alpha
Journal of Engineering and Applied Sciences
(AJEAS)
e-ISSN: 3023-6002

Alfa Mühendislik ve Uygulamalı Bilimler Dergisi
(2025) 3/1: 15 – 34
Research Article
DOI: 10.70988/ajeas.1591897

Received: 12/03/2024
Accepted: 01/22/2025
Published: 04/30/2025

New Generation Minimized Flow Resistance Butterfly Valve Design

Erhan ÖZKAN [a,*]

[a] Department of Mechanical Engineering Department, Faculty of Engineering, Ege University, İzmir, TÜRKİYE
* Corresponding author's e-mail address: erhanozkan81@icloud.com

Abstract
Butterfly valves get this name because the visual of their working principle resembles the wing movements of a butterfly. In this way, flow control is more ergonomic, and it becomes possible to save water. However, in addition to these advantages, the difficulty of minimizing flow resistance is observed as a disadvantage. The costs of design improvements made in valve products, which are produced by the casting method and involve labour-intensive production stages such as assembly and welding, are quite high. In these cases, computer-aided design and test simulation become more important. In this study, the designs of the products were carried out by using computational fluid dynamics and finite element methods. Details of the simulations and design verification stages were included, the designs were made with Solidworks and these data were validated by ANSYS. As a result of the study, a body and ring designs were achieved with an improvement of 26% compared to existing products and 32% compared to the industry average. Finally, national and international patent applications have been made for the unique curved body and gradual shaft designs obtained during the development phase of the product.
Keywords: *Valve design, Simulation, Flow resistance, Water save.*

Yeni Nesil Minimize Edilmiş Akış Dirençli Kelebek Vana Tasarımı

Özet
Kelebek vanalar, çalışma prensibine ait görselliğin bir kelebeğin kanat hareketine benzemesinden dolayı bu ismi almıştır. Bu sayede akış kontrolü daha ergonomik olmakla birlikte su tasarrufu da sağlanmış olur ancak bu avantajların yanı sıra akış direncini en aza indirmenin zorluğu da bir dezavantaj olarak görülmektedir. Döküm yöntemiyle üretilen ve montaj, kaynak gibi emek yoğun üretim aşamalarını içeren vana ürünlerinde yapılan tasarım iyileştirmelerinin maliyetleri oldukça yüksektir. Bu gibi durumlarda bilgisayar destekli tasarım ve test simülasyonu daha da önem kazanmaktadır. Bu çalışmada hesaplamalı akışkanlar dinamiği ve sonlu elemanlar yöntemleri kullanılarak ürünlerin tasarımları gerçekleştirilmiştir. Simülasyon ve tasarım doğrulama aşamalarının detaylarına yer verilmiş olup, tasarımlar Solidworks ile gerçekleştirilmiş ve bu veriler ANSYS ile geçerli kılınmıştır. Çalışma sonucunda mevcut ürünlere göre %26, sektör ortalamasına göre ise %32 iyileşme sağlayan bir gövde ve klape tasarımı elde edilmiştir. Son olarak ürünün geliştirme aşamasında elde edilen özgün kavisli gövde ve kademeli mil tasarımları için ulusal ve uluslararası patent başvuruları yapılmıştır.
Anahtar kelimeler: *Vana tasarımı, Simülasyon, Akış direnci, Su tasarrufu.*

Citation: E. Özkan, "New Generation Minimized Flow Resistance Butterfly Valve Design", AJEAS. (2025) 3(1): 15-34. http://dx.doi.org/10.70988/ajeas.1591897

1. Introduction

According to current research, it was determined that annual water consumption the ratio is 800 m^3/person [1]. 2.3 billion people are trying to continue their lives by completely deprived of drinking water [2]. According to the similar studies, the number of countries that will have problems regarding to reach the water and survive under these conditions up to 54 by 2050 [3]. By considering these data, the world population will be expected to reach approximately 9.4 billion and it is predicted that 40% of them will not be able to benefit from the drinking water [4]. Therefore, it is inevitable to use water resources effectively and prevent waste [5]. For this purpose, fluid transportation and storage systems reducing water loss to minimum levels become mandatory [6]. The valve designed to ensure this decreasing flow resistance in systems is of importance [7]. Making these designs is a quite costly process [8]. For this reason, computer-aided design and simulation programs are used [9].

Computational Fluid Dynamics (CFD) is considered a sub-branch of fluid mechanics and caused from liquids problems using numerical algorithms by providing analysis [10]. CFD method is used as an identifiable simulation tool [11], especially in processes involving flow such as liquid/gas [12]. This method is often used to determine the operating modes of all types of functions and convection [13], which includes flow, heat-mass movements [14]. But every fluid by using time analytical methods may not be possible to determine the behaviour, and numerical methods can be used to solve this problem [15]. Analytical methods are generally based on mathematical functions [16]. Numerical methods are used when producing solutions in the form generally provides numerical results [17]. The results calculated using these methods are very close to the real values and can be adjusted to the desired level [18]. In cases where it is desired to increase the sensitivity of the results the number of operations that need to be solved by the computer is increasing [19]. Hence this takes a long time, that causes both calculation times to increase and the computer to work inefficiently [20]. On the other hand, these simulations are used in the design of butterfly valves, where fluid behaviour is investigated and increasingly used in the industry, have become more important [21].

Butterfly valves have been designed to control fluid systems within the time specified in the parameters on preventing the movement [22, 23]. Possible damage to the valve carries risks such as leakage, cracks and explosion with material contamination and loss [24]. More importantly leaks in the system, such as poisoning and fire cause dangerous situations to occur [25]. Butterfly valves should be preferred to prevent such situations [26, 27]. The biggest advantage of butterfly valves on the application is providing high flow rates compared to its dimensions [28]. This issue becomes important especially when there is a lack of space in controlling chemical fluids [29]. Therefore, these products are commonly preferred in the systems, that contain chemicals, wastewater and distribution networks [30-34].

The motivation that started this study was reducing the friction on the ring surface of the butterfly valves with an original design that cause to achieve energy efficiency. For this purpose, first the target properties of the products exposed to the fluids have been defined, then the dynamics calculations, simulation studies, computer-aided design and prototype manufacturing processes have been carried out, respectively. The valves used in this study were designed to comply with EN GJS 400-18 LT standard. Moreover, the raw materials were selected to fulfil the TS EN 1267 standard requirements. The prototype with process and flow line design simulation was adjusted according to the EN 1074 with the scope of drinking water, distribution lines, industrial applications, water treatment facilities, pump stations, industrial waste systems. When literature studies were evaluated, pipelines against high stresses that may occur resistant must meet the EN GJS 400-15 (also known as GGG 40) standard requirements, so simulation-supported analyses with the defined materials and prototype production activities were carried out with this purpose. Finally, the original curved for body and stepped shaft designs have been performed, successfully. Moreover, a national and an international patent application have been made.

2. Methodology

2.1. Computational fluid dynamics

CFD is an analysis method, which is widely used in the valve industry. In this way, the flow conditions of the valve are able to simulate by using various equations that describe 3D flows. It is very important for designers to have all analyses results carried out before prototype production. It is possible to do all in a virtual environment and causing to do everything right in the first time with less labour and material loss. Thanks to this, it can be transferred economically in a short time to mass production of the product. The three-dimensional designs of products have been carried out by using Solidworks drawing program. ANSYS computer aided simulation program has been used for CFD analysis and FEA. Butterfly valve with a diameter of 200 mm, and 1000 mm were chosen as sample to do the experimental verifications of the study. As in applications 16 bar value most chosen, the nominal pressure was selected as the same and defined like PN16. While CFD analysis carried out, K-epsilon turbulence model was solved by keeping the y+ value at 3 and below. The limit in the layers (boundary layers) and the mesh improvements have been made during the process.

Flow coefficient and pressure loss coefficient formulas were defined according to EN 1267:2012 (E) standard. Accordingly, the flow coefficient (Kv) is calculated with Equation 1.

$$Kv = Q\sqrt{\rho \div (\Delta p \times \rho_0)} \tag{1}$$

In the equation, Kv (m^3/hour) is the flow coefficient, Q is the flow rate in m^3/hour, ρ is the density of water in kg/m^3, ρ_0 is the density of water in kg/m^3 at 15 °C, Δp expresses the pressure loss in the valve (difference between inlet and outlet pressure).

2.2. Finite element analysis

The valve was designed in FEA complies according to the EN 1704 standard, where the boundary conditions defined in the strength values. During the FEA phase the parts were presented in two different numerical formats such as single and assembled. However, Rayleigh-Ritz method was performed for the interpolation solution. The specified boundary conditions were applied exactly as defined in the standard. One and a half times the nominal pressure value for the body, ten percent of the nominal pressure value for the ring part defined as excess. As a result, obtained from this part, the material analysis and selection stage should take place.

