

Tomos Blizzard

1402861

BEng Mechanical and Manufacturing Engineering

Kelvin Lake

Structural & Fluid Analysis

Designing a Butterfly Valve: Structural Design

Contents

1.0 Introduction – Page 3
2.0 Finite Element Analysis – Page 3 - 5
3.0 Creating a Plate Model and Optimising – Page 6 - 10
4.0 Experiment – Page 11
5.0 Conclusion – Page 11
6.0 References – Page 12
7.0 List of Figures – Page 13

1.0 Introduction

This project will be concentrating on finite element analysis, known as FEA for short. FEA is a method used to optimise a component. Generally, a component which goes through FEA will be looking to reduce the weight of the component whilst maintaining a specific structural integrity. This project will specifically be looking at a butterfly valve plate. The objective of the project is to have a maximum deflection on the component of 0.1mm. Whilst minimising the deflection, the weight must be reduced as much as possible to reduce the cost to manufacture.

Firstly, the project will be looking at the history of FEA. Where it began and the possibilities that it could hold in the future. The valve plate in question will then be put through FEA analysis to gain an understanding of what FEA is capable of. The 3 steps which will be taken are the baseline carbon fibre reinforced plastic component. This component will then be optimised to achieve the 0.1mm deflection criteria, whilst keeping the weight down as much as possible. Finally, the discrete optimisation will be looked at to make the component more manufacturable when taking it from a computational method to the real world.

Lastly, a suitable method to test this butterfly valve will need to be determined. The correct machinery to test the product will be critical to confirm the computational results. The testing method must be capable of obtaining data from. This will be vital to gain a comparison of the FEA and practical results.

2.0 Finite Element Analysis (FEA)

FEA History

During this section of the brief, finite element analysis will be discussed. The optimisation techniques will be looked at and how they have developed over time. Also, how these methods may develop further in the future and what these possibilities will allow us to do. This will overall improve the design process.

Firstly, looking at the development of FEA from the beginning. FEA was first developed in 1950 in the aerospace industry by Boeing and Bell Aerospace in USA and Rolls Royce in the UK. It took 6 years before a paper was published which laid out the method of completing FEA (Tipton, 2017). This was written by M.J. Turner, R.W. Clough, H.C. Martin and L.J. Top. R.W. Clough was a professor at Berkeley University where the term 'finite elements' was first used. This led Berkeley University to be the centre of a larger proportion of FEA development. The first FEA programme was developed out of Berkeley University by E. Wilson in the 1960's. (Reddy, 1993)

The next step was made by NASA. They invested \$3,000,000 into making a more accomplished FEA program. This program is what we know it as today, which is NASTRAN. The \$3,000,000 which NASA invested in 1965 is the equivalent to \$30,000,000 in today's world. The program was freeware to begin with but over time it was released for a profit. The lead developer on this program was D. MacNeal (Brauer, 1993).

Current Method

When looking at finite element analysis, the biggest advantage is design optimisation. Using FEA allows you to gain a strong understanding of your product and also, the performance. FEA can remove bottlenecks, decrease material, prevent stress concentrations and help gain the desirable performance qualities. There are 6 steps which FEA takes. These are;

- Simplifying the model.
- Adding boundary conditions.
- Generating the mesh.
- Refining the mesh.
- Analysing results.
- Optimising the design.

Starting with simplifying the model. What is meant by simplifying the model is removing all parts of the product which are essential towards the analysis. This means that any components or structures which aren't essential should be removed prior to starting the analysis. (Costen, 2019)

Next, it's time to add boundary conditions to the essential products. This is where loads and constraints will be added. You can define a specific load which is directly placed on one face or multiple points. The constraints can include certain parts of the product being fixed or sliding a certain amount.

Once the boundaries are added, the mesh should be generated. Generally, the mesh which is automatically selected is suitable for the analysis, unless you desire a specific mesh type.

After the mesh is generated, you can refine it if the automated mesh isn't correct to your specific needs. The refining part allows for local mesh controls to optimise the mesh and create a better result. Tight angles and rounded areas which are areas where stress concentrations occur should have a smaller mesh defined compared to the rest of the product. This completes the optimisation of the mesh and after this has occurred you should run the model.

