

Structural & Fluid Analysis Assignment Two AMCA6010

Steffan Rhys Tubb (1910092)

Contents

Introduction	2
History	2
Optimisation techniques.....	3
Baseline structure	4
Optimised structure	6
Prototype and testing	8
Conclusion	10
Bibliography	11

Introduction

I was tasked to design the lay-up of the composite material so that when the hydrostatic forces seen from a previous assignment were applied, a maximum deformation of 0.1mm would occur. This was to ensure a good seal between the plate and the outer housing of the valve. The plate needed to use as little material as possible to reduce the cost of manufacture of the part.

To get an understanding of how to do this I wanted to research different Finite Element Analysis (FEA) techniques and learn a bit more about the optimization of them. This would help me understand how to implement them when it came to using software to design the material layup of the plate.

History

Finite element analysis (FEA) is the study and analysis of a phenomenon using the Finite element method (FEM). The first paper on FEM was published in 1956 by M.J.Turner, R.W.Clough, H.C.Martin and L.J.Topp. The paper stated that by breaking a large, complicated structure down into smaller structures and then re analysing those structures would simplify the problem without losing sufficient accuracy. As long as basic conditions of continuity and equilibrium are established at selected points (nodes) throughout the structure. Also, by increasing the number of nodes the accuracy of the results could be improved greatly (TURNER, CLOUGH, MARTIN and TOPP, 1956).

In 1965 NASA submitted a request for a proposal for the development of a structural analysis software tool so that results could be generated through simulation and not complex and laborious hand calculations. The resulting software was called NASTRAN (NASA Structural Analysis) and was used to solve structural problems using the available FEM techniques at the time. NASTRAN was further developed in 1968 by the computer sciences corporation. It wasn't until 1969 that NASTRAN was made commercially available by the MacNeal-Schwendler corporation. It was sold as MSC/NASTRAN and was known as the first generation of FEA software.

In the first few years after the software being commercialised, users were unaware of errors that could arise from combining 3-D and 1-D theories into the same mathematical model. In 1972, the first mathematical papers on FEM were published (Strang, 1972). Up until that point FEM was implemented but not mathematically proven.

A few years later in 1977 the first professional p-version FEA software was beginning to develop under Dr.Alberto Peano. The purpose of the software was to perform numerical analysis of dams and other large scale civil structures. The software was called FIESTA and would go on to be the second generation of FEA software.

As the software developed into the 1980's it began to be used for a broader range of applications. These included being used for designing and testing of nuclear pressure vessels, machined components and piping systems. Then in 1983 a new software called PROBE was developed at the center for computational mechanics in Washington University, St. Louis. The reason for the development of the software was to provide a platform for research and design capabilities for aerospace applications. PROBE is considered the third generation of FEA software.

In 2008, with FEA software being more widely used NASA released a standard for the development of FEA models and simulations. The NASA standard consists of two parts, the first being the performance of the software. This includes meeting programmatic, documentation, development, validation and reporting requirements. The second part is a scale for evaluating the validity of the simulation and model using 8 key factors, each having to meet a minimum level of merit. NASA

introduced this standard to bring better compliance and regulation to the field of FEA (Brief History of FEA | ESRD | Engineering Software Research and Development, Inc., 2021).

Optimisation techniques

There are several optimization techniques users can utilise from software. There are 3 main types of structural optimisations, these being Topology optimisation, shape optimisation and sizing optimisation. I decided to research the 3 to get a better understanding of how they work.

Sizing optimisation

Sizing optimisation is the simplest form of the three types. It is seen as the simplest form as there are only minor changes made. In sizing optimisation, the objective is to optimise the structure by adjusting the size of the features. For example, a component can be made thicker to increase the strength of the structure. It is used at the end of the design process when the layout of a component is fixed and cannot be altered in its shape (2021).

Topology optimisation

Topology optimization is renowned for being the most difficult of all three types of structural optimization. It usually starts with an initial design which gets labelled with tailored conditions such as loads and boundary conditions. The optimisation will then begin to distribute and remove material while ensuring that the tailored conditions are still met, and that performance isn't compromised. With this type of optimisation, you can also freeze sections that are critical to the makeup of the structure, and they will not be included for optimisation (About structural optimization, 2021).