2.3. Materials selection

The valve model includes 2 parts. First part is body that is in contact with the fluid and material selection of the body is defined in TS EN 12266 and TS EN 1074 standards. According to these standards body part must be high strength against the one and half times pressure. As a result of analyses made according to standards the body material that can withstand the damage was chosen. The material of the ring was also chosen by the same method with one small difference. As defined in the standards, the selection materials for the ring are that can withstand ten percent more than pressure value. The ring strength analysis has been made according to stress and deformation results. As a result of evaluation for all these data EN GJS 400-15 (also known as GGG 40) material has been defined and selected. This material matched according to the finite element mechanical analysis outputs, too. The stresses in the system were discussed and analysed in a virtual-simulated environment. Hence, product and process design were carried out. 24 bar pressure was defined on the surfaces of body fluid contact as a result of finite element analysis. The maximum amount to be applied to body surfaces pressure values are shown in Fig.1.

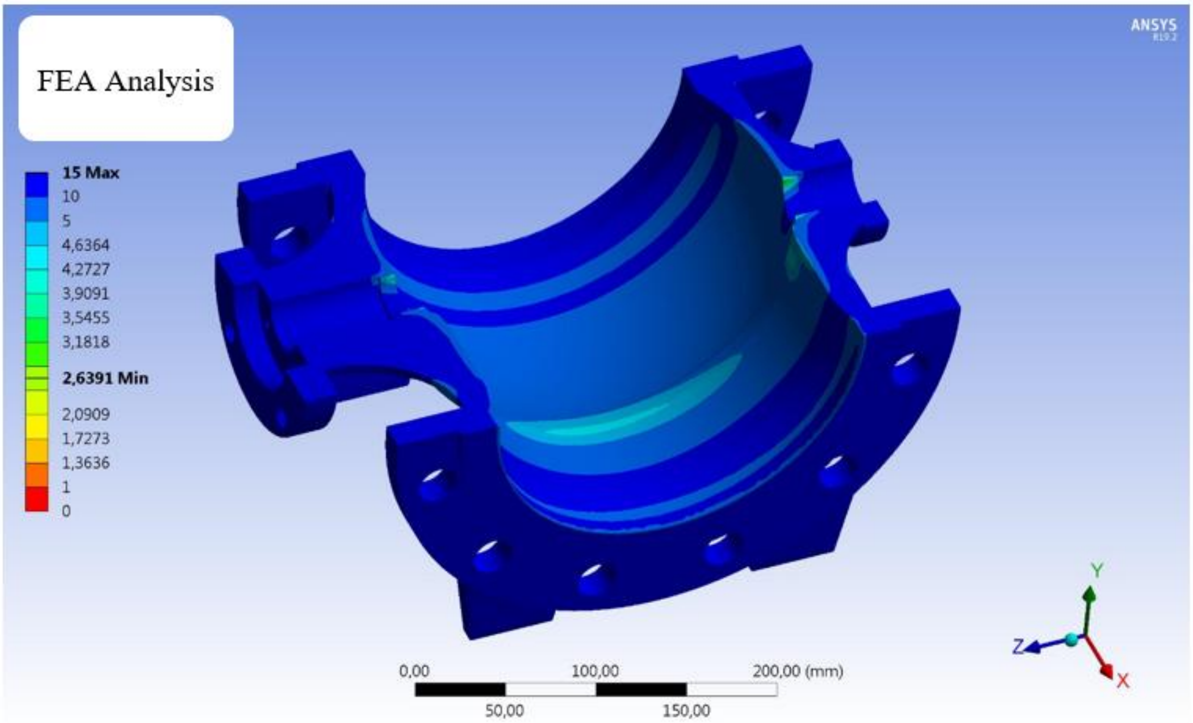

Figure 1. The highest-pressure load defined on the surface for finite element analysis

As the low melting temperature, suitable for prototype production fluidity, easy casting, ease of machinability, high strength with high wear resistance, and high ductility properties, EN GJS 400-15 material caused to accelerate the process. Thanks to numerical simulation applications, material load-stress distributions on operations dimensional changes and similar metallurgical and mechanical properties at different parameters can be revealed before the prototype production. According to the analysis of equivalent stresses on body design, verification process has been carried out. One and a half of the nominal pressure of 16 bar for the solid that can operate under 24 bar pressure was applied on the body by the mechanical analysis as shown in Fig. 2.

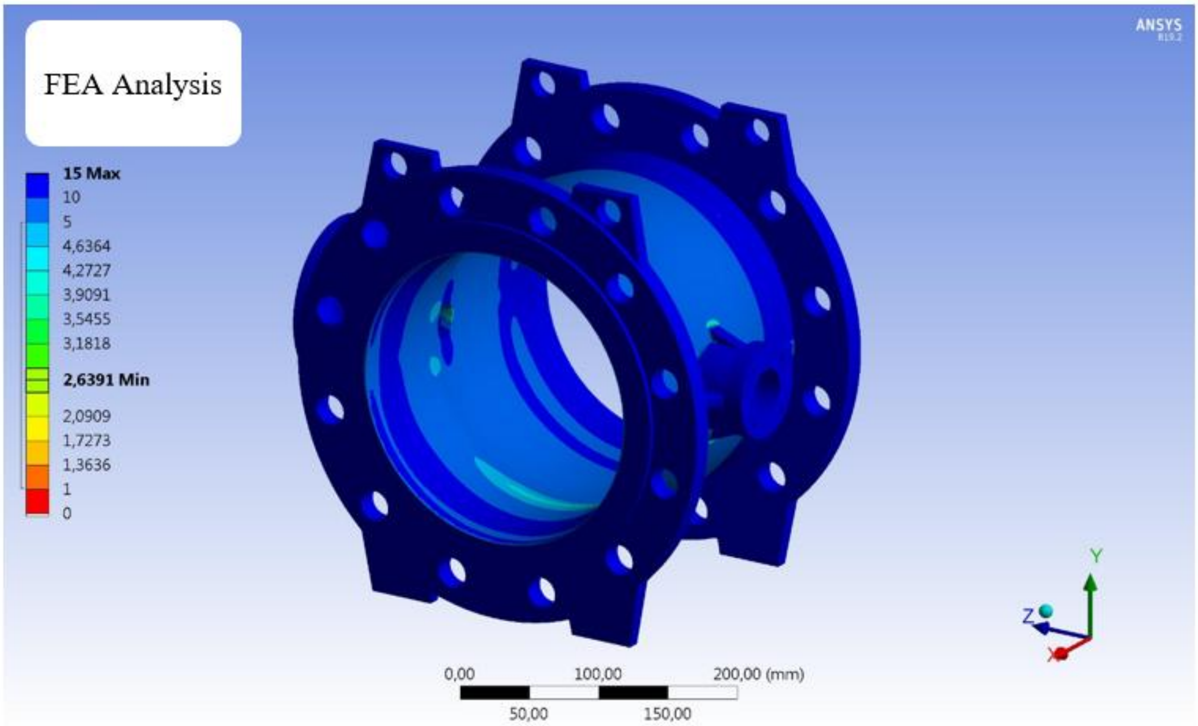

Figure 2. Mechanical analysis of the material selected for the body design under 24 Bar pressure

240 MPa yield strength value has been considered as ten percent more than nominal stress for realized ring materials mechanical tests by using numerical analysis method. Accordingly, 1.1 times more of the maximum working pressure (16 bar) was defined as 18 bar for the ring as shown in Fig. 3.

