SIMULATION OF FLUID FLOW THROUGH A STAINLESS STEEL GLOBE VALVE USING ENGINEERING FLUID DYNAMICS

N.H. Tho[1], N.H. Vinh[2], L.T. Hung[1], N.V.A. Duy[1]*
[1]Faculty of Automotive, Mechanical, Electrical and Electronic Engineering, An Phu Dong Campus, Nguyen Tat Thanh University, Ho Chi Minh City 729800, Vietnam
[2]Faculty of Mechanical Engineering and Technology, Ho Chi Minh City University of Food Industry (HUFI), Ho Chi Minh City 700000, Vietnam
Email: nhtho@ntt.edu.vn, *nvaduy@ntt.edu.vn

Abstract

Globe valves are often used in many practical application of industry with the aim of controlling the fluid flow through the valve. Valve design ensuring an operation capacity plays an important role for engineers. This paper presents the design and simulation of fluid flow through the globe valve using CAD/CAE tools such as Autodesk CFD. Besides, design of experiment (DOE) technique Box-Behnken is used to integrate the CFD simulation to identify the optimal design parameters of the globe valve. These parameters are inlet pressure, orifice diameter, and outlet clearance. The results showed that the optimal parameters are appropriate to improve the valve design to ensure the maximum fluid flow rate. Moreover, the integration of Box-Behnken based DOE and CAD/CAE based simulation contributes the significant improvement of the product geometry, and it highlighed that the proposed approach is potential applicability in product design and development.

Keywords: globe valve; fluid dynamics; CAD/CAE; design of experiment; simulation

Introduction

Nowadays, the construction of high and multiple-floor buildings is being widely popular because the land used for housing is shrinking while the demand for housing is increasing due to the growing population. Therefore, the use of apartments will be more and more urgent. The water pipes will have an increase in pressure during the process of ultilization. Thus, devices used for apartments with a height of five floors or more will face a big obstacle.

To ensure that the device is safe to use and not damaged by water pressure, we should install a device called a pressure relief valve to reduce the water pressure to a safer value. In Vietnam today, high-rise buildings will include a pump system to push water from the basement to the rooftop water tank, and the water from the rooftop tank will be distributed to apartments, rooms, floors of the building. Globe valves are commonly used to conduct and distribute fluid flows as shown in Figure 1.

Figure 1. Stainless steel globe valve with inlet diameter of 42 mm.

However, the investors usually save money, so only install the total pressure reducing valve on certain floors, or certain floors, so the water pressure in the floors is still not stable and safe. In this paper, the simulation of water flow through the globe valve based on engineering fluid dynamics to determine the operational parameters of the valve.

Operational principle of valve

Valves are used to regulate the flow of fluids in industrial piping systems. Globe valves have the ability to control the flow rate and reduce the pressure of the fluid flow inside the pipeline. Valves operate thanks to a transmission system consisting of a steering wheel, a valve shaft, and a valve disc. When the valve disc (like a piston) is in the position of sealing the septum between the two halves in the valve body, the valve is closed. When turning the steering wheel, it will act on the lever to lift the valve disc out of the baffle position, when the valve is in an open state, allowing the flow of fluid to pass through the gap on the baffle. The valve structure is as shown in Figure 2.

