JURNAL POLIMESIN
Unit Penelitian dan Pengabdian Kepada Masyarakat (P2M)
Politeknik Negeri Lhokseumawe
ISSN Print: 1693-5462, ISSN Online: 2549-1199
Website: http://e-jurnal.pnl.ac.id/index.php/polimesin

Geometry improvement and flow simulation in the water level control valve based on the CAD/CAE and DOE integrated system

Nguyen Huu Tho, Phan Hoang Phung, Huynh Van Nam and Nguyen Vu Anh Duy*
Faculty of Mechanical Engineering and Technology, Ho Chi Minh City University of Food Industry (HUFI)
140 Le Trong Tan st., Tay Thanh Ward, Tan Phu Dist., Ho Chi Minh city 700000, Vietnam
*corresponding author: duynva@hufi.edu.vn

Abstract

Floating valve is a type of valve which can open and close automatically based on the level of water in the tank by Archimedes force without using electric-based control signals. However, there are no engineering reports of the influences of factors to the liquid flow through the valve with the considerations of multiple objective optimization based on design of experiments (DOE) in the context of our country until now. Many products in the domestic market have been paid more attention by the customers but still have several disadvantages such as the practical malfunctions of floating valve. Thus, this study presents an integrated approach of CFD simulation and DOE-based multi-criteria optimization for proposed valve design. This technique allows to identify the influencing factors to the valve's goals of volume flow rate and the force which exerts into the rubber rubber to create a gap distance in order that the float valve can operate. In this study, several factors used as inlet pressure, a gap distance of rubber sheet, pressure vent hole diameter and orifice diameter. CFD (computational fluid dynamics) based simulation technique integrated into experimental design assists us to determine the most suitable dimensional and operational parameters of water level-control float valve. The numerical simulation results confirmed that design valve are appropriate and potential in the practice. The integrated simulation framework can be considered a general and effective way to analyze several types of other valves.

Keywords: level-control floating valve; fluid flow simulation; CFD; volume flow rate; CAD/CAE; DOE.

1. Introduction

Vietnam is a developing country and being in process of industrialization and modernization. Therefore, the mechanical engineering occupies a very important position for the development of industry. Product design and development occupy a very pivotal position in mechanical production, contributing to innovation and creativity, improving the competitiveness of businesses. Application of CAD/CAE system (Computer Aided Design/Computer Aided Engineering) to support the design and technical analysis process helps to reduce product time, further reduce technical risks and production costs. The thriving plastics, home appliances, and other supporting industries... need to apply the most advanced prototyping techniques to meet high production demands and to be economically efficient. Plastic products are diverse in size, shape, and complex designs to meet the needs of customers. CAD/CAE is capable of reducing the complexity of the design process as well as managing the product geometry dimensions at many different levels. [1].

Under the pressure of market globalization and international competition in the 21st century, manufacturing firms should strive to compete effectively by reducing product-to-market time and manufacturing cost while remaining ensure high quality product and service. The ability to respond quickly to business opportunities is considered as an important factor to ensure the dramatic competitiveness of businesses. Additive manufacturing has highly potential to reduce cycle time and cost for product development, being considered as a tool for in the modern digital manufacturing era for effective and rapid product development [2]. Additive manufacturing is a new technology, capable of shaping extremely complex surfaces, meeting the needs of the design and the aesthetics of the product. In particular, the method of material accretion has been successful in creating plastic products and invested in modeling in small and medium enterprises (SMEs).

Finite Element Analysis (FEA) has been developed over the past decades as a very useful tool for design analysis, evaluation, and improvement. Advances in the development of cheap and efficient computer technology, and the adoption of a user-friendly finite element method (FEM), has brought this technology forward. Producing better quality, reducing production costs and rapidly distributing products, has now become the three main targets for the manufacturing industry. To achieve this, manufacturing companies must adopt strategies based on new technologies. Strategies and technologies still need to be developed to reduce manufacturing costs and shorten production times associated with product development. FEA applications with the aim to better understand product properties in terms of hardness, durability, and potential failure [3].

CAD/CAE simulation provides a significant benefit on cost and time by providing a detailed portfolio into the product model development process before tool selection and manufacturing decision-making. Process data such as material deformation, pressure and temperature are easily collected by the user during the simulation, as well as at any location during the operation. The simulation results are obtained through comparison with experimental data based on the industrial standards. This method can be used effectively to optimize fabrication to increase mechanical strength, minimize scrap, and thereby reduce total production costs. CAD is the use of computers to draft the designers' ideas, the ideas are synthesized and drafted and modeled in 3D space, a CAD design software usually has 3 main modules such as 3-dimensional modeling. CAD through inter-process communication to perform tasks such as updating models, calculating geometrical properties, exporting geometry for mesh creation, tracking changes in shape, etc. CAE uses a computer system to analyze CAD geometry objects, thereby optimizing the CAD design process [4]. CAD/CAE system is applied next to CAD process to analyze, evaluate product quality, forecast risks, defects in the manufacturing process. There is currently only one scientific paper optimally analyzing the structure of valves serving the oil and gas drilling process with the assistance of computational fluid dynamics (CFD) [5].

In fact, at the current manufacturing factories, many CAD/CAE systems have not been applied in the design, simulation, and analysis process of plastic products, leading to time consuming in the process of product design and development. Using this system, it is also possible to determine the optimum product geometry for goals such as light weight, good resistance to tensile and compression [6]. Technological development of CAE has become commonplace in many industrial applications, and is essential to improving global competitiveness. However, although CAE technology is required, design engineer accessibility and analysis also takes a long time to train [7]. Therefore, the construction of the CAD/CAE system application process for the fabrication process by the additive method is extremely necessary to solve potential jobs. The formation of this process will be very suitable for many small and medium enterprises in manufacturing sector.

During the past few years, we have always spent a lot of effort and time in communicating and solving complex design optimization problems. Moreover, the best results obtained for complex problems are only approximate, macroscopic, discrete, and symbolic. Nowadays, with the development of information technology, mechanics has had effective support tools to overcome the above disadvantages. It is a computer-simulating technique of mechanical phenomena based on CAD/CAE software such as ABAQUS, ANSYS, Nastran, Solidworks simulation, FloEFD, etc... This technique helps to solve design problems with high precision, micro, continuity and vivid visualization. On the other hand, computer simulation brings economic efficiency, faster than experiment, very meaningful in education, training and actual working process of engineers in the field of system analysis. Seeing the role of actively capturing and catching up with the world science level, CAE calculation automation program is one of the indispensable contents in the automation strategy in Vietnam, included in the research and teaching at many universities, establishing many research institutes and centers for applied computation research such as DASI Industrial Software Application and Development Center, CAD/CAM/CAE Center of the University of Transport, Hanoi University of Technology, Ho Chi Minh City University of Technology, the institute of mechanics, the institute of mechatronics... and encourage research and transfer of technology to serve social needs. In which, most notably Solidworks simulation, a very powerful finite element analysis program, is used for solving mechanical problems of deformed solid body such as determining stress, displacement, deformation of bar under load... The interaction between the two advantages of Solidworks' CAD/CAE system and Flow Simulations allows to handle very complicated design optimization problems. Therefore, this is the main goal of the topic in this study.

Currently, the float valve market in Vietnam is vibrant with diverse structures. Optimization of the valve structure to satisfy the flow and pressure requirements has not been implemented for the water-level control float valve. Studies of applying experimental design to determine the parameters of the sizes/dimensions and optimal operation of the float valve are still left behind. Therefore, this topic will be a gap bridge between theory and experiments to the optimal design of float valves. Research and development results are also the direction of interest and application for Bach Khoa Mechanical Engineering Company (Ho Chi Minh, Vietnam).

In optimum design, the method supports the technical designs, the objective function combined with the constraints by varying the set of design variables such as part's dimensions and characteristics of materials. The objective function and constraints are often derived from the physical problems of the engineering design. Solving the design optimization problem takes a lot of time, sometimes we cannot get the solution. Modeling complex mathematical equations is not the strength of design engineers who need quick solutions to implement ideas. Therefore, the integration of CAD and CAE systems to build the optimal solution for multi-criteria mechanical product design is a necessary issue to assist engineers and managers in making engineering decisions quickly.

From all the reasons, the objective of the study is to research and optimize the design of the floating valve structure based on the integrated CAD/CAE system. This is the necessary applied research to support decision-making in product design for mechanical engineers.

The experimental method can only provide limited data at predetermined positions, and the interaction between the valve structure and the flow characteristics is not revealed, so it is difficult to optimize the valve design according to the experimental results.

2. Related works

With the recent development of computational fluid dynamics (CFD), it is possible to perform simulation and representation of complex flow field and then help designers learn more interesting details about flow field. Hence, CFD simulations can be an effective solution for experimental methods of evaluating valve performance, and many attempts have been made with the help of different softwares.

CFD simulation is used to test valve dynamics. It shows excellent workability to adapt to different types of valves and their geometries. CFD simulation provides a useful tool for valve design, optimization, and analysis.

Zhou [8] has formulated a flow efficiency factor associated with valve energy efficiency, and an efficient method for evaluating this coefficient for the valve industry. They argued that the evaluation is based on less accurate theory, while carrying out experiments were costly in terms of time and equipment. Therefore, they suggested using CFD simulation to obtain a simple and accurate assessment of the flow coefficient of the valve. They built a CFD simulation that included valve cross sections, front and rear valve. Adaptive mesh strategy is applied to improve simulation accuracy for the determination of the total system pressure loss through the valve, the overall system pressure drop and the pressure drop in the pipe estimated through linear regression models. The simulation results were also tested experimentally by Zhou et al. [8] to accurately predict the flow coefficients of different valve types, to confirm potential models that could be used in industry.

