Design of a mechanical pedal valve to improve the use of water in plumbing systems

L F López Uribe¹, C Sánchez Rincón¹, M Díaz Jiménez¹, and E Flórez Solano¹
¹ Grupo de Investigación en Ingenierías Aplicadas para la Innovación, Gestión y Desarrollo, Universidad Francisco de Paula Santander, Seccional Ocaña, Colombia

E-mail: lflopezu@ufpso.edu.co, cssanchezri@ufpso.edu.co

Abstract. The use of water is a very important economic and environmental factor in all industrial sectors since the improvement in its use offers an optimization of this resource. The waste of water usually causes damages firstly to the environment by the supply sources, then to the industries by its high demand and to the hydrosanitary sources that the houses have. The valves have some control that allows a more comfortable use of this resource in sinks that provides some savings. Currently, there are valves that usually replace the traditional ones due to their automatic and/or mechanical operation, which allows for innovation and savings of this resource. In this sense, this research is oriented to the design of a mechanical pedal valve that allows a more comfortable and controlled control, thus granting the operator control in an immediate way, without wasting the fluid in the middle of the washing and avoiding a manual contact that in a certain way dissipates the risk of contagion of diseases such as Covid-19 by direct manipulation. An analytical methodology supported by computer aided drafting programs such as Ansys Fluent and SolidWorks was used. Finally, it was determined that, for the dynamic and static studies, the designed valve complies with the Colombian technical standard and the static design factors.

1. Introduction
Hydraulic valves have brought a great development to hydraulic systems in most industrial applications and in homes with plumbing systems, since their use is not only based on the control of water flow, but also on the way in which this valuable resource can be saved. A large waste of water leads to large environmental, economic and energy losses. Therefore, many authors have developed research oriented to the innovation of these, in order to take them to another level of production and optimization [1,2]. On the other hand, the field of computational fluid dynamics (CFD) is a methodology of great help for the study of the behavior of the variables involved in a hydraulic system, these variables are the velocity and pressure of the liquid stream flowing around or inside the object under study [3-7]. In this way, the use of specialized software for the study of these dynamic processes of fluids and static processes of valve design becomes more acute, since continuous and discrete particle systems are easily simulated by means of these tools such as MATLAB® and Ansys, since their programming language presents a high complexity of equations for the analysis providing a low percentage of error.

Factors such as the inadequate use of traditional domestic valves have caused a disproportionate loss of this resource in the basic manual washing process, since the faucet is usually kept open while the operation is being completed. In view of the situation posed by the need for a more controlled use, this research is based on the design of a mechanical pedal valve, where the operator has the facility to complete the task with a moderate use of the surrounding fluid, thus avoiding waste, and providing
comfort in the process. For this, a descriptive methodology was used which was based on the static and dynamic analysis of the valve supported by SolidWorks and Ansys Fluent, which took as design support the parameters imposed by the “Norma Técnica Colombiana (NTC), NTC 1500” [8]. Finally, this design represents a positive impact in the adequate use of water and in the reliability of its operation under real and ideal conditions.

2. Methodology
This is a descriptive type of research with a quantitative approach based on different calculations and measurements that allowed structuring the design of a spring-return mechanical pedal valve as an alternative to improve the use of water through faucets in plumbing systems. In a pedal valve the user controls with the foot the flow of fluid through a pedal actuator that pushes a stem allowing the circulation of water through a pipeline, for this purpose the loads and pressures that interact in the body of the valve, the stem and the pedal must be analyzed. Therefore, these parts are studied as a discrete system of particles, which is related to physical principles such as Newton's second law with the development of the equations to perform the relevant static and dynamic analysis [9-11]. In addition, this process was developed in the environment of computer aided design (CAD) software that is based on finite element analysis (FEM).

2.1. Model of mechanical elements
Next, the modeling of the mechanical elements of the system was developed by the SolidWorks® software that presides high precision by the complexity of the programming language that supports it. Figure 1 shows the parts that make up the established design, such as the valve body, cover, spring, stem, pedal base, and pedal; then Table 1 shows the dimensions of the critical parts of the valve.