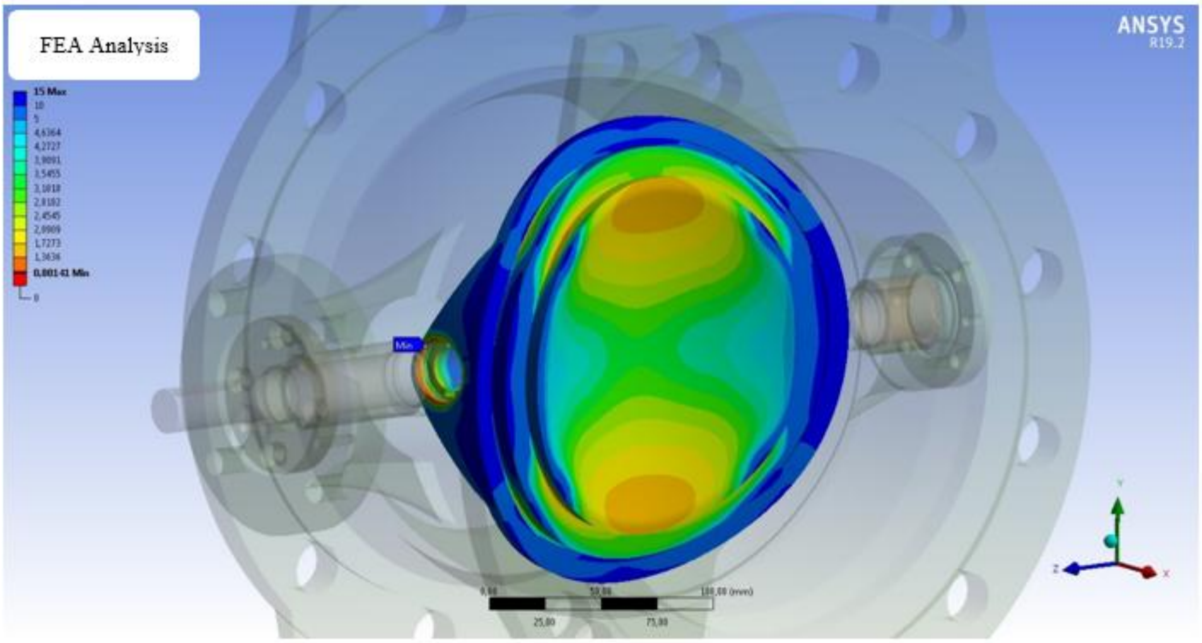

Figure 3. Mechanical analysis of the ring assembled on the body and operating under 1.1 times the nominal pressure value

2.4. Flow simulations

CFD analyses for the products designed with EN GJS 400-15 material were carried out under 16 bar pressure, 4m/s flow rate and with the valve in the fully open position by using finite element method. Fig. 4 shows the general view of CFD for disc component.

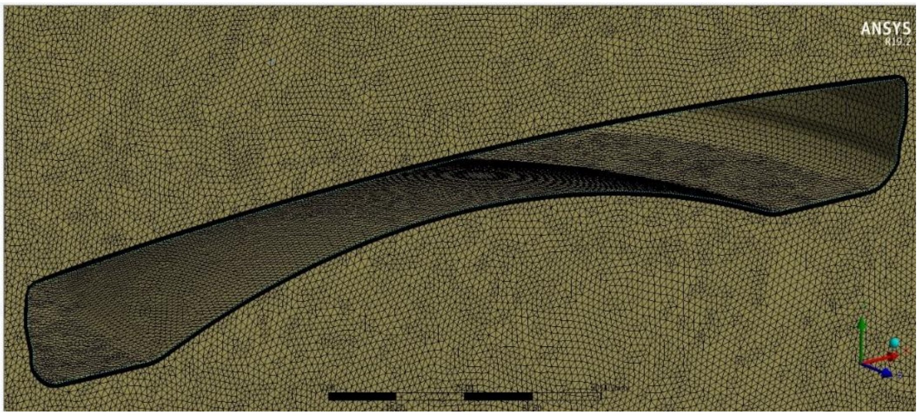

Figure 4. Mesh analysis of the disc performed by CFD

A mesh independence test was performed to prove that the results of the mesh analysis performed were reliable. Fig. 5 shows the corresponding graphic. Accordingly, the volume flow rate was determined as 116,14569 at element number 8.000.000 and the value was 116,15249 at element number 9.234.500.

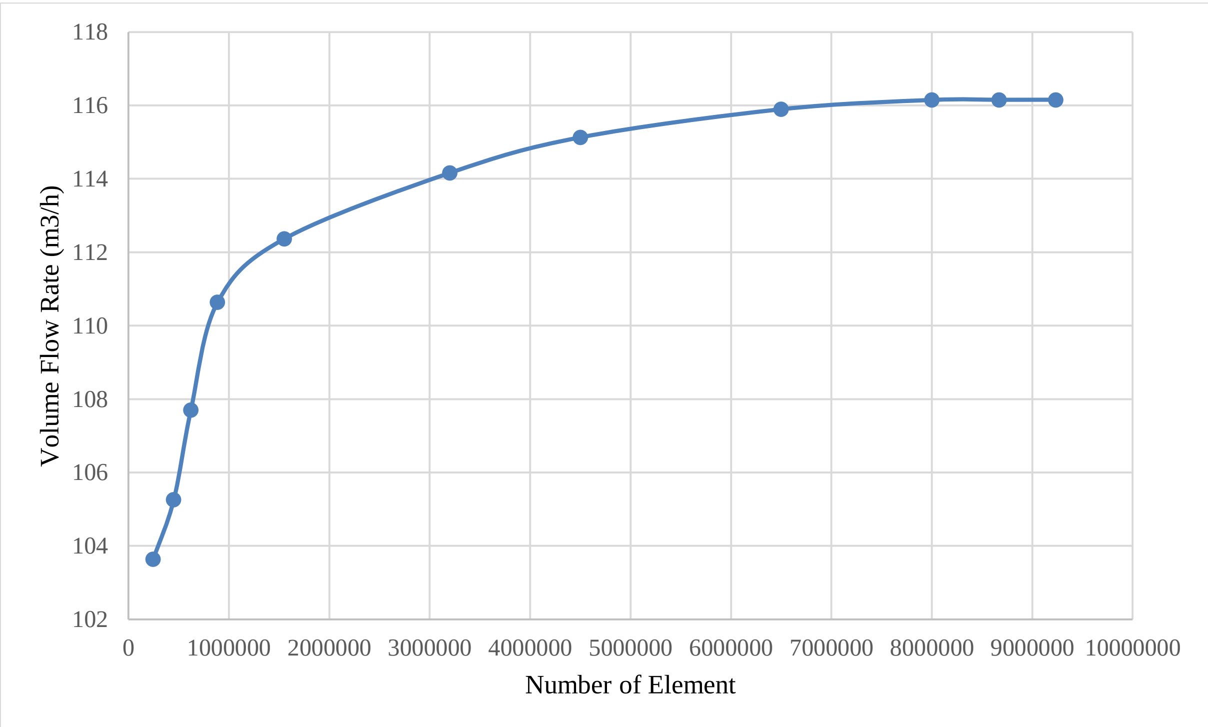

Figure 5. Mesh independence study curve

In order to better interpret the characteristics of this behaviour, velocity vector analysis was performed. Accordingly, when the valve is in the fully open position, the resistance of the valve and its body against flow can be characterized more clearly. Velocity vector analysis of the system in Fig. 6 is given for a better understanding of the subject.

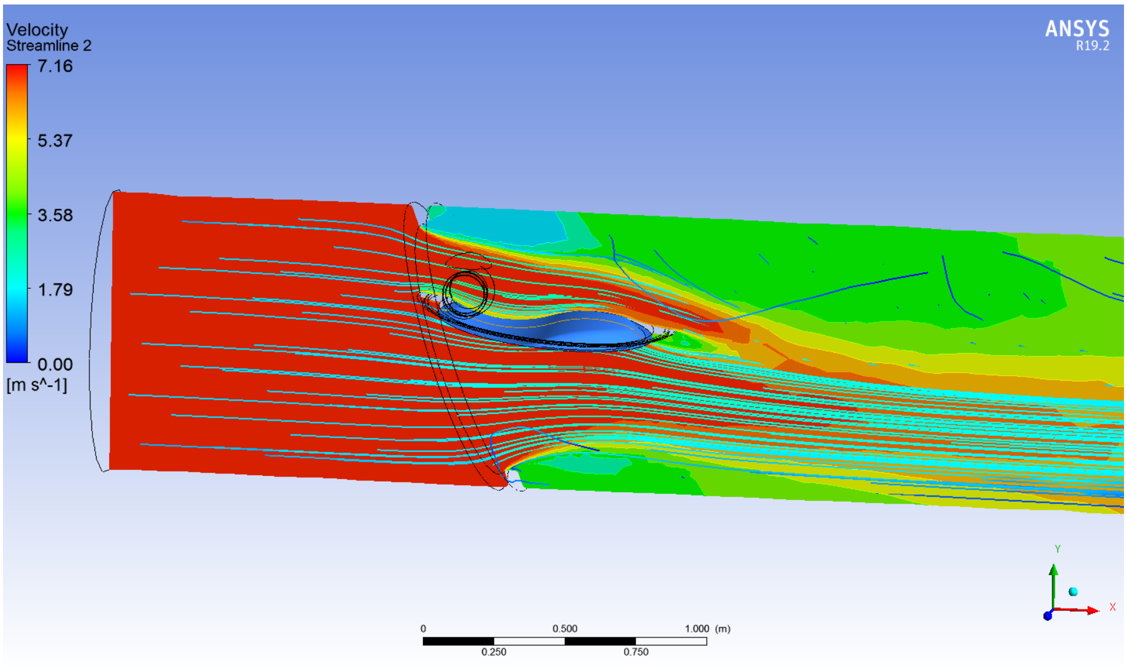

Figure 6. Flow velocity vector analysis of the developed design

The other parameter that will support the velocity vector analysis of this system will be its characteristic behaviour under 16 bar pressure. The pressure distribution of the system is shown in Fig. 7.

