

PAPER • OPEN ACCESS

Simulation-enabled Design and Analysis of Globe Valve

To cite this article: Nu'man Al-Basyir Mohd Nuri Al-Amin *et al* 2021 *J. Phys.: Conf. Ser.* **2051** 012025

View the [article online](#) for updates and enhancements.

You may also like

- [Experimental analysis of reed valve movement for different reed valve designs tested in an impact fatigue test system](#)
Muhammad Waqas Tofique, Alexander Löf and Chris Millward
- [Numerical Simulation on Hydraulic Characteristics of Spray Regulator Valve](#)
ZHOU Long-yu, Hao Si-ja, YANY Zhi-da et al.
- [Failure Mechanism and Prevention Research of Adhesive Wear in the Guide Part of Globe Valve](#)
Shaojiang Chen, Qing Zhang, Yilong Sun et al.

Simulation-enabled Design and Analysis of Globe Valve

Nu'man Al-Basyir Mohd Nuri Al-Amin¹, Ahmad Majdi Abdul Rani¹, Mohd Norhisyam Awang², M Nasir A Rani², Nur Athirah Ishar², Harvey M. Thompson³, M Hafizun Hassan¹, Bushra Mohamad Zakir¹ and Ahmad Tarmizi Hassan¹

¹ Department of Mechanical Engineering, Universiti Teknologi PETRONAS, 32610 Seri Iskandar, Perak Darul Ridzuan, Malaysia

² PETRONAS Chemicals Fertiliser Kedah Sdn Bhd, Jln Jeniang, 08300 Gurun, Kedah Darul Aman, Malaysia

³ School of Mechanical Engineering, University of Leeds, Woodhouse, Leeds LS2 9JT, United Kingdom

albasyirnuman@gmail.com

Abstract. Globe valves serve many purposes in oil and gas facilities as a tool to control fluid flow in pipelines. However, older designs of globe valves suffer many problems and faults, mainly from valve leakage and valve passing. In mitigating these problems, this paper will improve the design of a globe valve to enhance its fluid flow performance and reduce the pressure difference value throughout the valve. The improvements on the valve will be validated from the original design with the help of Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA). The project concluded that the improved valve design performs better than the original design in all aspects except in terms of valve passing at the contact seal between the seat and the disk in a fully closed position. Implementation of smart sensors and prototyping will be explored in future studies.

1. Introduction

1.1. Project Background

Valves are devices that functions to regulate, direct, or control the flow of a fluid by opening, closing, or partially obstructing various passageways. In piping systems, valves are installed to halt fluid flow through pipe sections, redirect flow of fluid from one pipeline to another, for maintenance by isolating the section of piping where the maintenance needs to be done and to avoid leakage in emergencies by shutting down the flow of fluid to the effected piping section. In the oil and gas industry, various types of valves are used depending on the operation type as different types of valves have different use cases. Globe valves are an example of a common type of valve to be used in offshore platforms and onshore oil and gas terminals. Globe valves have a distinct geometry compared to other type of valves. The flow of fluids through the valve will change from horizontal flow to vertical up flow following the valve's geometry before returning to the normal horizontal flow of the fluid. Most globe valves are manually operated by hand. Furthermore, globe valves can be used to regulate flow or pressure as well as completely halting flow, unlike gate valves. However, when compared to gate or ball valves, in fully open position, globe valves suffer from more exponential pressure loss because the geometry of the valves dictates the flow of fluids to change direction when passing through the valve [1].

1.2. Problem Statement



Content from this work may be used under the terms of the [Creative Commons Attribution 3.0 licence](https://creativecommons.org/licenses/by/3.0/). Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

PETRONAS Chemicals Fertiliser Kedah (PCFK) is a facility under the PETRONAS brand which mostly produces nitrogenous fertilisers for industrial and general use. The PCFK facility utilises many types of valves in their plant for various specific functions related to their operations. However, they have been facing problems with many of their manually operated valves especially globe valves as they tend to fail without prior warning resulting in huge losses due to downtime for repairs and replacements. The major problems related to these valves are leakage and passing due to wear and tear. Valve passing occurs when fluid can still flow through a valve even though the valve is in a fully closed position in which all fluid flow should be halted. Valve leakage however is when fluid flowing through the valve escapes to the environment, usually from the flange, gland packing and gasket area of the valve. With that, this project aims to solve these problems faced by PCFK by running Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) simulations on previous globe valve designs and improving on that design based on the results of the simulation, identifying optimal location on the globe valve to attach sensors for early fault detection and fabricating the prototype of the improved valve using 3D printing.