Shape optimisation

Shape optimization is performed similarly as the topology optimization. The main difference is in how the design variables are defined. The design variables used in shape optimisation are the coordinates of the boundary. The nodal coordinates are chosen as design variable because it concentrates on minimising stress concentrations. This is done by optimising the geometry of the component rather than its physical properties (About structural optimization, 2021).

Software

The software I am using for the optimisation of the disk is Design studio for Genesis. It is a design oriented pre- and post-processor graphical interface for the GENESIS Structural Analysis and Optimization Software. The main reason for using this software is that it has an easy to use interface and it has all the features I require to optimise the disk. These features include minimisation of mass and volume. The software can also assist with the reduction of displacement, stress and buckling issues.

Baseline structure

To be able to get an optimised version of the valve plate to meet the displacement requirement, I first had to simulate a baseline structure of a given lay up of plies and layers. This would then give an accurate idea of how the plate would perform under the stated conditions. The baseline later was to be made up of carbon fibre reinforced plastic with the following material properties:

Young's Modulus in the longitudinal direction (E_1): 132 GPa

Young's Modulus in the lateral direction (E_2): 10.8 GPa

Poissons Ratio (ν_{12}): 0.24

Inplane Shear Modulus (G_{12}): 5.65 GPa

Transverse Shear Modulus (1-Z plane)(G_{1z}): 5.65 GPa

Transverse Shear Modulus (2-Z plane)(G_{2z}): 5.65 GPa

Mass Density (ρ): 1540 kg/m³

Allowable longitudinal stress in tension (X_t): 1513 MPa

Allowable longitudinal stress in compression (X_c): 1000 MPa

Allowable transverse stress in tension (Y_t): 43.4 MPa

Allowable transverse stress in compression (Y_c): 43.4 MPa

Allowable inplane shear stress (S): 43.4 MPa

The baseline was structured with 25 Unidirectional plies, all of them being 0.1mm thick. They were arranged 5 plies per layer. The first layer was placed at -90° to the direction of the theta vectors. The second was placed at -45°. The third was running parallel to the vectors at 0°. The fourth and fifth were then placed at 45° and 90° respectively.

Before I could run the simulation I had to make some assumptions so that the software knew what I wanted it to do. The first assumption made was that the material in line with the pin wouldn't deflect as the material in that area of the plate would be thicker and therefore assumed to be supported. For the software to ignore this area of the plate for deflection I had to obtain the width of the pin which was 56mm and select the nodes that occupied that space and tell the software to fix them so that they couldn't deflect.

The second assumption made was that the theta vectors would run vertically. This assumption was made because that is the direction the pin runs so that's where the material on either side of the pin would deflect the most. As a good seal was required for the design I wanted a worst case scenario for deflection so that we could achieve the best possible result and this assumption gave me that.

The third and final assumption was that the pressure would be distributed evenly over the surface of the plate. While it is a reasonable assumption to make, it may not be strictly true in a practical situation due to effects such as turbulence.

Before inputting my simulation conditions, I wanted to calculate the force acting on the back of the plate. I took the values from the scenario of the previous assignment to get my answer. The density of the petroleum was 800kg/m^3 , the height of the pipe was 5m and the radius of the pipe was 97mm . I used the following calculations to solve:

$$\text{Pressure} = \text{Fluid density} \times \text{acceleration of gravity} \times \text{height of liquid}$$