![Figure 1. Pedal valve parts.](image)

| Piece                        | Area (mm²) |
|------------------------------|------------|
| (1) Spring                   | --         |
| (2) Valve body inner conduit | 7642       |
| (3) Internal surface of the rod | 81.68   |
| (4) External surface of the rod | 132.11   |
| (5) Body cover               | --         |
| (6) Pedal base               | --         |
| (7) Valve joint              | --         |
| (8) Pedal face               | 2243.26    |

2.2. Simulation of fluid behavior through the valve
The geometric design of this valve was subjected to a fluid analysis with the Ansys Fluent tool to observe the behavior of the pressures and velocities in conditions of daily use established in the range of the NTC 1500 [8] varying the obstruction of the water flow. For this purpose, the hydraulic duct of a dishwasher system used in most homes was represented as an example, in which the conventional
handles were replaced by the proposed design and results were obtained from the idealized evaluation in the software; K-EPSILON standard was the evaluation model used in the simulation. The kinetic energy of turbulence, k, and its dissipation rate, ε, are obtained from the following transport Equation (1) and Equation (2) [12].

$$\frac{\partial}{\partial t} \rho k + \frac{\partial}{\partial x_i} \left( \rho k u_i \right) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k, \quad (1)$$

$$\frac{\partial}{\partial t} \left( \rho \varepsilon \right) + \frac{\partial}{\partial x_i} \left( \rho \varepsilon u_i \right) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} \varepsilon) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon, \quad (2)$$

where, $G_k$ represents the turbulence kinetic energy generation due to mean velocity gradients, $G_b$ is the turbulence kinetic energy generation due to buoyancy, $Y_M$ represents the contribution of fluctuating dilatation in compressible turbulence to the overall dissipation rate, $C_{1\varepsilon}, C_{2\varepsilon},$ and $C_{3\varepsilon}$ are constants. $\sigma_k$, $\sigma_\varepsilon$ are the turbulent Prandtl numbers for $k$ and $\varepsilon$, respectively. $\rho$ is the fluid density, $u_i$ is the flow velocity, $x_i, x_j$ are the positions, $S_k$ and $S_\varepsilon$ are user-defined source terms and $\mu_t$ is the turbulent viscosity.

This method uses standard wall functions based on the analysis of the boundary layer that the fluid has with the internal walls of the device, point to point operations are performed depending on the mesh to be used. The logarithmic law of mean velocity valid and reliable results must meet in a predetermined range "Yplus" or "Y*" between $30 < Y < 300$ [13,14]. Table 2 shows the parameters and variables taken for the simulation, such as mesh type and number of elements. According to the standard in its second update, it indicates that the operation of the service of the water fluid for hydro-sanitary devices consists of a minimum pressure of 5 kPa and a minimum flow rate of 0.19 L/s. However, for real conditions of use, an experimental method of measurement of specific pressures was carried out for this research, in homes in different areas of the city of Ocaña, Norte de Santander, Colombia, it was obtained that the highest pressure found in the homes was 242 kPa.

| Table 2. Dynamic simulation parameters in Ansys Fluent. |
|-------------------------------------------------------|
| Type of mesh                                           |
| Tetrahedral with edge tuning                           |
| Number of nodes                                     | 88880 |
| Number of elements                                   | 404775 |
| Pipe diameter                                        | 1/2"  |
| Inlet pressure                                       | 242 kPa |
| Outlet pressure                                      | Atmospheric |
| Type of fluid                                        | Water (H₂O) |
| Fluid characteristic                                 | Turbulent (k-epsilon) no-slip condition |
| Iterations                                           | 100    |

2.3. Static analysis of the valve

Following the dynamic study, a static analysis was performed in SolidWorks by means of FEM in the design against maximum pressure conditions. The simulation was developed considering the valve closed, this implies pressure of 242 kPa in the internal walls, fastenings with screws that fix the device to the floor and a bolt that joins the base of the valve and the pedal. Most valves and faucets are made of metallic and polymeric materials, the most commonly used being copper, steel, polyvinyl chloride (PVC) and high-density polyethylene (HDPE) [15,16]. Based on the above, it was decided to use a material with excellent resistance to traction, compression, low density, chemically inert, high impermeability, and ductility. In this sense, the selection was oriented towards (HDPE).

2.3.1. Meshing. The Jacobian points are used to determine the degree of confidence of the mesh, they verify that the contours of the elements do not intersect, if a Jacobian point is negative the software stops immediately. Table 3 shows the static simulation parameters; another way with which the software makes a quality check on the mesh elements is by means of the aspect ratios, it does this by making sure
that the tetrahedral elements are perfect and uniform whose edges are of equal length [17]. Figure 2(a) shows the fixed points for the analysis and in Figure 2(b) the total meshing.