1.3. Objectives and Scope of Studies

To objectives of this project are listed as follows:

1. To improve the design of globe valve to mitigate leaking and passing using CFD and FEA simulation.
2. To identify the optimal location on the globe valve to attach sensors for early fault detection.
3. To fabricate the prototype of an intelligent globe valve using metal additive manufacturing.

This project is a continuation of a previous project where PCFK supplied a globe valve used in their facility to UTP. In the previous project, the valve was CNC wire-cut in half so that the internal geometry of the valve can be scanned using a 3D laser scanner [2]. Ultimately, the process is done to reverse engineer a geometrically accurate CAD drawing of the valve so that improvements on the design of the valve can be done. Moreover, the finished CAD drawing of the valve is also sent to the University of Leeds as a collaboration with UTP as their students will run their own version of the project to improve the design of the globe valve. This improved design from the University of Leeds will further be utilized in this project. The paper focuses on the 3/4" 150lb Raised Face (RF)/Bolted Bonnet (BB)/ A105 Full Bore globe valve obtained from PCFK. The standards and specifications of this valve are given from the assembly drawing from PCFK, some of which include references such as ASME B16.34 [3], API 602 [4], and BS 5352 [5]. This project will focus on the comparative study between the original and the improved globe valve model using CFD and FEA. Objectives relating to implementation of sensors and prototyping will be addressed in future studies because of project constraints and the COVID-19 pandemic.

2. Methodology

2.1. Globe valve design improvement

The provided globe valve contains several potentially problematic geometries where high-pressure loss of fluid can occur. These geometries have faces that can potentially block part of the flow and constraint the flow chamber, especially around the seat area.

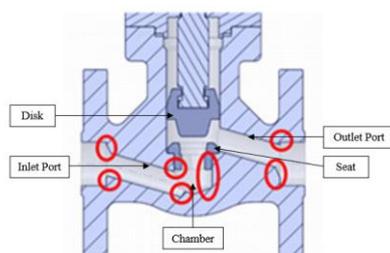


Figure 1. Problematic geometry in original globe valve design.

The problematic geometries highlighted in red in figure 1 contain faces that are perpendicular to the flow which can cause high levels of stagnation around the inlet and outlet port of the valve and the chamber at the bottom of the seat of the valve. These stagnations equate to high pressure loss across the valve and also creates high pressure spots along the valve walls that can be the cause of leakage or passing due to high pressure. These faces can also potentially cause flow separation as it produces sharp corners for fluids when passing over these areas [6]. In optimizing the problematic geometry in the valve, the valve design is improved. In figure 2, the improvements are shown where the inlet and outlet ports are widened and extruded according to the initial diameter of the inlet and outlet at the flange area to remove perpendicular faces surrounding the port sections. The bottom chamber is also optimized to have a pipe-like bend and is more naturally merged with the inlet port to remove the sharp edge protruding from the bottom chamber. The curvature introduced to the bottom chamber will remove the perpendicular wall from the original design and will help redirect flow of fluids better and more naturally to the direction of the flow. Furthermore, the seat is merged with the valve body to remove the threaded connection between the seat and the body from the original design which will eliminate the possibility of valve passing from unsecure connection of the two parts.

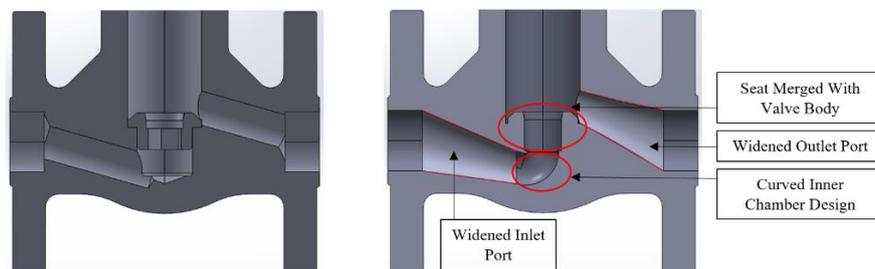


Figure 2. Original structure (left) vs optimised structure (right).

