



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

University of Wales Trinity
Saint David

School of Engineering

Structural analysis of a propeller blade using carbon fibre reinforced plastics

Matthew Johnson (2012415)

1 Content

2	Introduction	3
3	Literature review.....	4
3.1	Historical development.....	4
3.2	Basic concepts:.....	4
3.3	Possible future developments	5
3.4	Improving the design process	5
3.5	Optimisation Techniques	6
4	Design Studios optimisation of propeller using CFRP.....	7
4.1	Stainless steel.....	8
4.2	Carbon fibre reinforced plastic (CFRP).....	8
4.3	Optimised CFRP.....	9
4.4	CFRPs optimised for manufacturing	10
4.5	Results Comparison	11
4.5.1	Stainless steel.....	11
4.5.2	CFRP	11
4.5.3	CFRP optimised	11
4.5.4	CFRP optimised for manufacturing.....	11
5	Prototype experimentation	12
6	Conclusion.....	13
7	Bibliography	14

Figure 1 - (Engineering design and analysis, 2021).....	4
Figure 2 – change in force from the front of the blade to the back.	7
Figure 3 – Area held in propeller hub	7
Figure 4 – Stainless steel deflection.....	8
Figure 5 – CFRP deflection	8
Figure 6 – Layer build-up optimised	9
Figure 7 – optimisation of CFRP's	9
Figure 8 – optimised for manufacturing	10
Figure 9 - optimisation for manufacturing of CFRP's.....	10
Figure 10 – results table.....	11

2 Introduction

This report covers the weight optimisation for manufacturing a carbon fibre reinforced plastic propeller blade (CFRP); this will follow the stages from the base stainless-steel model through to the final optimised manufacturing stage. A supporting literature review will be carried out relating to basic concepts of finite element analysis (FEA), the evolution of the process, possible future development, and the optimisation techniques of FEA. Models will be optimised by using different layers of CFRP in a single model, in which the carbon fibre is orientated at different angles. This process will increase layers until it has met the objective, and then change the layers of CFRP until it has the lowest weight. Once the model has been finalised, a discussion of a possible experiment and how data should be obtained, with a comparison to the simulation.

3 Literature review

3.1 Historical development

The concept of FEA was pioneered by the works of Hrenniffoff (1941) and Courant (1943) whilst solving complex elasticity and structural analysis problems for the development of aeronautical engineering (Fadiji, et al. 2018). The term FEA was ultimately coined by Ray William Clough in his book (National academy of engineering, 2019); which relied on the work done by R.W Clough and M.J. Turner in 1956 in the development finite element method (FEM), whilst working at Boeing (Pradhan, 2019). FEA continues to be the predominant strategy employed by engineers to conduct structural analysis. FEA is the application of FEM; FEM was developed with no efficient implications. As computers developed and advanced the complexity and scope of ability of FEA software will grow.

3.2 Basic concepts:

FEA is the application of FEM on a physical phenomenon, which is a computational technique used to obtain approximate solutions to boundary value problems in engineering. FEM is the procedure of breaking an element down into a finite number of elements (Pradhan, 2019). An object is broken down by geographically similar models consisting of multiple-linked, simplified representations of discrete regions. The system consists of points called nodes, forming a mesh grid. (Hearn, 1977)The mesh contains the material and structural properties which will govern how the structure will react in certain conditions (Fadiji, et al. 2018).

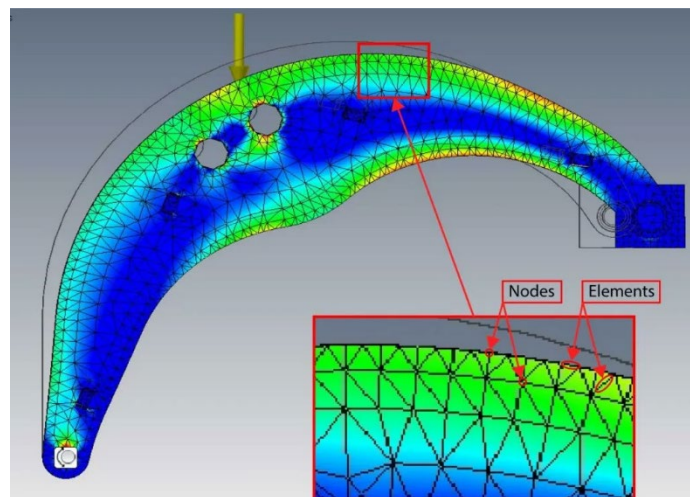


Figure 1 -showing Meshing, node points and elements (Engineering design and analysis, 2021)

3.3 Possible future developments

Future advancement of FEA relies on rectifying the limitations of current software. As computers advance further programs will have to reduce cycle time for the analysis, as computers have more ram (random access memory). Therefore, programs will have the capability to have a higher number of nodes allowing for more accurate results. (De Borst & Vermeer, 1984)

The software relies on there being no defects in the material or the manufacturing process which there inevitably will be, therefore parts are rigorously tested before use. The program is used as a basis for the design but not as a final test. (Lowe & Laitinen, 2021)

Another limitation is the number of nodes in terms of cycle time. Nodes are the average force across the elements, if a large, unexpected force were to hit part of the model it could fail. It also does not consider changing pressures from natural disasters. Although more nodes this force more accurately it is impossible to know the force which could be applied. (Hearn, 1977)

When trying to manipulate static force on an area, if the line passes through the area, it will select the nearest element to where a line has been drawn. To correct the issue there should be a way to place lines down to allow the program to create nodes around to increase the accuracy.

