

DESIGN AND STRUCTURAL EVALUATION OF A LARGE-DIAMETER BUTTERFLY VALVE USING ELASTIC STRESS ANALYSIS

SANDEEP V¹, ANUPAMA ROUSTRAY², LENKA CEPOVA³, J. JAYAPRAKAR⁴, ARNAB DAS SARMA⁵, ELANGO VAN MUNIYANDY⁶, ANKUR BANSOD⁷, TOMAS DVORACEK³

¹Department of Mechanical Engineering, School of Engineering and Technology, JAIN (Deemed to be University), Bangalore, India
v.sandeep@jainuniversity.ac.in

²Department of Mechanical Engineering, Siksha 'O' Anusandhan (Deemed to be University), Bhubaneswar 751030, India
anupamaroustray@soa.ac.in

³Department of Machining, Assembly and Engineering Metrology, Faculty of Mechanical Engineering, VSB-Technical University of Ostrava, 70800 Ostrava, Czech Republic
lenka.cepova@vsb.cz

⁴Department of Mechanical Engineering, Sathyabama Institute of Science and Technology, Chennai, Tamil Nadu, India
jayaprabakar.mech@sathyabama.ac.in

⁵Department of Mechanical Engineering, Brainware University, Kolkata, India
ads.me@brainwareuniversity.ac.in

⁶Department of Biosciences, Saveetha School of Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai 602105, India
muniyandy.e@gmail.com

⁷Design Department, Petro Valves Private Limited, Ahmedabad 382433, India
ankur.1754@gmail.com

DOI: 10.17973/MMSJ.2026_10_2025089

lenka.cepova@vsb.cz

This study presents the design, structural analysis, and validation of a 1200-mm-diameter butterfly valve developed in accordance with ASME and MSS standards for industrial fluid-control applications. The valve body wall thickness was calculated using ASME B16.34 Appendix VI and MSS SP-67, including corrosion allowances, resulting in an effective thickness of 21.58 mm. The flange and body neck were dimensioned in accordance with ASME B16.47 to ensure adequate load-carrying capacity, while the disc was optimized with an elliptical profile (80–180 mm thick) to minimize deformation under pressure loading. The stem and pin were designed using torque-based equations, yielding a 100-mm-diameter stem capable of sustaining the required operational torque. Finite Element Analysis (FEA) was conducted to assess deformation and stress distribution. A maximum deformation of 3.54 mm was observed in the disc under a pressure of 2.94 MPa, while the valve body remained structurally stable. Stress Classification Line (SCL) analysis confirmed that the valve satisfies the requirements of ASME Section VIII, Division 2 up to the design pressure of 1.96 MPa, beyond which the stress levels exceed the allowable limits. The proposed design offers a production-ready, code-compliant

solution suitable for large-scale applications in water treatment, petrochemical processing, and power generation systems, where structural integrity and long-term reliability are critical. The findings provide an industrially applicable framework for the design verification and optimization of large-diameter butterfly valves.

Keywords

Butterfly Valve, Elastic Stress Analysis, Design, Structural Evaluation, Large diameter

INTRODUCTION

Butterfly valves are extensively used in fluid-control systems owing to their compact structure [Wang 2024], lightweight design [Yuvaraj, 2022], and efficient flow regulation [Sotoodeh 2018]. Their widespread application in the water distribution, petrochemical, and power industries demands high structural reliability, especially for large-diameter valves subjected to complex loading conditions such as pressure fluctuations, cyclic stresses, and external forces [Sotoodeh, 2022] [Song, 2008].

The design of such pressure-retaining components is guided by standards including ASME B16.34 and B16.47 for wall thickness and flange dimensions, and MSS SP-67 for butterfly valve specifications [Sozinando 2025]. Structural integrity under operational loading conditions is further ensured through the ASME Boiler and Pressure Vessel Code (BPVC), Section VIII, Division 2, which provides criteria for plastic collapse, fatigue, and deformation control [Hossain et al., 2009]. Stress classification methods—encompassing membrane, bending, and secondary stresses—are commonly used to evaluate code compliance [Iyer, 2004].

Recent advances in finite element analysis (FEA) have enabled the detailed prediction of the behaviour of mechanical systems under realistic pressure and boundary conditions, offering a powerful complement to analytical design procedures [Earl 2018] [Thakare, 2019] [Pal 2025]. Integrating FEA with standard-based analytical methods enhances design accuracy and enables optimization for strength and material efficiency [Yuvaraj, 2025] [Pal 2024a, b].

This study aims to design and structurally evaluate a large-diameter butterfly valve in accordance with ASME and MSS standards to ensure its safety, reliability, and compliance under operational pressure conditions. The primary objective is to establish a validated design framework by integrating analytical design calculations with FEA for assessing deformation, stress distribution, and failure limits.

MATERIALS AND METHODS

The material used for the outer shell/body and the disc of the butterfly valve was ASTM A216 WCB [Goldman, 1976]. ASTM A216 WCB is a cast carbon steel grade widely used for the manufacture of flanges and valves. The major elemental composition consists of carbon (0.28 wt.%), manganese (0.65 wt.%), phosphorus (0.02 wt.%), and sulphur (0.030 wt.%). The minimum tensile strength of the material was 485 MPa, with a minimum yield strength of 250 MPa and a total elongation of 25%, as specified in the ASME Boiler and Pressure Vessel Code (BPVC), Section II, Parts A and D. Additionally, the yield strength of the material was 138 MPa at room temperature.