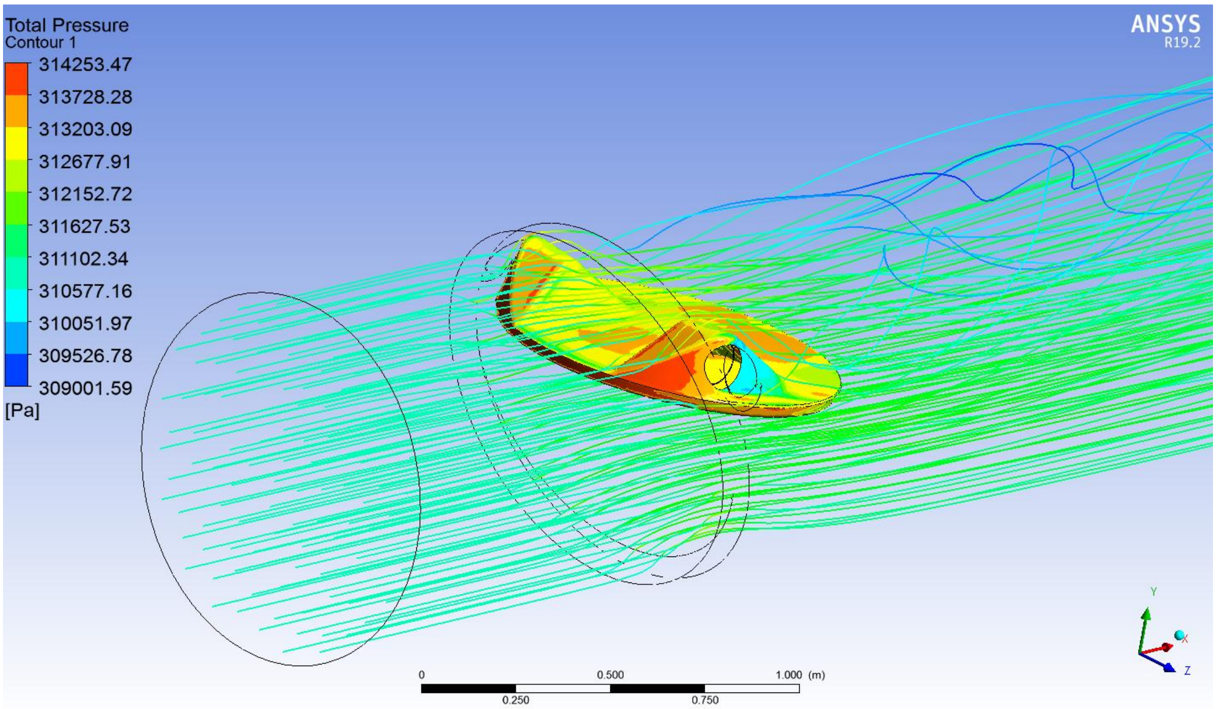

Figure 7. Total pressure distribution on the surface

When the flow occurs at a certain speed and pressure, the distribution analysis of the flow filaments of the system is performed and the total strength is analysed. Fig. 8 shows the flow filament distribution analysis of the system under 16 bar pressure at 4m/s vector speed.

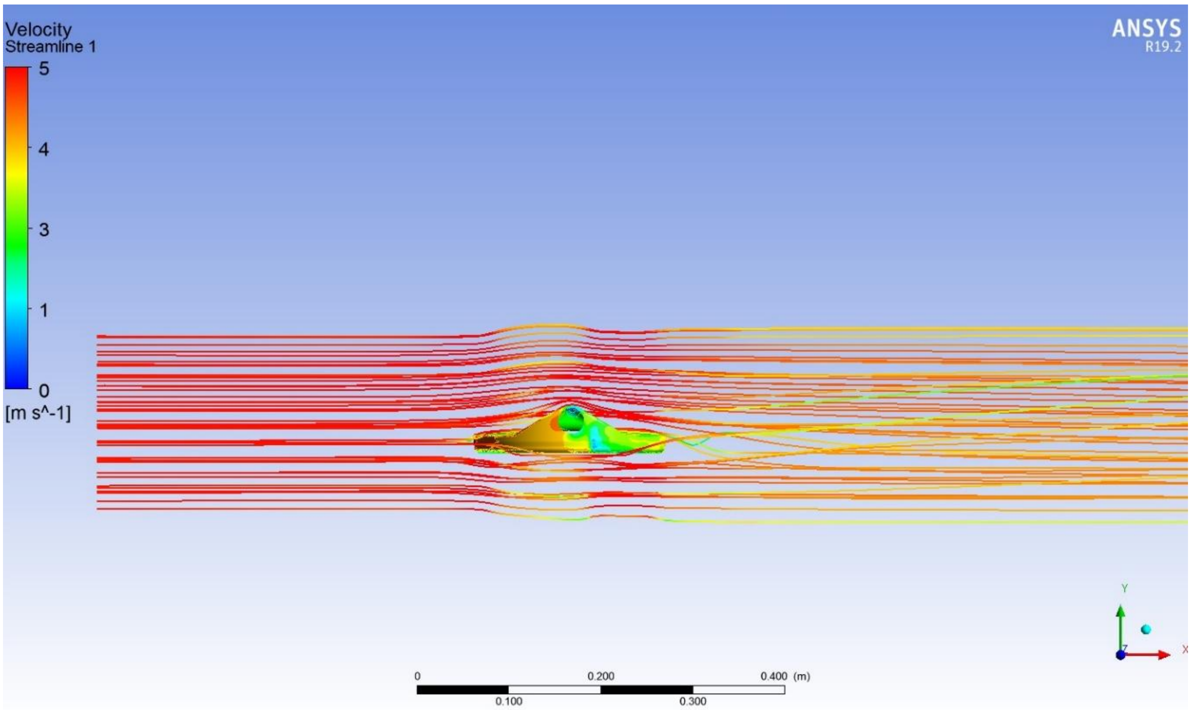

Figure 8. Flow filament distribution analysis

After all flow and pressure analyses were carried out, hydrodynamic forces on the valve were analysed. Fig. 9 shows the CFD analysis visual of the hydrodynamic forces on the ring.

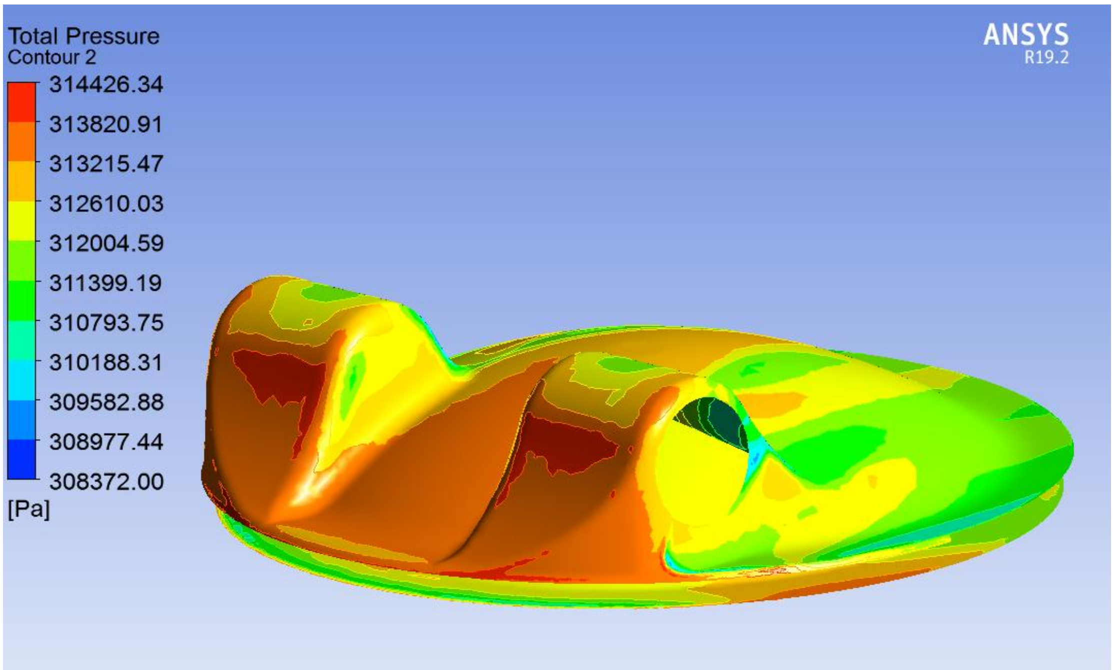

Figure 9. Total pressure distribution analysis applied on the surface on the ring

2.5. Design verification, validation and prototype production

The casting model and core box of the valve, whose hydrodynamic properties were analysed and designed, were produced on Computer Aided Manufacturing (CAM) software and Computerized Numerical Control (CNC) benches. Fig. 10 shows the wooden pattern production of the ring.