Check valves are widely used as flow monitoring equipment in many industrial fields. It is designed to accurately monitor the volume flow rate. The parameter to evaluate the performance of valve is volume flow rate coefficient. An experimental method to calculate this coefficient is not really available because the valves work at high pressures. Therefore, the CFD simulation tool is very useful in evaluating the volume flow rate efficiency. Yevale and Gharge [9] presented a small overview of several studies using CFD to simulate valve flow, and ANSYS/Fluent application for simulation. The device's volume flow rate is a relative measure for the liquid flow to go through, describing the relationship between a pressure drop passing through a valve or other assembly hole and the flow rate. This coefficient C_v depends on geometry, flow pattern, pressure and temperature, etc. [2], and can be expressed as follows.

$$C_v = Q\sqrt{\frac{\delta}{\Delta P}} \quad (1)$$

where Q is the volume flow rate (gallon/min); δ is specific gravity (for water, $\delta = 1$); Δp is pressure drop (in psi).

Budziszewski and Thoren [10] presented CFD simulation study of safety pressure reducing valve to improve one-way valve model using RELAP5. They tested safety valves modeled with CFDs to define correlation environments among parameters deployed in RELAP5 to achieve more realistic results of the forces exerted in the pipeline system. CFD simulations are performed in ANSYS/FLUENT to compare the results between 2D and 3D CFD models to come up with improved design for valve. The safety valve in their project is a proportional valve. It starts to open at a set pressure of 31 bar (g) and is fully opened at a 10% excess pressure, i.e. 34.1 bar (g), where a maximum lift distance of 8.5 mm is reached. The movement of the spindle is determined by the different forces acting on it. In their project hydraulic forces, spring forces and gravity were considered.

Andhale and Deshmukh [11] examined the (unidirectional) ball valve design to improve productivity. They used CPVC (Chlorinated Polyvinyl Chloride) ball valve to study the flow through the valve at a relatively low pressure. CATIA V5 software is used to design a block model of the valve. ANSYS FLUENT 14.5 software is used to conduct CFD simulations to evaluate the working flow energy through the valve. Simulations with two boundary conditions such as inlet pressure 36 kg/cm^2 and 26 kg/cm^2, temperature 82 ℃ and flow direction normal to boundary.

Yang, Zhang, Liu and Hu [12] modeled and simulated flow characteristics in stop valves. Flow in this valve has a complex structure and nonlinear properties due to hydraulic phenomena and the interaction between solid mass structures and fluid occurring inside the valve. A three-dimensional numerical simulation was also given by the author to observe the flow patterns and measure the flow-through-valve coefficients, flow variations when different valve flow is used. The simulation results are also used in the design to reduce noise and improve valve efficiency for industry.

Many industrial engineering problems require consideration of the fluid-structural interaction during analysis and dynamic properties to improve performance and operation of the valve. Song, Wang, Park and Sun [13] studied safety valves subjected to springs to prevent overpressure in the pressure vessel. The authors have used CFD simulation to overcome the difficulties in determining the dynamic properties of the valve disc and the properties of the flow. The results of their studies confirmed the importance of simulation up to the design stage and demonstrated the availability and efficiency of simulating harmonic CFDs in analysis of safety valves under pressure caused by spring.

Del Toro [14] analyzed computational fluid dynamics for the butterfly valve yield factors, a type of valve commonly used in industrial applications to monitor the internal flow of compressable fluid and incompressable. When the valve opening angle is increased from 0 degrees (fully closed) to 90 degrees (fully opened), the liquid can more easily flow through the valve. Characteristics of valve performance factors such as pressure drop, hydraulic torque, flow factor, loss factor and torque factor are essential for the fluid system designer to solve the system requirements, to operate the valve properly and prevent permanent damage. Their study compared the experimental yield factors of a 48-inch butterfly valve using CFD in an incompressable fluid with a Reynolds number ranging from 10^5 to 10^6 for the middle open positions ($30 \div 60$ degrees). The CFD model can approximate common yield factors for butterfly valves. For lower valve angle cases ($10 \div 20$ degrees), the CFD simulations fail to predict the same values while larger angles ($70 \div 90$ degrees) give mixed results.

Eriksson [15] studied CFD on pump stroke in a pump check valve system. Pressure waves can be caused by valve operation, pumping, hose shutdown and pressure loss. Therefore, it is of interest to check the behavior of the centrifugal pump and the torsion check valve. An existing pump model will be improved by considering the losses. Their purpose is to simulate the pump stroke scenario both with 1-dimensional model (1D) and to analyze 3-dimensional CFD (3D) and compare the results. The main focus will be to see how the centrifugal pump rotates and how the torsion check valve closes.

Bandari [16] tested the flow control valve using CFD simulation for the water distribution system, especially at the pipe connections. They studied for fixed set flow control valves with volume flow rate from 5-6 liters / min. A ball check valve geometry is realized, in which the ball is used to stop the fluid return flow from the valve. The details of the dynamic changes in the pressure, the flow velocity in the valve are also conducted by simulation. Study of fluid properties describes the desired design and determines the flow structure in the valve. The results demonstrate the designed capacity of the valve and meet technical specifications.

Safety relief valves are well-established components that prevent catastrophic failure of the pressurization systems when abnormal operation occurs. However, only recent development in the CFD technique, the ability to predict the complex flow conditions occurring in valves has been carried out by many studies. Dempster and Elmayyah [17] presented the experimental and theoretical research applicable to safety discharge valves, designed for the refrigeration industry but extended to consider compressed air systems since air is the compressible fluid. The discharge flow rate and valve force are determined both theoretically and experimentally for different valve lift conditions and with respect to detailed flow conditions (pressure, temperature and Mach number) in the projected valve, guess by CFD technique. The CFD/FLUENT program has been used with a two-way symmetric RANS method using a turbulent flow model k-e to predict the highly compressible flow through the valve. The model has been confirmed by comparison with test measurements and predictive results showing good consensus, giving confidence in using CFD techniques to design and improve valves.

Computational fluid dynamics play an important role in investigating complex flow in a product. With the help of optimization algorithms, CFD-based optimization has been increasingly applied in product development to improve product design. Although this approach is increasingly maturing, the problem is that the CFD solver cannot correctly respond to design changes under the batch mode, resulting in inaccurate optimization and simulation results. Also, there is no dedicated work to handle physically invalid design points during optimization. Li, Cheng and Lange [18] proposed an intelligent CFD solver that is used to analyze the flow at each design point and to establish the solver with the most suitable simulation models. An intelligent CFD-based optimization system is proposed to conduct design optimization with automatic invalid design point removal. This system is able to analyze the various design points identified by the DOE (surface response method) and to configure the CFD solver with suitable simulation models. Based on exact simulation results, invalid design points are automatically removed from the design space. Metamodeling is used to process the valid design space with the simulation results provided by the smart solver tool and give the optimal results. Optimization of the design of the steam control valve is used as a case study to demonstrate how the proposed system performs. Optimization is conducted based on the meta-model constructed by the response surface model and radial base function to verify the effectiveness of the proposed method. Furthermore, many of the latest studies have successfully combined CFD simulation and experimental planning to optimally design the geometry of valves, flows, and hydraulic machines using automotive electric pump shown in research [19, 20]. In addition, Nikhil [21] also applied CFD simulation method combined with numerical method to model 3D flow of check valve in boiler. They used the Solidworks Flow Simulation tool to understand the nature of the internal flow and to determine the flow efficiency coefficients such as pressure drop, the efficiency coefficient of valves subjected to high water pressure. In addition, Li et al. [22] and Olovetti et al. [23] also studied the valve geometry optimally and simulated CFD flow through the valve.

3. Design of water level-control floating valve.

3.1 Operational principle

Figure 1 shows the principle of the float valve. The flow of water with volume flow rate Q_v, pressure p_v enters the pipe has cross section A_v on the left side, then the water flow will pass through the hole of diameter $Ø_1$, creating pressure to push the rubber sheet down to open the door distance for moving water into the tank with pressure pr, volume flow rate Q_r to a certain level. When the water reaches a certain level, the water will push the float with Archimedes thrust, close a hole with a diameter of $Ø_2$ and push the rubber sheet that closes the door (zero distance). At that time, the water level in the tank is reached as the initial setting. Currently, current float valves are usually $Ø_1$ about 0.8 mm and $Ø_2$ about 1.5÷2 mm, with the water inlet pressure is quite low but not stable. The problem is that if you want the valve to operate stably or to withstand the maximum inlet pressure, what is the size of the holes $Ø_1$, $Ø_2$ and the diameter of the float d_3 to satisfy the output volume flow rate Q_r at the same time is most appropriate.

The volume flow rate enters through the valve's input pipe, the fluid passes through the mesh to create a steady flow, into the valve's inner chamber. The fluid fills the inner chamber. Now there is the pressure difference because the pressure under the rubber sheet is smaller. The pressure difference Δp above and below the rubber pad creates a thrust to push the rubber sheet to move downwards, creating a gap for fluid to overflow and into the outlet pipe to enter the tank. According to Bernoulli equation, the pressure exerted on the upper surface of the rubber sheet will change with the flow through the valve, and at the same time, the rubber sheet will move downwards to create a corresponding gap. The rubber sheet will remain in a stationary state until a new equilibrium of force is reached. As a result, the float valve is fully open to let the flow of flow through.

