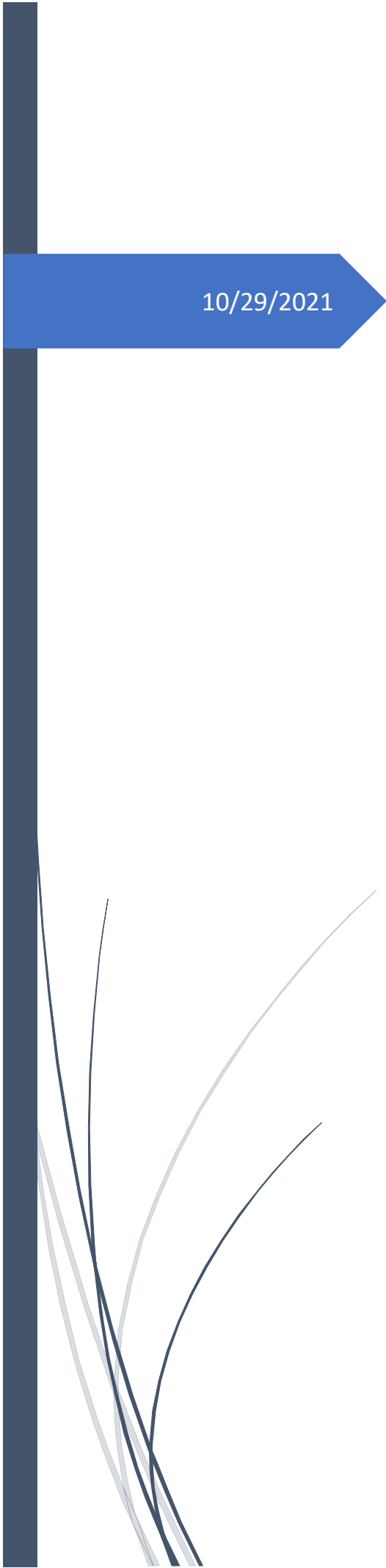




10/29/2021

Designing a Butterfly Valve: Structural Design

Structural & Fluid Analysis

Joseph John Garland 1707750
TATA STEEL

Contents

Summary 2

Introduction 3

Finite Element Analysis (FEA)..... 4

 History of FEA..... 5

 FEA analysis techniques 7

 Linear static analysis 7

 Non-linear static analysis 8

 Thermal engineering analysis 9

 Dynamic analysis..... 9

 Benefits of FEA 11

 Future development of FEA 12

Finite Element Analysis and Optimization of Valve Plate 14

Prototyping 16

Bibliography 18

List of Figures 20

..... 20

Summary

The following is a study into what finite element analysis is and its uses during the design phase of a structure or product.

Firstly, the report takes a brief look into what finite element analysis is and its uses within the engineering and design community. Research into the history of FEA analysis was then carried out. An in-depth discussion into some of the earliest forms of FEA and how it has been developed over the years was then carried out.

Following this the report looks at the numerous different types of FEA analysis available to engineers such as linear and non-linear analysis techniques and how they benefit the design process.

The report also looks at the future development of FEA analysis, its proposed use and how it aims to further improve the design process.

FEA software design studio was then used in-order to carry out optimization of a butterfly valve plate. The plate was allowed a maximum of 0.1mm deflection when subjected to an applied load of 1161.6 newtons and was optimised for mass.

Finally the report discusses how prototype testing of the butterfly valve should be carried out, specifically looking at the influence of operation on the valve plate and any instrumentation that could be implemented to aid the data gathering.

Introduction

As part of this assignment a detailed study into different methods of FEA optimisation will be carried out. The study will aim to look at past, current and potential future methods of carrying out FEA analysis and how they can be implemented during the design process and manufacturing of items/products.

Following the creation of a butterfly valve using solid works CAD software the plate component of the valve must be exported as a file into the FEA optimisation software, design studio. With the use of design studio, the desired manufacturing specifications and component conditions i.e. material, layers of ply and angle, will then be applied to the butterfly valve plate before carrying out an FEA of the component. Once this is done design studio will then be used to optimise the structural make up of the component to minimise the mass of the valve plate.

As a result of the initial optimisation of the valve plate the desired material properties may not have been met, therefore further optimisation is then carried out to ensure the manufacturing requirements are met and the component is fit for purpose i.e. within tolerance.

Following the completion of the FEA and optimisation of the valve plate using design studio, a discussion into how to appropriately carry out experimental FEA analysis of a prototype of the component is then carried out. The aim of this experiment would be to gather a better understanding of the deformation experienced by the valve plate when subjected to its proposed hydrostatic pressure. How data from the experiment is obtained and the comparison of data with that gathered from the simulation will also be discussed.

Finite Element Analysis (FEA)

Finite element analysis is a method used in engineering to simulate the behavioural characteristics of a component of assembly under any specified conditions. The data gathered from finite element analysis is then assessed using FEM, finite element method. FEA is able to simulate physical phenomena experienced by engineering components during use or prior to being used during the design phase of a part or project.

Finite element analysis implements the use of mathematical models to process and translate the effects of external conditions a part is exposed to in the real world i.e. loads, stress, strain etc. The simulations carried out for finite element analysis are performed using specialised software which allows for engineers to pin point potential design flaws or areas of weakness withing the item in question prior to putting it into production. This allows engineers to reduce the number of physical prototypes needed before giving the green light for the part to be produced on a large scale or put into use in high-cost projects.

Finite element analysis not only identified potential causes for concern within the design of an item but allows engineers to optimise the components so that they are as efficient as possible when in use. The use of FEA software for the optimisation of the design of a component allows the developer to save money in production of prototypes or re-designing.