The valve stem was manufactured from SA-182 F316 material. The major alloying elements of the material are carbon (0.08 wt.%), manganese (2.0 wt.%), phosphorus (0.045 wt.%), sulphur (0.030 wt.%), Silicon – wt.%, nickel (12 wt.%), chromium (18 wt.%), and molybdenum (2–3 wt.%). The valve was designed for a temperature range of –29°C to 38°C with a design pressure (P)

of 19.6 bar (1.96 MPa). However, in the present investigation, a maximum pressure of 2.94 MPa (1.5×1.96 MPa) was considered. The operating pressure is lower than the design pressure of the valve. In this study, pressures both below and above the design pressure were evaluated to compare the resulting stress levels. The valve was analysed in the fully closed position, where the maximum internal pressure is experienced. SOLIDWORKS 2024 software was used to create the CAD model. The CAD model was exported as a STEP file and imported into ANSYS for analysis. The end flanges were designed in accordance with ASME B16.47 Pipe Flanges and Flanged Fittings. Various simulations of the valve body were performed using ANSYS Workbench 2023 R1. The model geometry was modified to remove unnecessary geometric features in ANSYS Workbench 2023. A tetrahedral mesh was generated to accurately capture the complex valve geometry. The mesh was refined near high-stress regions such as the disc-stem junction and flange seat to ensure the accuracy of the results. The final mesh size was determined based on a mesh convergence study, in which successive refinements were performed until variations in maximum stress and deformation were within 5% between iterations.

Boundary conditions were applied as follows:

1. Pressure Load: A uniform internal pressure ranging from 0.98 MPa to 2.94 MPa was applied to the inner surface of the valve body and to one side of the disc to simulate operational conditions.
2. Fixed Support: The flange face was fully constrained to represent the restraint provided by the bolted connection.
3. Gravity: Standard Earth gravity (9.81 m/s^2) was applied in the vertical direction to account for the weight of the components. A static structural analysis was performed to evaluate the structural behaviour of the valve body under the applied loading conditions.

RESULTS AND DISCUSSION

Body wall thickness

Mandatory appendix VI for 150 class standards:

Table 1 shows the minimum valve wall thickness as per ASME B16.34 Mandatory appendix VI for 150 class.

Valve pressure class, P_c	Inside diameter in mm	Metric Equation t_m (mm)
150	$3 \leq d < 50$	$t_m(150) = 0.064d + 2.34$
150	$50 \leq d \leq 100$	$t_m(150) = 0.020d + 4.5$
150	$100 < d \leq 1300$	$t_m(150) = 0.0163d + 4.7$

Table 1. Minimum valve wall thickness equations

For Class 300 valves with a diameter of 48 inches (1200 mm), Equation (3) from Table 1 was used. According to the equation, the minimum wall thickness was calculated to be 24.26 mm. The bore diameter was specified according to the API 6D standard as 1200 mm. A corrosion allowance of 3 mm was added to the calculated thickness in accordance with the API standard, resulting in a required wall thickness of 27.26 mm.

MSS SP-67 standard :

For the design of butterfly valves, MSS SP-67 is a widely accepted standard. Equation 1 provides minimum wall thickness for making the valve.

$$t_m = \frac{1.5 \times P \times d}{2S - 1.2 P} \quad (1)$$

where:

P = Pressure rating at temperature under consideration, psi

d = Inside diameter of valve body (or shaft bore in body for the body neck of flangeless valves), in.

S = Allowable stress, psi

The wall thickness of the material as per MSS SP-67 was calculated to be 12.9 mm for making butterfly valve. Adding 3 mm corrosion allowance the wall thickness becomes 15.9 mm. By taking the average value obtained from both standards, the final wall thickness was determined to be 21.58 mm.

Designing and flange body

The flange on the body was designed as per ASME B16.47 Pipe Flanges and Flanged Fittings. Figure 1 (a) shows the design of the flange, including the outer diameter, inner diameter, number of bolt holes, and bolt pitch circle diameter (PCD). The raised face of the valve was kept as 2 mm, face to face dimension was 656.40 mm and flange thickness was found to be 106.40 mm as per ASME B16.47, 150 class body as shown in Figure 1 (b).

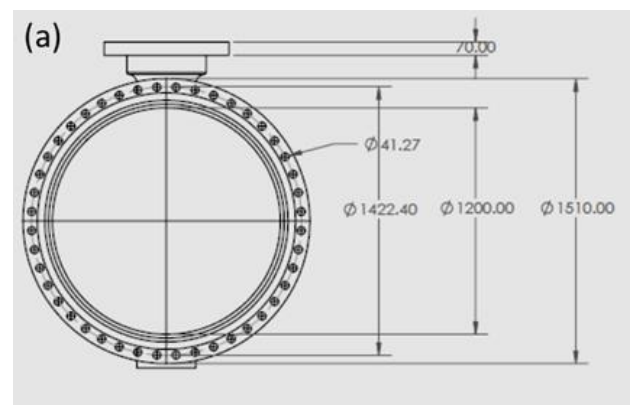


Figure 1a. Designing of the flange with outer and inner diameter (all dimensions are in mm)