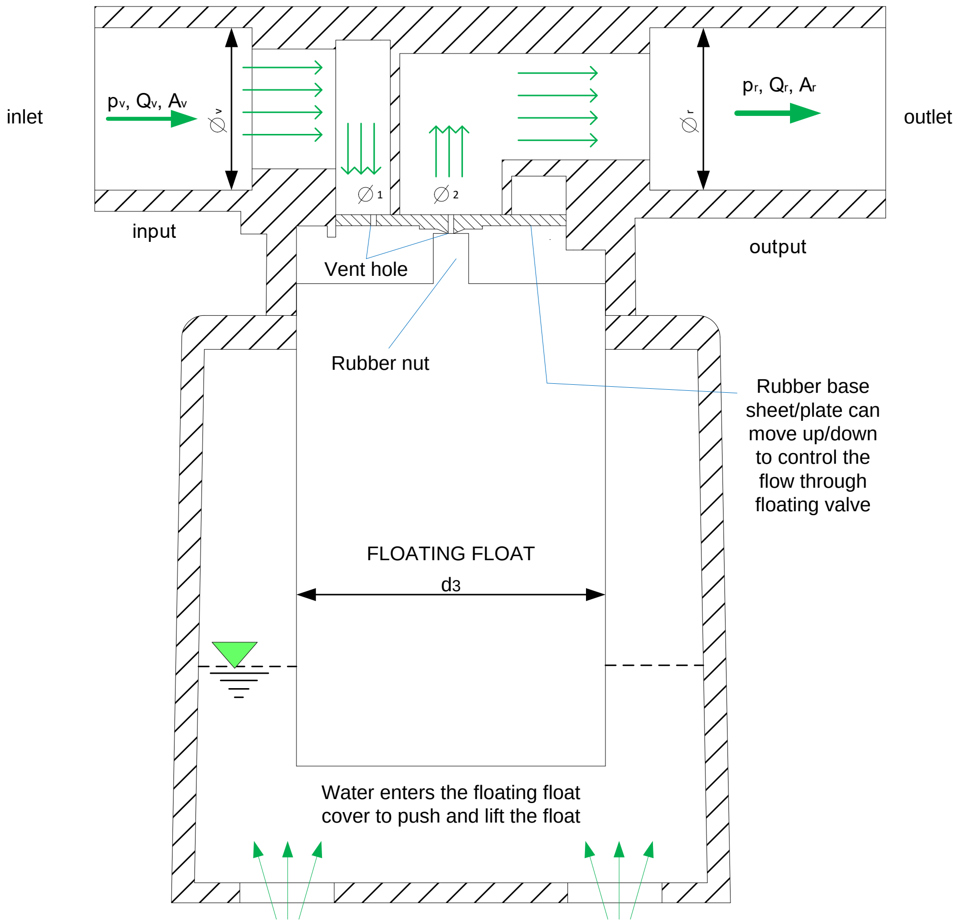

Figure 1. Water level-control floating valve

When the tank is full of water, it will lift the float. The valve opens and closes thanks to the float, operating due to the effect of Archimedes thrust, the float will exert force on the swing-arm to close the pressure vent hole. The pressure vents are closed, causing the pressure in the inner cavity under the rubber sheet to increase until it equilibrates with the pressure on the top. When the pressure above and below the rubber sheet is balanced, the spring creates an instantaneous push the rubber nut up, closing vent hole to prevent fluid from flowing through the valve. Because the bearing area on the lower surface is larger than the bearing area of the upper surface of the rubber sheet, a stronger and more durable force will form to close the gap and the fluid cannot continue to flow into the tank. At that time, the valve is closed fully.

3.2 Primary design

Pump has a power P_0 (kW) and its efficiency as shown in Eq. (2), so we calculate the pump power which supplies more energy into the flow inside the pipeline through the float valve.

$$\eta = \frac{P}{P_0} \tag{2}$$

where P_0 is a pump power and P is a power supplying to the flow into the pipeline.

Exerted force on the rubber sheet (follow the formula of Robert L. Mott, 2014) as shown in Eq. (3).

$$F_y = \rho Q \Delta v_y = \rho Q v_2 = \rho Q\left(\frac{Q}{A_2}\right) = \frac{4\rho Q^2}{\pi d_2^2} \tag{3}$$

Force F_y push the rubber sheet to move down in order to open the valve. It plays as a piston moving down to make a gap distance y.

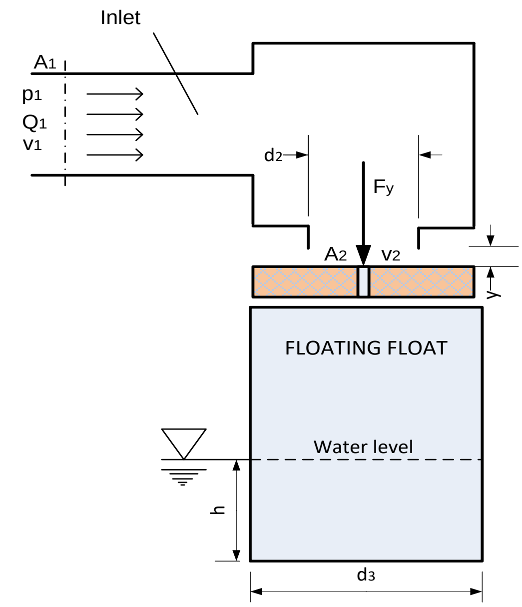

Figure 2. Principle of float valve

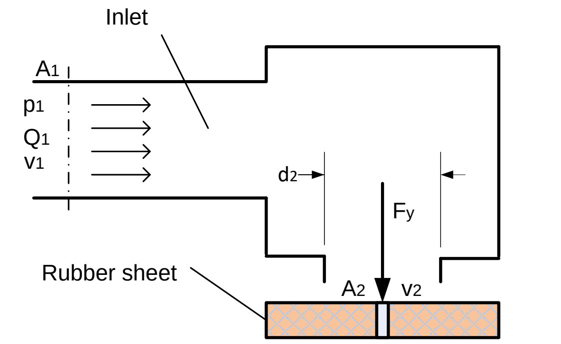

Figure 3. The rubber sheet moves down gererating the gap distance *y*.

Force F_y is assumed as concentrated force, average value, exerts on the rubber sheet which is considered as a rigid body. Rubber sheet moves up/down to create the gap and is considered as a spring with its stiffness *k*. The spring has total stiffness *k* is considered as a synthesis of paralleled two springs. In particular, one is represented as elastic rubber sheet and another is represented for a spring supporting the more force to close the float valve.

According to Hooke's law, gap distance y of the rubber sheet is determined as shown in Eq. (4).

$$y = \frac{F_y}{k} = \frac{4\rho Q^2}{\pi k d_2^2} \qquad (4)$$

Thus, if the gap reaches the value y_{max}, the area around the gap is large and the flow flows faster, releasing high pressure concentrated in the float valve. Diameter d_2 depends on the structure of the float and is the main dimension parameter. If the gap reaches the value y_{max} desired by the designer, then the diameter d_2 may be determined in Eq. (5).

$$d_2 = \sqrt{\frac{4\rho Q^2}{\pi k y_{max}}} \qquad (5)$$

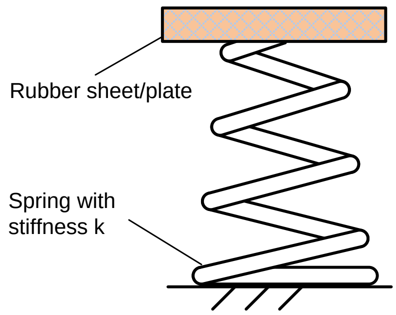

Figure 4. Principle of spring with stiffness *k*

When the gap y is reached, the fluid enters the valve and exits the outer pipe to enter the tank. Part of the fluid, because of the high pressure, passes through a small opening (due to the differential pressure). When the water in the tank rises, flooding all or part of the float volume will generate the Archimedes thrust to push the float to close the gap of vent hole (diameter is about 0.5 mm ÷ 1.0 mm). When the vents are closed by the swing-arm, the pressure is then balanced on the upper and lower sides of the rubber sheet. As a result, the thrust of the spring closes the orifice hole, and the fluid does not continue to enter the tank. When the rubber pad is closed, the area of the upper part is smaller than the bottom of the pad with the same balanced pressure, so the force from the bottom is greater and the rubber sheet is tightly closed.

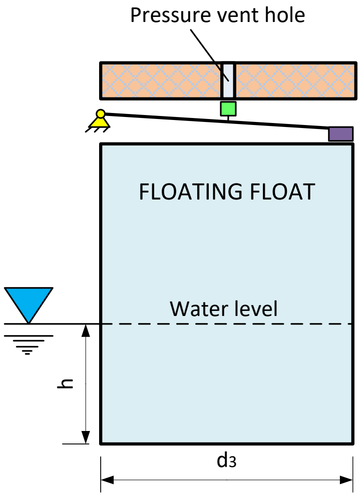

Figure 5. The principle of closing the float valve

Archimedes thrust is determined in Eq. (6):

$$F_A = \gamma V = \frac{\pi \gamma h d_3^2}{4} \tag{6}$$

For the float to close the pressure vent, we need Archimedes F_A force greater than the F_y force if we don't use the swing arm. If using a swing arm, $0.5F_A$ should be greater than the F_y force.

$$\frac{F_A}{2} \geq F_y \Rightarrow \frac{\pi \gamma h d_3^2}{8} \geq \frac{4\rho Q^2}{\pi d_2^2}$$
$$\Rightarrow d_3 \geq \frac{4Q\sqrt{2}}{\pi d_2 \sqrt{gh}} \tag{7}$$

Therefore, we need to choose the float diameter d_3 to satisfy Eq. (7), d_3 depends on the flow Q (m^3/s), diameter d_2 (m), and the flooding level on the float h (m).