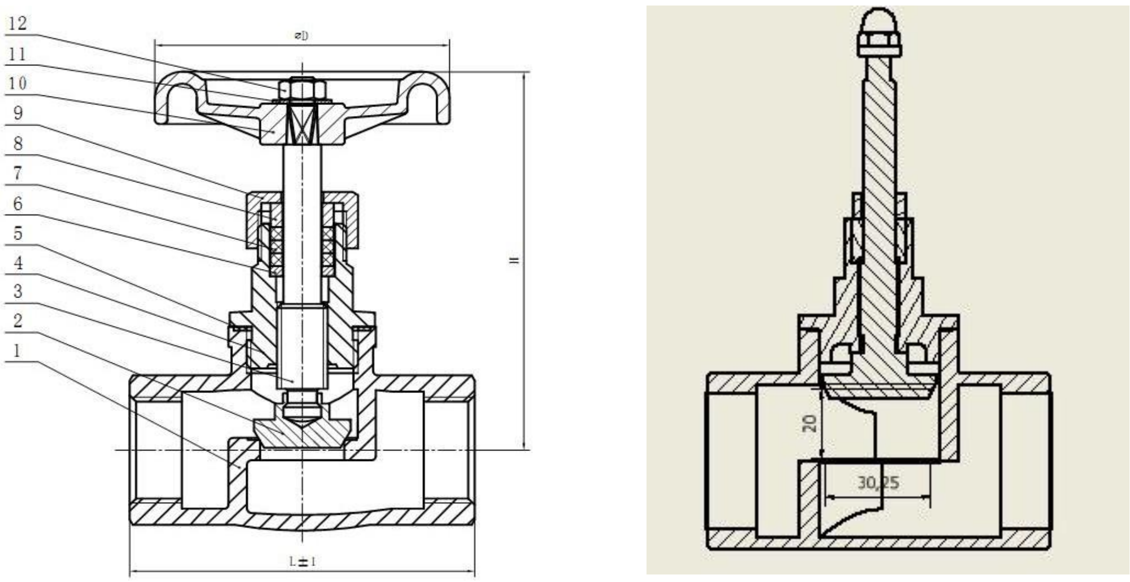

Figure 2. Structure of valve.

Common stainless steel globe valves usually have six sizes as in Table 1, and the parameters we prepared for the test are GLS-07 with inlet and outlet diameters of 11/4", converted to mm is 42 mm.

Table 1. Basic parameters of stainless steel globe valve.

Model	Dimension	L	H	D	Mass (kg)
GLS-04 (GLS-04)	1/2"	61	90	65	0,4
GLS-05 (GLS-05)	3/4"	74	96	72	0,5
GLS-06 (GLS-06)	1"	85	102	72	0,7
GLS-07 (GLS-07)	**1 1/4"**	**95**	**120**	**96**	**1,2**
GLS-08 (GLS-08)	1 1/2"	104	126	105	1,5
GLS-09 (GLS-09)	2"	123	140	105	2,2

CFD based simulation of fluid flow

To perform the CFD simulation for flow through the valve, we first need to assign materials to the valve components according to the procedure in (Autodesk CFD, 2020; Solidworks Flow Simulation, 2020). Flow with a fluid that is water at laboratory temperature. The valve body is made of steel. Then, we set the boundary condition for the inlet flow with a velocity of 1000 mm/s, the pressure at the valve inlet is the normal pipeline pressure from 1.5 to 4 bar. The boundary condition at the outlet cross-section of the flow is that the pressure there is zero, in contact with the atmosphere. Model's mesh is performed by default mode of the calculation mode. The simulation execution calculation iteration step is set to 100. The simulation results are shown in Figure 3 and Table 2.

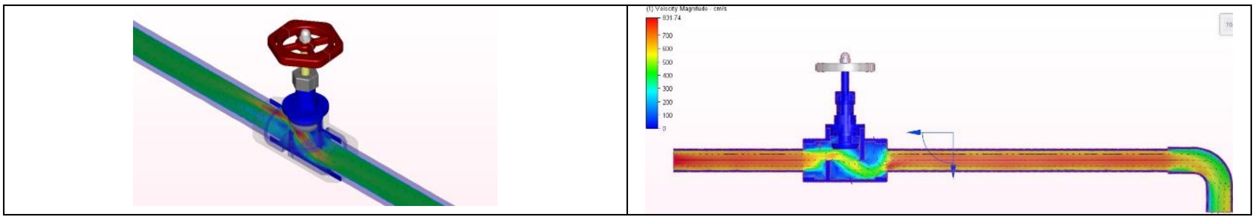

Figure 3. CFD based simulation of fluid flow through the valve.

Table 2. Results of fluid flow simulation through the valve.

Load case: 100; Last Iteration/Step	**Results**	**unit**
REGION # considered	1369.86	cm^2
Flow rate	2.49E-05	m^3/s
Vx-Velocity	−6.97132	m/s
Vy- Velocity	0.0410434	m/s
Vz- Velocity	0.0001816	m/s
Pressure	−31.6307	bar
Force	5.55E+09	dyne

Results on simulation experiments

To find out the valve design parameters, we need to set up an experimental table consisting of factors as shown in Figure 4. The three parameters considered with the corresponding levels are: Closing diameter: 29 – 30.25 – 32 mm; Inlet pressure: 5 – 7.5 – 10 bar; and Clearance: 30%; 50%; 70%.