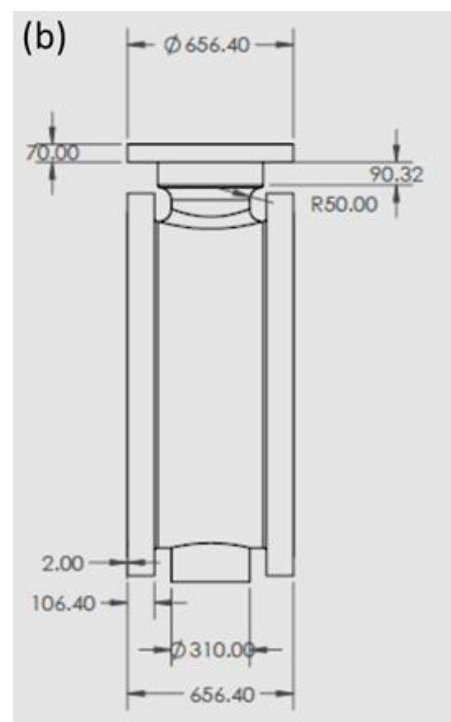


Figure 1b. Face to face and raised face dimensions (all dimensions are in mm)

Figure 2 shows the various body neck dimensions as per ASME B16.34. The empirical equations were used to calculate the value of m , n and L . The value of m and n should be greater than body valve thickness also it should withstand the dead weight of

external bodies like actuator, gearbox etc. Hence, the values were calculated to be 105 mm, 146 mm respectively.

$$m = 6 \times t_m \quad (2)$$

$$n = 8 \times t_m \quad (3)$$

$$L = t_m \left(1 + 1.1 \sqrt{\frac{d}{t_m}} \right) \quad (4)$$

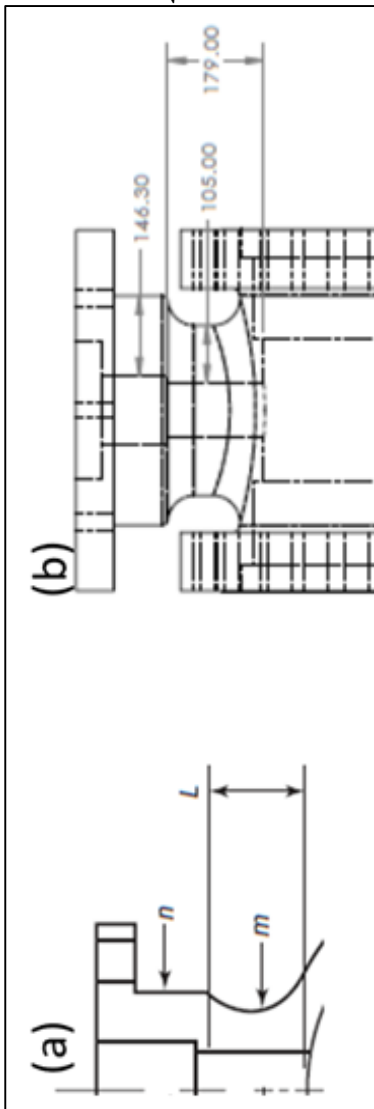


Figure 2. Calculation of various body neck parts as per ASME B16.34 (all dimensions are in mm)

Designing of disc

Figure 3 shows the designed disc of the butterfly valve. The diameter of the disc was calculated such that the 98% of the water could pass through the valve when it was fully open. Therefore, an outer disc diameter of 1180 mm was adopted. The disc was designed with an elliptical profile so that it could withstand high-pressure loading without experiencing excessive deformation. The minimum thickness of the disc was calculated as per the equation 5. The minimum thickness was calculated to be 69.36 mm. After adding a 10-mm allowance, the minimum thickness was determined to be 80 mm. A 1:2 thickness ratio between the outer edge and the central region of the disc was adopted. The middle portion thickness was calculated to be 160 mm. After considering an additional allowance of 20 mm, the central thickness was determined to be 180 mm. Since no reinforcement was provided in the disc design, an additional thickness was incorporated at the centre of the disc to improve its structural rigidity under pressure loading.