The improvements done to the valve complies with ASME B16.34 [3] standards where a class 150 globe valve with 21 mm inner diameter must comply to the minimum wall thickness of 3.5 mm for the valve body. When measured, the spot with the thinnest wall thickness for the improved valve body is at 7.7 mm, which proves the safety and compliance of this design improvement.

2.2. CFD workflow

In comparing the improvements of fluid flow between the original and the improved valve design, ANSYS R19.1 Fluent is used as the CFD software of choice. The valve is configured in the fully open position as the software requires fluid flow to be in motion to simulate the flow through the valve body and the simulation cannot be done if there is zero flow. The inlet port entrance and outlet port exit are also extended by 100 mm to simulate the initial flow of fluid from the pipeline into the valve and the properties of the flow in the pipeline after exiting the valve.

2.2.1. Volume extraction and meshing. The process starts with extracting the volume of the valve CAD drawings to be used as the model in the simulation. The process then continues in the meshing stage where firstly the inlet and outlet of the valve is defined. Then the meshing is done where the target skewness of the mesh is set to 0.85. In selecting the proper element size for the mesh, mesh independence study is done to find the middle point between mesh accuracy and computational demands. From the study, 0.6 mm is chosen because the margin of error is below 1% [6] while still be able to be computed without issues in a short time duration. Furthermore, inflation layer is used to enhance the mesh quality near the valve walls where the boundary layer flow is. In order to reduce the number of elements and minimize computational needs in solving the simulation, hex mesh is used for simpler geometries at the cylindrical shape of the pipeline at the extended inlet port entrance and outlet port exit. Meanwhile, the complex geometry of the valve body is applied with tetra mesh for more accurate results.

2.2.2. Turbulence model and boundary conditions. The turbulence model used for the simulation is k- ϵ realizable with enhanced wall treatment enabled to complement the inflation mesh layer for accurate

boundary layer calculation. This model shows superiority over standard and RNG k- ϵ model where it is able to capture the mean flow of complex structures and contains a new formulation for turbulent viscosity [7]. Ethylene glycol is the fluid chosen to replace sulphuric acid which is used in the PCFK plant because of its unavailability in the software database as it has approximately the same density and dynamic viscosity as sulphuric acid. Boundary conditions are then applied to the inlet of the extracted volume of the valve. Firstly, the velocity of the fluid used is 6 ft/s. This value is extracted from API RP 14E [8] standard where the average of 3 ft/s minimum and 15 ft/s maximum velocity is used from the standard. Next, gauge pressure of 156800 Pa is applied to the inlet fluid as it is 80% of the rated pressure for 150 class valves from ASME B16.34 [3] standard and it is assumed that the standard working pressure of the valve is 80% of the rated pressure.

2.3. FEA workflow

For finite element analysis, ANSYS 19.1 Static Structural software is used to simulate and compare the total equivalent (von-Mises) stress and total deformation on the valve body between the original and the improved valve design when in a fully closed position. The analysis is done to simulate contained fluid pressure when the valve is in fully closed position to compare the resistance of the two models in terms of valve passing.

2.3.1. Material definition and meshing. The material of the body, seat, disk, stem, gland, and bonnet of the valve CAD drawing is defined where most of the parts are made of nickel-copper alloy except the body and bonnet which are made of carbon steel. Then, meshing is applied on the valve parts. Tetra mesh is fully utilized for the meshing of the model because of the complex shape throughout the valve body. The target quality value for the meshing is set to 0.05 and 0.8 mm element size is used instead of 0.6 mm because of hardware computing limitations. Inflation layer is also applied throughout the walls of the valve parts so that fluid pressure transferred to the walls will be simulated more accurately.

2.3.2. Input parameters and solution. In simulating the total equivalent (von-Mises) stress and deformation of the valve, input parameters are needed to feed required information to the software in order to numerically calculate the solution. Fixed supports are assigned to the faces in the bolt holes at the flange at both ends of the valve that supports and connects the valve to the pipeline. Pressure load is also then applied to all the inner faces of the valve from the inlet into the disk and seat connection where the contact seal is situated while in fully closed condition. The magnitude of the pressure used is the same as the one used in the CFD analysis which is 156800 Pa. Finally, before starting the calculation, equivalent stress and total deformation solution types are selected to be displayed as the result after the simulation has finished.