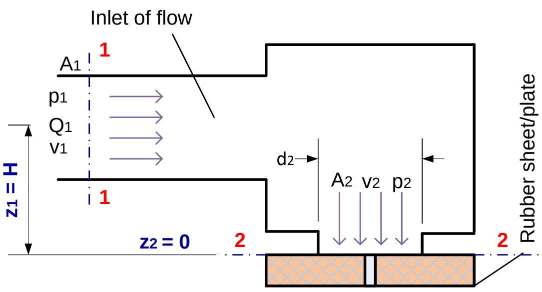

Figure 6. Float valve is fully closed due to rubber sheet under Archimedes force

Since there is a steel mesh sheet, the inlet flow is a laminar flow pipe, it can be approximated to simplify the engineering problem that the energy loss (local loss) is due to sudden open section is considered insignificant (h_L = 0). From there, applying Bernoulli's equation for the liquid flow between cross-section 1-1 and 2-2, we obtain Eq. (8).

$$\frac{p_1}{\gamma} + z_1 + \frac{v_1^2}{2g} - h_L = \frac{p_2}{\gamma} + z_2 + \frac{v_2^2}{2g} \tag{8}$$

Average pressure p_2 as determined in Eq. (9):

$$p_2 = \frac{F_y}{A_2} = \frac{16\rho Q^2}{\pi^2 d_2^4} \tag{9}$$

Thus, we have formula which describes the relationship of p_1, d_1, d_2, and Q as shown in Eq. (10).

$$\frac{p_1}{\gamma} + H + \frac{\left(\frac{4Q}{\pi d_1^2}\right)}{2g} = \frac{\frac{16\rho Q^2}{\pi^2 d_2^4}}{\gamma} + \frac{\left(\frac{4Q}{\pi d_2^2}\right)}{2g}$$
$$\Rightarrow \frac{p_1}{\gamma} = \frac{\frac{16\rho Q^2}{\pi^2 d_2^4}}{\gamma} + \frac{\left(\frac{4Q}{\pi d_2^2}\right)}{2g} - \frac{\left(\frac{4Q}{\pi d_1^2}\right)}{2g} - H \tag{10}$$
$$\Rightarrow p_1 = \frac{16\rho Q^2}{\pi^2 d_2^4} + \frac{2\rho Q}{\pi}\left[\frac{1}{d_2^2} - \frac{1}{d_1^2}\right] - \gamma H$$

From the Eq. (10), if the desired inlet pressure is measured, or if we want the inlet pressure p_1 and the volume flow rate Q to be a certain value, the diameters d_1 and d_2 can be completely determined to form a preliminary structure of float valves. After determining the value of d_1 and d_2, the value y_{max} and float diameter d_3 can also be determined.

3.3 CAD model of floating valve

The plunger has an arrow direction showing the inlet and outlet. The inner valve structure is designed to limit local energy loss when flow through the valve. The thin, light float shell structure has holes for pressure and the water level in the tank rises to push the float to close the valve with Archimedes force. This part has a vertical axis to guide the float so that the float moves up and down, evenly and without being skewed or stuck on the float. The base plate has a vent hole of about 0.5 mm in diameter to open the inside of the hose. This base plate will be fitted tightly on the underside of the hose, creating a chamber when the flow passes. This lever receives the Archimedes force from the float when the water level rises. The cavity inside the valve structure was equal to the pressure, the spring force pushed the rubber base plate to close the valve, not allowing the flow to pass through. Springs are inserted into the inside of the valve, acting as a thrust, supporting the process of switching flow through the valve. This rubber sheet acts as a piston moving up and down thanks to the force of the fluid, the spring's elastic force and its own resilience. The plastic bottom plate increases the stiffness of the rubber sheet. Therefore, the horizontal surface of the rubber sheet moving up and down can be considered as absolute stiffness during the calculation. The structure of the float valve after being assembled is illustrated in Figure 7.

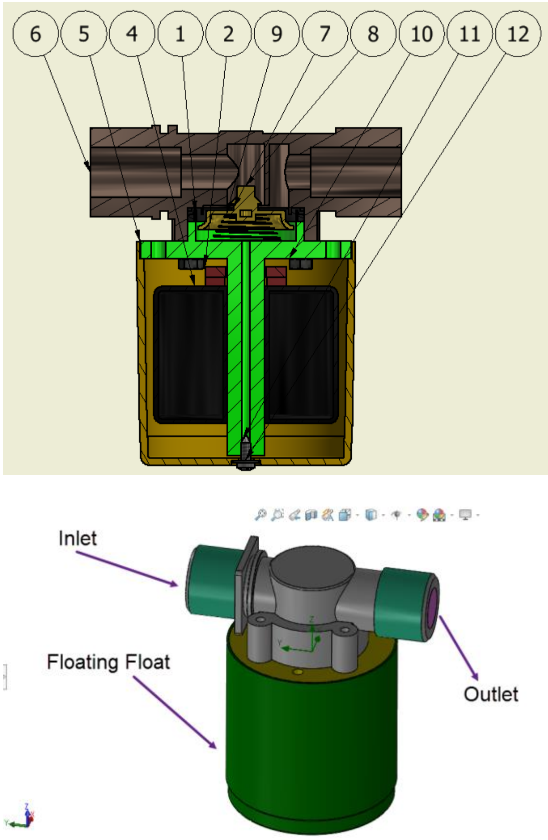

Figure 7. CAD model of the float valve (1. Spring locating shaft; 2. Tightening screw; 3. Shaft; 4. Floating float; 5. Valve seat cap; 6. Water pipe hose; 7. Plastic base plate rising hard; 8. Flow guide bolt; 9. Taper spring; 10. Screw; 11. Screw; 12. Washer)

3.4. Numerical simulation of CFD

The preliminary design is an important stage to build the CAD model for the float valve. Based on the reference structure of products of Bach Khoa Mechanical Company and some valves available on the market, this study uses Solidworks design, modeling, CAE tools using Solidworks Flow Simulation/FloEFD. to optimize the valve parameters with the target function of minimum force acting on the rubber sole and maximum flow rate at the outlet. After modeling and evaluation of CAE, the float valve drawing set was improved and finalized. Then, the research models the valve nozzle by material accretion method through geometrical data *.STL file. Finally, we carry out numerical and practical simulation experiments to test the proposed CAD model. Research procedure as described in Figure 8. CAE technical analysis demonstrates relationship and indicates valve behavior between liquid flow, pressure, and dimensional parameters. CAD/CAE systems such as Solidworks Flow Simulation/FloEFD will assist in evaluating the optimal structure based on the selected criteria.

In this study, float valves are simulated with boundary conditions used as follows: Inlet and outlet pressure constraints. The inlet pressure is the pressure entering the valve, according to manufacturers' inlet pressure from 0.1 to 10 bar. However, in the practice, the valve typically operates at a pressure of 4.137 bar (flow is about 0.32 liters/s) And a pressure of 2.758 bar (volume flow rate of about 0.189 liters/s) The maximum average temperature is in the range of 70 ÷ 120 °C. This study tests boundary conditions for inlet pressure in terms of 1,5; 3.0; and 4.5 bar. The outlet pressure is equal to the atmospheric pressure, since the water flows directly into the water tank. The pressure at the vent hole with the float assembly is assigned equal to the atmospheric pressure. Thus, we have a total of three boundary conditions that are applied to the CFD simulation problem for the floating float valve.

Next, we set up the Goal factor that needs to be monitored for the float valve. For valves, pressure drop and volumetric flow expressed as mean flow velocity are important for evaluating valve efficiency. Therefore, this study will analyze the mean velocity, and the distribution of pressure on the upper surface of the rubber sheet. Pressure distribution on the rubber sole plate surface is the basis of determining the thrust force that creates the gap between the rubber sheet and the outlet, and the basis for determining spring stiffness and reasonable rubber parameters. The implementation procedure with Solidwork Flow Simulation software is shown in Figure 9. Mesh generation is done by default to reduce computation time with acceptable accuracy.

Meshing for a calculation model depends on the computer configuration and the version used by the calculation engine. The finer the model mesh, the more accurate the results, the longer calculation time, can be up to several tens of hours if the machine configuration is not strong enough. In this study, we use a mesh resolution of 6 (Tools >>

Flow Simulation >> Global Mesh… slide the Result resolution ring to 6), setting 0.001 m for the Minimum gap size. In the Manual settings, set the Number of cells per X to 40, Number of cells per Y to 74 and similar for Z to 236. Click OK to finish meshing. After completing the meshing of the model, we run the simulation: select Tools >> Flow Simulation >> Calculation Control Options…, click the Refinement tab and hide the function from the Value menu. Click OK to finish. Select Tools >> Simulation >> Solve >> Run to calculate the model. Click Run and wait for results.