Finite element analysis software implements the use of partial differential equations (PDEs) in order to solve complicated mathematic equations and generate relevant data for structural behaviour characteristics. Structural behaviour such as stresses and strains created by the given load the item is subjected to are then displayed for the engineers to see on the FEA software (What is Finite Element Analysis (FEA), 2021).

History of FEA

The earliest form of FEA analysis was developed during the 1800 by John William Strutt Rayleigh more commonly known as Lord Rayleigh. Rayleigh was able to develop a method for estimating a natural frequency for simple structures. By estimating the deformation that occurred within natural structures Rayleigh was able to quantify the shape using minimization of the energy distributed over the structure.

Walter Ritz then further developed Rayleigh's method for predicting the stress and displacement within natural structures which later became known as the Rayleigh-Ritz method. Due to the assumption of the size of the shapes using this method it made the accuracy of the results gathered very poor. This was because as the shapes in question became more complex the number of different possible deformities or displacement the structure experienced increased exponentially. However, the predictive model of the Rayleigh-Ritz method later became critical to the development of algorithms within FEA analysis.

During the 1940's, Alexander Hrennikoff established a numerical method for predictable behavioural characteristics of more general structures when they were subjected to varying types of loads. The method developed by Hrennikoff was based off of typical frame and truss structures. Hrennikoff later made use of energy theories established by Italian mathematician Alberto Castigliano and Irish mathematician William Rowan Hamilton to further develop and improve the method he had established.

Following the development of Hrennikoff's method mathematician Richard Courant proposed that the continuous system for analysis should be broken into triangular subsections. Whereas, the continuous system developed by Hrennikoff utilized the used of

lattice analogy of structures. Courant's approach divided a given structure into finite triangular subsections in order to solve second order elliptic partial differential equations.

As a result of technological developments analogue computers became far more capable and were now able to process far more complex structural analysis. Promise of further development of more powerful computers analytical methods also progressed to include matrix based analysis of frame and truss structures.

Following the use of FEA analysis by Boeing in the 1950's the benefits of carrying out in-house FEA became apparent to a number of large scale, highly profitable industries, particularly in the manufacturing world.

The 1960's signalled the true beginning FEA analysis with the replacement of analogue computers with digital computers capable of carrying out thousands of operations in a second. The 1960's saw US space agency NASA decide to award small analogue computer manufacturer, MacNeal-Schwendler Corporation a contract to specifically develop and create general purpose FEA code. Following this the commercial use of FEA software became widely adopted on a large scale throughout the manufacturing industry globally. MacNeal-Schwendler Corporation labelled the code they had created as NASTRAN.

Originally the NASTRAN code was created with a limit of 68,000 degrees of freedom, which at the time was believed to be more than what would ever be needed for FEA analysis.

With the development of computers during the 1970's FEA analysis became more and more frequently used by analysts and mathematicians. In 1972, the first mathematical proof of finite element methods was published by Gilbert Strang. This meant FEA data was finally able to be proven with the use of manual mathematical equation.

The 1980's saw graphical development within computers allow analysts to see a visual representation of which areas of a structure came under stress when under load with the use of colours. The graphical developments in FEA analysis were better known as "pre" and "post" processor. The development in technology allowed for designers to see contours created by the stress's static and dynamic loads put on structures. It was only after this development that the incorporation of FEA analysis was seriously considered as an integral part of the product design process.

The PC platform of computing became extremely prevalent in the 1990's and was used broadly in FEA analysis. The average computer was now capable of processing models with up to one million degrees of freedom with the use of software such as "StressCheck".

The continuous development and updates, for software such as "StressCheck and Design Studio" have allowed enabling more and more complex FEA analysis and optimization to be carried out on structure or product during both design and improvement phases.

FEA analysis techniques

Finite element analysis is used broadly in all forms of engineering for many different purposes. Each form of FEA analysis utilizes optimization software and continuous research to aid engineers with improving/solving problems in their given professional field. Different type of FEA include the following:

Linear static analysis

Linear static analysis is a form of FEA where linear relation is held in by applied forces and displacements. Linear static analysis is best used for structures that are subjected to linear

stresses and have material that experience linear elastic deformation. Materials used in linear static analysis have a constant stiffness matrix and the process of solving is comparatively short compared to non-linear static analysis of a similar structure. Static analysis aims to calculate the effects constant loads have on mechanical structures irrespective of effects phenomena such as inertia and shock may have on structures that experience varying applied loads. However, static analysis does still take into consideration constant inertia loads such as gravity or varying loads that can be estimated as a static equivalent. Static linear analysis is often done first before carrying out full non-linear finite element analysis.

Non-linear static analysis

Unlike linear static analysis, Non-linear analysis is carried out when non-linear relation occurs between applied forces and displacement. Abnormalities or irregularities like geometrical non-linearity and non-linear material (elasto-plastic) in a structure can result in non-linear behavioural characteristics when carrying out FEA analysis. The stiffness matrix of non-linear structures is not constant during loading as a result of the non-linear geometric and material characteristics. As opposed to the stiffness matrix exhibited in linear analysis where it is constant. Non-linear analysis requires a different solving method to that of linear analysis.

Modern FEA analysis software such as design studio are capable of processing and solving most non-linear problems encountered by structures under load. The software is also able to recommend amendments that could be made to the structures design to improve the item in areas of concern and better the functionality of the product.

Although useful FEA analysis is often difficult to use and requires past experience of using similar or identical software to fully benefit from its use. This said, when using Software such as Design Studio to carry out FEA analysis the specific parameters for the solution must be specified prior to solving in-order to get an accurate and logical solution.

Thermal engineering analysis

Thermal analysis is a form of static analysis The use of FEA for thermal analysis allows engineering to quantify to heat transfer that occurs in or between and object (Martin,2021). Heat transfer phenomena such as conduction, convection and thermal radiation all able to be calculated by the analysts. However, in-order to effectively use FEA software for thermal analysis convection and thermal radiation must be defined by the analyst and are unable to be calculated using the FEA software (Martin, 2021).