$$\frac{D}{2} \times \sqrt{\frac{P \times K}{S_y}} \quad (5)$$

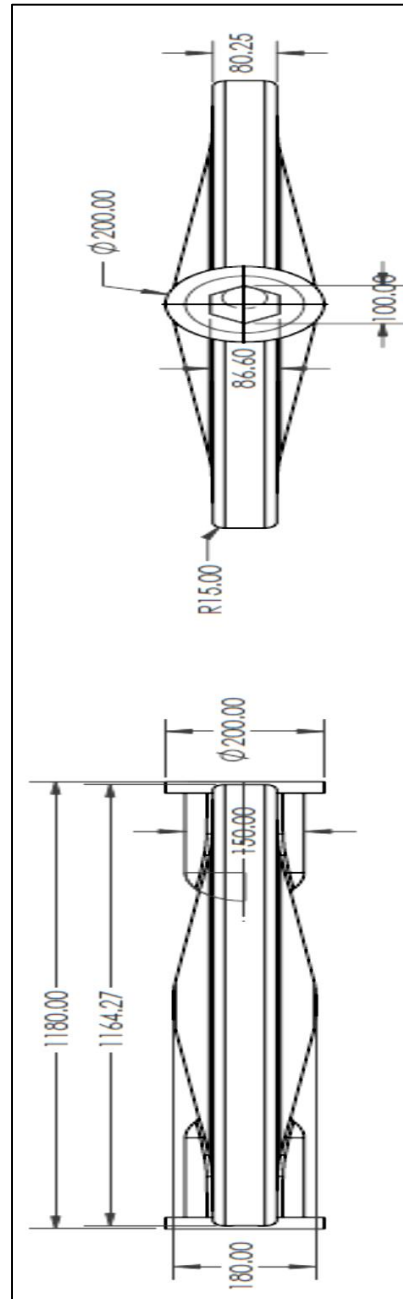


Figure 3. Designed Disc of Butterfly Valve (all dimensions are in mm)

Designing for stem and Pin

Figure 4 (a) shows the designed dimensions of the butterfly valve. The diameter of the stem was calculated using Equations (6) and (7).

$$T = C_t \times \text{bore diameter}^3 \times P \times 10^6 \quad (6)$$

$$d_{\text{stem}} = \left(\frac{16 \times T}{\pi \times \tau \times 10^6} \right)^{1/3} \quad (7)$$

Where, T is the torque in N-m, C_t is the torque coefficient (0.005), P is the design pressure in MPa, τ is the allowable shear stress ($0.6 \times$ allowable yield stress), bore diameter is the inner diameter of the body. The minimum diameter of the stem (d_{stem}) was calculated to be 82.67 mm. Hence, 100 mm was considered to be the stem diameter as shown in Figure 5. Figure 5 shows the actual image of the butterfly valve after assembly. The length of the stem (L_{stem}) was calculated to be 9 times the diameter of the stem.

$$L_{\text{stem}} = 9 \times d_{\text{stem}} \quad (8)$$

For the supporting pin located at the bottom of the valve, a similar diameter was adopted with a different length, as shown

in Figure 4(b). Figure 5 shows the complete assembly of the butterfly valve rendered in SolidWorks.

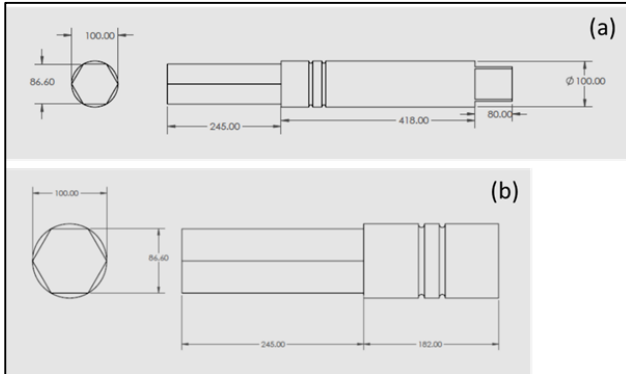


Figure 4. (a) Designed Stem of Butterfly Valve (b) Designed Pin of Butterfly Valve (all dimensions are in mm)



Figure 5. Rendered 3D image of a butterfly valve

CAD and FEA analysis of Butterfly Valve

Figure 6 shows the 3D model of the valve developed in ANSYS SpaceClaim. It can be observed that the studs and nuts were not included in the numerical model. The mesh consisted of 396,022 nodes and 238,111 elements. A refined mesh was applied in the bolt-imprint regions using face sizing to improve the accuracy of the results. Tetrahedral mesh was used throughout the valve body. Static structural analysis was performed for the elastic stress analysis. The fatigue design procedure is described in Section 5.5 of ASME VIII-2, entitled "Protection Against Failure from Cyclic Loading". It includes three detailed fatigue design methods: elastic stress analysis (equivalent stress), elastic-plastic stress analysis (equivalent strain) and elastic stress analysis (structural stress).

The effective total equivalent stress amplitude is used to assess fatigue damage based on the results obtained from linear elastic stress analysis. For fatigue evaluation, the governing stress is the effective total equivalent stress amplitude, which is defined as half of the effective total equivalent stress range ($S_a = P_L + P_b + Q + F$) calculated for each cycle in the loading histogram. The equivalent stress, also referred to as the primary + secondary + peak equivalent stress (see Figure 9), represents the

maximum value through the thickness of a section obtained from the combined effects of primary, secondary, and peak stresses. These stresses arise from the specified operating pressures, other mechanical loads, and general and local thermal effects, and they also account for the influence of both gross and local structural discontinuities.