Next comes the analysis of the results. This will give you an idea of the areas of your product which are under a larger amount of stress than other parts. Firstly, setting the contour bar to have a maximum value which equals the materials yield stress or a particular stress value which you are looking to achieve. The simple colour system which is in place show the areas which are under a large amount of stress in red. Areas which are under large amounts of stress should be looked at to improve. For instance, taking away sharp corners as these areas cause stress concentrations. You can also look at the areas which have little to no stress. These areas may allow you to remove material if this is what's desired. The disadvantage of removing material is that this could negatively affect the overall performance of the product as FEA looks at the entire geometry. So, removing material in one area may cause another area to be put under more stress.

Finally, the last step is optimisation. Once the design has been optimised, this is the time where the product is meeting your entire criteria. There may have been a specific weight target which you were looking to achieve whilst maintaining a certain yield stress. The only consideration to take is the usability of a design after optimising. Sometimes the product can meet all requirements but may be too difficult to manufacture. It is important to have the balance between optimisation and manufacturability.

Future Possibilities

Currently, the biggest issue with FEA tools and programs is that the mesh process is an arrangement of linear elements. This makes complex designs and curves difficult to mesh so they must often be simplified. This is known as defeaturing which allows the mesh to be more accurate. A precise mesh is dependent on more elements, which in turn requires more computational time. Defeating is used to reduce the amount of computational time but this results in a less accurate result.

It is a fine balance between the number of elements included to get an accurate result but a large computational time and defeaturing a product, but getting an accurate result. The overall goal is that the designer knows which section of a product that can require less elements but also create a good mesh overall which reflects the results accurately.

Future developments will allow meshfree software which will require no time to be wasted with the above tasks. Having a meshfree program will allow the designer to have the most complex designs without the task of refining the mesh quality and defeaturing. Meshfree would allow a CAD model to be uploaded, set the boundary conditions required for the product and the computer would calculate the rest. This would save hours of pain staking trial and error with different mesh types and also save computational time. A meshfree program would allow the elements to not be shape-dependant. There are currently developments being made by (Roade, 2018) where the program uses structure orthogonal grids which are independent to the boundaries of the model. The program will divide the space occupied by the model. #

3.0 Creating a Plate Model & Optimising

During this part of the project, design studio was physically used to create a model. The first steps to take was calculating the amount of force the plate was under. The calculations for this are below. From the previous assignment we know that the fluid that we are dealing with the petrol which is held 5m above the valve in a tank. Petrol has a density of 800kg. Firstly, we calculate the mass. This is shown below.

$$\begin{aligned} \text{Mass} &= \text{Density} \times \text{Area} \\ \text{Mass} &= 800 \times 5 \\ \text{Mass} &= 4000\text{kg} \end{aligned}$$

After the mass has been calculated, it will allow the potential energy to be calculated.

$$\begin{aligned} PE &= mgh \\ PE &= 4000 \times 9.81 \times 5 \\ PE &= 196,200 \text{ J} \end{aligned}$$

Once the potential energy is calculated, we move on to the pressure.

$$\begin{aligned} P &= pgh \\ P &= 800 \times 9.81 \times 5 \\ P &= 39,240 \text{ N} \end{aligned}$$

After the pressure is calculated, we must work out the area of the plate we are looking to analysis in design studio.

$$\begin{aligned} \text{Area} &= \pi r^2 \\ \text{Area} &= \pi \times (97 \times 10^{-3})^2 \\ \text{Area} &= 0.0296\text{m}^2 \end{aligned}$$

Using the area, we are able to calculate to total force that the plate is coming under when the valve is fully closed.

$$\begin{aligned} P &= \frac{F}{A} \\ \therefore F &= P \times A \\ F &= 39240 \times 0.0296 \\ F &= 1161.5 \text{ N} \end{aligned}$$

Finally, using the total force, we are able to divide that by the total number of nodes on our plate which gives us the force per node.