Figure 10. Model production of the ring

Details of the pattern dimensions are given in mm in Fig. 11. to explain along with its dimensions.

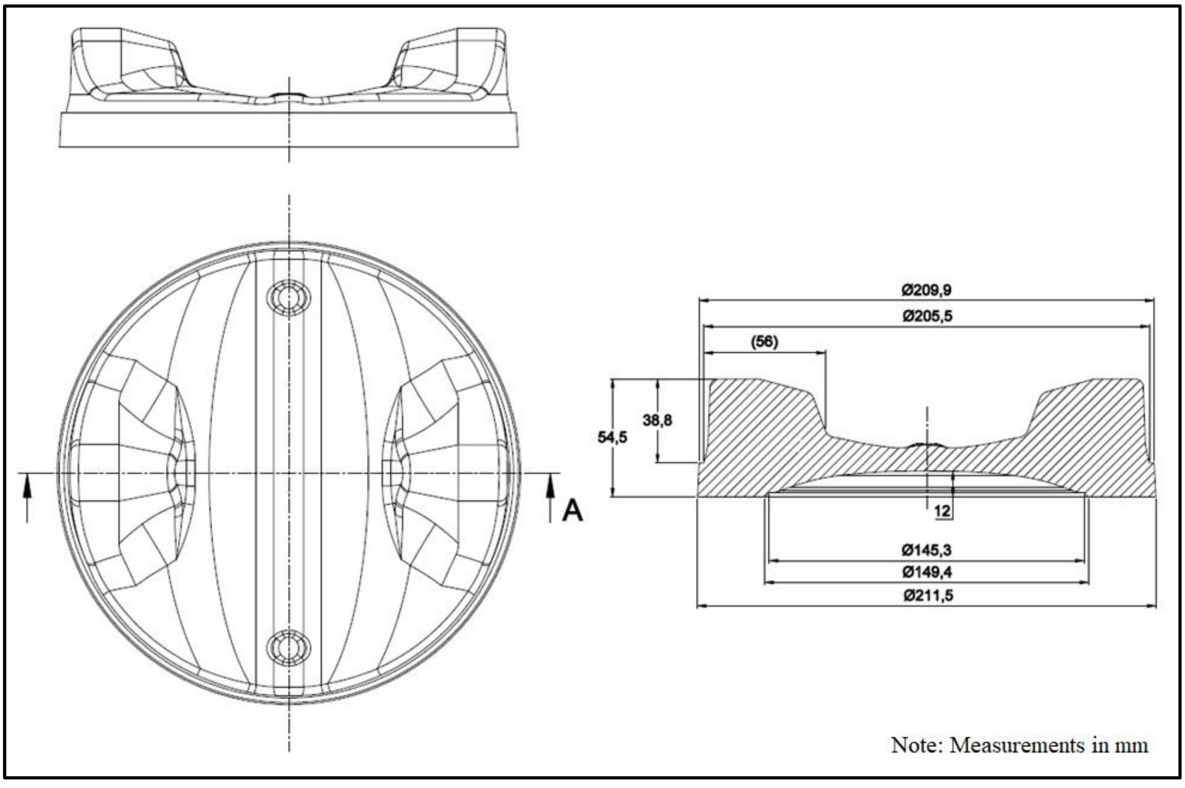

Figure 11. Model explanation along with its dimensions

The model design of the body on which the ring would be mounted, where the designed mechanical strength analyses were verified by simulation, is shown in Fig. 12.

Figure 12. The finish pattern of the ring for the body assembling

The dimension control of the cast body was carried out with the Faro Arm Quantum device and the report of the measurements was obtained by Faro CAM2 software. Fig. 13 shows the report of the precise measurements of the body casting.

		Real	Nominal	Deviation	-tol	+tol	Result
erler:12.	Diameter	43,871mm	44,000mm	-0,129mm	-0,200mm	0,200mm	+
erler:7.	Diameter	1.111,874mm	1.112,000mm	-0,126mm	-5,500mm	0,000mm	+
Okunan De erler:21.	Diameter	1.009,932mm	1.010,000mm	-0,068mm	-0,100mm	0,100mm	+
erler:24.	Surface	20,033°	20,000°	0,033°	-1,000°	1,000°	+
erler:16.	Diameter	230,816mm	231,000mm	-0,184mm	-0,500mm	0,500mm	+
erler:16.	Diameter	114,182mm	114,000mm	0,182mm	0,000mm	0,200mm	+
erler:5.	Diameter	21,945mm	22,000mm	-0,055mm	-0,100mm	0,100mm	+
erler:16.	Diameter	350,930mm	350,000mm	0,930mm	-1,000mm	1,000mm	+

Figure 13. Dimension CMM report

Simulated pressure tests of the produced body and the mounted ring were carried out in accordance with EN 1074. Pressure tests were evaluated according to whether the valve was leaking or damaged under 24 bar pressure for the body. The design in which the pressure test was carried out is shown in Fig. 14.

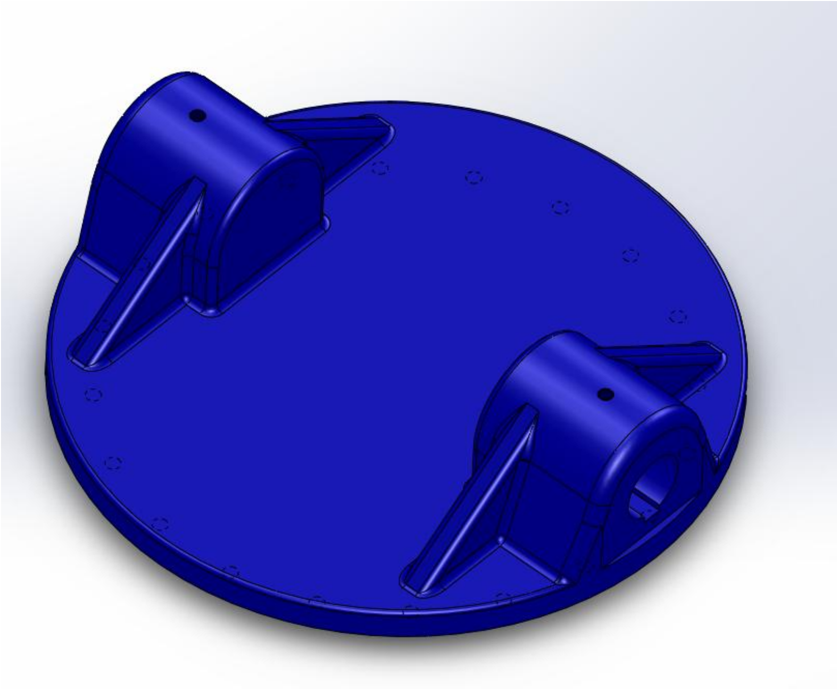

Figure 14. The ring final design for the pressure test

According to the TS EN 1267 flow resistance test standard, the valve was modeled in the Solidworks drawing program in a fully open position. The modeling performed is shown in Fig. 15.

Figure 15. Three-dimensional valve test verification model

Defining fluid inlet and outlet boundary conditions was of critical importance in this modelling. According to the standard the inlet boundary conditions were defined at a distance of 2xDN from the valve, and outlet boundary conditions were defined at a distance of 10xDN. Fig. 16 shows the fluid direction design for CFD.