The most beneficial use for FEA analysis within heat transfer problems is in cases where the convection is able to be assumed as a constant throughout the object or when the coefficient of heat is known to the analyst. Other quantities such as thermal expansion and structural stresses caused by thermal gradients within the part.

Dynamic analysis

Dynamic analysis is then study of the dynamic properties a mechanical object has when subjected vibration. Dynamic FEA analysis utilises the vibrational simulation to determine characteristics like natural frequencies, noise and vibration problems within the design and the effect of transient loading of the part.

Dynamic FEA analysis is most beneficial when carried out during the design phase of a physical structure as it can significantly reduce the costly and need for rigorous prototype testing. Failures associated with dynamic load have a tendency to be extremely costly and highly detrimental to the structure of an item. This therefore makes thorough dynamic FEA analysis critical during the design phase of mechanical parts, in-order to avoid the potentially catastrophic failure that could prove dangerous, costly and damaging to the developer's reputation.

There are several different types of dynamic analysis used within engineering and design, these include:

- Modal analysis. This type of dynamic analysis is used by engineers to determine the natural frequencies within an item. Modal analysis allows products or structures to be designed in such a way that the vibration it is exposed to never matches the item's natural frequency. This is done in-order to eliminate or significantly reduce the chances of critical failure due to dynamic loads i.e excessive vibration.
- Harmonic response analysis. This is used to evaluate the effects of dynamic loading on a structure or part and allows engineers to predict the stress, deflection and fatigue life of a part. Based on the types of load the part/structure is subjected to allows engineers to predict how it will behave.
- Transient dynamic analysis allows the designer/engineer to determine how a part will behave when exposed to loads that are a function of time. Transient dynamic analysis is similar to harmonic response analysis however the loading of the part can be carried out non-periodically and is usually used under seismic or shock loading conditions.

Benefits of FEA

Finite element analysis is carried out within engineering and design to help improve and predict structural behaviour of parts when subjected to loads prior to being produced. The benefits of carrying out FEA on a design or structure are endless however, the following are some of the main benefits associated with the use of FEA:

- Optimized Design. FEA allows engineers or designers to see potential flaws in their design that are created when under load and allows them to amend the design for optimum effectiveness.
- Increased accuracy. During early stages of manufacturing designs were sketched and prototypes were made and tested as a form of FEA. This process allowed for design restriction to be overlooked or missed. However, FEA software now allows the designer or engineer to model the physical stresses and strain a part is subjected to with the use of inputting the material parameters. Carrying out FEA allows for an instant increase in accuracy of material behavioural characteristics, which helps with the design of the item.
- Increased knowledge of critical design parameters. Being able to determine what critical factors may affect the design of an item the designer is able to avoid designing/manufacturing a structure that may fail to be able to carry the jobs its intended for. This is seen as one of the most beneficial factors of FEA.
- Virtual prototyping. The simulation of use of a part carried out by FEA software allows designers and engineers to rectify design flaws in a much period of time in comparison to the time that would be needed to rectify design flaws in hard prototyping models.

- More efficient and cost-effective design cycle. Simulations within FEA software like design studio do not require repetition of cycles which can prove to be costly if carrying out hard prototyping.
- Improved Productivity and revenue. The use of FEA means the design process of an efficient product or structure is much shorter. It also creates less material waste and saves time.

Future development of FEA

The process of making mesh when using FEA software is essential to the accuracy of the results gather in simulation. The process of meshing feature can sometime prove to be difficult when trying to mesh shapes the software is not familiar with.

The meshing process in current software focuses on the detailed arrangement of linear elements. This mean curve or complex designs often have to be re-designed or simplified. This process is called defeaturing and is done in-order to achieve an accurate and effective mesh. The more detailed the mesh the greater the number of elements needed in the design which requires either further defeaturing or longer computational time. Defeating a design often results in an output of less accurate data during simulation.

The amount of element and amount of defeaturing done by an engineer/designer is critical to the how an object is designed. Ensuring the design has a mesh of good quality is an integral task carried out during FEA analysis.

Future development in FEA software aim to rid engineers and designers of time-consuming meshing process that limits their designs by creating mesh-free FEA software.

The software uses mathematical algorithms that allow FEA to be carried out without the need of meshing, unlike regular FEA software. The algorithm is a combination of traditional FEA techniques and an iteration of the Dirichlet function.

When reaching a boundary, the Dirichlet function has a value of zero when boundaries, this enables the shape to be mapped using a mathematic algorithm if it is integrating for the area volume.

As the boundaries are not met the grid block entirely within the shape and will be analysed with the use of general FEA formulae. However, the grid blocks that are contained by boundaries will be analysed proportionally to the Dirichlet function as it describes the limits within the boundaries.

Where traditional FEA software relies on defeaturing to create mesh of nodes to the area of a shape in-order to allow the boundary conditions of the nodes to be applied. Meshfree, is capable of creating boundaries on both nodes of the shape and those found using the Dirichlet function.

Civil and mechanical engineering software company Midas IT and Samsung worked side by side in the development of “meshfree” software and aim for it to be broadly used throughout engineering and design processes worldwide.

Finite Element Analysis and Optimization of Valve Plate

Prior to completing the FEA butterfly valve the following equation had to be completed in-order to gather relevant data for the FEA.

