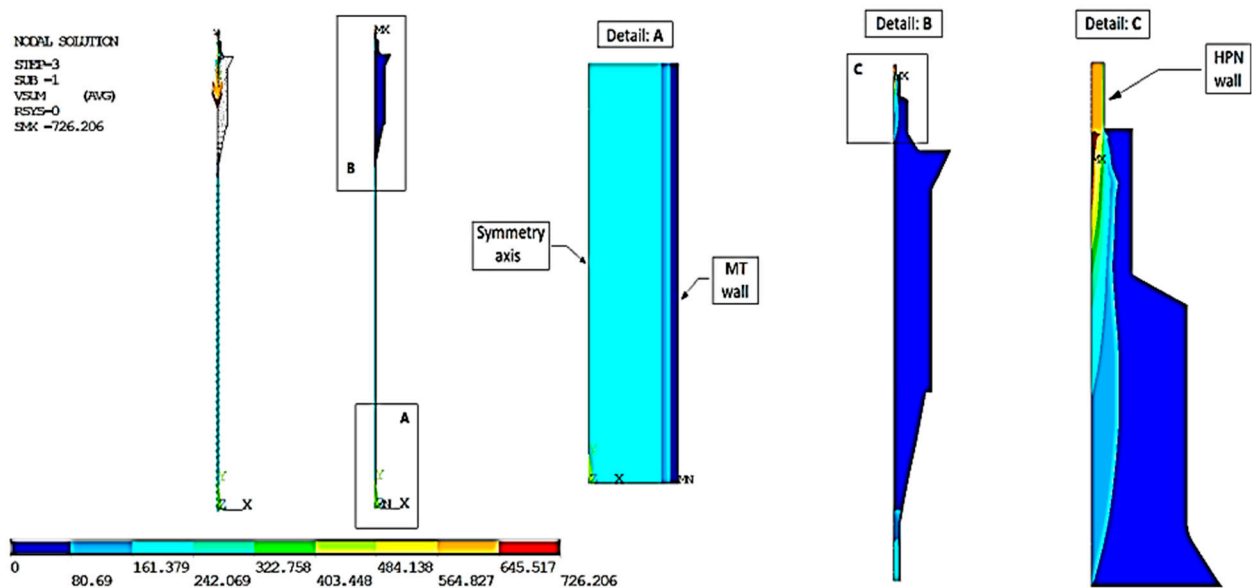


Figure 19. Velocity distribution in case of free TA.