3. Results and discussion

3.1. CFD results and discussion

Table 1 shows the inlet pressure, outlet pressure and pressure drop between the original and improved valve obtained from the CFD simulation. In general, the higher the pressure drop value throughout the valve, the less the fluid pressure is transferred into the valve body and parts from pressure stagnation as the result of perpendicular faces obstructing the flow of the fluid. However, due to the difference in inlet pressure between the original and improved design given by the simulation, the pressure drop comparison between the two design is irrelevant.

Table 1. Original and improved design pressure difference.

Valve Design	Inlet Pressure (Pa)	Outlet Pressure (Pa)	Pressure Drop (Pa)
Original	114058	1.65	114056.35
Improved	109118	0.82	109117.18

The original design has a 4% higher inlet pressure value compared to the improved design. This is likely due to the constraint at the inlet port of the original design due to the perpendicular walls surrounding the inlet port which constricts flow and builds up pressure of the fluid along the pipeline inlet. This is shown in the pressure contour in figure 3 where for the original design, the pressure drops after entering the inlet port while the improved design maintains the same pressure until the fluid reaches the inner chamber of the valve. The higher inlet pressure and the pressure build-up around the constraint area around the flange entering the inlet port of the original design may cause more strain to the inlet flange connection between the valve and the pipeline over time, increasing the probability of leaks occurring in the original valve due to wear and tear from high pressure. Because of the inlet port constraint on the original valve, fluid flow also becomes more turbulent and erratic, creating high-pressure spots on the inner valve walls. Among them, the ones directly under the seat connection to the body of the original valve wall can be a factor in contributing to valve passing through continuous high-pressure fluid flow to the spot causing wear and tear leading to leak spots around the threaded connection between the seat and the body. The improved valve however gradually decreases in pressure along the chamber and up into the seat and disk cavity and because the improved design's seat is merged with the body, the high-pressure dissipation along the bottom of the seat contributes no internal passing issues.

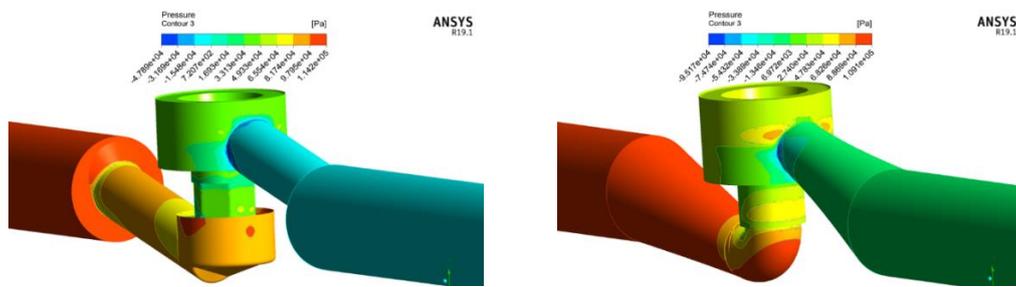


Figure 3. Inner wall pressure contour of original (left) and improved (right) design.

3.2. FEA results and discussion

3.2.1. Equivalent stress. In general, the overall stress contour shows that the original valve design has higher maximum equivalent (von-Mises) stress and more high stress spots compared to the improved design, especially around sharp edges at the bottom of the chamber and at the edge of the polygonal shaped orifice of the seat of the original design. However, the maximum stress found on the improved drawing comes from the contact seal between the seat and the disk, the main source of internal valve passing. A further look of the stresses around the contact seal between the two designs are presented in figure 4.

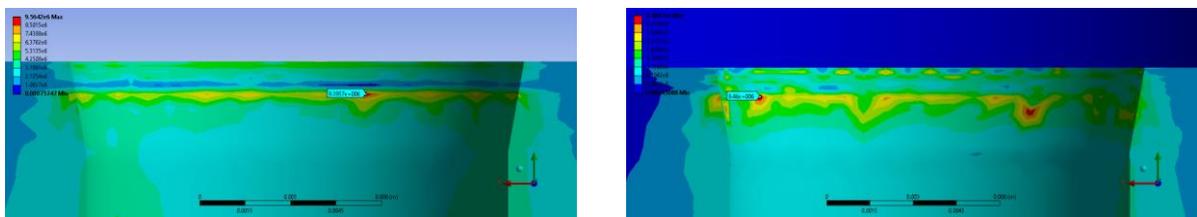


Figure 4. Contact seal stress contour of original (left) and improved (right) design.