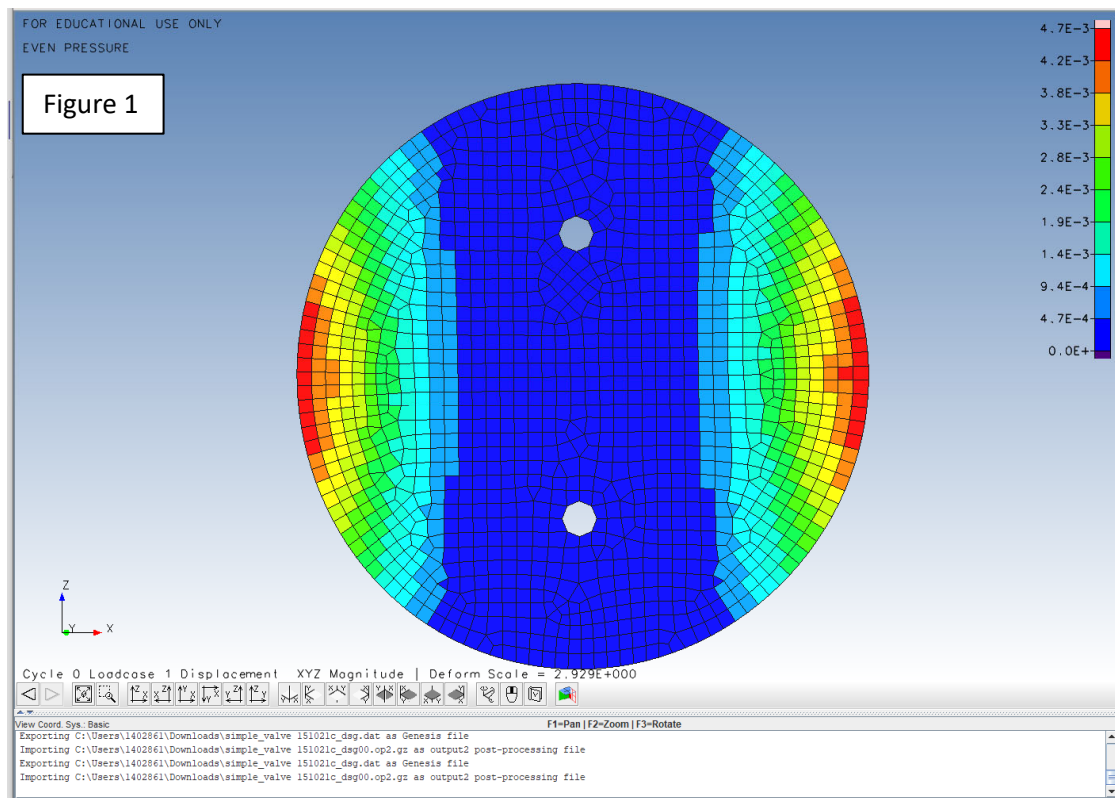
$$\begin{aligned} \text{Force per node} &= \frac{\text{Force}}{\text{No. of Nodes}} \\ \text{Force per node} &= \frac{1161.5}{1438} \\ \text{Force per node} &= 0.8077 \text{ N} \end{aligned}$$

To generate these calculations, we have had to make some assumptions. These assumptions are that the pin which the plate is attached to has no movement or flexibility. This allows us to control this

variability in the design studio software. The next assumption is that the nodes are evenly distributed throughout the plate. This allows us to assume that each node is put under the same amount of force. Using these assumptions, we are able to work through 4 key steps. The first is the steel baseline structure. This is our starting point. We then move onto the carbon fibre reinforced plastic baseline plate. Once we have this baseline, we must then optimise this part and finally, we must discreetly optimise the part.