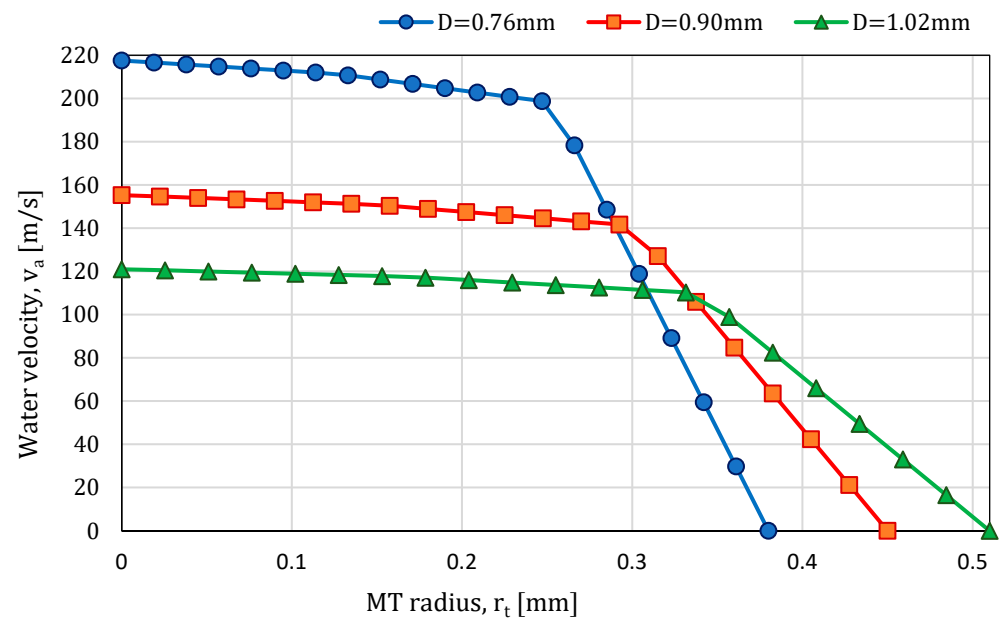


Figure 20. Variation in water velocity through MT in case of free TA.

The flow characteristics resulting from the CFD analysis through the active elements of the cutting head, in case of unlocking the TA circuit, are presented in Table 8. In order to highlight the amplification phenomenon produced by the CH assembly with the help of the TA circuit, the notion of amplification factor (ca) was introduced, both in the case of the average speed and in the case of the maximum speed. Their expressions are given by relations (24) and (25).

$$c_{a,m} = \frac{v_{m,BL}}{v_{m,DBL}} = 2.606 \quad (24)$$

$$c_{a,M} = \frac{v_{M,BL}}{v_{M,DBL}} = 2.595 \quad (25)$$

Table 8. Flow characteristics in the case of free TA.

MT diameter, d_t [mm]	0.76	0.90	1.02
CFD average velocity, v_{CFD} [m/s]	169.553	120.947	94.186
Maximum velocity, v_M [m/s]	217.57	155.31	121.01
Reynolds number, Re	1.276×10^5	1.078×10^5	0.951×10^5
Wall tangential stress, τ_p [Pa]	5.895×10^4	3.115×10^4	1.943×10^4
Friction velocity, v_f [m/s]	7.685	5.586	4.412
Reference length, y_f [m]	1.314×10^{-7}	1.808×10^{-7}	2.289×10^{-7}

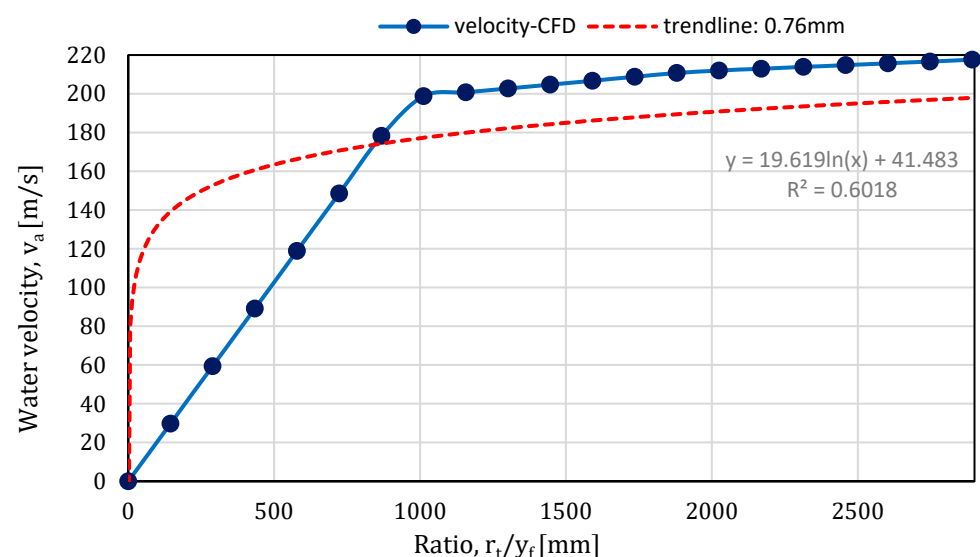
In the relations (24) and (25), with the speed amplification factor noted, the m/M indices refer to the average and maximum values of the velocities, and the BL/DBL indices refer to the case of TA in locked and free state, respectively.