Potential energy:

$$PE = Mgh$$

$$PE = 4000 \times 9.81 \times 5$$

$$PE = 196,200 \text{ Joules}$$

Pressure:

$$P = \rho gh$$

$$P = 800 \times 9.81 \times 5$$

$$P = 39,240 \text{ N/m}^2$$

Area:

$$A = \pi \cdot r^2$$

$$A = \pi(97 \times 10^{-3})^2$$

$$A = 0.0296 \text{ m}^2$$

Therefore:

$$P = \frac{F}{A}$$

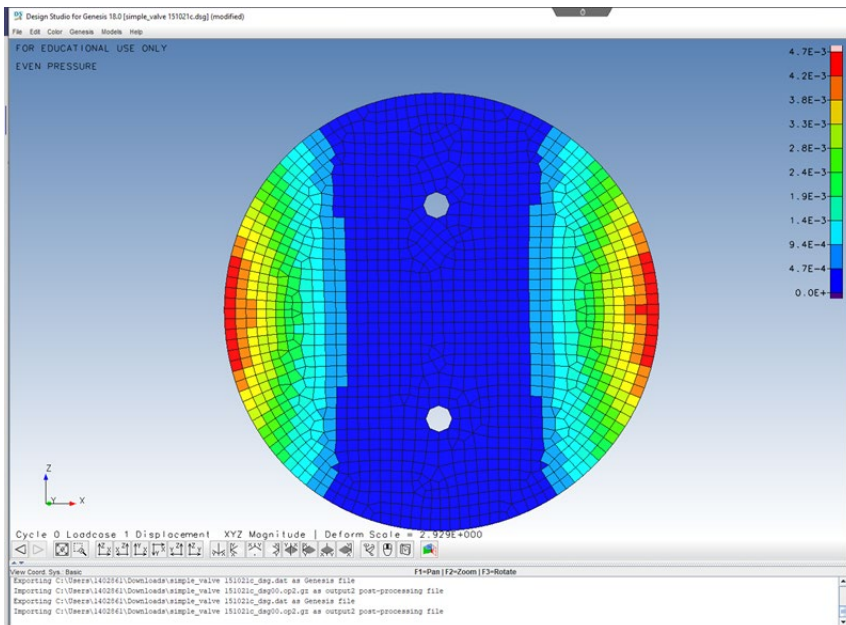
Hence:

$$F = P \times A$$

$$F = 39,240 \times 0.0296$$

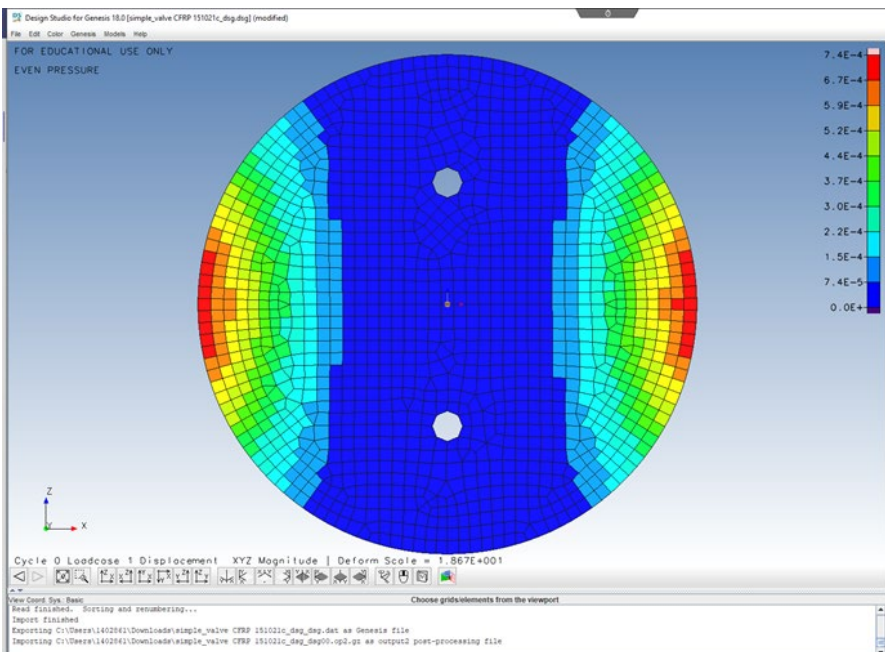
$$F = 1161.5 \text{ Newtons}$$

Figure 1 - FEA of steel valve plate



Above is the deformation that occurred in the steel baseline of the valve plate when subjected to the determined loads. A deformation 0.47mm occurred during loading of the valve plate.

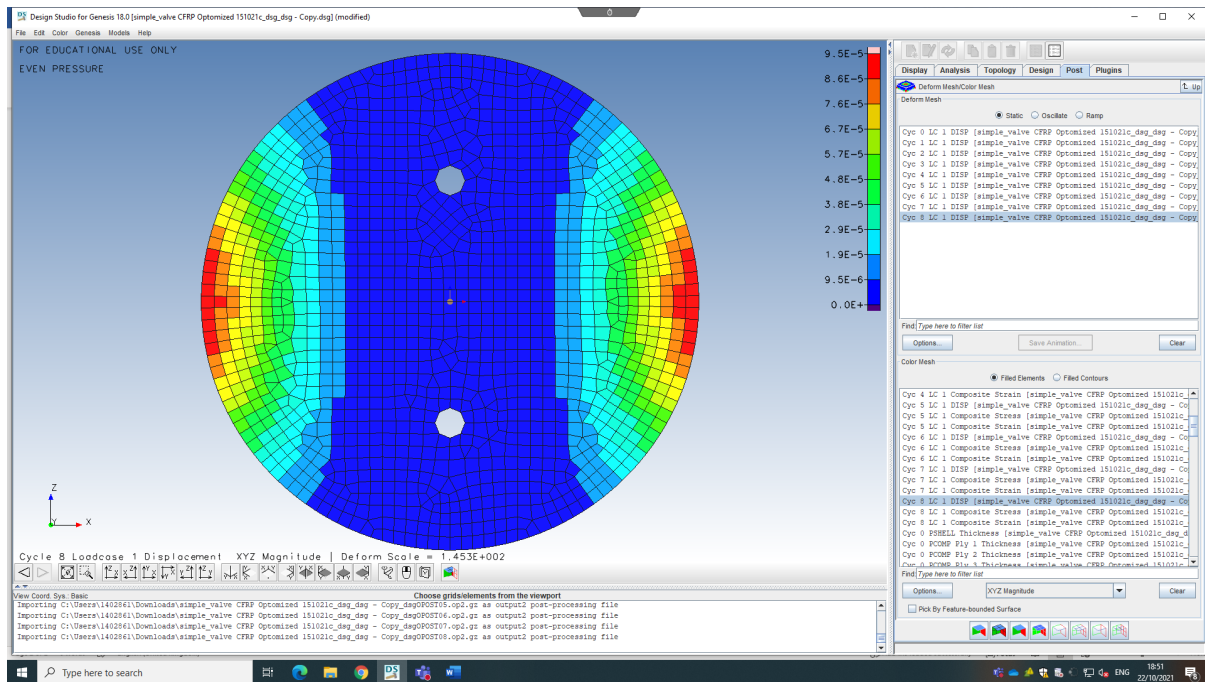
Figure 2 – FEA of valve CFRP valve plate