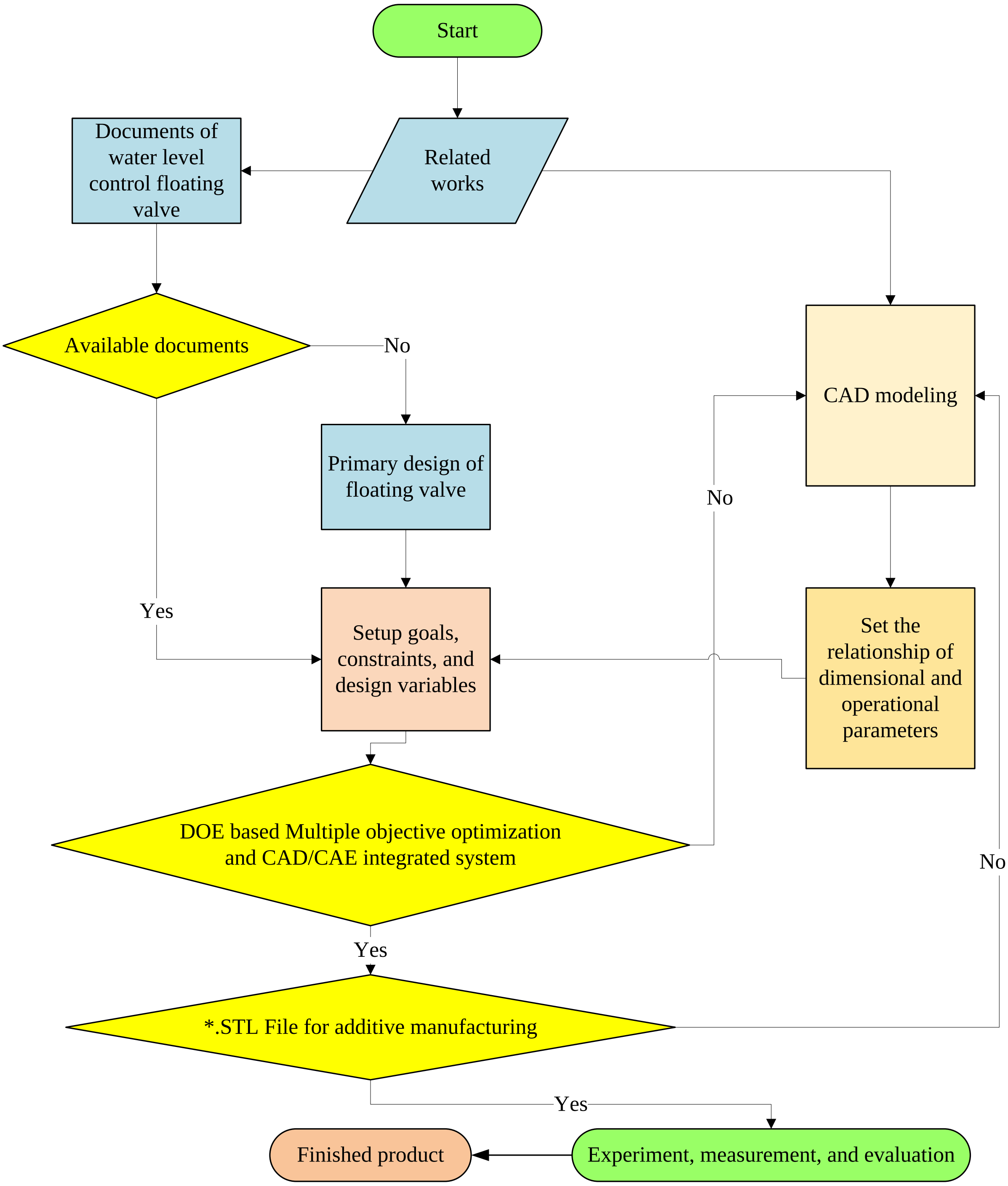

Figure 8. Research Flowchart

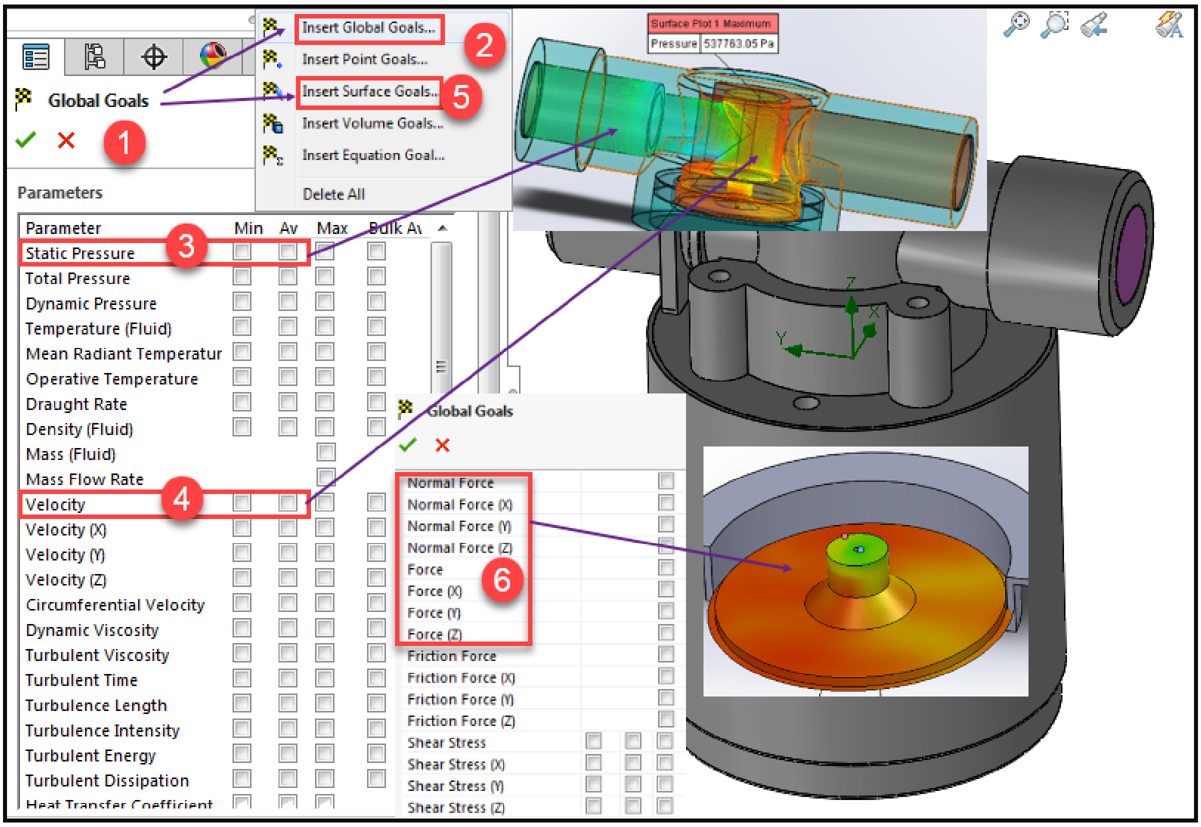

Figure 9. Goals of monitoring the flow through the floating valve

4. Results and discussion

Optimization based on a multi-variable response surface extends from the single-objective optimization problem (one dimension). It allows us to define simulation parameters and geometries and optimize performance (maximum, minimum or target value) of the objective function. The objective function can be defined as a single goal, or a combination of items with weights. The approach to achieve the optimal solution is called design of experiments. Figure 10 presents the solutions of multi-response surface based on DOE. In particular, Experiment 4 (distance d_1 = 3.56 mm, static pressure of 2 bar, SG Force Y = 24.65 N) is considered the most appropriate solution for representation of float valve model. Figure 11 shows the maximum and minimum pressures distributed into the flow through the float valve. The maximum pressure is 205472.82 Pa (larger than 2 bar), the minimum pressure is 62627.08 Pa (about 0.62 bar). Figure 12 show the maximum and minimum velocity in the flow through the float valve. The maximum velocity is 14.708 m/s and the minimum velocity is 0.044 m/s. Figure 13 presents the distributed pressure exerted on top face of the rubber sheet, the maximum and minimum pressures are 200645.49 Pa and 48579.95 Pa, respectively. Figure 14 describes the relationship between force exerted on rubber sheet and CPU time, it takes about 280 seconds.

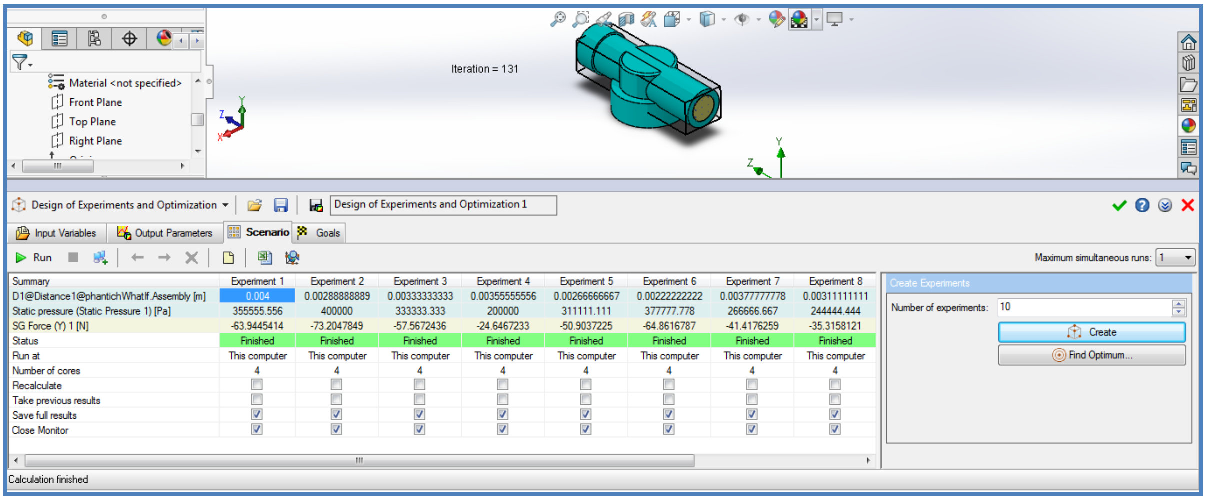

Figure 10. Solutions of multi-response surface based on DOE

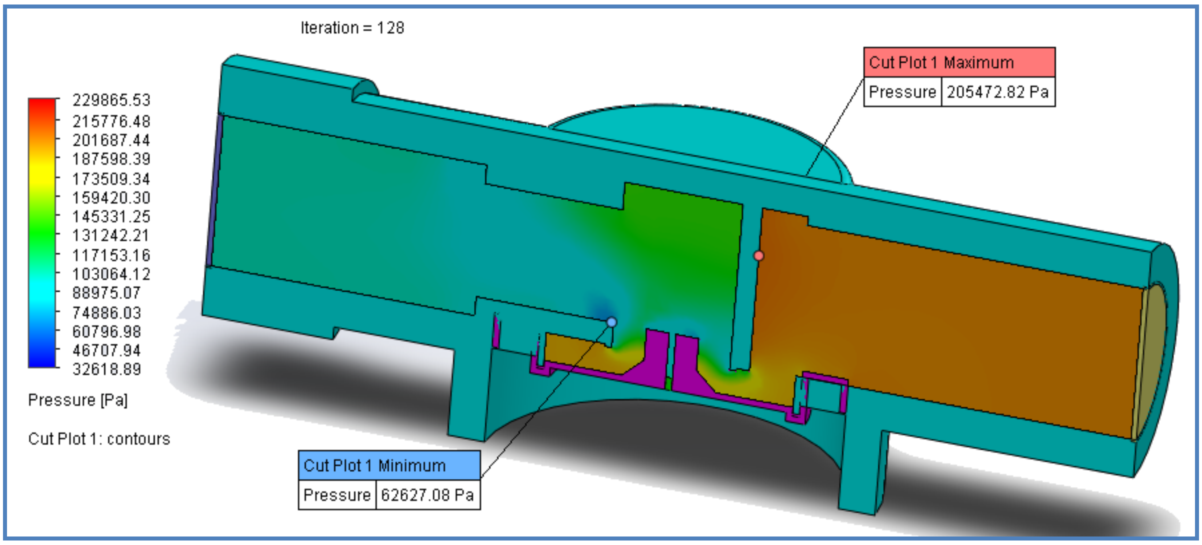

Figure 11. Distributed pressure into the flow of Experiment 4 (min pressure of 62627 Pa)