It is observed that the amplification factor remains the same for all adopted MT sizes, both in the case of the maximum speed and in the case of the average speed. This is due to the fact that the fluid inlet velocity through the TA is the same for all three adopted dimensions.

According to [8,31], water jets typically exhibit high Reynolds numbers, and as the Reynolds number increases, the influence of turbulence becomes more significant.

The values of the Reynolds number presented in Table 8 are somewhat lower than the range reported in other studies. For instance, [31] provides an overview of the experimental parameters for water jets, indicating Reynolds number values in the range of 163,000 to 180,000. This comparison highlights that while the Reynolds numbers in this study are high, they are still below the typical range reported in [31], suggesting that the turbulence effects in this study might be less pronounced than those observed in the higher range of Reynolds numbers.

Figure 21 graphically shows the distribution of water velocities at the exit of the CH, in the case of the mixing tube with a diameter of 0.76 mm, depending on the ratio between the radius of the tube and the reference length. As can be seen, the graphic form of the CFD velocity distribution is specific to turbulent flow, a fact also demonstrated by the value of the Re number.

**Figure 21.** Variation in water speed through MT with a diameter of 0.76 mm.

The distribution law of water velocities in the cross section of the tube is given by the relation (26), this being obtained under the assumption that the logarithmic flow law

applies to the entire MT-section. Additionally, this shape was obtained by dividing the trend curve of the flow velocities by the friction velocity v_f .

$$\frac{v}{v_f} = 2.552 \cdot \ln(x) + 5.397 \quad (26)$$

By analogy with the logarithmic law describing the turbulent motion, the constants can be identified: $A = 2.552$ and $B = 5.397$. As can be seen, the matching factor of the CFD results with the logarithmic velocity distribution law is 60.18%. This demonstrates that the logarithmic law is not quite appropriate over the entire flow section of the tube, and each turbulent layer in the flow constitution must be treated separately.

4. Conclusions

The analysis of water flow through the active parts of an AWJ machine using both analytical methods and CFD simulations yielded significant insights. The study demonstrated the complex nature of turbulent flow through the ruby orifice, highlighting the importance of considering multiple flow zones to accurately predict velocity distributions and pressure drops.

Since the flow regime through the orifice of the ruby nozzle is turbulent, the analytical study was conducted taking into account that in the cross section of the nozzle, there are four flow zones measured from the wall symmetry axis, the thicknesses of these layers, as well as the average velocities obtained, were determined. Using these values, the distribution curves of the average analytical and theoretical flow velocities were obtained. The theoretically determined logarithmic law within the analytical analysis has the constants $A = 2.352$ and $B = 5.002$. These values were obtained considering a nozzle wall roughness value of $0.012 \mu\text{m}$. The thickness of the wall layer has the value of 0.0181 mm , meaning 14.49% of the ruby nozzle radius, this value being very close to 15% according to [23]. In the analytical calculation of the stationary turbulent motion, the friction velocity was calculated obtaining the value of 27.59 m/s , as well as the tangential wall stress with the value of 0.76 MPa .

The pressure drop obtained in the numerical simulation shows two values: along the axis of symmetry, it is 9.649 MPa , and at the nozzle wall, it is 12.431 MPa . It was found that the pressure varies at the nozzle inlet following a power law. The same graphical configuration of the pressure was obtained in the work [32]. The maximum water velocity was determined to be 642.46 m/s , distributed over an interval corresponding to the central flow zone, measured at 0.038 mm from the nozzle wall. The average flow velocity distribution law obtained in this case is logarithmic, with the coefficients of this equation being $A = 2.375$ and $B = 5.013$. According to the logarithmic velocity distribution law from the CFD analysis, the average velocity at the end of the intermediate zone corresponds to a thickness of 0.018 mm is 544.65 m/s .