$$\text{Pressure} = 800 \times 9.81 \times 5 = 39240 \text{ N}$$

$$\text{Area} = \pi r^2$$

$$\text{Area} = \pi(0.097)^2 = 0.02956\text{m}^2$$

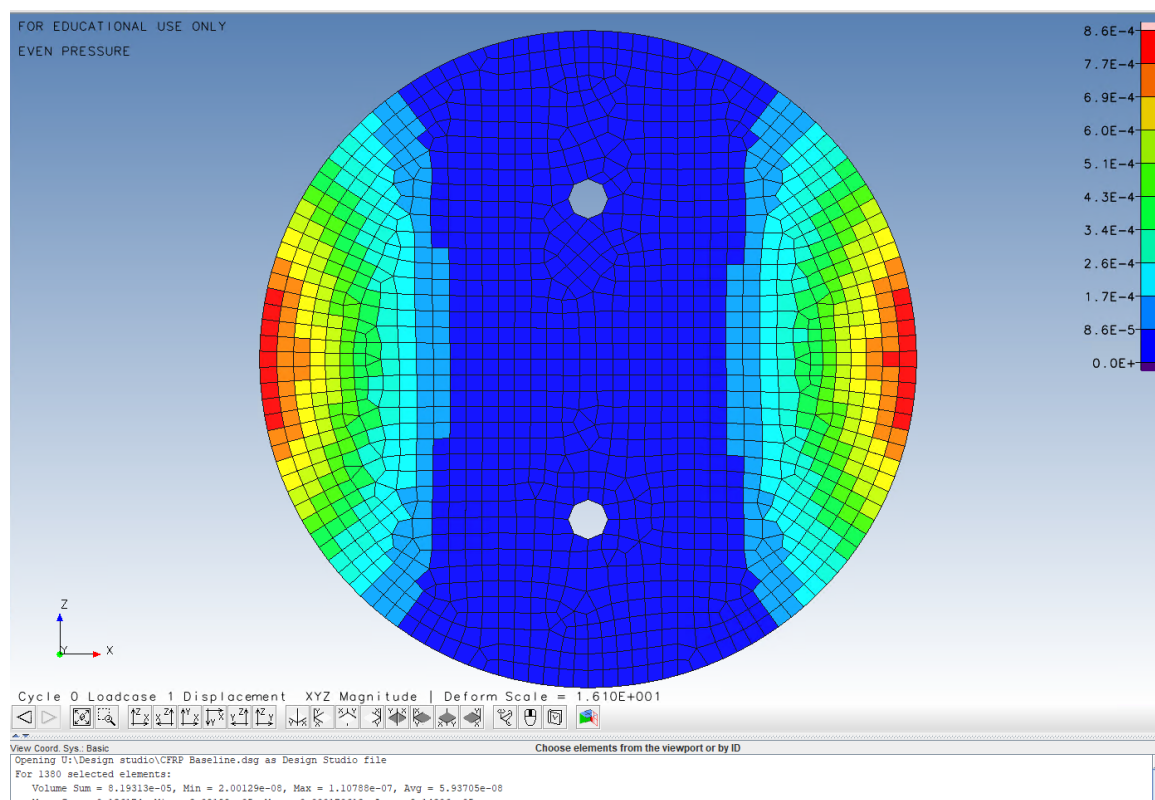
$$\text{Force} = \text{Pressure} \times \text{area} = 39240 \times 0.02956 = 1159.93 \text{ N/m}^2$$

With the force calculated I could then calculate the force on each grid. As there were 1438 grids all I had to do was divide the force by the number of grids.

$$\frac{1159.93}{1438} = 0.8066 \text{ N per grid}$$

Now I had all that I needed to compose the baseline structure in Design Studio. To begin, I loaded in the NASTRAN deck of the simplified plate. The NASTRAN deck already had the mesh generated for the structure. My next step was to set my assumptions up, such as fixing the grids in line with the pin and ensuring the theta vectors were running vertically. I then changed the material properties to the ones posed above. Once this was done, I applied my forces acting negatively in the Y axis so the force would bear down on the plate. My last step in the set up was to structure the plies of the layers in the correct orientation and thickness.

With the set up complete I then ran the simulation using the 'single analysis function'. The results showed that the edges of the plate were deflecting 0.86mm . This meant that as it was the plate wouldn't be fit for purpose as the petroleum would leak through the seals as the gap would be too large. The mass of the plate was 126.17g and it was 2.5mm thick due to the lay up of the 25 0.1mm thick plies.