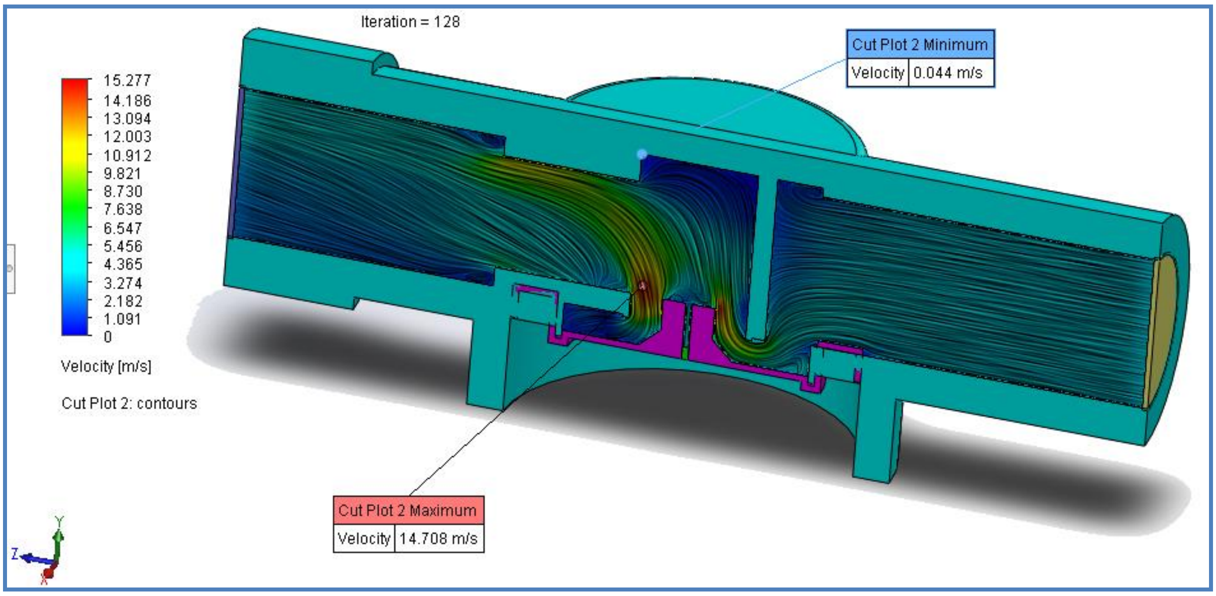

Figure 12. Velocity field in the flow of Experiment 4 (max velolity of 14.7 m/s)

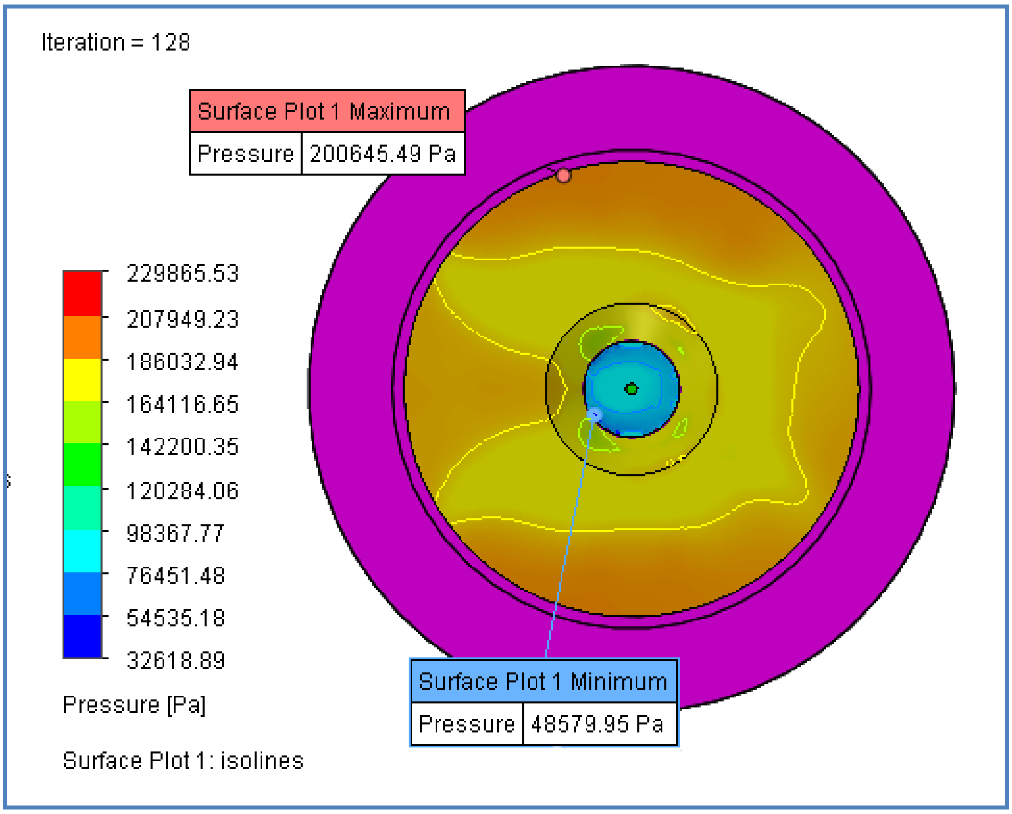

Figure 13. Distributed pressure exerted on top face of the rubber sheet

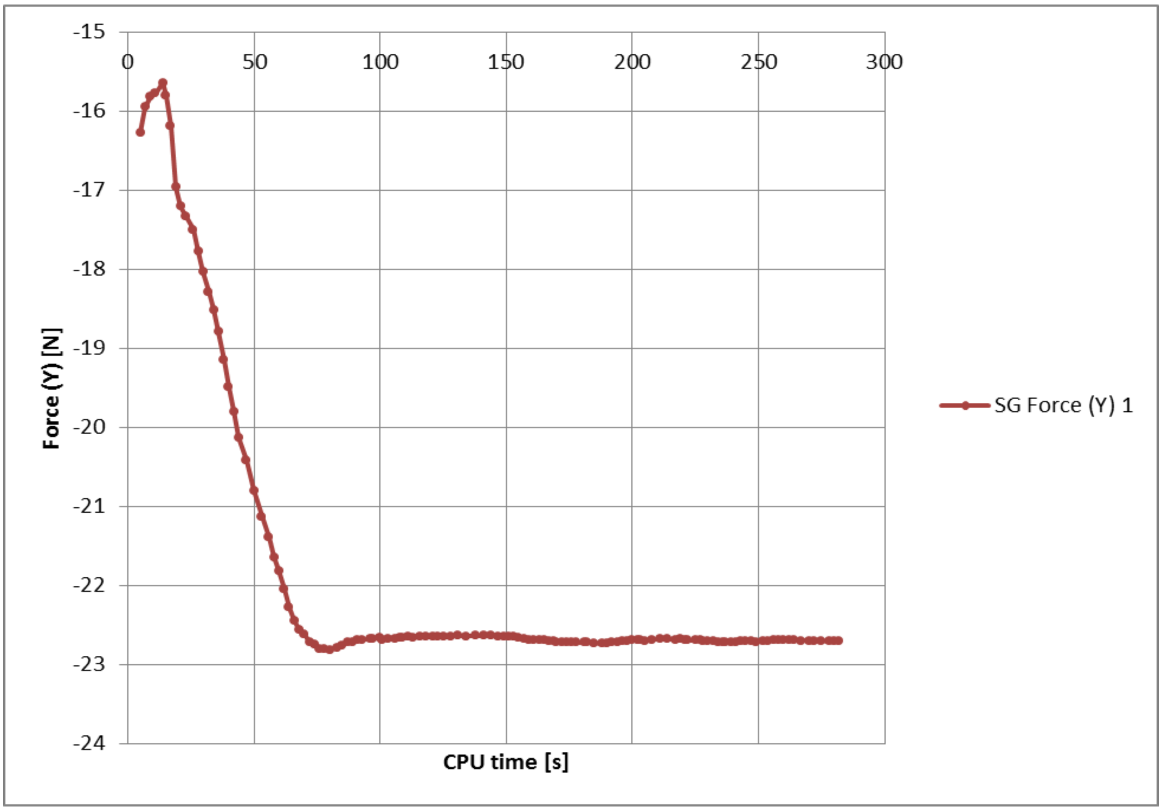

Figure 14. Relationship between force exerted on rubber sheet and CPU time

Now, based on the float valve structure, to understand the behavior of inlet pressure (p_1 = 2 bar ÷ 4 bar), the gap of rubber sheet (y = 2 mm ÷ 4 mm), the applied force on rubber sheet (this is an important but unknown quantity), orifice diameter d_2 (mm), and vent hole diameter. We considered nine more numerical experiments based on the 3-level change of the vent diameter (Ø0.5 mm; Ø0.75 mm; Ø1.0 mm) and 3 levels of the orifice diameter (Ø8 mm, Ø9 mm, and Ø10 mm). Each experiment consists of 10 sub-experiments. Therefore, a total of 90 sub-experiments were performed to investigate the changes of 4 parameters for the valve simulation, which are valve dimension parameters, geometry parameters and boundary conditions. Many cases of CFD simulation for float valve are summarized: Case 1: Pressure vent hole Ø0.5 mm, orifice diameter Ø9 mm. In this case we have surveyed above; Case 2: Pressure vent hole Ø0.75 mm, orifice diameter Ø9 mm; Case 3: Pressure vent hole Ø1 mm, orifice diameter Ø9 mm; Case 4: Pressure vent hole Ø1 mm, orifice diameter Ø10 mm; Case 5: Pressure vent hole Ø1 mm, orifice diameter Ø8 mm; Case 6: Pressure vent hole Ø0.75 mm, orifice diameter Ø8 mm; Case 7: Pressure vent hole Ø0.5 mm, orifice diameter Ø8 mm; Case 8: Pressure vent hole Ø0.75 mm, orifice diameter Ø10 mm; Case 9: Pressure vent hole Ø0.5 mm, orifice diameter Ø10 mm, in this case, pressure vent hole Ø0.5 mm and orifice diameter Ø10 mm: the force applied to the rubber pad reaches 21.84 N.