3.4 Improving the design process

FEA is the numerical technique that is used to predict the reaction of real-world forces e.g., vibration, heat, and fluid flow on an object. FEA can predict the functionality range of an object, or if it will break or wear out. FEA can be used to simulate most if not all environments which give it great utility. (Rao, 2010)

According to (Rao, 2010) FEA is a useful tool and can assist with:

- Provide comprehensive results while generating the physical response of a system at any instantaneous point.
- Allowance of multiple tests of the same model under different circumstances.
- The ability to run a multitude of tests in a shorter time scale with fewer resources as opposed to experimental tests.
- Allowing for simple safe simulation of potentially dangerous or impractical conditions of failures.
- Provide a rapid calculation for most applications with relatively low investments making it useful in a wide range of engineering problems such as solid mechanics, dynamics, and heat transfer.
- Make extrapolation of experimental results possible through parametric analysis of the validated models.
- Provide visual representations and simulation calculations for different parameters (for example stress or temperature distribution) making it possible for rapid performance evaluation and modification.
- Provide a rapid calculation for most applications with relatively low investments making it useful in a wide range of engineering problems such as solid mechanics, dynamics, and heat transfer

3.5 Optimisation Techniques

For the software to work effectively, certain parameters need to be met. These parameters allow for the software to work efficiently whilst containing all the necessary information to allow for the correct results. This can be related to mesh creation which determines the time taken to apply forces to the medium. Mesh is the breakup of a model into many sections, at the crossroad of these sections are known as nodes. Nodes are where forces are applied, for each node a calculation is carried out, the more nodes the longer the sum of the calculations takes, however, increases accuracy (Hearn, 1977). There is a stress value that the software knows, the more nodes the closer to the known stress value, thus a more accurate result.

To construct FEA then SIMPLE should be applied, which stands for:

- Split – deciding how to break down the model into elements (creating a mesh)
- Interpret – removing small surfaces and simplifying geometry without causing major changes to the part to reduce cycle time.
- Material properties – materials may contain defects that could lead to the part being at the minimum/ maximum limit.
- Physical properties – the implication that the part shape would have, the thickness and orientation.
- Loading and Boundary conditions – Direction and magnitude of the force, forces can be static or dynamic, changes of properties at boundary conditions
- Evaluation – evaluating the results, scrutinising and drawing conclusions.

Simplifying the model is to reduce the direction of freedoms or the number of nodes, The distance between each node can also be changed, if the distance from one node to the other is increased, there are fewer nodes required equating to fewer calculations. If the design can be done in a 2D plane then it should, 2D shapes simpler have fewer nodes. Removing fillets and changing complex shapes into more square shapes, simplifying the shape. Removing fillets and changing parts to more simplified shapes should only be done if it doesn't heavily change the physics of the part.

4 Design Studios optimisation of propeller using CFRP

A provided model has been provided to be analysed by Design studios. The point of the report is to create a blade that has a displacement of less than 100mm with the lowest weight. Using weight and stress from the results stiffness can be calculated. The results from each test will be displayed in a table after all models have been made.

The propeller blade will be placed under a force of approximately 125kN. This has been determined to be the force that would be experienced by the tanker propeller blade, which was calculated in a separate report.

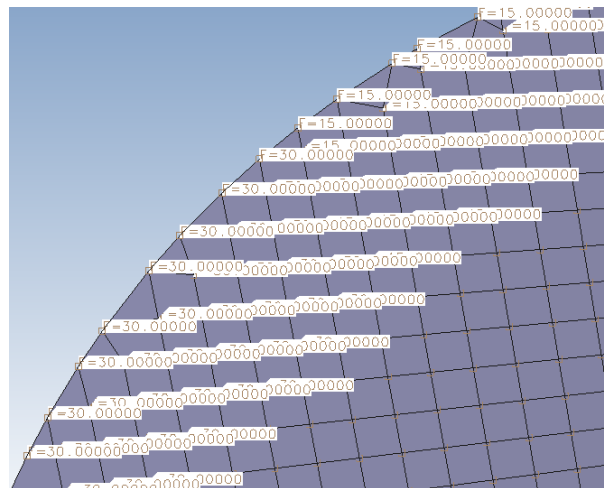


Figure 2 – change in force from the front of the blade to the back.

It was determined that force acted more prominently at the leading edge of the part which then dissipated as the fluid moved over the surface of the propeller blade. This has been illustrated in figure 2 by the change from the average 30 newtons of force at the front half of the blade to 15 newtons at the back half of the blade.