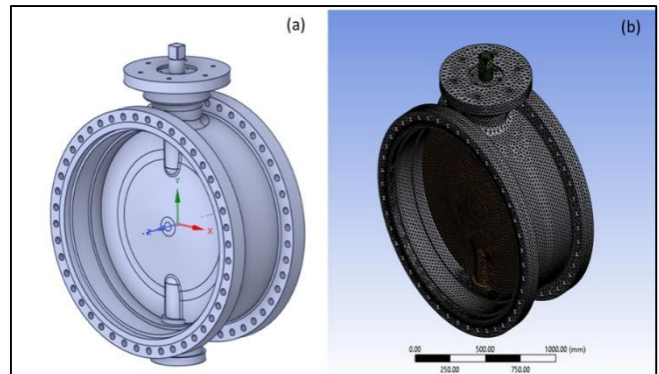


Figure 6. (a) 3D model of a butterfly valve made developed in SpaceClaim (b) Mesh graph of a Butterfly valve

The various stress categories considered in the analysis are as follows:

- General Primary Membrane Equivalent Stress (P_m):** The uniform membrane stress distributed across the entire section.
- Local Primary Membrane Equivalent Stress (P_L):** The membrane stress localized within a specific region of the section.
- Primary Bending Equivalent Stress (P_b):** The bending stress resulting from primary loads.
- Secondary Equivalent Stress (Q):** The stress produced by displacement-controlled conditions such as thermal gradients or constraints.
- Peak Equivalent Stress (F):** The additional stress arising from local stress concentrations or localized thermal effects occurring above the nominal ($P + Q$) stress level.

In the present study, primary membrane and primary bending stresses contribute to the dominant stress field, while secondary stresses are negligible due to the absence of thermal loading. (No thermal loading was applied in the numerical model).

Loads and Boundary Conditions

- Symmetry constraints were applied on the symmetry plane.
- To prevent rigid body-motion, displacement was restricted in the Z-direction at the nozzle end-face.
- Internal pressure was applied to the inner surface of the model.
- The flange-end joints were fixed.
- Standard gravitational loading was applied in the downward direction.

Finite Element Analysis Results

Deformation

Figure 7 shows the total deformation of the valve body. The valve was kept at closed condition where the maximum pressure on the valve is experienced. From the figure it is evident that the maximum deformation occurs in the valve disc. The valve body exhibited negligible deformation, which may be attributed to the sufficient wall thickness adopted in the design. The maximum deformation was found to be 3.4 mm. The sealing material EPDM (ethylene propylene diene monomer), was used between the body and the disc. This helps in reducing the deformation of the disc and sealing the gaps between them. Therefore, the design can be considered safe under the given pressure load.

Stiffeners can be added at the back side of the disc to reduce more deformation. The absence of significant deformation on the valve body confirms its structural stability for subsequent analyses. Table 2 shows the deformation at various pressure levels below 2.94 MPa. As evident from the analysis, the deformation decreases with decreasing pressure magnitude. Figure 8 shows the equivalent stress (Huber–Mises stresses) at the valve body. It can be observed that the stresses are primarily concentrated in the stem and pin regions of the valve. The stress distribution in the valve body indicates that the stresses remain below the yield strength of the material. However, higher stresses were observed in the bottom neck region, which was therefore selected for further analysis.

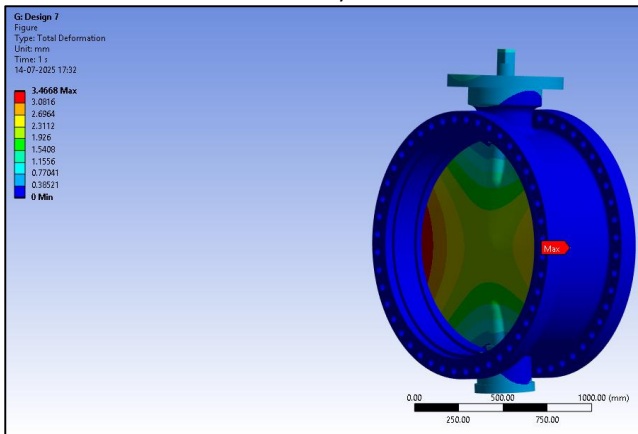


Figure 7. Deformation analysis of valve

Pressure Magnitude [MPa]	Total Deformation Maximum [mm]	Equivalent Stress Maximum [MPa]
2.94	3.54	1165
1.96	2.36	776
0.9	1.08	356

Table 2. Total Deformation and Maximum Equivalents stress for various internal pressures values.

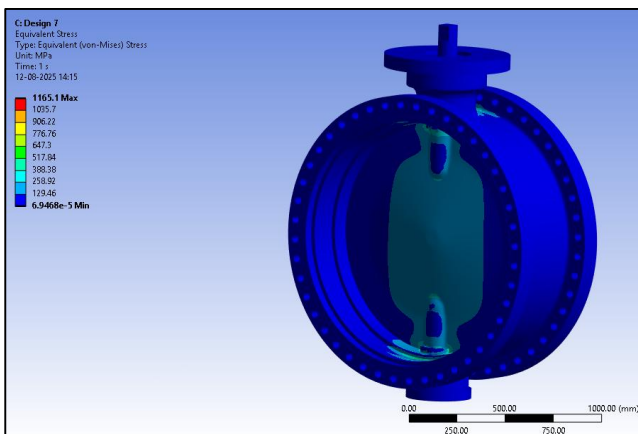


Figure 8. Huber–Mises's stress analysis of valve

Stress Intensity Analysis

- Protection against plastic collapse shall be demonstrated by performing an elastic stress analysis of the component under the specified loading conditions. The resulting stresses shall be classified and compared to the applicable limiting values.
- The three categories of equivalent stress and the corresponding allowable limits required for protection against plastic collapse are defined in Figure 9.
- The stress classifications used for elastic analysis, including general primary membrane stress, local primary membrane