Steel Baseline

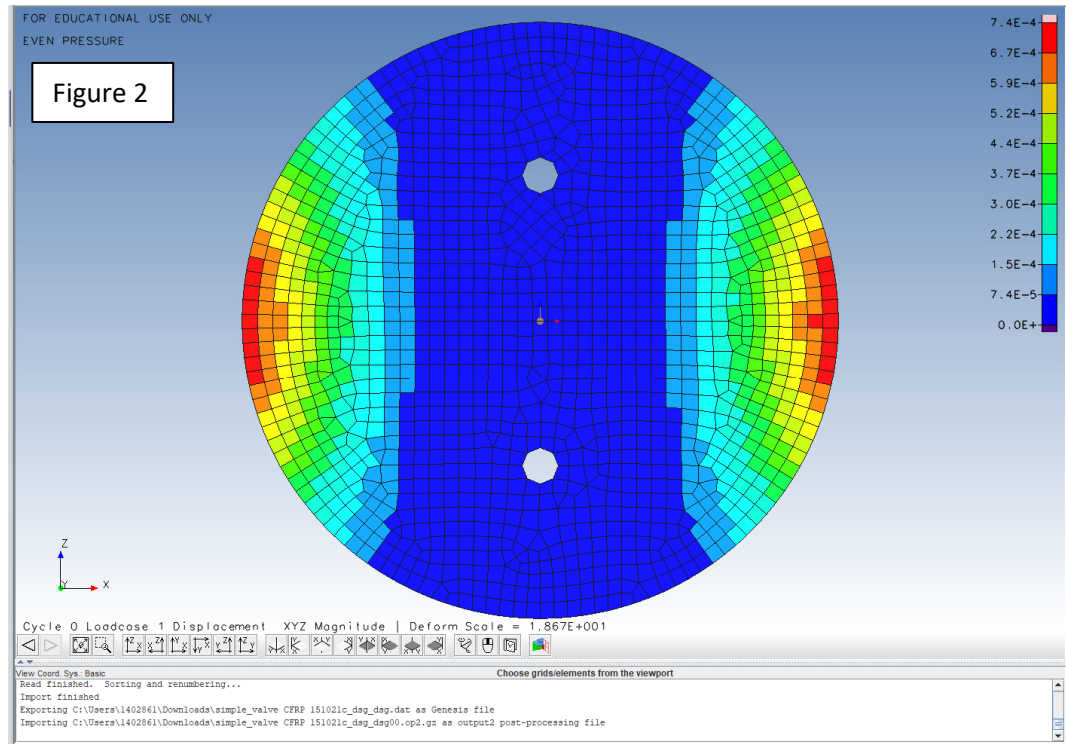
The steel baseline below has had the 0.8077N of force per node added to the static loads. This baseline is only used as reference to what we originally had. We will use this to show the difference after using the design studio tools.



The deflection seen on the steel baseline structure was 0.47mm. You can see from the above image, where the main stress points are. The aim of this practical is to achieve a deflection of less than 0.1mm.

Carbon Fibre Reinforced Plastic Baseline

The photo you can see below is the same as the steel structure with the same load applied. The difference in this is the material that is being used. This material is carbon fibre reinforced plastic. As you can see from the scale on the top right-hand corner, this material change, in exactly the same set up as the steel has caused the deflection to increase to 0.74mm. The weight of the baseline carbon fibre reinforced plastic component was 126g.