It was observed that at the exit of the ruby nozzle orifice, the water velocity is lower along the axis of symmetry compared to the interval toward the wall. At a certain distance from the ruby orifice exit, the water velocity along the axis of symmetry increases to values of 1100 m/s for the nozzle with a 0.2 mm orifice. This leads to the divergent nature of the jet, which entrains the surrounding medium (it can act as a velocity amplifier in the CH assembly).

From the flow analysis through the tubes in the blocked/free TA situations, an increase in water velocities is observed in the free TA case with an amplification factor of 2.606 for the average velocity and 2.595 for the maximum velocity. Due to the extreme velocities (approx. 725 m/s) developed by the water nozzles, a vacuum pressure is produced in the MC, which entrains the connecting media (e.g., air and abrasive particles).

For the mixing tube with a diameter of 0.76 mm (usual size for the WUXI YCWJ-380-1520 machine), the average water velocity at the exit was 170 m/s , and the maximum was around 217 m/s . In some materials [33], the velocities adopted in the AWJ impact

study are in the range of 180–220 m/s, confirming the validity of the previously presented calculations and assumptions.

The tangential stresses developed during turbulent flow were determined, not exceeding the critical stress values of the MT wall material.

The coefficients and distribution law of the velocities in the cross section of the MT with a diameter of 0.76 mm were established. Due to the poor fit of the trend curve Formula (24) over the CFD results, it follows that the velocity distribution law is not logarithmic across the entire MT section.

Our findings show that the theoretical models, validated by CFD simulations, can effectively describe the flow characteristics in different zones, from the viscous substrate to the turbulent core. The results also underscore the necessity of incorporating detailed flow analyses in the design and optimization of AWJ machine components to enhance cutting efficiency and precision.

Future work should focus on extending this analysis to include the effects of abrasive particle interactions and varying operational conditions. Additionally, experimental validation with a broader range of nozzle geometries and flow rates would further refine the predictive capabilities of the models developed in this study. By advancing our understanding of water flow dynamics in AWJ machines, we can contribute to the development of more efficient and precise machining processes, ultimately benefiting a wide range of industrial applications.