Figure 2. Meshing process. (a) fixed points established for the static analysis, (b) total valve meshing.

| Table 3. Static simulation parameters in SolidWorks. |
|-----------------------------------------------|
| Type of mesh | Tetrahedral with edge tuning |
| % Of distorted elements | 0 |
| Jacobian points | 4 |
| Element size | 2.39 mm |
| % Of elements with aspect ratio > 10 | 0.02 |
| Total number of nodes | 72235 |
| Total number of elements | 45012 |

2.3.2. Von Mises stress and factor of safety. To calculate the element stresses, the program averaged the corresponding nodal stresses for each element. In most cases, the yield stress (which is defined by the selected material) is used as the stress limit. However, SolidWorks allows the use of the tensile/rupture stress limit. The software offers criteria with which the safety factor distribution is calculated, the recommended one for linear static analysis and ductile materials is the Von Mises Max stress criterion. This says that the safety factor (SF) will be equal to the quotient between the yield stress of the material and the Von Mises Max stress, since the material will start to fail when the yield stress and the stress are equal [18]. Concerning the principal stresses $\sigma_1$, $\sigma_2$, $\sigma_3$, the Von Mises stress is expressed by Equation (3); where, $\sigma_e$ is the maximum Von Mises stress.

$$
\sigma_e = \sqrt{\frac{(\sigma_1-\sigma_2)^2 + (\sigma_1-\sigma_3)^2 + (\sigma_2-\sigma_3)^2}{2}},
$$

3. Results
In this section it reports the detailed analysis of the data in two sections governed by the methodological order established from the beginning, in which the analysis of the dynamic behavior of the fluid and the static behavior of the device are specified, respectively.

3.1. Dynamic analysis in the valve
The following graphs illustrated in Figure 3, in conjunction with Table 4 show the result of the variation of pressure and velocity as the stem obstructs the conduit, the condition reports that the fully open stem has moved 0 mm and when closed, its displacement will have been 10 mm as shown in Figure 3. Then, it is observed that in Figures 3(a) and 3(b) the variation of pressure and velocity respectively, does not present considerable fluctuations between 0 mm and 5 mm of opening.
Figure 3. Representation of variables. (a) pressure drop growth as a function of stem opening, (b) velocity drop growth as a function of stem opening.

Table 4. Simulation results in Ansys Fluent.

| Opening in obstruction (mm) | Inlet velocity (m s⁻¹) | Inlet pressure (kPa) | ΔVelocity (m s⁻¹) | ΔPressure (kPa) | Output velocity (m s⁻¹) | Minimum output parameter NTC 1500 [8] |
|---------------------------|------------------------|---------------------|-------------------|----------------|-------------------------|--------------------------------------|
|                           |                        |                     |                   |                |                         | Flow rate (L s⁻¹) | Pressure (kPa) |
| 0                         | 3.10                   | 242.00              | 1.40              | 236.62         | 1.69                    | 0.19                  | 5.00          |
| 3                         | 3.10                   | 242.00              | 1.43              | 236.67         | 1.66                    | 0.21                  | 5.38          |
| 5                         | 3.10                   | 242.00              | 1.36              | 236.84         | 1.73                    | 0.21                  | 5.33          |
| 7                         | 3.10                   | 242.00              | 0.88              | 139.91         | 2.21                    | 0.28                  | 102.09        |
| 10                        | 3.10                   | 242.00              | 3.10              | 242.00         | 0.00                    | 0.00                  | 0.00          |

Likewise, based on the previous data, the minimum pressure at which the fluid will leave the tap and the minimum flow rate at which the service will be obtained were tabulated, thus ensuring that it complies with the parameters of the NTC 1500 [8]. Next, Ansys Fluent evaluated the effectiveness of the simulation with the Yplus parameter. Figure 4 corroborates that the dynamic simulation of the continuous system complies with the established range.

Figure 4. Simulation of the surrounding fluid in the valve duct.

3.2. Static analysis of the valve
The unit deformations were calculated from the displacements, which show that the critical deformations are in the contact zone in the valve body inner conduit. On the other hand, the largest deformation is 2.461x10⁻⁵ mm and the minimum deformation value, which is distributed in most of the model, is negligible; the static simulations acquired from the isolation of the critical parts showed the effectiveness of the maximum unit deformation, Von Mises Max stress and minimum SF, as shown in Table 5.
Table 5. SolidWorks static simulation summary.

| Piece   | Max deformation (mm) | Von Mises max effort (Nmm\(^2\)) | Safety factor min |
|---------|----------------------|-----------------------------------|------------------|
| Stem    | 0.02                 | 23.67                             | 20.93            |
| Pedal   | 0.02                 | 2.69 x106                         | 10.58            |