Optimised structure

With the baseline determined I now wanted to optimise it so that I could meet the design objective. To do this I needed to set an objective, a constraint and some design variables so that the software knew what I wanted to achieve through the optimisation. The first thing I did was set an objective. I told the software that my objective was to keep the mass as low as possible while meeting my constraint. My constraint was then set so that the deflection couldn't exceed 0.1mm. Finally, I set the variables, I did this by telling the software to lay the plies up however it wanted to but within the boundaries I set. Those boundaries were that the plies were to still be 0.1mm thick but layers could be any where from 1-15 plies thick and following the orientations previously arranged. With all my requirements given I used the optimised function on the software. The results were as follows:

Maximum displacement – 0.095mm

Mass – 257.77g

Layer 1 – 11.421 plies = 1.1421mm thick

Layer 2 – 7.566 plies = 0.7566mm thick

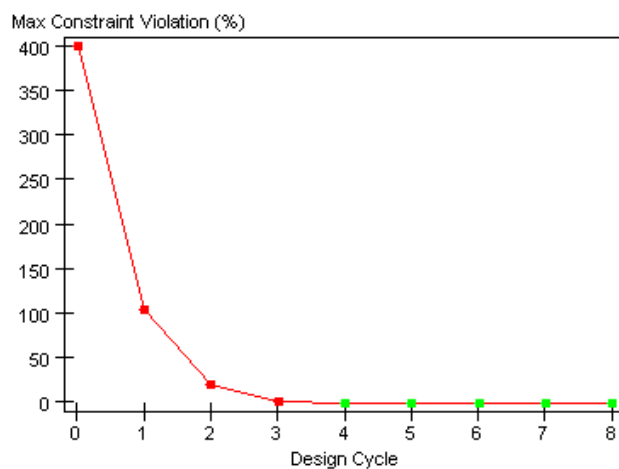
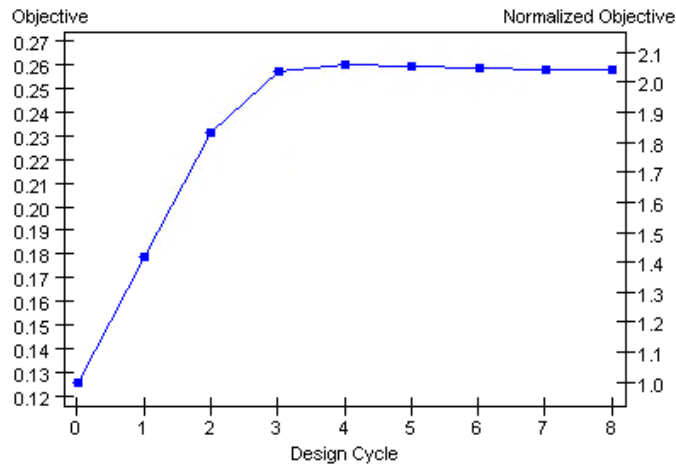
Layer 3 – 7.455 plies = 0.7455mm thick

Layer 4 – 7.557 plies = 0.7557mm thick

Layer 5 – 14.282 plies = 1.4282mm thick

Total – 48.281 plies = 4.8281mm thick

D E S I G N V A R I A B L E H I S T O R Y					
DESIGN CYCLE	DESIGN VARIABLE 1	DESIGN VARIABLE 2	DESIGN VARIABLE 3	DESIGN VARIABLE 4	DESIGN VARIABLE 5
	L1THK	L2THK	L3THK	L4THK	L5THK
0	5.0000E-04	5.0000E-04	5.0000E-04	5.0000E-04	5.0000E-04
1	7.6438E-04	6.7083E-04	6.7661E-04	6.7075E-04	7.6634E-04
2	1.0515E-03	8.2253E-04	8.3675E-04	8.2255E-04	1.0550E-03
3	1.2319E-03	8.7122E-04	8.9380E-04	8.7152E-04	1.2360E-03
4	1.2774E-03	8.6082E-04	8.7592E-04	8.6097E-04	1.2816E-03
5	1.3079E-03	8.4680E-04	8.2612E-04	8.4520E-04	1.3189E-03
6	1.3602E-03	8.0500E-04	7.9070E-04	8.0379E-04	1.3692E-03
7	1.4036E-03	7.6987E-04	7.6106E-04	7.6901E-04	1.4113E-03
8	1.4214E-03	7.5667E-04	7.4550E-04	7.5570E-04	1.4282E-03



It took 8 cycles for the software to reach the solution. It achieved a 0% design violation after 4 cycles but kept adjusting the layers after that to achieve the minimum weight while still meeting the displacement constraint. The mass increased by 104.3% but the deflection decreased significantly. Even though the disk was now thicker (4.8281mm) and heavier than the baseline version, the displacement was brought under the maximum value which meant the disk was now fit for purpose. From a manufacturing point of view though it wouldn't be possible to machine as the accuracy of the make up of the layers was too high. It would be almost impossible and very expensive to produce a layer from plies that have an accuracy of 4 decimal points. So, while this was the most optimised version from the information I gave the software it wouldn't be the most practical to use.