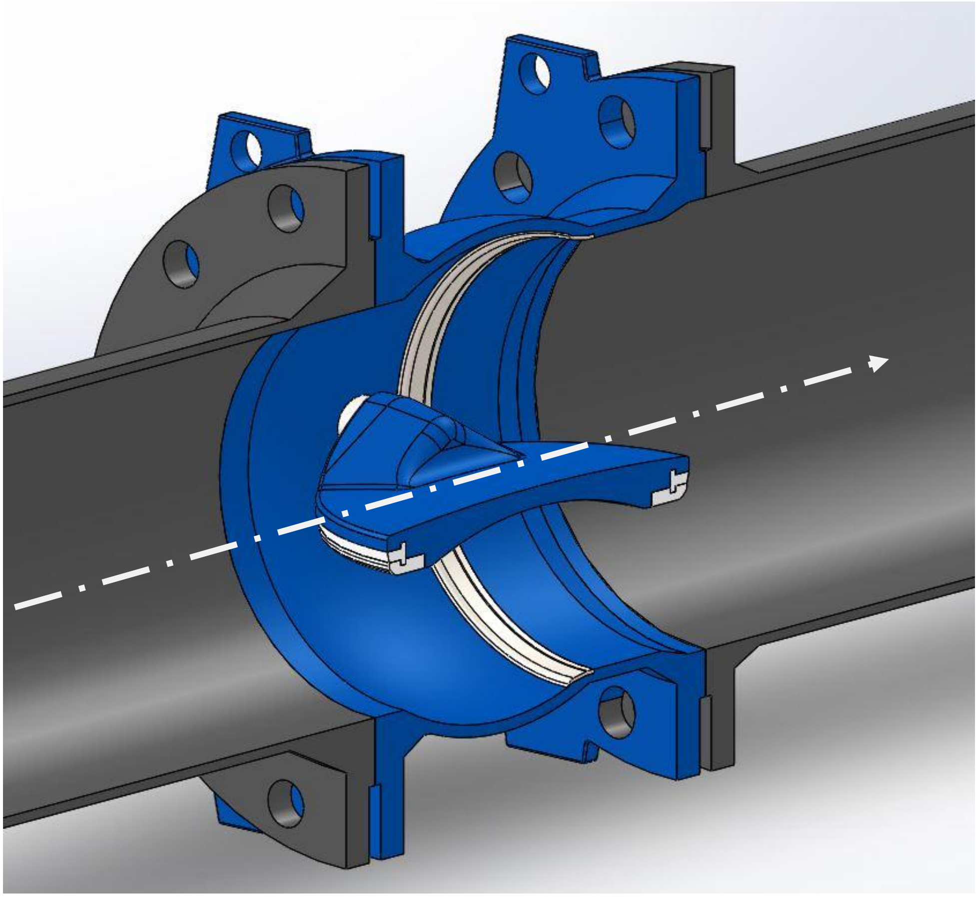

Figure 16. Determining fluid directions for CFD

According to the conservation principle, fluid movement is by trying to define the flow structure in detail at every point of the field or by using a finite by working with the region and establishing the balance between the incoming flow and the outgoing flow. If the continuity equation is applied to circular cross-section pipe flows, the continuity equation will be cylindrical due to the geometric shape of the flow field and the control volume selected accordingly. It should be applied in coordinates (r,θ,z). Incompressible flows in cylindrical coordinate system and the continuity equation for is as follows

$$\frac{1}{r}\frac{\partial(rV_r)}{\partial r}+\frac{1}{r}\frac{\partial(rV_\theta)}{\partial\theta}+\frac{1}{r}\frac{\partial(rV_z)}{\partial z}=0 \tag{2}$$

Here, r, θ and z are the radial (radius), tangential (circumferential) and axial directions, respectively, and vr, vθ and vz refers to the flow velocities in radial, tangential and axial directions, respectively.

Flow capacity measurements of the body and ring, which did not detect any problems in the simulation studies and were produced accordingly, were made in the Kv test device. Fig. 17 shows the prototype Kv test setup.

Figure 17. Kv test setup for the designed prototype

3. Results and Discussion

Product and experimental designs were carried out with a line pressure of 16 bar, and no plastic deformation or leakage was observed in the product under 16 bar line pressure.

Simulation studies were carried out so that the fluid coefficient Kv was 3009.9 m^3/hour when the valve was in the fully open position under 16 bar internal pressure and 4 m/s fluid speed. The values obtained as a result of the tests and flow analysis were measured as 2985 m^3/hour. This measurement corresponds to a margin of error of 5% when compared to the design, and in EN 1074 standards this deviation is expected to be below 10%. For this reason, it was determined that the flow analysis and physical test results were compatible with each other and no nonconformities were encountered. The results are included in the physical test report given in Table 1.

Table 1. Physical test report

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS REPORT												
VALVE TYPE	:	DOUBLE ECCENTRIC BUTTERFLY VALVE / ***Dikkan Ultra Series***										
DN	:	200	(	200	)	/	(	Default fluid velocity	:	4	m/s	)
PN	:	16										

OPENED POSITION OF OBTURATOR (%)		:	100		
Inlet Pressure (bar)	Outlet Pressure (bar)	Velocity from CFD Result (m/s)	Fluid Flow Rate (m^3/h)	Differantial Pressure (ΔP) (bar)	Kv (m^3/h)
15,99999	15,97740	4,096	452,4	0,02259	3009,9

Resistance to 1.5 x 16 bar = 24 bar body internal pressure and 1.1 x 16 bar = 18 bar leakage tests of product components specified in TS EN 12266 standard have been successfully carried out. The values in most valves might vary due to losses caused by friction resulting from the actual length of the flow path, sudden changes in the flow direction, and surface roughness, but when the prototype manufacturing real test results and simulation results were compared, no difference was detected in the Kv values of the product, computer-aided design and prototype production, so the data overlapped almost exactly.

Fig. 18 shows images of the old and new body designs. The curved body, where flow properties were improved, can be observed more clearly in this picture.

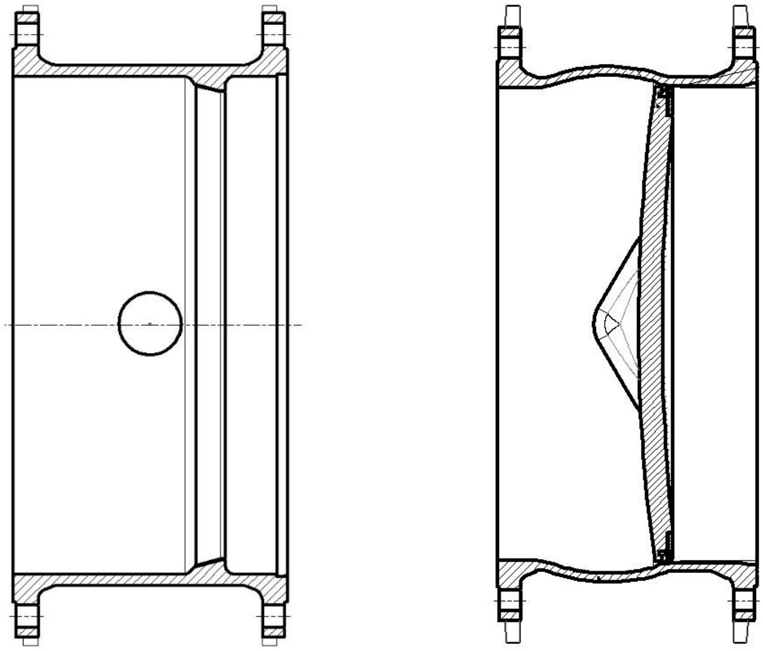

Figure 18. Images of old (left) and new (right) body designs

Fig. 19 shows the visual of the old and new disc designs mounted on the valve. The green colour represents the old design, and the red colour represents the new design. The gain in surface area, which provides 20% more and homogeneous flow, can be noticed in this picture.