CFRP was then applied to the valve plate and analysis of deformation was then carried out again at the same layer thickness of 0.5mm. Following applying the desired number of layers and ply's to the valve

plate the deformation that occurred was 0.74mm.

Figure 3 – FEA of CFRP of optimised valve plate



The CFRP valve plate was then optimised for mass with a maximum allowable deflection of 0.1mm. once optimised the deflection experienced was 0.095mm.

Prototyping

It is proposed that testing of a prototype design is first carried out in a laboratory. The aim of the testing is to better understand the that deformation that occurs in the valve plate when subjected to a given load. First, it would be best to carrying out FEA simulation using FEA software such as design studio in order to gather a set of comparative data (Roade, 2021).

Assume no cost limitations, the prototype should be produced true to design specification and conditions of use should be replicated exactly i.e use of petroleum and height of tank.

As a result, conditions during the test will directly imitates conditions in its intended

environment. By doing this the testing team should expect there be no anomalies in data caused by scaling down the valve.

The valve should then be operated through all potential positions i.e fully open, fully closed and when in transition (Team, 2021), this should be done to ensure the valve is fully operational. Doing this will allow for the plate to be subjected to all possible types of loading it is intended for therefore resulting in more accurate results (Roade, 2021).

In order to gather data to better understand deformation in the plate, the plate should be fitted with strain gauges. The strain gauges will provide data of exactly how much load the plate is actually subjected to during use and can be compared to data gathered from the simulation. The strain gauges will let the engineers know if the load exceeds what is expected and whether to expect greater deformation than shown in the simulation. The valve should also be fitted with flow meter on either side of the plate. The flow meters will allow the engineers and designers know whether the deformation exceeds what was established in the FEA simulation (0.1mm), as when fully close zero flow should pass through the valve. Flow through the valve when fully closed would suggest excessive deformation has occurred in the valve plate causing the seal to become ineffective (Roade, 2021).

Analysis of the results gather will allow the engineers to deem whether the valve is then fit for purpose or requires design amendments i.e optimization.

Bibliography

- Jacob Fish and Ted Belytschko, "A First Course in Finite Elements by Jacob Fish and Ted Belytschko", Wiley, 2007
- SimScale. 2021. *What Is FEA | Finite Element Analysis? Documentation | SimScale*. [online] Available at: <<https://www.simscale.com/docs/simwiki/fea-finite-element-analysis/what-is-fea-finite-element-analysis/>> [Accessed 27 October 2021].
- Twi-global.com. 2021. What is Finite Element Analysis (FEA)?. [online] Available at: <<https://www.twi-global.com/technical-knowledge/faqs/finite-element-analysis>> [Accessed 26 October 2021].
- Martin, F., 2021. Thermal Analysis with FEA and CFD | Quadco Engineering. [online] Quadco.engineering. Available at: <<https://www.quadco.engineering/en/services/thermal-analysis.htm>> [Accessed 29 October 2021].
- Goudsmit.co.uk. 2021. *Magnetics Archives - Page 2 of 7* -. [online] Available at: <<https://www.goudsmit.co.uk/category/magnetics/page/2/>> [Accessed 29 October 2021].
- Meadows Analysis. 2021. Understanding FEA: These Are the Different Types of Finite Element Analysis. [online] Available at: <<https://meadowsanalysis.com/understanding-fea-these-are-the-different-types-of-finite-element-analysis/>> [Accessed 29 October 2021].
- ESRD. 2021. *Brief History of FEA | ESRD | Engineering Software Research and Development, Inc.*. [online] Available at: <<https://www.esrd.com/simulation-technology/brief-history-of-fea/>> [Accessed 29 October 2021].

- Femto Engineering. 2021. *In short explained: Linear and nonlinear structural analysis*.
[online] Available at: <<https://www.femto.nl/stories/linear-non-linear-analysis-explained/>> [Accessed 29 October 2021].
- Roade, G., 2021. *The Future of Finite Element Analysis: MeshFree*. [online]
Midasoft.com. Available at: <<https://www.midasoft.com/mechanical/blog/the-future-of-finite-element-analysis-meshfree>> [Accessed 29 October 2021].

List of Figures

Below is a list of screenshots supporting the finite element analysis carried out of the butterfly valve plate.