In 1960, Box and Behnken devised a design that allows us to perform quadratic models directly. All survey factors have three levels: -1, 0, and 1. These designs are easy to implement and are sequential in nature. It is possible to study the *k* factors and still have the option to add new features without losing results from the tests performed..
Box-Behnken design for three elements built on a cube. For the four elements, the design is built on a four-dimensional tetrahedron. The test points are placed not at the corners, but also in the middle of the edges, in the centers of the faces (squares), or at the center of the cube. This arrangement means that all test points are placed equivalently from the center of the studied domain, that is, on a sphere depending on the number of dimensions. Center points are added to the sphere center.

The Box-Behnken design for three elements is shown in Figure 4. The cube has 12 edges. Traditionally, three (possibly five) experimental sites have been placed at the center of the study. Box-Behnken designed for three factors so there are 12 + 3 = 15 or 17 trials. Note that with four central points instead of three, and the design adheres to the orthogonality criterion.
Figure 4 shows Box-Behnken design illustration for three elements. There are twelve test points at the center of each edge, and three at the center of the cube.

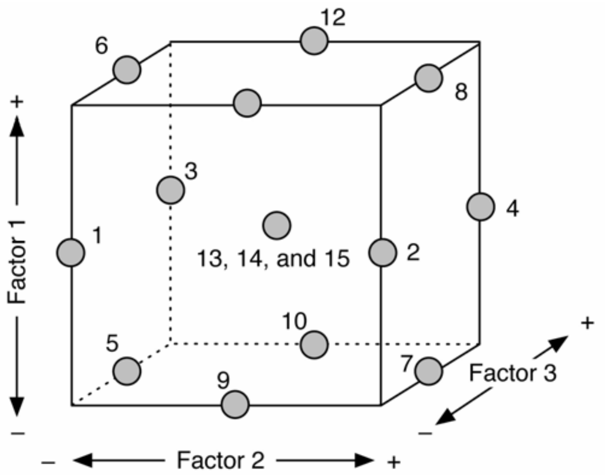

Figure 4. Factor levels and number of experiments to consider (Jacques Goupy, 2007)**.**

DOE analysis

In an industrial laboratory, a researcher wanted to study the effect of several dimensional design factors on valve opening force and flow rate. Inlet pressure is an operating parameter, the valve clearance to allow flow through and the size of the orifice diameter are two size parameters that need to be considered. The purpose of this multi-objective study is to reduce the force applied to the piston face to open the valve, thereby reducing the valve opening and closing force through the threaded joint, and at the same time, the maximum flow through the valve.

The first step of this process is to investigate the relevant factors, especially dimensional parameters and operating parameters such as diameter size, clearance and flow pressure into the valve. The second stage of this process is to conduct fluid dynamics simulations based on modern *CAD/CAE* systems such as Autodesk Inventor Professional and Autodesk CFD 2020 (Autodesk, 2020). The simulation results are used in the analysis and optimal experimental evaluation, in which two outputs are selected, namely valve opening force and valve flow rate. In this study, the three factors used in this test are:

Factor 1: Inlet pressure p_1. This is the factor that contributes to the displacement of flow through the valve.

Factor 2: Valve opening clearance, this gap is large or small depending on the pressure difference between the inlet and outlet of the valve. The larger the clearance, the greater the flow through the valve and vice versa.

Factor 3: The orifice diameter D_0 for water flow through the valve, this diameter with the highs and lows of each factor is shown in Table 3. The two responses to be selected are valve opening force and valve flow. In which, the valve opening force is desired minimum and the flow through the valve is maximum. The two output parameters are shown in Table 4.

Table 3. Simulation experimental level for factors

Factor	**Name**	**Unit**	**Min**	**Max**	**Low**	**High**	**Mean**	**Standard deviation**
A	Clearance	mm	5.40	12.60	−1 ↔ 5.40	+1 ↔ 12.60	9.00	2.55
B	Hole Diameter	mm	29.00	32.00	−1 ↔ 29.00	+1 ↔ 32.00	30.50	1.06
C	Inlet pressure p_1	bar	5.00	10.00	−1 ↔ 5.00	+1 ↔ 10.00	7.50	1.77

Table 4. The output parameters to be considered

No.	**Name**	**Unit**	**Experiment**	**Min**	**Max**	**Mean**	**Standard deviation**
R1	Force (to open the valve)	N	17	171.014	472.15	306.36	79.33
R2	Volume Flow Rate	L/min	17	472.8	840.98	642.57	112.55

This study determines the output responses according to the quadratic regression model. After thinking about this problem in detail, the team chose a Box-Behnken design, as it allows to estimate the quadratic regression model without limiting the number of experiments. The results of 17 trials are shown in Table 5. These trials are rearranged into the classic blueprint of the Box-Behnken design. Experimental planning in Table 4 shows the order of 17 CFD simulation experiments for valve flow.