stress, primary bending stress, secondary stress, and peak stress, are shown in Figure 9. Figure 8 shows the distribution of Huber–Mises equivalent stress across the valve surface. It is evident that stress concentrations occur primarily near the stem and the bottom neck regions. At 2.94 MPa, the equivalent stress exceeds the allowable stress limits, indicating potential structural vulnerability under this load case. However, elastic stress analysis confirms that the selected wall thickness is safe under the specified loading conditions. Figure 9 shows the Stress Classification Line (SCL) drawn across the highly stressed regions of the valve, perpendicular to the surface. Linearized equivalent stresses were analysed over the SCL and values of P_m and P_b were obtained. Table 3 shows the value obtained by the Linearized equivalent stress over SCL. The maximum value of $P_m + P_b$ was found to be 288.14 MPa. To evaluate whether the stress state satisfies the safety criteria, ASME BPVC Section VIII, Div. II provides criteria under which the pressure vessel (valve body) can be considered safe for design.

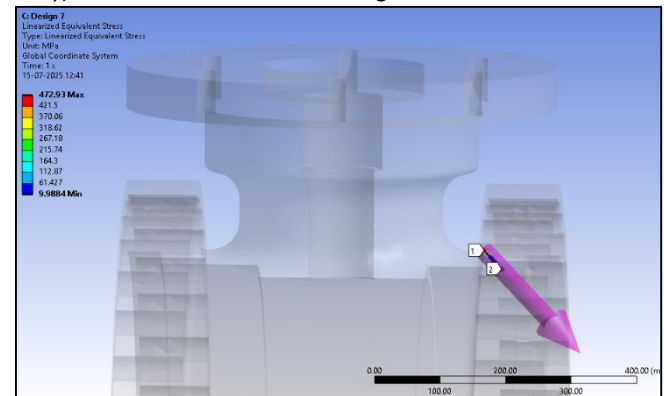


Figure 9. Stress classification line drawn over the highly stressed area

Design condition

$$P_m \leq S_y \quad (15)$$

$$P_m + P_b \leq 1.5S_y \quad (16)$$

The above equation provides the design criteria for wall thickness of a pressure vessel defined by minimum valve wall thickness as per ASME B16.34 Mandatory appendix VI.

Test condition

Primary Membrane Stress, P_m

$$P_m \leq \beta T S_y \quad (17)$$

Primary Membrane Stress Plus Primary Bending Stress $P_m + P_b$.

Case 1

$$P_m \leq S_y/1.5 \text{ then } P_m + P_b \leq \gamma_{\min} S_y \quad (18)$$

Case 2

$$\frac{S_y}{1.5} < P_m < \beta T S_y \text{ then } P_m + P_b \leq \left(\frac{1 - \gamma_{\min}}{\beta T - \left(\frac{1}{1.5} \right)} \right) P_m - \left[\left(\frac{1 - \gamma_{\min}}{\beta T - \left(\frac{1}{1.5} \right)} \right) \beta T - 1 \right] S_y \quad (19)$$

Sl. No.	Length	P_m , Mpa	P_b , Mpa	$(P_m + P_b)$, MPa
1	0	62.98	236.53	288.14
2	0.9012	62.98	226.67	278.39
3	1.8024	62.98	216.82	268.64
4	2.7036	62.98	206.96	258.89
5	3.6048	62.98	197.11	249.16
6	4.506	62.98	187.25	239.44
7	5.4072	62.98	177.39	229.72
8	6.3084	62.98	167.54	220.02
9	7.2096	62.98	157.68	210.34

10	8.1108	62.98	147.83	200.67
11	9.012	62.98	137.97	191.02
12	9.9132	62.98	128.12	181.39
13	10.814	62.98	118.26	171.79
14	11.716	62.98	108.41	162.22
15	12.617	62.98	98.553	152.68
16	13.518	62.98	88.697	143.19
17	14.419	62.98	78.842	133.75
18	15.32	62.98	68.987	124.37
19	16.222	62.98	59.132	115.07
20	17.123	62.98	49.276	105.88
21	18.024	62.98	39.421	96.815
22	18.925	62.98	29.566	87.921
23	19.826	62.98	19.711	79.255
24	20.728	62.98	9.8553	70.9
25	21.629	62.98	5.9632	62.98
26	22.53	62.98	9.8553	55.682
27	23.431	62.98	19.711	49.282
28	24.332	62.98	29.566	44.172
29	25.234	62.98	39.421	40.839
30	26.135	62.98	49.276	39.734
31	27.036	62.98	59.132	41.037
32	27.937	62.98	68.987	44.536
33	28.838	62.98	78.842	49.771
34	29.74	62.98	88.697	56.26
35	30.641	62.98	98.553	63.619
36	31.542	62.98	108.41	71.581
37	32.443	62.98	118.26	79.966
38	33.344	62.98	128.12	88.654
39	34.246	62.98	137.97	97.564
40	35.147	62.98	147.83	106.64
41	36.048	62.98	157.68	115.84
42	36.949	62.98	167.54	125.15
43	37.85	62.98	177.39	134.53
44	38.752	62.98	187.25	143.97
45	39.653	62.98	197.11	153.47
46	40.554	62.98	206.96	163.01
47	41.455	62.98	216.82	172.59
48	42.356	62.98	226.67	182.19
49	43.258	62.98	236.53	191.82

Table 3. Values obtained by the Linearized equivalent stress over SCL for 2.94 Mpa Pressure

The elastic stress classification results indicate that the valve body satisfies the plastic collapse criteria of ASME Section VIII Division 2 up to the design pressure of 1.96 MPa. At 2.94 MPa ($1.5 \times$ design pressure), the linearized primary membrane plus bending stresses exceed the allowable limits prescribed by the Code. Therefore, 2.94 MPa is considered a verification load case only and does not represent an allowable operating condition.