Figure 4

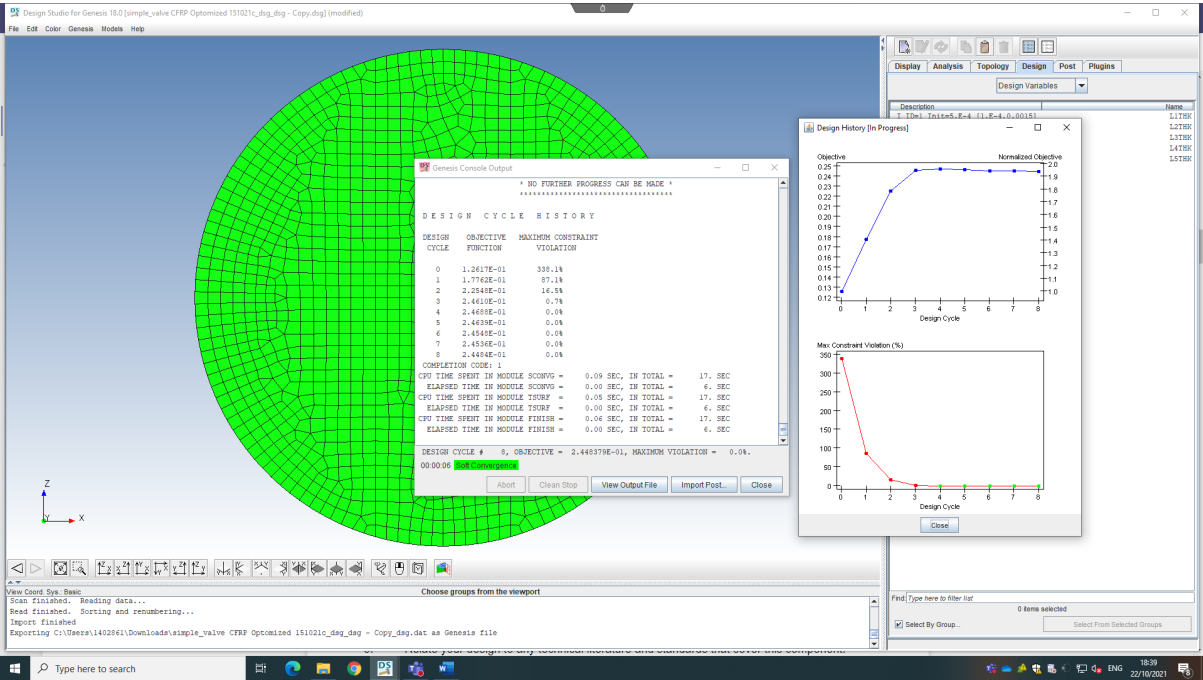


Figure 5

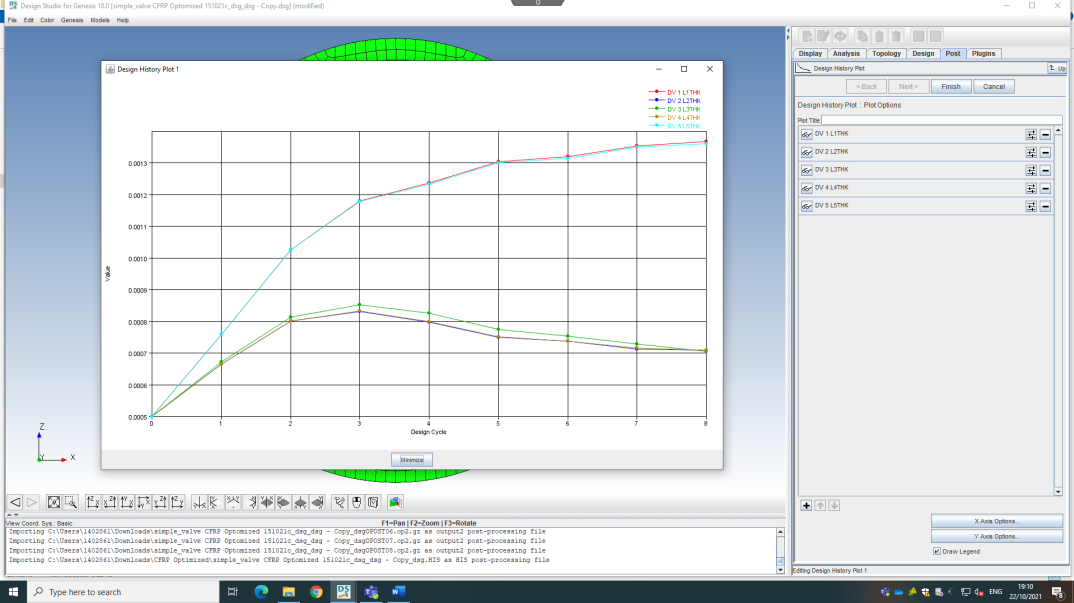


Figure 6