The figure shows that the improved design has many high stress spots near the contact seal of the valve in fully closed position when compared to the original valve. The stress magnitude of these spots is also slightly higher than the original valve, with the highest being at 9.46×10^6 Pa. However, because the original design's seat is threaded and mounted to the valve body, stress concentrations also occur around the edges of the threaded connection between the two parts as shown in figure 5.



Figure 5. Equivalent stress contour at seat-body connection of original design.

The original design is more prone to valve passing through the seat-body connection and it is unlikely for it to happen for the improved design as there is no seat-body connection because the seat is merged and integral with the body of the valve.

3.2.2. Total deformation. Figure 6 shows the total deformation between the two designs. The original valve design has very high total deformation around the seat area of the valve compared to the improved design. Combined with the fact that the seat of the original design is connected to the valve body via thread connection, the possibility of the seat loosening and deforming from the valve body over time can be a huge factor leading to internal valve passing. The improved valve design however is less deformed at the seat and shows maximum total deformation at the bottom of the inlet port. In spite of that, because the inlet port modification complies with ASME B16.34 [2] valve standard in terms of wall thickness, the effect of the deformation leading to valve leakage is considered to be minimal.

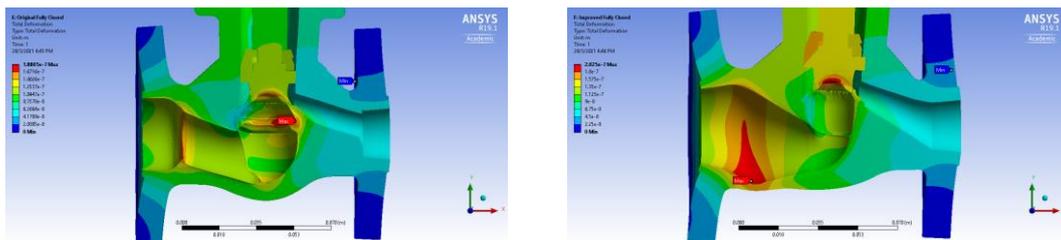


Figure 6. Total deformation of original (left) and improved (right) design.

4. Conclusion

The use of CFD proves that the improved globe valve design performs better and has less risk of valve leakage when compared to the original valve design. FEA simulation however gives mixed results. Unfortunately, equivalent stress analysis shows that the improved design is more prone to valve passing at the contact seal between the disk and the seat compared to the original design. The original design however has the disadvantage of stress contours around the edge connecting the seat to the valve body that can lead to passing from the threaded connection. In terms of total deformation, the original valve performs far worse than the improved valve at the seat area, increasing the risk of internal leakage.

References

- [1] Shashi Menon E. Chapter Twelve - Meters and Valves. In: Shashi Menon E, editor. Transmission Pipeline Calculations and Simulations Manual. Boston: Gulf Professional Publishing; 2015. p. 431-71.
- [2] Hassan MH. Enhancing Globe Valve Performance Through Modelling And Simulation Analysis: Universiti Teknologi PETRONAS; 2020.
- [3] ASME B16.34. The American Society of Mechanical Engineering; 2017.
- [4] API Standard 602 Steel Gate, Globe, and Check Valves for Sizes NPS 4 (DN 100) and Smaller for the Petroleum and Natural Gas Industries: American Petroleum Institute; 2009.
- [5] BS 5352. Specification for Steel wedge gate, globe and check valves 50 mm and smaller for the petroleum, petrochemical and allied industries: British Standards Institution; 1983.
- [6] Bland JA. Design Modification of Globe Valve Using CFD Analysis. University of Leeds. 2020.
- [7] Realizable k-epsilon Model: ANSYS; 2009 [Available from: <https://bit.ly/3gLUiiB>].
- [8] Institute AP. API RP 14E. 2019.