Table 5. Test matrix and test results

	Factor 1	Factor 2	Factor 3	Response 1	Response 2
No.	A:Clearance	B:Hole Diameter	C: Inlet pressure p_1	Force	Volume Flow Rate
Unit	mm	mm	bar	N	Liters/min
1	12.6	30.5	5	171.014	600.62
2	5.4	30.5	5	235.726	472.8
3	9	30.5	7.5	296.9	660.98
4	9	32	5	200.257	600.25
5	9	29	10	407.611	720.68
6	5.4	30.5	10	472.15	660.14
7	9	30.5	7.5	296.9	660.98
8	9	30.5	7.5	296.9	660.98
9	12.6	29	7.5	354.609	540.64

	Factor 1	Factor 2	Factor 3	Response 1	Response 2
No.	A:Clearance	B:Hole Diameter	C: Inlet pressure p_1	Force	Volume Flow Rate
Unit	mm	mm	bar	N	Liters/min
10	9	30.5	7.5	296.9	660.98
11	5.4	29	7.5	313.648	480.3
12	5.4	32	7.5	358.082	600.3
13	9	32	10	400.382	840.45
14	12.6	30.5	10	345.383	840.98
15	9	30.5	7.5	296.9	660.98
6	9	29	5	204.27	480.98
17	12.6	32	7.5	260.505	780.6

The quadratic regression model has been written in the form as follows:

$$y = a_0 + a_1x_1 + a_2x_2 + a_3x_3 + a_{12}x_1x_2 + a_{13}x_1x_3 + a_{23}x_2x_3 + a_{11}x_1^2 + a_{22}x_2^2 + a_{33}x_3^2 + e \quad (1)$$

Table 6. ANOVA analysis for valve opening force

C- Inlet pressure p1	82877.21	1	82877.21	235.37	< 0.0001	
AB	4798.19	1	4798.19	13.63	0.0077	
AC	962.71	1	962.71	2.73	0.1422	
BC	2.59	1	2.59	0.0073	0.9341	
A²	810.55	1	810.55	2.30	0.1730	
B²	503.60	1	503.60	1.43	0.2707	
C²	93.26	1	93.26	0.2649	0.6226	
Residual	2464.80	7	352.11			
Lack of Fit	2464.80	3	821.60			
Pure Error	0.0000	4	0.0000			
Cor Total	1.007E+05	16				
Source	Sum of Squares	df	Mean Square	F-value	p-value	
Model	98230.52	9	10914.50	31.00	< 0.0001	significant
A-Clearance	7693.89	1	7693.89	21.85	0.0023	
B-Hole Diameter	463.78	1	463.78	1.32	0.2888	

The analysis in Table 6 shows that the *F*-value of the model is 31.00 indicating that the model is statistically significant. There is only a 0.01% chance that such a large F value can occur due to noise. P value less than 0.0500 shows that the model's variable groups are statistically significant. In this case *A*, *C*, *AB* are groups of significant variables. Values greater than 0.1000 indicate that the groups of variables in the model are not statistically significant. If there are many groups of variables in the model that are not significant (excluding those necessary to support the model structure), reducing the model can improve the regression model.