Author Contributions: Conceptualization, I.P., R.G.R., C.P. and M.T.; methodology, I.P. and R.G.R.; validation, I.P. and M.T.; formal analysis, I.P., R.G.R., C.P. and M.T.; investigation, I.P., R.G.R. and M.T.; resources, C.P.; data curation, I.P. and R.G.R.; writing—original draft preparation, I.P., R.G.R., C.P. and M.T.; writing—review and editing, I.P., R.G.R., C.P. and M.T.; visualization, I.P. and M.T.; supervision, C.P. and M.T.; project administration, C.P.; funding acquisition, C.P. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: All data are contained within the article.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Patirnac, I.; Ripeanu, R.G.; Laudacescu, E. The Influence of the AWJM Working Parameters on Manufactured Surfaces Microgeometry. *MATEC Web Conf.* **2020**, *318*, 01011. [\[CrossRef\]](#)
2. Baha, V.; Pavlenko, I.; Židek, K.; Ciszak, O. Ensuring the Abrasive Jet Machining Efficiency Using a Nozzle with a Perforated Insert. *Machines* **2024**, *12*, 347. [\[CrossRef\]](#)
3. Baha, V.; Mižáková, J.; Pavlenko, I. An Increase in the Energy Efficiency of Abrasive Jet Equipment Based on the Rational Choice of Nozzle Geometry. *Energies* **2023**, *16*, 6196. [\[CrossRef\]](#)
4. Zou, X.; Fu, L.; Wu, L.; Zuo, W. Research on Multiphase Flow and Nozzle Wear in a High-Pressure Abrasive Water Jet Cutting Head. *Machines* **2023**, *11*, 614. [\[CrossRef\]](#)
5. Deepak, D.; Anjaiah, D.; Karanth, K.V.; Sharma, N.Y. CFD Simulation of Flow in an Abrasive Water Suspension Jet: The Effect of Inlet Operating Pressure and Volume Fraction on Skin Friction and Exit Kinetic Energy. *Adv. Mech. Eng.* **2012**, *4*, 186430. [\[CrossRef\]](#)
6. Der, O.; Ordu, M.; Başar, G. Multi-Objective Optimization of Cutting Parameters for Polyethylene Thermoplastic Material by Integrating Data Envelopment Analysis and SWARA-Based CoCoSo Approach. *Osman. Korkut Ata Üniversitesi Fen Bilim. Enstitüsü Derg.* **2024**, *7*, 638–661. [\[CrossRef\]](#)
7. Hu, R.; Shahzad, F.; Abbas, A.; Liu, X. Decoupling the Influence of Eco-Sustainability Motivations in the Adoption of the Green Industrial IoT and the Impact of Advanced Manufacturing Technologies. *J. Clean. Prod.* **2022**, *339*, 130708. [\[CrossRef\]](#)
8. Shao, C.; Ge, Z.; Zhou, Z.; Liu, W.; Li, Z.; Tian, C.; Chang, W. Experimental and Numerical Investigation of Abrasive Water Jet Nozzle Erosion. *Powder Technol.* **2023**, *430*, 119031. [\[CrossRef\]](#)
9. Pătrînac, I.; Ripeanu, R.G.; Laudacescu, E. Abrasive Flow Modelling through Active Parts Water Jet Machine Using CFD Simulation. *IOP Conf. Ser. Mater. Sci. Eng.* **2020**, *724*, 012001. [\[CrossRef\]](#)
10. Liu, H.; Wang, J.; Kelson, N.; Brown, R.J. A Study of Abrasive Waterjet Characteristics by CFD Simulation. *J. Mater. Process. Technol.* **2004**, *153–154*, 488–493. [\[CrossRef\]](#)
11. Venugopal, S.; Chandrasekaran, M.; Muthuraman, V.; Sathish, S. Computational Fluid Dynamics Analysis of Nozzle in Abrasive Water Jet Machining. *IOP Conf. Ser. Mater. Sci. Eng.* **2017**, *183*, 012021. [\[CrossRef\]](#)