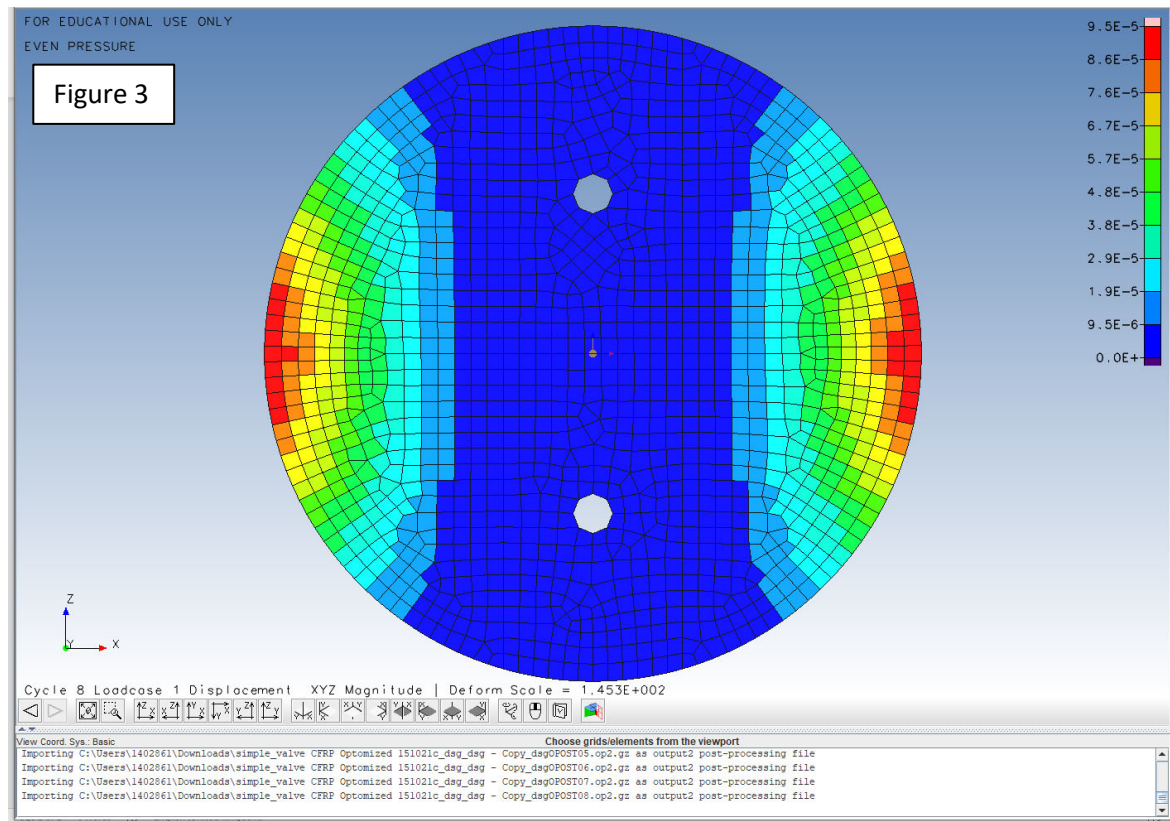


Carbon Fibre Reinforced Plastic Optimised

This stage of the program is critical. This optimisation stage is where we look to set our goals of the program. During this stage the maximum deflection of 0.1mm is input into the program. The number of layers is included in this stage too, along with the overall thickness. The overall thickness was set to 2.5mm, this spread across 5 layers allowed for 0.5mm for each layer. Furthermore, each layer allowed for 5 ply's set at 0.1mm in thickness. This gave a total number of 25 ply's. Each layer was different in one key way. The angle at which they were laid was different. The angles for each layer were;

- Layer 1 - 90°
- Layer 2 - 45°
- Layer 3 - 0°
- Layer 4 - -45°
- Layer 5 - -90°

These layers in different directions allows the material to be stronger without increasing the overall thickness of the component. After this program had been run the below results were achieved.



This image shows a maximum deflection of 0.095mm which achieves the target set at the beginning of the project. The main stress areas have remained the same throughout but using the design studio software, we have been able to gather a good result. The weight of the product after optimising has increased the weight from 126g to 245g. This is expected because we were looking to add strength which generally requires more material or size to be added. We have also increased the overall thickness of the plate to 4.8514mm. This thickness is the minimum amount required to have a maximum deflection of less than 0.1mm.

Layer 1	Layer 2	Layer 3	Layer 4	Layer 5
13 Ply	7 Ply	7 Ply	7 Ply	13 Ply
1.3675 mm	0.7091 mm	0.7053 mm	0.7086 mm	1.3609 mm

The table above shows the thickness of each layer. Layers 1 & 5 are almost double the thickness of the other 3 layers. This is down to layers 1 & 5 running 90° and -90°. These directional supports are critical to stop the outer edges of the plate deflecting. This is why these have increased in size. The central 3 layers which are running at 0°, 45° and -45° aren't required to be as strong because these layers will also support the top and bottom outer edges, as well as some support to the left and right outer edges. Due to the pin supporting the top and bottom edges, these layers aren't required to be as thick.