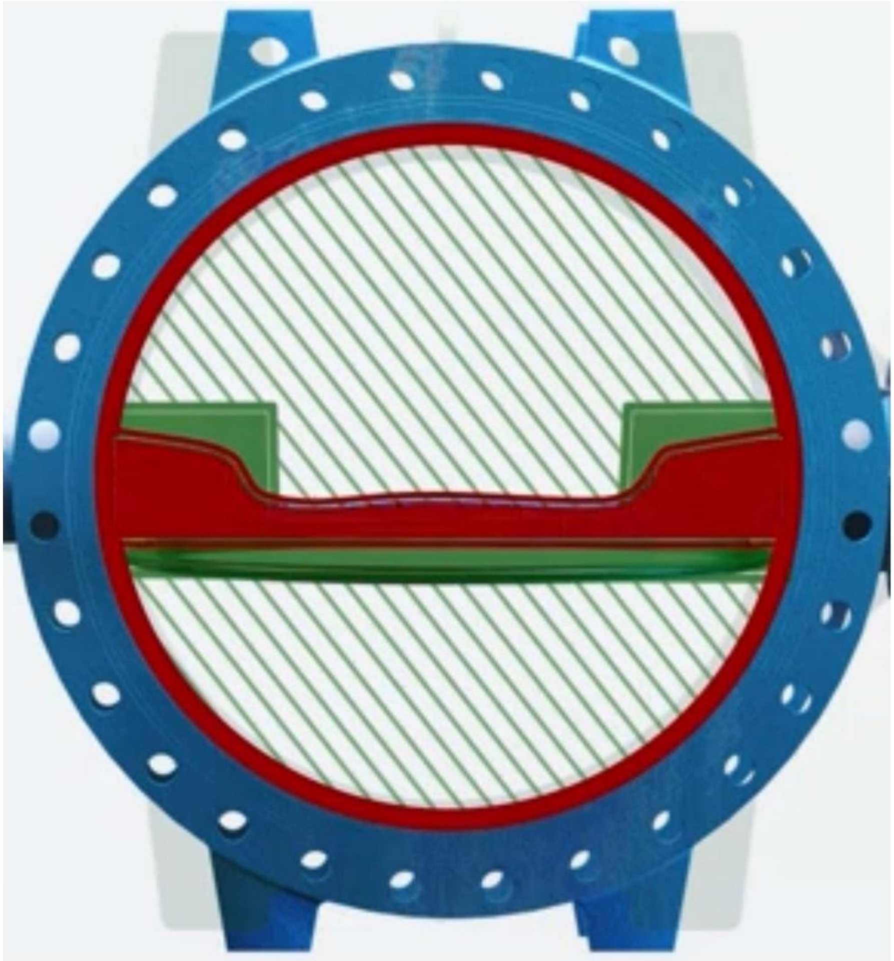

Figure 19. Superimposed image of old and new disc designs

In the flow tests performed according to the minimum 240 N/mm^2 yield strength in the body design, no negative effects were detected under 16 bar and no plastic deformation occurred in the product under 24 bar pressure.

Resistance to 1.5 x 16 bar = 24 bar body internal pressure and 1.1 x 16 bar = 18 bar sealing tests of product components specified in TS EN 12266 standard have been successfully carried out. In most valves, these values vary due to losses caused by friction resulting from the actual length of the flow path, sudden changes in the flow direction, obstacles in the flow direction and surface roughness, but when the prototype manufacturing test results and simulation results are compared, no difference was detected in the Kv values of the product and computer-aided design and prototype production. The data overlapped almost exactly.

In the tests of the ring material, no problems were detected under stresses exceeding 1.1 times, no deformation was observed under loads twice the maximum pressure, and the product successfully met the requirements of the EN 1074 standard.

With the Solidworks sustainability report, a carbon footprint of 140 kg CO_2 per product was achieved, and 26% improvement in existing products. The design and prototype manufacturing of the product, which would achieve 32% energy saving with a total energy consumption of 1400 MJ/product with material, production and product life, has been carried out (Fig. 20). This chart was obtained by making a systematic comparison of 24 national and international products in the sector. Finally, national and international patent applications have been completed for the unique curved body and gradual shaft designs obtained during the development phase of the product.

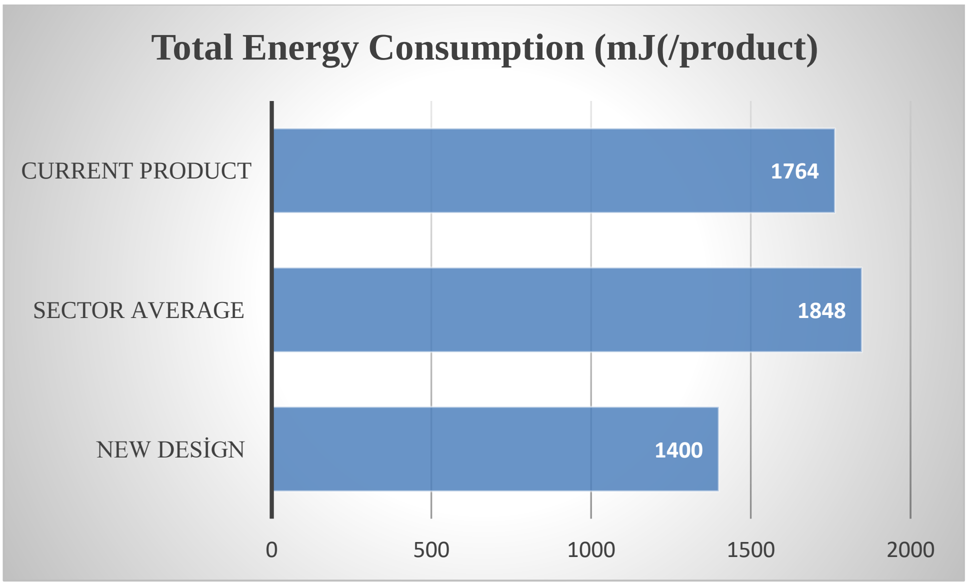

Figure 20. Total energy consumption graph

4. Conclusion

Efficient use of water resources has become inevitable within the scope of combating the climate crisis. Efforts continue to reduce the carbon footprint, which is one of the main reasons for the climate crisis and a measure of global warming. As a result of this study, energy losses were reduced by 32% and carbon footprint was reduced by 26% with improvements in flow coefficients in the butterfly valve product designed. The 95% overlap between simulation-supported tests and real test data in body and ring designs has once again demonstrated the importance of simulation-supported designs. As a result of the studies carried out in the article, national and international patent applications were made for the unique curved body and stepped ring structure. The first patent in the valve industry was published in 1839, and patents for almost all valve components were obtained in the mid-1900s. Therefore, all national and international patents were scanned, and it was determined that no patents were received regarding the developed curved body and gradual ring design. This result made the study more original. Finalization of the commercialization of the studies carried out in the article, a completely domestic and national product that is long-lasting, highly energy efficient, and aims to protect environmental and human health values has been introduced to the valve industry.

List of Symbols

CFD	Computational Fluid Dynamics
FEA	Finite Element Analysis
CAD	Computer Aided Manufacturing
CNC	Computerized Numerical Control

Declarations and Ethical Standards

The author declared no potential conflicts of interest with respect to the study, authorship, and/or publication of this article. The author of this article declares that the materials and methods used in this study do not require ethical committee permission and/or legal-special permission.

Author Contribution

The all stages including computer aided design, simulations, prototype production and validation tests of this study were carried out by Erhan Özkan.

References

[1] D. I. Pomoni, M. K. Koukou, M. G. Vrachopoulos, L. Vasiliadis, "A review of hydroponics and conventional agriculture based on energy and water consumption, environmental impact, and land use", Energies 16:4 (2023) 1690. Doi: https://doi.org/10.3390/en16041690.

[2] A. S. George, A. H. George, A. G. Martin, "The environmental impact of AI: a case study of water consumption by chat GPT", Partners Universal International Innovation Journal 1:2 (2023) 97–104. Doi: https://doi.org/10.5281/zenodo.7855594.

[3] D. Jaffee, "Unequal trust: Bottled water consumption, distrust in tap water, and economic and racial inequality in the United States", Wiley Interdisciplinary Reviews: Water 11:2 (2024) e1700. Doi: https://doi.org/10.1002/wat2.1700.

[4] J. Li, Z. Shen, G. Liu, Z. Jin, R. Liu, "The effect of social economy-water resources-water environment coupling system on water consumption and pollution emission based on input-output analysis in Changchun city, China", Journal of Cleaner Production 423 (2023) 138719. Doi: https://doi.org/10.1016/j.jclepro.2023.138719.

[5] N. Antonova, J. Mendoza-Jiménez, I. Ruiz-Rosa, "Determinants of water consumption in hotels: New insights obtained through a case study", Water 15:17 (2023) 3049. Doi: https://doi.org/10.3390/w15173049.

[6] R. Wang, X. Zhao, H. Qiu, X. Cheng, X. Liu, "Uncovering urban water consumption patterns through time series clustering and entropy analysis", Water Research 262 (2024) 122085. Doi: https://doi.org/10.1016/j.watres.2024.122085.

[7] L. Lai, X. Wang, G. Kefayati, E. Hu, K. C. Ng, "Optimisation of cooling performance and water consumption of a solid desiccant-assisted indirect evaporative cooling system using response surface methodology", International Journal of Refrigeration 168 (2024) 376–388. Doi: https://doi.org/10.1016/j.ijrefrig.2024.09.023.

[8] S. S. Abdujalilova, R. S. Zukhridinovna, "Measuring water consumption in fittings", Central Asian Journal of Mathematical Theory and Computer Sciences 4:5 (2023) 29–33. Available: [No DOI provided].

[9] M. Żyłka, N. Marszałek, W. Żyłka, "Numerical simulation of pneumatic throttle check valve using computational fluid dynamics (CFD)", Scientific Reports 13:1 (2023) 2475. Doi: https://doi.org/10.1038/s41598-023-29457-4.