Table 7. Coefficient of quadratic regression model for valve opening force

Factor	**Coefficient Estimate**	**df**	**Standard Error**	**95% CI Low**	**95% CI High**	***VIF***
Intercept	296.90	1	8.39	277.06	316.74	
A-Clearance	−31.01	1	6.63	−46.70	−15.32	1.0000
B-Hole Diameter	−7.61	1	6.63	−23.30	8.07	1.0000
C- Inlet pressure p_1	101.78	1	6.63	86.09	117.47	1.0000
AB	−34.63	1	9.38	−56.82	−12.45	1.0000
AC	−15.51	1	9.38	−37.70	6.67	1.0000

Factor	Coefficient Estimate	df	Standard Error	95% CI Low	95% CI High	*VIF*
BC	−0.8040	1	9.38	−22.99	21.38	1.0000
A^2	13.87	1	9.14	−7.75	35.50	1.01
B^2	10.94	1	9.14	−10.69	32.56	1.01
C^2	−4.71	1	9.14	−26.33	16.92	1.01

Table 7 shows an estimate of the coefficient that represents the expected change in output response per unit change in input factor value when all other factors are held constant. The intercept in an orthogonal design is the overall mean output response of all the experiments. The coefficients are adjustments around that average based on the input settings. When the factors are orthogonal, the *VIFs* are 1; *VIF* greater than 1 shows multi-colinearity, the higher the *VIF*, the stronger the correlation of the factors. As a general rule, a *VIF* less than 10 is acceptable. Finally, we build the quadratic regression equation as described in Table 8.

Table 8. Regression model for valve opening force

Force	=
+2863.85887	
+180.66419	Clearance
−242.24111	Hole Diameter
+74.06120	Inlet pressure p_1
−6.41380	Clearance * Hole Diameter
−1.72375	Clearance * Inlet pressure p_1
−0.214400	Hole Diameter * Inlet pressure p_1
+1.07057	$(\text{Clearance})^2$
+4.86061	$(\text{Hole Diameter})^2$
−0.753020	$(\text{Inlet pressure } p_1)^2$

Similar to previous analysis for valve opening force, We analyze the ANOVA for the output response in terms of flow (liters/min), the coefficients and construct the quadratic regression model for the flow rate in Table 9.

Table 9. The quadratic regression mode of flow rate

Volume Flow Rate	=
−8797.38590	
−107.55880	Clearance
+590.23111	Hole Diameter
−20.23367	Inlet pressure p_1
+5.55370	Clearance * Hole Diameter
+1.47278	Clearance * Inlet pressure p_1
+0.033333	Hole Diameter * Inlet pressure p_1
−2.98900	$(\text{Clearance})^2$
−9.68111	$(\text{Hole Diameter})^2$
+3.42280	$(\text{Inlet pressure } p_1)^2$

Table 10. Statistical testing

Std. Dev.	19.81	R^2	0.9864
Mean	642.57	Adjusted R^2	0.9690
C.V. %	3.08	Predicted R^2	0.7832
		Adeq Precision	24.7919

Table 10 shows that a Predicted R^2 value of 0.7832 is a reasonable acceptance for an Adjusted R^2 = 0.9690; the deviation is less than 0.2. Adeq Precision measures signal-to-noise ratio. Ratios greater than 4 are expected. The 24.792 ratio shows sufficient signal of the model. This model can be used to fully describe the design space.
In order for the valve to work well, both the minimum valve opening force and the maximum flow rate are achieved. We investigate the multi-objective optimal empirical model with weight 1:1 for two objective functions. The obtained results are shown in Figure 5.