Optimisation is a critical part in the design studio process, but there is one flaw in this. The thickness of each layer isn't a multiple of 0.1mm thick. This causes manufacturability to become very difficult. The next step which is discreet optimisation takes the manufacturability of a product into consideration.

Carbon Fibre Reinforced Plastic Discreet Optimisation

This final stage of design studio is looking to make the product easily manufacturable. This work on the numbers which were generated in the optimisation stage and looks to make these into whole numbers. As you can see in the table above, with layers 0.1mm thick it is impossible to create a layer which is 1.3675mm. This is where the discreet optimisations come in. The discreet optimisation will look at the layers thickness and simply put, round them to nearest whole numbers which comply with the 0.1mm thick ply rule set. Below you will see how this effects the weight and ply thickness.

Layer 1	Layer 2	Layer 3	Layer 4	Layer 5
14 Ply	8 Ply	8 Ply	8 Ply	14 Ply
1.4 mm	0.8 mm	0.8 mm	0.8 mm	1.4 mm

This small change is so critical when going from a design side of a project and actually being able to manufacture something which has been designed. The weight has increase from 245g to 262g during this discreet optimisation stage. The overall thickness has increased to 5.2mm but this product is now fully optimised in meeting the less than 0.1mm maximum deflection and also being made from minimal material to reduce cost.

The final results are displayed in the table below to show the changes which occurred during the FEA. These results show how the part was optimised whilst achieving the required targets.

Run	Material	Thickness	Deflection	Mass	Load	Stiffness	Stiffness/Weight
CFRP Baseline	Composite	2.5mm	0.74mm	126g	1161.5N	1,570 N/mm	1270.2 N ² /mm
CFRP Optimised	Composite	4.8514mm	0.1mm	245g	1161.5N	11,615 N/mm	4832.6 N ² /mm
CFRP Discreet	Composite	5.2mm	0.1mm	262g	1161.5N	11,615 N/mm	4519.1 N ² /mm

The above results clearly show that the target of 0.1mm deflection has been achieved. This achievement has come at a 136g increase in weight and a 2.7mm thickness increase but this was to be expected with an original deflection value of 0.74mm. As you can also see, the stiffness to weigh rate has increased dramatically with the final figure being just below three times stronger.

4.0 Experiment

At this point in the project, an experiment will be discussed to show in a practical setting, what hydrostatic forces are seen and how the results correlate to the FEA analysis carried out. FEA is used prior to practical experiments because it helps predict the safety of the design. The problem with a butterfly valve is that it has a large number of non-linearities. This is the reason why a practical experiment is carried out prior to releasing a design. (X.G. Song, 2011) believes that the Kriging Model is the most effective to do this. This method allows a cheap meta-model to be created in substitute to an expensive simulation model. Before using the Kriging method, a model of a butterfly valve plate is created and FEA is used to optimise the component. This has already been completed in the previous section. The next step is conducting several experiments with a variety of different combinations. This is done by using orthogonal array. The last step is using the Kriging model. The Kriging model finds the optimum results using both practical results and computational results. Once the optimum result is found, this information will be fed back into the FEA simulation again to confirm the results. This method is very effective in maintaining the correct level of deflection whilst reducing the weight as much as possible (Jo, 2008).

If a practical experiment was to be conducted, the method that I would use would be a butterfly valve test machine. The particular one for this experiment would be a BV 80/500. The BV stands for butterfly valve, the 80 is 80 tonnes of clamping pressure which is capable. This clamping pressure holds the valve in place whilst the test is ongoing and the 500 is the maximum diameter in millimetres which can be tested on this machine. The smallest butterfly valve which can be tested on this machine is 50mm. This machine allows us to set the target pressure to 1,161.5N to replicate the results in the FEA program.

The butterfly valve is clamped between two plates to the correct pressure for the valve's diameter. The butterfly valve should be closed when completing this test. The correct pressure is selected and a valve is opened to allow the pressure to increase gradually until the desired test pressure is reached. This machine has an integrated bubble counter which identifies if there is a leak in the valve. If there are no bubbles being formed this confirms that the plate is sealing properly whilst under pressure. Once the test has been carried out, the pressure can be released (Metrus, 2019).