[10] S. Li, G. Deng, Y. Hu, M. Yu, T. Ma, "Optimization of structural parameters of pilot-operated control valve based on CFD and orthogonal method", Results in Engineering 21 (2024) 101914. Doi: https://doi.org/10.1016/j.rineng.2024.101914.

[11] M. K. Neyestanaki, G. Dunca, P. Jonsson, M. J. Cervantes, “A Comparison of Different Methods for Modelling Water Hammer Valve Closure with CFD”, Water 15:8 (2023) 1510. Doi: https://doi.org/10.3390/w15081510.

[12] H. Yao, Q. Ye, C. Wang, P. Yuan, “CFD Dynamic Mesh-Based Simulation and Performance Investigation of Combined Guided Float Valve Tray”, Chemical Engineering and Processing-Process Intensification 193 (2023) 109523. Doi: https://doi.org/10.1016/j.cep.2023.109523.

[13] Q. Wu, J. Xue, N. Hu, Y. Lai, H. Zhao, Q. Li, J. Gu, “Open Balance Point and Dry Pressure Drop of a Rectangular Float Valve Tray: Experiment and CFD Simulation”, Industrial & Engineering Chemistry Research 62:48 (2023) 20844–20858.

[14] S. Evangelista, G. Tortora, G. Viccione, “Experimental and Numerical CFD Modelling of the Hydrodynamic Effects Induced by a Ram Pump Waste Valve”, Sustainability 15:17 (2023) 13104. Doi: https://doi.org/10.3390/su151713104.

[15] S. D. Khamankar, N. K. Ade, P. H. Sahare, M. A. Kumbhalkar, K. S. Rambhad, S. A. Choudhari, M. M. Sardeshmukh, “CFD Analysis of a Butterfly Valve to Optimize Its Design”, AIP Conference Proceedings 2839:1 (2023). Doi: https://doi.org/10.1063/5.0167691.

[16] M. M. Said, H. AbdelMeguid, L. Hassan Rabie Sakr, “A Comparison Study between 3-D CFD and Experimental Data of Butterfly Valve Coefficients”, MEJ-Mansoura Engineering Journal 39:3 (2020) 10–22.

[17] S. W. Choi, H. S. Seo, H. S. Kim, “Analysis of Flow Characteristics and Effects of Turbulence Models for the Butterfly Valve”, Applied Sciences 11:14 (2021) 6319. Doi: https://doi.org/10.3390/app11146319.

[18] K. Alkhulaifi, A. Alharbi, M. Alardhi, J. Alrajhi, H. H. Almutairi, “Comparative Analysis of the Performance Characteristics of Butterfly and Pinch Valves”, Processes 11:7 (2023) 1897. Doi: https://doi.org/10.3390/pr11071897.

[19] L. Wang, S. Zheng, X. Liu, H. Xie, J. Dou, “Flow Resistance Optimization of Link Lever Butterfly Valve Based on Combined Surrogate Model”, Structural and Multidisciplinary Optimization 64:6 (2021) 4255–4270. Doi: https://doi.org/10.1007/s00158-017-1694-4.

[20] Z. Lin, D. Yin, J. Tao, Y. Li, J. Sun, Z. Zhu, “Effect of Shaft Diameter on the Hydrodynamic Torque of Butterfly Valve Disk”, Journal of Fluids Engineering 142:11 (2020) 111202. Doi: https://doi.org/10.1115/1.4047795

[21] Q. Wen, Y. Liu, Z. Chen, W. Wang, “Numerical Simulation and Experimental Validation of Flow Characteristics for a Butterfly Check Valve in Small Modular Reactor”, Nuclear Engineering and Design 391 (2022) 111732. Doi: https://doi.org/10.1016/j.nucengdes.2022.111732.

[22] B. Xu, Z. Zhu, Z. Lin, D. Wang, G. Ma, “Numerical and Experimental Research on the Erosion of Solid-Liquid Two-Phase Flow in Transport Butterfly Valve Based on DEM Method”, Industrial Lubrication and Tribology 73:4 (2021) 606–613. Doi: https://doi.org/10.1108/ILT-12-2020-0454.

[23] G. Su, J. Zhang, L. Yan, "Study on the Occurrence and Development Mechanism of Pipeline Corrosion Behind Butterfly Valve", CIESC Journal 73:12 (2022) 5504. Doi: 10.11949/0438-1157.20221209.

[24] S. Li, B. Zhang, L. Yang, J. Zhang, Y. Wang, W. Kang, "Study on Wear Properties of the Graphite-Sealing Surfaces in a Triple Eccentric Butterfly Valve Based on EDEM-Fluent Coupling", Machines 11:4 (2023) 463. Doi: https://doi.org/10.3390/machines11040463.

[25] J. Tao, Z. Lin, G. Zhang, J. Su, Z. Zhu, "A Numerical and Experimental Study of the Time-Averaged and Transient Flow Downstream of a Butterfly Valve", Journal of Fluids Engineering 144:5 (2022) 051202. Doi: https://doi.org/10.1115/1.4052632.

[26] G. L. Kyriakopoulos, Y. Aminpour, O. A. Yamini, A. Movahedi, S. H. Mousavi, M. R. Kavianpour, "Hydraulic Performance of Howell–Bunger and Butterfly Valves Used for Bottom Outlet in Large Dams under Flood Hazards", Applied Sciences 12:21 (2022) 10971. Doi: https://doi.org/10.3390/app122110971.

[27] K. Kumar, S. Komble, S. Thorat, P. Korgaonkar, V. Bartakke, P. Sonje, K. Kavire, "Validation of Installation Effects of an Ultrasonic Flow Meter on Butterfly Valve Using Numerical Analysis", Materials Today: Proceedings 46 (2021) 6692–6699. Doi: https://doi.org/10.1016/j.matpr.2021.04.164.

[28] S. J. Oh, J. H. Yoon, S. P. Kim, "A Study on the Flow Characteristics of Butterfly Valve with Baffles", Journal of Mechanical Science and Technology 35 (2021) 1065–1073. Doi: https://doi.org/10.1007/s12206-021-0220-1.

[29] Q. K. Nguyen, K. H. Jung, G. N. Lee, S. B. Park, J. M. Kim, S. B. Suh, J. Lee, "Experimental Study on Pressure Characteristics and Flow Coefficient of Butterfly Valve", International Journal of Naval Architecture and Ocean Engineering 15 (2023) 100495. Doi: https://doi.org/10.1016/j.ijnaoe.2022.100495.

[30] S. Wen, Y. A. Wei, H. W. Wang, "Optimization of Flow Characteristics of Superheated Steam Triple-Eccentric Butterfly Valve", Journal of Xihua University (Natural Science Edition) 41:3 (2022) 15–20. Doi: 10.12198/j.issn.1673-159X.4067.

[31] S. B. Lee, "A Study on Reduction of Cavitation with Orifice on High Differential Pressure Control Butterfly Valve", Journal of the Korean Society of Industry Convergence 25:1 (2022) 131–139. Doi: https://doi.org/10.21289/KSIC.2022.25.1.131.

[32] S. H. Yu, S. H. Park, J. G. Hwang, H. J. Yang, "A Study on the Fluid Flow According to the Opening Angle of a Butterfly Valve with High Control Performance", Journal of the Korean Society of Industry Convergence 24:5 (2021) 617–623. Doi: https://doi.org/10.21289/KSIC.2021.24.5.617.

[33] M. S. Kim, W. L. Seong, W. C. Sung, "Effect of the Turbulence Model in the Flow Analysis of Triple Offset Butterfly Valve", Journal of Advanced Marine Engineering and Technology 46 (2022) 356–361. Doi: https://doi.org/10.5916/jamet.2022.46.6.356.

[34] W. Wu, B. Qiu, G. Tian, X. Liao, T. Wang, "CFD-Based Cavitation Research and Structure Optimization of Relief Valve for Noise Reduction", IEEE Access 10 (2022) 66356–66373. Doi: 10.1109/ACCESS.2022.3184449.

Author Biography

 Erhan Özkan	Dr. Erhan Özkan completed his undergraduate education in ITU Metallurgical and Materials Engineering Department in 2004 and graduated from Dokuz Eylul University Metallurgical and Materials Engineering Department in 2006 with a master's degree and in 2015 from Ege University Mechanical Engi-neering Department with a doctorate degree. Dr. Erhan Özkan, who started his career as an academician, then he worked as R&D Manager at Oerlikon, Strategic Planning and Project Coordinator at Kocaer Haddecilik, Process Quality Supervisor at Eldor, Assistant General Manager at Tescom, General Manager at Aryom, and worked as R&D Center Director at Dikkan respectively. He continues his studies on digitalization, artificial intelligence, advanced technological materials, and advanced production techniques.