12. Kamarudin, N.H.; Prasada Rao, A.K.; Azhari, A. CFD Based Erosion Modelling of Abrasive Waterjet Nozzle Using Discrete Phase Method. *IOP Conf. Ser. Mater. Sci. Eng.* **2016**, *114*, 012016. [\[CrossRef\]](#)
13. Cană, P.; Ripeanu, R.G.; Pătrînac, I.; Diniță, A.; Tănase, M. Investigating the Impact of Operating Conditions on Relief Pressure Valve Flow through CFD and Statistical Analysis. *Processes* **2023**, *11*, 3396. [\[CrossRef\]](#)
14. Lospa, A.M.; Ripeanu, R.G.; Dinita, A. Erosion Modelling: A Systematic Review of Available Models and Equations. *IOP Conf. Ser. Mater. Sci. Eng.* **2020**, *724*, 012037. [\[CrossRef\]](#)
15. Ri, J.; Ripeanu, R.-G.; Dinita, A. Erosion Modeling in Parallel Gate Valve. *FME Trans.* **2020**, *48*, 808–815. [\[CrossRef\]](#)
16. Di Nucci, C.; Russo Spena, A. Mean Velocity Profiles of Two-Dimensional Fully Developed Turbulent Flows. *Comptes Rendus. Mécanique* **2012**, *340*, 629–640. [\[CrossRef\]](#)
17. Afzal, N.; Seena, A.; Bushra, A. Power Law Velocity Profile in Fully Developed Turbulent Pipe and Channel Flows. *J. Hydraul. Eng.* **2007**, *133*, 1080–1086. [\[CrossRef\]](#)
18. Fletcher, D.F.; Maskell, S.J.; Patrick, M.A. Theoretical Investigation of Chilton-Colburn Analogy Using a Modified Form of the Van Driest Eddy Viscosity Hypothesis. *Trans. IChemE Part A* **1982**, *60*, 122–125.
19. Nanduri, M.; Taggart, D.G.; Kim, T.J. The Effects of System and Geometric Parameters on Abrasive Water Jet Nozzle Wear. *Int. J. Mach. Tools Manuf.* **2002**, *42*, 615–623. [\[CrossRef\]](#)
20. Yu, Y.; Sun, T.; Yuan, Y.; Gao, H.; Wang, X. Experimental Investigation into the Effect of Abrasive Process Parameters on the Cutting Performance for Abrasive Waterjet Technology: A Case Study. *Int. J. Adv. Manuf. Technol.* **2020**, *107*, 2757–2765. [\[CrossRef\]](#)
21. Li, Z.; Yao, S.; Yun, F.; Wang, X.; Wang, L.; Wu, Y. Simulation and Optimization of the Nozzle Section Geometry for a Suspension Abrasive Water Jet. *Machines* **2021**, *10*, 3. [\[CrossRef\]](#)
22. Wang, H.; Yuan, R.; Zhang, X.; Zai, P.; Deng, J. Research Progress in Abrasive Water Jet Processing Technology. *Micromachines* **2023**, *14*, 1526. [\[CrossRef\]](#) [\[PubMed\]](#)
23. Ibarra, R.; Zadrazil, I.; Matar, O.K.; Markides, C.N. Dynamics of Liquid–Liquid Flows in Horizontal Pipes Using Simultaneous Two–Line Planar Laser–Induced Fluorescence and Particle Velocimetry. *Int. J. Multiph. Flow* **2018**, *101*, 47–63. [\[CrossRef\]](#)
24. *NORSOK Standard M-506*; CO₂ Corrosion Rate Calculation Model. Norwegian Technology Center: Oslo, Norway, 2005.
25. Rugozitati. Available online: <https://www.rezmat.ro/rugozitati/?cmd=rugozitati> (accessed on 15 June 2024).
26. Çengel, Y.A.; Cimbala, J.M. *Fluid Mechanics: Fundamentals and Applications*, 3rd ed.; McGraw Hill: New York, NY, USA, 2014; ISBN 978-0-07-338032-2.
27. Absi, R. A Simple Eddy Viscosity Formulation for Turbulent Boundary Layers near Smooth Walls. *Comptes Rendus. Mécanique* **2009**, *337*, 158–165. [\[CrossRef\]](#)
28. Ansys 56 Manual | JOEL AGUILAR ROSALES-Academia.Edu. Available online: https://www.academia.edu/34791911/Ansys_56_manual (accessed on 17 June 2024).
29. McComb, W.D. *The Physics of Fluid Turbulence*; Oxford Engineering Science Series; Clarendon Press: Oxford, UK, 1992; ISBN 978-0-19-856256-6.
30. Zadrazil, I.; Markides, C.N. An Experimental Characterization of Liquid Films in Downwards Co-Current Gas–Liquid Annular Flow by Particle Image and Tracking Velocimetry. *Int. J. Multiph. Flow* **2014**, *67*, 42–53. [\[CrossRef\]](#)
31. Jasper, S.; Hussong, J.; Lindken, R. PIV Investigation of High Reynolds Number Submerged Water Jets at High-Pressure Ambient Conditions. *Exp. Fluids* **2021**, *62*, 97. [\[CrossRef\]](#)
32. Yang, F.C.; Shiah, S.W.; Heh, T.Y.; Jen, C.Y.; Chang, W.I. Study of Numerical Simulation Applying to the Design of an Orifice with High-Velocity Waterjet. *J. Appl. Sci. Eng.* **2008**, *11*, 145–154. [\[CrossRef\]](#)
33. Junkar, M.; Jurisevic, B.; Fajdiga, M.; Grah, M. Finite Element Analysis of Single-Particle Impact in Abrasive Water Jet Machining. *Int. J. Impact Eng.* **2006**, *32*, 1095–1112. [\[CrossRef\]](#)

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.