4. Conclusions
The static study of the maximum deformations under critical working conditions according to the standards shown in this work provided high reliability in the design executed. At the same time, it was shown that the pedal has the lowest safety factor, which is an approved value for the application to be fulfilled in the design. On the other hand, the valve body shows the least uniform distribution of the applied stresses, however, it is congruent to state that the stresses do not generate a significant risk of fracture in the body structure. In the hydraulic analysis, it was demonstrated that the operation of this device fits the minimum parameters stipulated by Colombian technical standards since the minimum outflow of the tap is 0.21 Ls\(^{-1}\) and the minimum outlet pressure is 5.16 kPa being compared with the standard: 0.19 Ls\(^{-1}\) and 5 kPa respectively. Finally, it is plausible to mention that the use of this kind of device represents a hygienic alternative to the Covid-19 sanitary contingency, due to the low manual interaction in comparison with traditional taps.

Acknowledgments
To the Universidad Francisco de Paula Santander, Ocaña, Colombia, the INGAP research group of the “Programa de Ingeniería Mecánica” and the “División de Investigación y Extensión” for supporting the exploration of new technological advances.

References
[1] Sun X, Kim H S, Yang S D, Kim C K, Yoon J Y 2017 Numerical investigation of the effect of surface roughness on the flow coefficient of an eccentric butterfly valve J. Mech. Sci. Technol. 31 2839
[2] Mu Y, Liu M, Ma Z 2019 Research on the measuring characteristics of a new design butterfly valve flowmeter Flow Meas. Instrum. 70 101651
[3] Aabid A, Khan S A 2021 Investigation of high-speed flow control from CD nozzle using design of experiments and CFD Methods Arab. J. Sci. Eng. 46 2201
[4] Bhuwan D, Kumar K 2017 Computational fluid flow analysis of base valve for twin tube shock absorbers 5th International Conference of Materials Processing and Characterization (ICMPC 2016) ed Swadesh K (Rachi: Materialsstoday) p 95
[5] Mohamed A, Kchaou H, Abid M S, Driss Z 2018 Impact of stern design on hydrodynamic drag of auv’s hull Indian J. Geo-Marine Sci. 47 89
[6] Dong X, Zhang Z, Liu D, Tian Z, Chen G 2018 Numerical investigation of the effect of grids and turbulence models on critical heat flux in a vertical pipe Front. Energy Res. 6 58:1
[7] White F M 2011 Introduction to Fluid Mechanics, 8th edition (New York: McGraw-Hill)
[8] Instituto Colombiano de Normas Técnicas y Certificación (ICONTEC) 2004 Código Colombiano de Fontanería, Norma Técnica Colombiana, NTC 1500 (Colombia: Instituto Colombiano de Normas Técnicas y Certificación)
[9] Cekus D, Gnatowska R, Kwiatoń P, Šofer M 2019 Simulation research of a wind turbine using SolidWorks software J. Phys. Conf. Ser. 1398 012001:1
[10] Zhou X M, Wang Z K, Zhang Y F 2017 A simple method for high-precision evaluation of valve flow coefficient by computational fluid dynamics simulation Adv. Mech. Eng. 9 1
[11] Li Z, Zhu H, Zhang J 2020 Design and online calibration methods of pressure-independent intelligent regulating valve based on hydrodynamic resistance characteristics Energy Build. 224 110227
[12] Ansys, Inc 2009 Ansys (Pensilvania: ANSYS, Inc)
[13] Xu Y, Jiang F, Chen J, Li S 2018 Design and optimization of the faucet flow channel structure Mech. Mach. Sci. 55 595
[14] Zou Y, Zhao X, Chen Q 2018 Comparison of STAR-CCM+ and Ansys Fluent for simulating indoor airflows Build. Simul. 11 165
[15] Fan J, Njuguna J 2016 An Introduction to Lightweight Composite Materials and Their Use in Transport Structures (Cambrige: Woodhead Publishing)
[16] Mahakul R, Nath Thatoi D, Choudhury S, Patnaik P 2019 Design and numerical analysis of spur gear using SolidWorks simulation technique Mater International Conference on Recent Advances in Mechanical Engineering Research and Development (ICRAMERD-20) ed Seshadev S (Odisha: Materialstoday) p 145

[17] González O, González C, López A 2019 Introducción a análisis de elemento finito: SolidWorks y MATLAB Ideas en Ciencias la Ing. 1 28

[18] Mott R L, Vavrek E M, Wang J 2018 Machine Elements in Mechanical Design, 6th edition (New York: Pearson Prentice Hall)