Based on 90 experiments of numerical calculation and CFD simulation for the float valve, we found that the good structure has pressure hole Ø1 mm, orifice diameter Ø10 mm, the pressure of 2 bars pushes the gap of rubber sheet y = 2 mm gives the minimum force applied to the rubber sheet, about 21.4 N. This is the best operating condition for a float valve. In addition, with a larger clearance y, we also find that some of the following cases are also a reasonable mode of operation for the valve as follows:

- Pressure vent Ø0.75 mm, orifice diameter Ø9 mm, clearance y = 3.56 mm, inlet pressure 2 bar, the force exerted on the rubber sheet reaches 24.58 N.

- Pressure vent Ø1 mm, orifice diameter Ø9 mm, clearance y = 3.56 mm, inlet pressure 2 bar, the force exerted on the rubber sheet reaches 24.50 N.

- Pressure vent Ø1 mm, orifice diameter Ø10 mm, clearance y = 3.56 mm, inlet pressure 2 bar, the force exerted on the rubber sheet reaches 24.33 N. With this mode, we It is found that the diameter of the pressure hole Ø1 mm has two most reasonable operating modes. This is considered the best case.

- Pressure vent Ø1 mm, orifice diameter Ø8 mm, clearance y = 3.56 mm, inlet pressure 2 bar, force exerted on the rubber sheet reaches 28.21 N.

- Pressure vent Ø0.75 mm, orifice diameter Ø8 mm, clearance y = 3.56 mm, inlet pressure 2 bar, the force exerted on the rubber sheet reaches 28.38 N.

- Pressure hole Ø0.5 mm, orifice diameter Ø8 mm, clearance y = 3.56 mm, inlet pressure 2 bar, the force exerted on the rubber sheet reaches 28.47 N.

- Pressure vent Ø0.75 mm, orifice diameter Ø10 mm, clearance y = 3.56 mm, inlet pressure 2 bar, the force exerted on the rubber sheet reaches 24.44 N.

- Pressure vent Ø0.5 mm, orifice diameter Ø10 mm, clearance y = 3.56 mm, inlet pressure 2 bar, the force exerted on the rubber sheet reaches 24.5 N.

So the larger the orifice diameter, the better it is and must be smaller than the inlet diameter. A larger vent diameter will reduce the force exerted on the rubber sheet, but if the vent is too large it will reduce pressure loss and flow energy loss, and thus affect the volume flow rate. In the above cases, we find that the most reasonable pressure vent hole diameter is Ø1.0 mm, the valve operates at an inlet pressure of 2 bar, orifice diameter of Ø10 mm, clearance y = 2 mm or 3.56 mm is the most reasonable.

Next, we will present more in depth about the results of flow characteristics through the float valve for case of p_1 = 2 bar, y = 2 mm or 3.56 mm, d_2 (orifice) = 10 mm, and d_O (pressure vent hole) = 1 mm. We built a flow characteristic model through the float valve and simulated with the fine mesh mode to observe the behavior of the valve by integration of DOE-based optimization and CAD/CAE Solidworks flow simulation. The maximum and minimum pressure in the water flow through the valve is respectively 206908.06 Pa and 63022.68 Pa. The maximum and minimum velocities of the flow are 14.305 m/s and 0.251 m/s, respectively. One of the most important parameters is the pressure exerted on the top face of the rubber sheet. In this case, the maximum mean pressure is 200803.04 Pa and the minimum is 87772.74 Pa. This will create great pressure to push the pad to move downwards to increase the clearance for the flow of passing the valve. The volume flow rate through the valve has an average value of 0.00043 m^3/s, converging after 91 iterations. The average applied force on the rubber pad was 22.45 N. If the clearance y = 3.56 mm then the volume flow rate through the float valve in the numerical simulation model is 0.0008651 m^3/s.

To test again, we conducted the CAD/CAE simulation for flow through the float valve with the following parameters: inlet pressure p_1 changes from 1.3 bar to 2 bar, gap distance y = 2 mm ÷ 3.56 mm, the value of the orifice diameter and the pressure vent hole were kept the same. The goal is to monitor the behavior of the force exerted on the top surface of the rubber sheet and the volume flow rate through the float valve through 20 digital experiments automatically generated via CAD/CAE system (Figure 15). Comparison between volume flow rate by numerical model and theoretical model is performed in Table 1. The unit of measurement is m^3/s which is used to measure the flow of liquid through the valve.

Table 1. Comparison of volume flow rate through the water level-based control floating valve

No.	**Inlet pressure (bar)**	**Volume flow rate Q_v (m^3/s)**	
		Numerical experiment	Analytical method
1	1,3 (y = 2,493)	0,0002435	0,0002435
2	2,0	0,00043	0,00043
3	2,5	0,00053	0,00053
4	3,0	0,00061	0,00061
5	3,5	0,00069	0,00069
6	4,0	0,00075	0,00075

The experimental results of computational simulation using DOE-based optimization integrated CAD/CAE system confirmed that the proposed float valve model is reasonable. Figure 16 presented three multiple criteria optimal solutions. Those are minimum, maximum, and target design points, responding to Optimum 2, 3, and 1, respectively.

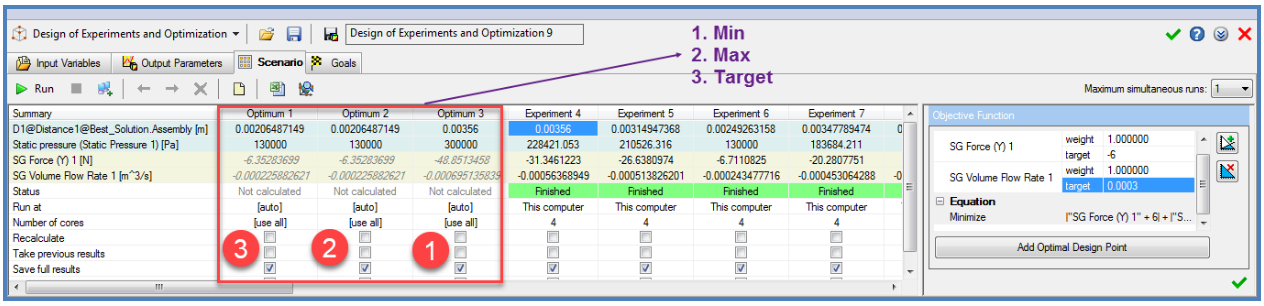

Figure 15. Implementation of more numerical experiments on DOE based optimization and CAD/CAE integrated system

Summary	RSM Errors [%]	Optimum 1	Optimum 2	Optimum 3
D1@Distance1@Best_Solution.Assembly [m]		0.00206487149	0.00206487149	0.00356
Static pressure (Static Pressure 1) [Pa]		130000	130000	300000
SG Force (Y) 1 [N]	0.51	-6.56582816	-6.56584769	-48.9518072
SG Volume Flow Rate 1 [m^3/s]	1.05	-0.000232315957	-0.000232312925	-0.00070536648
Status		Finished	Finished	Finished
Run at		This computer	This computer	This computer
Number of cores		4	4	4

Figure 16. Multiple criteria optimal solution using DOE-based optimization integrated CAD/CAE system