5.0 Conclusion

At the start of this project, I set out to try and create a valve plate which had a maximum deflection of 0.1mm whilst keeping the weight down to as little as possible. I believe this project has clearly shown that these targets have been achieved. The baseline model was far from achieving our original goal but through the use of FEA, this goal was accomplished. Taking the steps of optimising the valve plate to get the component to the necessary requirements was the first step. The next step was making the optimised part manufacturable. This was done but using discreet optimisation which made the ply's rounded numbers, making it easier to create pro-types and manufacturable parts.

Overall, I believe the FEA is a powerful tool which can help in not only reaching deflection challenges but also to reduce the cost of parts currently being used in manufacturing plants around the world. It would allow companies to possibly reduce the sizes of their components or look at a variety of materials which could possibly be stronger in the long term or obtain better properties for the purpose of the component. I feel that there are a large number of companies out there not using this technology to their advantage. This tool could be utilised more often than it currently is. How many companies out there are currently making a product which could be optimised using FEA? I feel its larger than we'd like to think.

6.0 References

- Belimo, 2014. *Avoid Common Errors When Installing Butterfly Valves*. [Online]
Available at: <https://blog.belimo.com/Blog/bid/74793/Avoid-Common-Errors-When-Installing-Butterfly-Valves>
[Accessed 5 July 2021].
- Brauer, J., 1993. *What Every Engineer Should Know About Finite Element Analysis*. 2nd ed. New York: Marcel Dekker.
- Costen, E., 2019. *Engineering Design and Analysis*. [Online]
Available at: <https://asrengineering.com/2019/05/15/design-optimization-with-fea/>
[Accessed 2 October 2021].
- Engineering360, 2020. *Butterfly Valve Information*. [Online]
Available at:
https://www.globalspec.com/learnmore/flow_control_flow_transfer/valves/butterfly_valves
[Accessed 6 July 2021].
- Hemphill, J. E., 1996. *Fisher Governor Co.*. [Online]
Available at: https://gaz.wiki/wiki/pl/Butterfly_valve
[Accessed 05 07 2021].
- Jeon, S. Y., 2010. Earth and Environmental Science. *Flow Characteristics and Performance Evaluation of Butterfly Valves Using Numerical Analysis*, 5 July, pp. 3-5.
- Jo, Y., 2008. Optimization of Valve Disk Using Orthogonal Array and Kriging Model. *AIP Conference Proceedings*, 1052(1), pp. 315-320.
- Metrus, 2019. *How To Test A Butterfly Valve*, Dusseldorf: Metrus.
- Reddy, J., 1993. *Introduction to the Finite Element Method*. 2nd ed. New York: McGraw Hill.
- Road, G., 2018. The Future of Finite Element Analysis. *Meshfree*, 8 July, pp. 4-9.
- Solken, W., 2008. *Explore The World of Piping*. [Online]
Available at: http://www.wermac.org/valves/valves_butterfly.html
[Accessed 5 July 2021].
- Song, X., 1996. Proceedings of the Institution of Mechanical Engineers. *Journal of Process Mechanical Engineering*, 223(2), pp. 81-89.
- Takeyoshi, K., 1995. Hydrodynamic characteristics of a butterfly valve — Prediction of pressure loss characteristics. *ISA Transactions*, 34(4), pp. 319-326.
- Tipton, R., 2017. *Northern Lights Solutions*. [Online]
Available at: <http://nlsde.com/2017/01/04/a-brief-history-of-finite-element-analysis/>
[Accessed 2 October 2021].
- X.G. Song, L. W. & Y. P., 2011. Analysis and Optimization of a Butterfly Valve Disc. *Proceedings of the Institution of Mechanical Engineers*, 223(8), pp. 81-89.

7.0 List of Figures

Figure 1 – Steel Baseline.

Figure 2 – Carbon Fibre Reinforced Plastic Baseline.

Figure 3 – Carbon Fibre Reinforced Plastic Optimised.