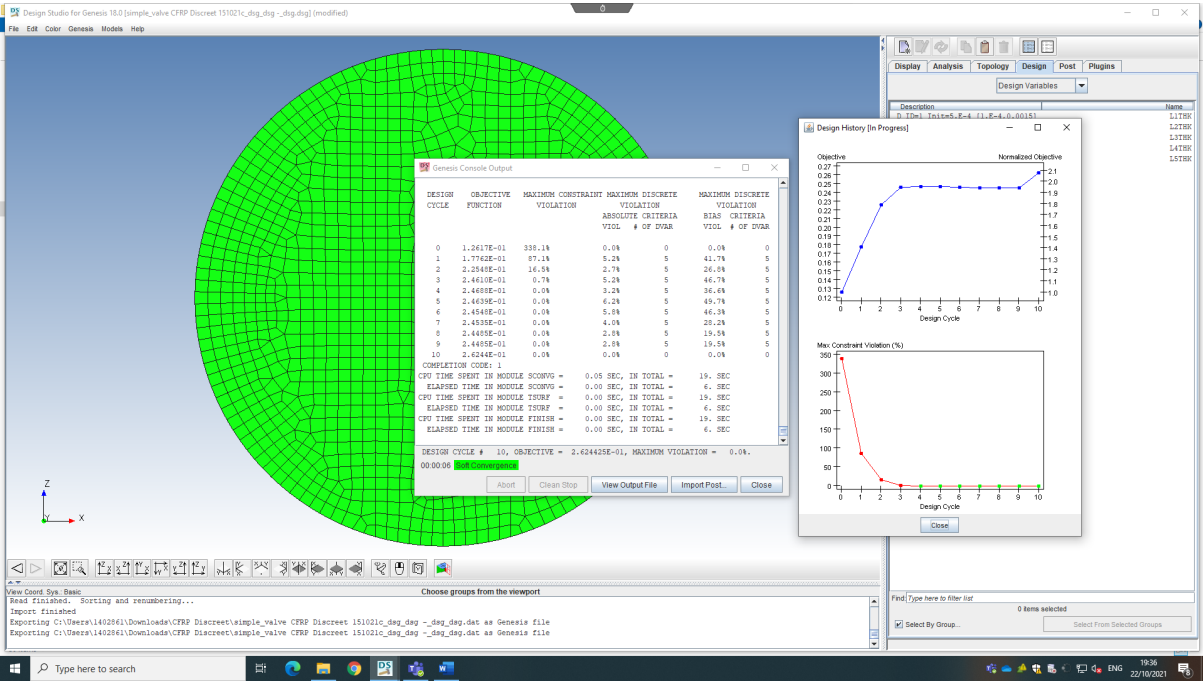


Figure 7

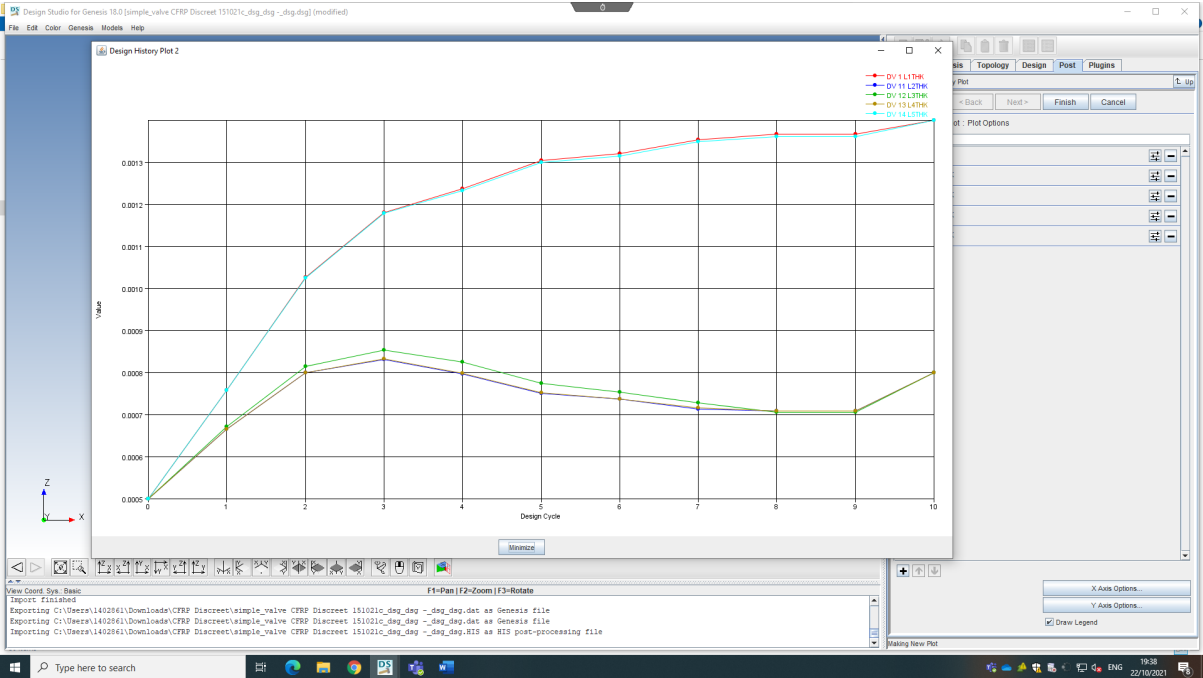
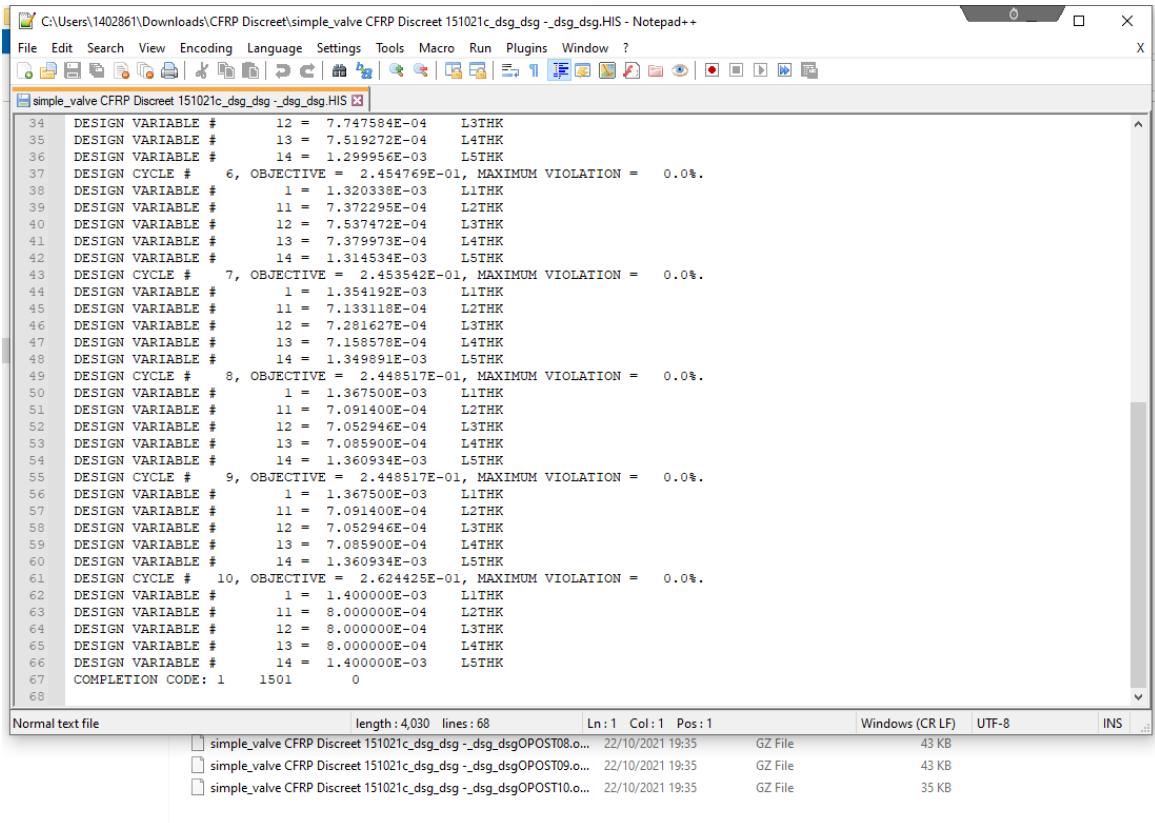