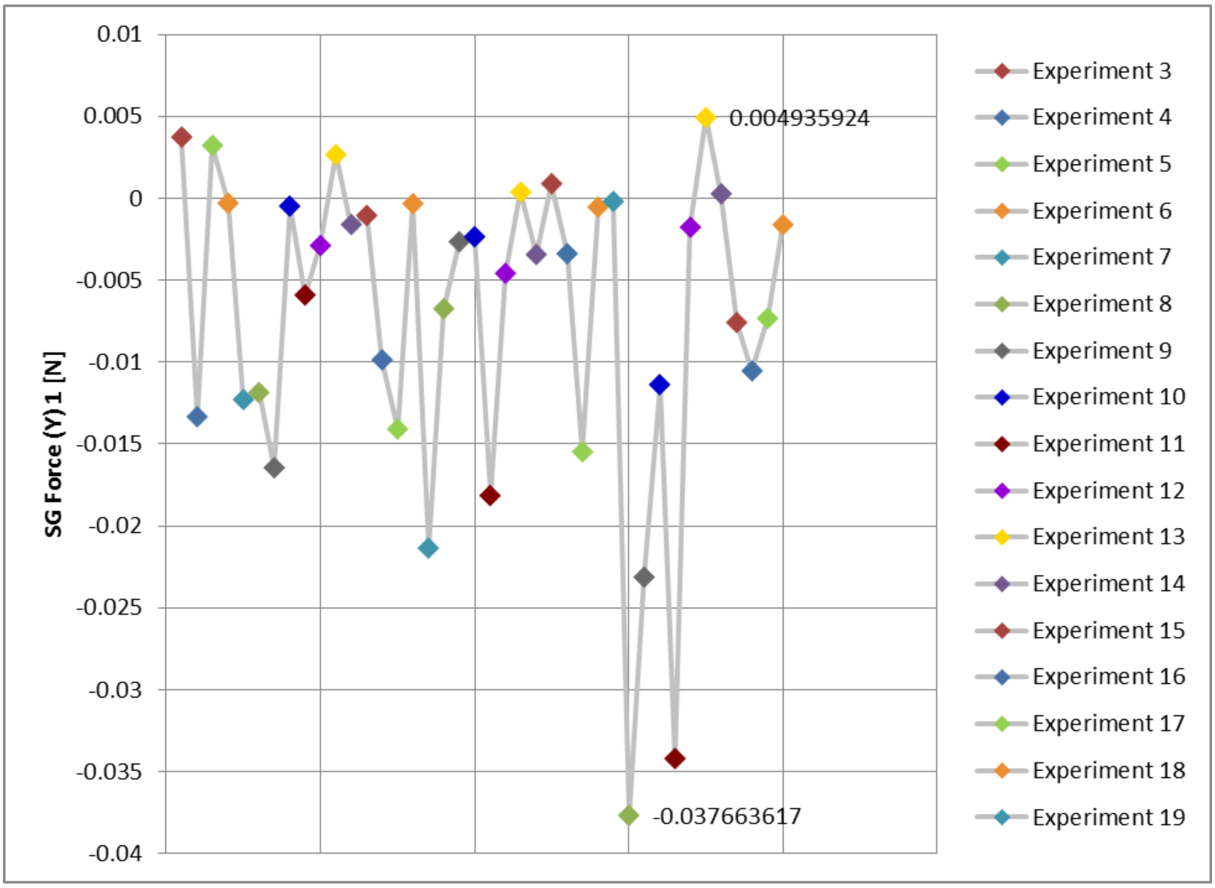

Figure 17. Convergence graph of exerted force on elastic rubber cushion vs. iterations

Based on the results of the float valve structure with the parameters: p_1 = 2 bar, y = 2 mm or 3.56 mm, d_2 (orifice) = 10 mm, and d_O (vent hole) = 1 mm, we proceed simulation with fine mesh mode to observe the force exerted on the pressure vent position in case of this hole closed by the lever from the float due to the Archimedes force. We implemented 40 numerical simulations with the pressure p_1 changing from 2 bar to 4 bar, the gap y changes continuously from 2 mm to 4 mm, the results show that the force exerted at the pressure hole is valid, the minimum value is 0.005 N and the maximum value is 0.038 N (see Figure 17). If the water flooding level of float is set to H = 20 mm (depending on the manufacturer/ supplier/ customer/ user), the float diameter shall have a value satisfying the Archimedes thrust greater than 0.038 N. From there, the float diameter is determined as below.

$$d_{\text{float}} = \sqrt{\frac{0.038 \times 4}{0.02 \times 9810 \times \pi}} = 0.016 \text{ (m)}$$

So, in order for the float to be moved upwards and close to the pressure vent hole, we need the float diameter to be greater than 16 mm. For the closure of the pressure vent, it is necessary to have an elastic rubber cushion to seal it, then we need the Archimedes force many times more. Therefore, it is possible to design a larger float diameter (d_{float} Ø42 mm) to ensure the ability to deform and close the pressure vent hole.

5. Conclusions

Study on the effects of the dimensions and operating parameters based on multi-criteria optimization is important when designing a float valve because it help us understand the valve's stable operating threshold to guide users. The results have identified the maximum force exerted on the rubber sheet, which will aim to help us choose the type of rubber and spring stiffness to improve and design with the goal of increasing the volume flow and reducing the force exerted on the rubber sheet. The effective parameters for valve operation are the most reasonable pressure vent hole diameter is Ø1.0 mm, the valve operates at an inlet pressure of 2 bar, orifice diameter of Ø10 mm, and clearance y = 2 mm or 3.56 mm are the most reasonable. At that time, the force exerted on the rubber sheet reaches 24.33 N. The proposed valve model is verified the correction with the previous result in the literature with the case of exerted force on rubber sheet of 6 N. In the future, this study will implement the simulation to identify the cavitation and use the simulation results to transfer the FEA loads into the structural analysis environment to analyze the material strengths and propose the optimum topology for float valve.

Acknowledgments

Authors would like to say many special thanks to Ho Chi Minh city University of Food Industry for their kind support of research fund for the annual research project with number 54/HĐ-DCT, signed on 03/09/2019.

References

[1] A. Mosavi, "Decision-making models for optimal engineering design and their applications," ed: *Doctoral Dissertation*, University of Debrecen, Hungary, 2013.

[2] H. Lan, "A web-based rapid prototyping manufacturing system for rapid product development," in *Collaborative design and planning for digital manufacturing*: Springer, 2009, pp. 245-264.

[3] M. Prabaharan, "Application of CAD/CAE, Reverse Engineering and Rapid Prototyping in New Product Development Industry," in *ASME 2013 International Mechanical Engineering Congress and Exposition*, 2013, pp. V015T16A026-V015T16A026: American Society of Mechanical Engineers.

[4] S. Yue, G. Wang, F. Yin, Y. Wang, and J. J. J. o. M. P. T. Yang, "Application of an integrated CAD/CAE/CAM system for die casting dies," *Journal of Materials Processing Technology*, vol. 139, no. 1-3, pp. 465-468, 2003.

[5] Y. Xu, Z. Guan, Y. Liu, L. Xuan, H. Zhang, and C. Xu, "Structural optimization of downhole float valve via computational fluid dynamics," *Engineering Failure Analysis*, vol. 44, pp. 85-94, 2014/09/01/ 2014.

[6] R. Rezaie, M. Badrossamay, A. Ghaie, and H. J. P. C. Moosavi, "Topology optimization for fused deposition modeling process," *Procedia CIRP*, vol. 6, pp. 521-526, 2013.

[7] Y.-C. J. J. o. M. P. T. Kao, "Development of a remote quick CAE system on sculptured metal extrusion die surface," *Journal of Materials Processing Technology*, vol. 140, no. 1-3, pp. 116-122, 2003.

[8] Zhou, X.-M., Wang, Z.-K., and Zhang, Y.-F., "A simple method for high-precision evaluation of valve flow coefficient by computational fluid dynamics simulation," *Advances in Mechanical Engineering*, 9(7), pp. 1-7, 2017.

[9] Yevale, S. R., and Gharge, K., "A Review on CFD Analysis of Control Valves," Gloabl Research and *Development Journal for Engineering*, 3(7), pp. 66-69, 2018.

[10] Budziszewski, A., and Thoren, L., 2012, "CFD simulation of a safety relief valve for improvement of a one-dimensional valve model in RELAP5," *Master Thesis*, Chalmers University of Technology, Gothenburg, Sweden.

[11] Andhale, V. A., and Deshmukh, D., "Investigation of ball valve design for performance enhancement," *Pratibha: International Journal of Science, Spirituality, Business and Technology* (IJSSBT), 4(2), pp. 105-112, 2016.

[12] Yang, Q., Zhang, Z., Liu, M., and Hu, J., "Numerical Simulation of Fluid Flow inside the Valve," *Procedia Engineering*, 23, pp. 543-550, 2011.

[13] Song, X. G., Wang, L. T., Park, Y. C., and Sun, W., "A Fluid-structure Interaction Analysis of the Spring-Loaded Pressure Safety Valve during Popping Off," *Procedia Engineering*, 130, pp. 87-94, 2015.

[14] Del Toro, A., 2012, "Computational fluid dynamics analysis of butterfly valve performance factors," *Master Thesis*, UTAH STATE UNIVERSITY, Logan, Utah.

[15] Eriksson, E., 2016, "CFD study of a pump trip in a pump-check valve system," *Master Thesis*, Umeå University, Sweden.

[16] Bandari, S. R., 2017, "Investigation on Flow Control Valve by CFD Simulation," *Master Thesis*, Blekinge Institute of Technology, Karlskrona, Sweden.

[17] Dempster, W., and Elmayyah, W., "A computational fluid dynamics evaluation of a pneumatic safety relief valve," *Proc. 13th International Conference on Applied*

Mechanics and Mechanical Engineering (AMME-13), pp. 92-106.

[18] Li, L., Cheng, Z., and Lange, C. F. J. M. P. i. E., "CFD-based optimization of fluid flow product aided by artificial intelligence and design space validation," *Mathematical Problems in Engineering*, 2018, p. 14, 2018.

[19] Olivetti, M., Monterosso, G. F., Marinaro, G., Frosina, E., and Mazzei, P., "Valve Geometry and Flow Optimization through an Automated DOE Approach," *Fluids*, 5(1), 2020.

[20] Si, Q., Lu, R., Shen, C., Xia, S., Sheng, G., and Yuan, J. , "An Intelligent CFD-Based Optimization System for Fluid Machinery: Automotive Electronic Pump Case Application," *Applied Sciences*, 10(1), 2020.

[21] Nikhil, R., "Design and Analysis of Feed Check Valve as Control Valve Using CFD Software," *International Journal of Technical Research and Applications*, 4(3), pp. 223-231, 2016.

[22]. Lei Li, Zhengrong Cheng, Carlos F. Lange, "CFD-Based Optimization of Fluid Flow Product Aided by Artificial Intelligence and Design Space Validation", *Mathematical Problems in Engineering*, vol. 2018, Article ID 8465020, 14 pages, 2018. https://doi.org/10.1155/2018/8465020

[23] Olivetti, M.; Monterosso, F.G.; Marinaro, G.; Frosina, E.; Mazzei, P. Valve Geometry and Flow Optimization through an Automated DOE Approach. *Fluids* 2020, 5, 17.