Prototype and testing

It was suggested that a prototype was to be made up for the purpose of testing and checking the accuracy of the results of the software. To do this a manufactured test piece would need to be made. I decided to run another optimised simulation through the Design Studio software. This time however I had to put discrete variables in so that the software had even more boundaries to work under. This would hopefully produce a manufacturable disk to use for testing. The new variables I entered stated that each layer could be still made up from 0.1mm thick plies ranging from 1-15 per layer, but it also restricted the software to only move up or down in 0.1mm increments meaning a degree of freedom was removed from the computing process. With this inputted I ran the simulation, and these were the results that were given:

Maximum displacement – 0.081mm

Mass – 272.54g

Layer 1 – 15 plies = 1.5mm thick

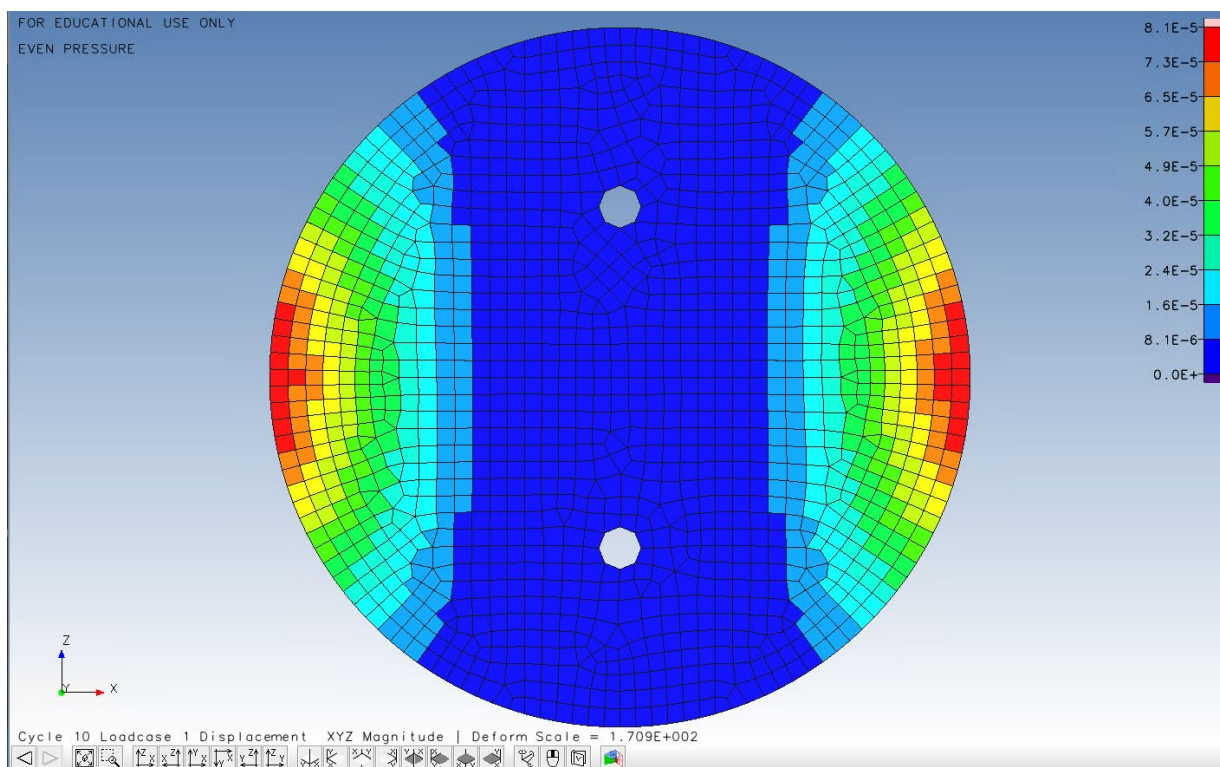
Layer 2 – 8 plies = 0.8mm thick

Layer 3 – 8 plies = 0.8mm thick

Layer 4 – 8 plies = 0.8mm thick

Layer 5 – 15 plies = 1.5mm thick

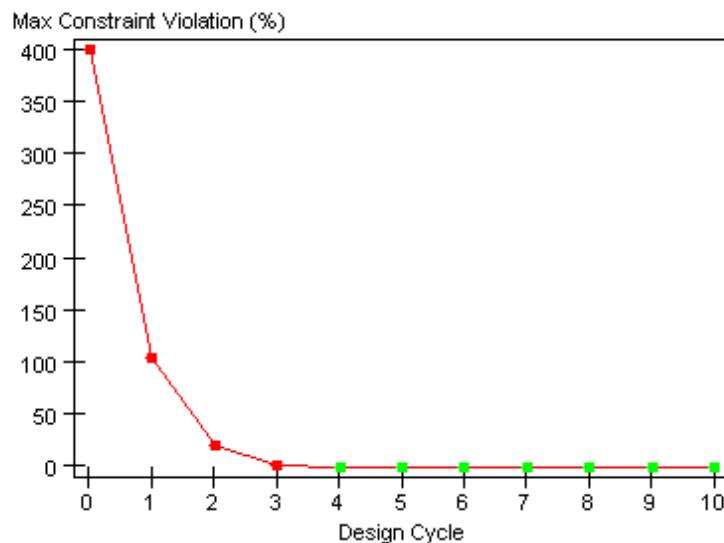
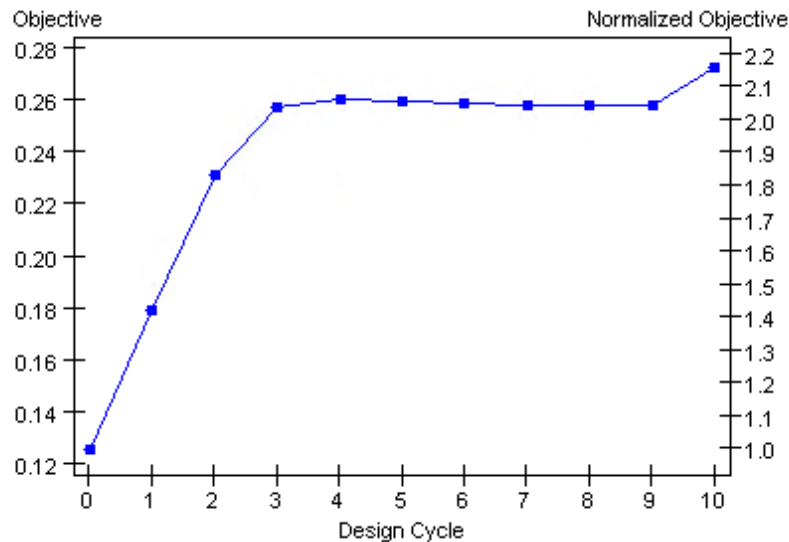
Total – 54 plies = 5.4mm thick



```

DESIGN CYCLE #    10, OBJECTIVE =  2.725364E-01, MAXIMUM VIOLATION =
0.0%.
DESIGN VARIABLE #      1 =  1.500000E-03    L1THK
DESIGN VARIABLE #      2 =  8.000000E-04    L2THK
DESIGN VARIABLE #      3 =  8.000000E-04    L3THK
DESIGN VARIABLE #      4 =  8.000000E-04    L4THK
DESIGN VARIABLE #      5 =  1.500000E-03    L5THK
COMPLETION CODE: 1    1501    0

```



The discrete values helped greatly in producing a result for a manufacturable disk. As you can see it mimicked the optimised simulation, but it then carried on for a further two cycles to determine the closest available 0.1 nominal ply to make the structure from. The graph shows that the weight rose from 257.77g to 272.54g, this is due to the software rounding the plies upwards. It has to do this because if it rounded down then the displacement constraint might be violated. The consequence of this rounding makes it heavier than the optimised version, but it also makes it displace less (0.081mm) so there is a fair trade off of weight to displacement. The disk is also 5.4mm thick instead of 4.8281. Now the disk is meeting the desired result and also is easily manufacturable so that testing can be done on it.