Fatigue Analysis

Figure 10 shows the fatigue analysis of a butterfly valve. Fatigue assessment was performed in accordance with Section 5.5 of ASME Section VIII, Division 2 using the elastic stress analysis

method (equivalent stress approach). The fatigue evaluation procedure consisted of the following steps:

1. The equivalent total stress range was extracted from the linear elastic FEA results at critical locations identified by the stress classification line (SCL).
2. The effective total equivalent stress amplitude was calculated as one-half of the equivalent stress range.
3. A fully reversed loading condition ($R = -1$) was assumed to represent the most conservative cyclic pressure fluctuation.
4. The ASME elliptical mean stress correction model was applied.
5. The design fatigue curve for carbon steel (ASME VIII-2) was used for life estimation.

The maximum equivalent stress amplitude at 1.96 MPa was found to be within the allowable fatigue strength limit corresponding to 10^6 cycles. The calculated fatigue damage factor was below unity ($D < 1$), satisfying the cumulative damage criterion. At 2.94 MPa, although static plastic collapse limits were exceeded, fatigue evaluation was not considered governing since the design pressure qualification was not met.

	Pressure [MPa]		Design condition	Test Condition		
	Linearized Equivalent Stress Membrane [MPa]	Linearized Equivalent Stress Membrane+Bending [MPa]		Condition 1 (eq.15)	Condition 2 (eq.16)	P_m (eq.17)
						$P_m + P_b$ Case 1 (eq.18)
						(eq.19)
2.94	62.98	288.14	Qualified	Not Qualified	Qualified	Not Safe
1.96	39.49	191.22	Qualified	Qualified	Qualified	Safe
0.9	18.01	87.19	Qualified	Qualified	Qualified	Safe
						Not Required

Table 4. Design and test condition for safe design

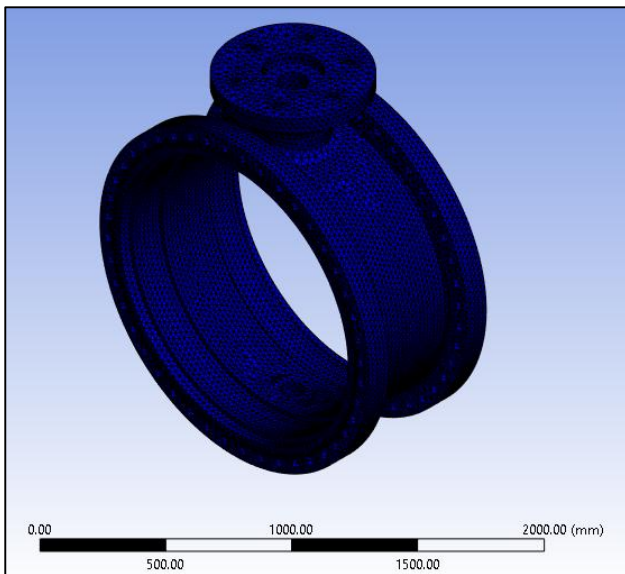


Figure 10. Fatigue analysis results of valve

Figure 10 illustrates the fatigue life contour plot of the valve body at 1.96 MPa internal pressure under fully reversed loading. The minimum fatigue life observed in the model exceeded 10^6 cycles, corresponding to infinite life criteria for carbon steel components under elastic conditions. The maximum cumulative fatigue damage was below 1.0, confirming compliance with ASME fatigue acceptance requirements.

CONCLUSION

The design and structural evaluation of a large-diameter butterfly valve were performed in accordance with ASME B16.34, ASME B16.47, MSS SP-67, and ASME Section VIII, Division 2 standards. Analytical calculations defined the minimum wall thickness (21.58 mm) and key geometric parameters for the body, disc, and stem, ensuring dimensional compliance with governing codes.

1. FEA verified the analytical design and revealed a maximum deformation of 3.54 mm in the disc under 2.94 MPa pressure, with stress concentrations primarily in the stem and pin regions. Stress classification analysis confirmed that the valve satisfies the ASME plastic collapse limits up to 1.96 MPa, beyond which allowable stress values are exceeded. Fatigue assessment further indicated an acceptable life under cyclic loading within the safe pressure range.

2. The valve design is structurally qualified for continuous operation up to the specified design pressure of 1.96 MPa. The 2.94 MPa case corresponds to a $1.5 \times$ design pressure verification load and is used solely for comparative stress evaluation. Since the stress limits exceed allowable values under this condition, operation above the design pressure of 1.96 MPa is not permitted.