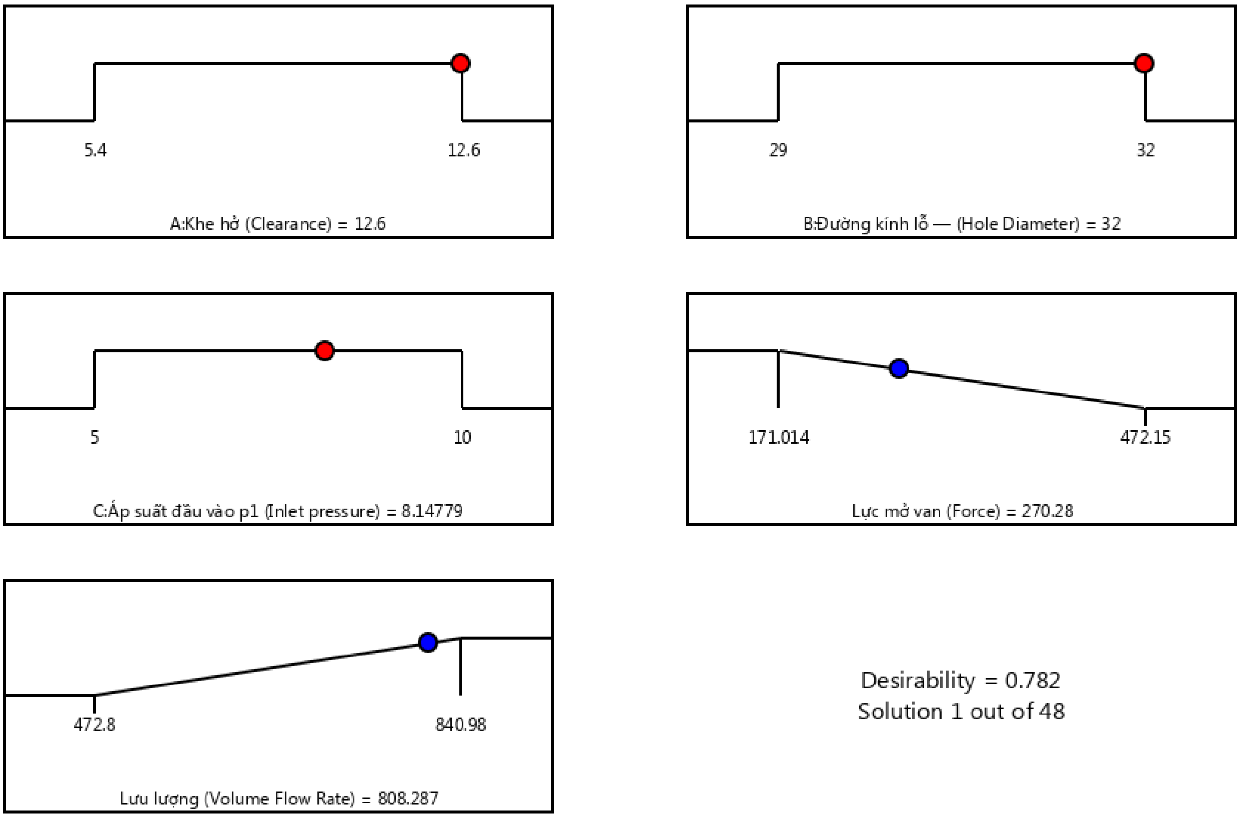

Figure 5: The optimal solution is achieved for the model

From Figure 5, we see that the optimal value satisfying two conditions is the small valve opening force and the largest valve flow, which are input parameter values as shown in Table 11 and response output values as in Table 12.

Table 11. Optimal input parameter results

Factor	Name	Level	Low Level	High Level
A	Clearance	12.60	5.40	12.60
B	Hole Diameter	32.00	29.00	32.00
C	Inlet pressure p_1	8.57	5.00	10.00

Table 12. Input and output values corresponding to optimal parameters such as 12.6 mm clearance; hole diameter of 32 mm; and inlet pressure p_1 = 8.57 bar

Two-sided Confidence = 95% Survey Population = 99%

Response	Predicted Mean	Predicted Median	Std Dev	SE Mean	95% CI low for Mean	95% CI high for Mean	95% TI low for 99% Pop	95% TI high for 99% Pop
Force (to open valve)	284.167	284.167	18.7647	17.061	243.824	324.51	164.979	403.356

Through the final results, we see that the valve works well with the following parameters: The reasonable pressure through the valve is 8.57 bar; The reasonable closing diameter is 32 mm; and reasonable clearance 12.6 mm. Based on these parameters, we apply it to build a new and improved design model for the valve.

Conclusion

This paper simulated the flow of fluid through a steam globe valve using *CAD/CAE* Autodesk CFD software. The fluid considered was water at laboratory temperature. The numerical experimental integration technique investigated the three input factors of the valve, namely inlet pressure, outlet diameter and clearance in order to identify a 2^{nd} order regression model for the smallest force acting on the piston's underside and maximu flow rate throughput of the valve. The results showed that the optimal parameter for design improvement is 12.6 mm clearance; 32 mm hole diameter; and inlet pressure p_1 = 8.57 bar. In the future, this paper will deal with valve cavitation by CFD simulation.

Reference

[1]. Autodesk, Autodesk CFD 2020 tutorials, website: https://www.autodesk.com/products/cfd/features, accessed on 25/12/2020.
[2]. Jacques Goupy, Lee Creighton, Introduction to Design of Experiments with JMP Examples, Third Edition, SAS Press (October 5, 2007).
[3]. Solidworks, Solidworks Flow Simulation, website: https://www.solidworks.com, accessed on 25/12/2020.