To conduct a fair test the disk would need to undergo the same conditions as the one ran in the simulation. It can be upscaled or downscaled to suit the equipment available, but all parameters would need to be scaled the same so for example if the disk was doubled and made 10.8mm thick the plys would also need to be double the thickness. Also, the force would need to be doubled and so on.

For this scenario I would recommend scaling it up so that the plys of 0.1mm would be thicker and easier to assemble. Assembly would take place through lamination of plys. Lamination is a process by which a material is laid in layers over each other and then cured under pressure in an autoclave to the desired thickness or strength (CFRP, 2021). The main advantage of upscaling this particular plate is that it's easier to obtain results for the displacement. Trying to obtain a displacement measure of 0.1mm or under is hard to do so using basic equipment, so by scaling it up you would be measuring a bigger displacement and making it easier to capture. The disadvantage to this method would be that any errors would be multiplied by the scale meaning potential readings could be out by a large margin. A more accurate but expensive way of taking a reading would be to place a strain gauge over the back of the plate while the test is conducted. The results should then be compared to those obtained in the simulation to verify its accuracy.

Conclusion

To conclude, for the optimisation I made reasonable assumptions that enabled me to be able to know my boundary conditions and parameters. I then set up the NASTRAN deck in design studio using the parameters given to me and the assumptions I made. After that, I ran the baseline structure through the software to get a benchmark to improve on. Once the benchmark was achieved I asked the software to tighten up the displacement by putting a constraint on it while also asking it to keep the mass to a minimum in the objectives section. I gave it freedom to set the plys within a specified parameter. Once everything was set the software optimised the disk to what I had asked. However, it was not a manufacturable solution as the plys had too many degrees of accuracy to be produced. I then set the conditions for the plys in increments and ran the optimisation again and it gave a slightly unfavourable result compared to the previous optimised one but this time it was easily manufacturable compared to the truly optimised one. It still met the displacement constraint however it was slightly heavier. Even though this wasn't favourable it meant it could be easily manufactured for the testing process.

Bibliography

TURNER, M., CLOUGH, R., MARTIN, H. and TOPP, L., 1956. Stiffness and Deflection Analysis of Complex Structures. *Journal of the Aeronautical Sciences*, 23(9), pp.805-823.

Strang, G., 1972. Approximation in the finite element method. *Numerische Mathematik*, 19(1), pp.81-98.

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/#1482981756494-23797cb3-ba9f>> [Accessed 27 October 2021].

Ijirset.com. 2021. [online] Available at: <http://www.ijirset.com/upload/2017/cotii/17_ME_COTII_Structural_Optimization_Methods_A_General_Review.pdf> [Accessed 29 October 2021].

Abaqus-docs.mit.edu. 2021. *About structural optimization*. [online] Available at: <<https://abaqus-docs.mit.edu/2017/English/SIMACAEANLRefMap/simaanl-c-optover.htm#simaanl-c-optover-topo>> [Accessed 29 October 2021].

Plyable. 2021. *CFRP*. [online] Available at: <https://www.plyable.com/capabilities/cfrp/?utm_term=carbon%20moulding&utm_campaign=CFRP+Capability&utm_source=adwords&utm_medium=ppc&hsa_acc=8496985291&hsa_cam=14938008856&hsa_grp=126736064045&hsa_ad=552691770402&hsa_src=g&hsa_tgt=kwd-311289148732&hsa_kw=carbon%20moulding&hsa_mt=b&hsa_net=adwords&hsa_ver=3&gclid=Cj0KCQjwt-6LBhDIARIsAIPRQcKuThW4FYleonnqhagtl-Gv4kGdvG0tB-YqUA6eal_S1A-E-EAoRG4aAvAOEALw_wcB> [Accessed 29 October 2021].