Figure 8



The screenshot shows a Notepad++ window with the title bar "C:\Users\1402861\Downloads\CFRP Discreet\simple_valve CFRP Discreet 151021c_dsg_dsg -_dsg_dsg.HIS - Notepad++". The menu bar includes File, Edit, Search, View, Encoding, Language, Settings, Tools, Macro, Run, Plugins, Window, and ?. The toolbar contains various icons for file operations and editing. The text area displays the following content:

```
34 DESIGN VARIABLE #      12 = 7.747584E-04  L3THK
35 DESIGN VARIABLE #      13 = 7.519272E-04  L4THK
36 DESIGN VARIABLE #      14 = 1.299956E-03  L5THK
37 DESIGN CYCLE #        6, OBJECTIVE = 2.454769E-01, MAXIMUM VIOLATION = 0.0%.
38 DESIGN VARIABLE #      1 = 1.320338E-03  L1THK
39 DESIGN VARIABLE #     11 = 7.372295E-04  L2THK
40 DESIGN VARIABLE #     12 = 7.537472E-04  L3THK
41 DESIGN VARIABLE #     13 = 7.379973E-04  L4THK
42 DESIGN VARIABLE #     14 = 1.314534E-03  L5THK
43 DESIGN CYCLE #        7, OBJECTIVE = 2.453542E-01, MAXIMUM VIOLATION = 0.0%.
44 DESIGN VARIABLE #      1 = 1.354192E-03  L1THK
45 DESIGN VARIABLE #     11 = 7.133118E-04  L2THK
46 DESIGN VARIABLE #     12 = 7.281627E-04  L3THK
47 DESIGN VARIABLE #     13 = 7.158578E-04  L4THK
48 DESIGN VARIABLE #     14 = 1.349891E-03  L5THK
49 DESIGN CYCLE #        8, OBJECTIVE = 2.448517E-01, MAXIMUM VIOLATION = 0.0%.
50 DESIGN VARIABLE #      1 = 1.367500E-03  L1THK
51 DESIGN VARIABLE #     11 = 7.091400E-04  L2THK
52 DESIGN VARIABLE #     12 = 7.052946E-04  L3THK
53 DESIGN VARIABLE #     13 = 7.085900E-04  L4THK
54 DESIGN VARIABLE #     14 = 1.360934E-03  L5THK
55 DESIGN CYCLE #        9, OBJECTIVE = 2.448517E-01, MAXIMUM VIOLATION = 0.0%.
56 DESIGN VARIABLE #      1 = 1.367500E-03  L1THK
57 DESIGN VARIABLE #     11 = 7.091400E-04  L2THK
58 DESIGN VARIABLE #     12 = 7.052946E-04  L3THK
59 DESIGN VARIABLE #     13 = 7.085900E-04  L4THK
60 DESIGN VARIABLE #     14 = 1.360934E-03  L5THK
61 DESIGN CYCLE #       10, OBJECTIVE = 2.624425E-01, MAXIMUM VIOLATION = 0.0%.
62 DESIGN VARIABLE #      1 = 1.400000E-03  L1THK
63 DESIGN VARIABLE #     11 = 8.000000E-04  L2THK
64 DESIGN VARIABLE #     12 = 8.000000E-04  L3THK
65 DESIGN VARIABLE #     13 = 8.000000E-04  L4THK
66 DESIGN VARIABLE #     14 = 1.400000E-03  L5THK
67 COMPLETION CODE: 1    1501    0
68
```

The status bar at the bottom indicates "Normal text file", "length: 4,030", "lines: 68", "Ln: 1", "Col: 1", "Pos: 1", "Windows (CR LF)", "UTF-8", and "INS". Below the status bar, a file list shows three files:

File Name	Date/Time	Type	Size
simple_valve CFRP Discreet 151021c_dsg_dsg -_dsg_dsgOPOST08.o...	22/10/2021 19:35	GZ File	43 KB
simple_valve CFRP Discreet 151021c_dsg_dsg -_dsg_dsgOPOST09.o...	22/10/2021 19:35	GZ File	43 KB
simple_valve CFRP Discreet 151021c_dsg_dsg -_dsg_dsgOPOST10.o...	22/10/2021 19:35	GZ File	35 KB