ACKNOWLEDGEMENT

The authors extend their acknowledgment to the financial support of the European Union under the REFRESH-Research Excellence For REgion Sustainability and High-tech Industries project number CZ.10.03.01/00/22_003/0000048 via the Operational Programme Just Transition and has been done in connection with project Students Grant Competition SP2026/062 „Specific research on progressive and sustainable production technologies“ and SP2026/063 „Specific research on innovative and progressive manufacturing technologies“

financed by the Ministry of Education, Youth and Sports and Faculty of Mechanical Engineering VŠB-TUO.

REFERENCES

- [Earl 2018] Earl, E. and Mohammadi, H. Improving finite element results in modeling heart valve mechanics. *Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine*, 2018, Vol.232, No.7, pp.718–725. <https://doi.org/10.1177/0954411918780150>
- [Goldman 1977] Goldman, H. L. U.S. Patent No. 4,038,734. Washington, DC: U.S. Patent and Trademark Office, 1977.
- [Hossain 2009] Hossain, M. M., Reinhardt, W. D., and Seshadri, R. Simplified Stress Categorization Using a Single Linear Elastic Analysis. *ASME Journal of Pressure Vessel Technology*, December 2009, Vol.131, No.6, pp.061204. <https://doi.org/10.1115/1.4000199>
- [Iyer 2004] Iyer, S. Stress classification and assessment of locally thinned class 1 piping components. In: *ASME Pressure Vessels and Piping Conference*, January 2004. pp.143–153. <https://doi.org/10.1115/PVP2004-2250>
- [Pal 2024a] Pal, S., Kalita, K., and Haldar, S. Comparison of nature-inspired algorithms in finite element-based metaheuristic optimisation of laminated shells. *Expert Systems*, 2024, Vol.41, No.9, pp.e13620. <https://doi.org/10.1111/exsy.13620>
- [Pal 2024b] Pal, S., Kalita, K., and Haldar, S. Grey wolf optimizer based-natural frequency optimization of laminated composite shells carrying distributed mass. *Mechanics Based Design of Structures and Machines*, 2024, Vol.52, No.11, pp.8509-8543. <https://doi.org/10.1080/15397734.2024.2321971>
- [Pal 2025] Pal, S., Kalita, K., and Haldar, S. Optimized Free Vibration Design of Laminated Composite Shells with Central Cut-outs: A Metaheuristic Approach. *European Journal of Mechanics - A/Solids*, 2025, 105944. <https://doi.org/10.1016/j.euromechsol.2025.105944>
- [Song 2008] Song, X., Wang, L., and Park, Y. Fluid and structural analysis of large butterfly valve. In: *AIP Conference Proceedings*, October 2008, Vol.1052, No.1, pp.311–314. American Institute of Physics. <https://doi.org/10.1063/1.3008687>
- [Sotoodeh 2018] Sotoodeh, K. Selecting a butterfly valve instead of a globe valve for fluid control in a utility service in the offshore industry (based on an industrial experience). *American Journal of Mechanical Engineering*, 2018, Vol.6, No.1, pp.27–31. <https://doi.org/10.12691/ajme-6-1-4>
- [Sotoodeh 2022] Sotoodeh, K. Safety and reliability analysis for butterfly valves in the offshore oil and gas industry. *Safety and Reliability*, April 2022, Vol.41, No.2, pp.121–143. <https://doi.org/10.1080/09617353.2022.2107254>
- [Sozinando 2025] Sozinando, D. F., Tchomeni, B. X., and Alugongo, A. A. FEA-Based Vibration Modal Analysis and CFD Assessment of Flow Patterns in a Concentric Double-Flange Butterfly Valve Across Multiple Opening Angles. *Vibration*, 2025, Vol.8, No.3, pp.42.
- [Thakare 2019] Thakare, A. J. and Keche, A. J. Optimization of Valve-Train Components for Durability Analysis by Using Finite Element Analysis Method. In: *Emerging*

Trends in Mechanical Engineering: Select Proceedings of ICETME 2018, Singapore: Springer Singapore, 2019, pp.11–24. https://doi.org/10.1007/978-981-32-9931-3_2

[Wang 2024] Wang, D., Wang, S., Gao, B., Guo, M., Wei, Z., and Feng, J. An in-position measurement method of point-replacing surface for triple-eccentric butterfly valve precision machining. The International Journal of Advanced Manufacturing Technology, 2024, Vol.133, No.5, pp.2607–2621.

[Yuvaraj 2022] Yuvaraj, K. and Arun Kumar, G. Study on characteristics and performance of butterfly valve using numerical methods – A review. In: AIP Conference Proceedings, August 2022, Vol.2460,

No.1, p.030004. AIP Publishing LLC. <https://doi.org/10.1063/5.0095622>

[Yuvaraj 2025] Yuvaraj, K. and Arun Kumar, G. Fatigue failure and structural integrity analysis of a 3-inch butterfly valve shaft using finite element modelling and microstructural evaluation. Advances in Science and Technology Research Journal, 2025, Vol.19, No.5, pp.440–453.

CONTACTS :

Lenka Cepova, Ing. PhD.

VSB-TUO, Faculty of Mechanical Engineering

17. listopadu 2172/15, 708 00 Ostrava, Czechia

lenka.cepova@vsb.cz, 59 732 4396

LICENSE CREATIVE COMMONS:

The article is published under the terms and conditions of the Creative Commons Attribution 4.0 International License (CC BY 4.0).