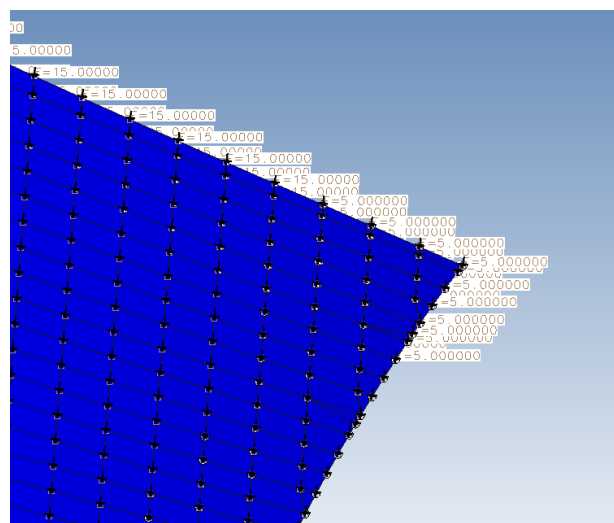


Figure 3 – Area held in propeller hub

In figure 3, the force has been reduced at the point at which the plate would meet the propeller hub to an average force of 5 newtons. Some of the blade would lay inside the propeller hub to allow a strong connection which is less likely to fail in comparison to other connecting techniques such as welding.

4.1 Stainless steel

In figure xx the propeller of mass 650kg has been exposed as previously stated, a maximum displacement of 5.4 metres is shown in figure 4. The objective is to reduce the displacement to 0.1 metre displacement.

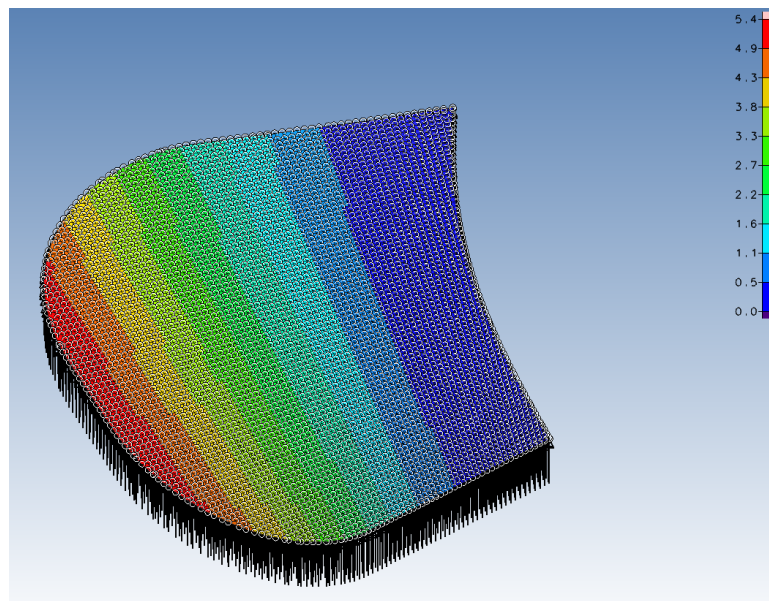


Figure 4 – Stainless steel deflection

4.2 Carbon fibre reinforced plastic (CFRP)

The displacement of the blade is 5737 metres with a mass of 32kg seen in figure 5, the displacement is to be expected for the structure due to the thinness of the blade.

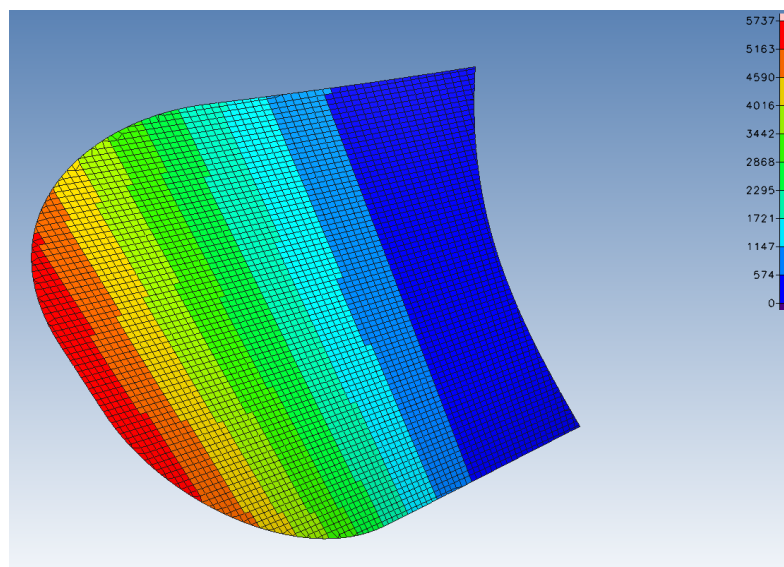


Figure 5 – CFRP deflection

4.3 Optimised CFRP

During the optimisation, the program increased the mass of the blade by increasing the layers of CFRP proportionally, until the model no longer compromises the constraint violation (figure 7). The model has a mass of 560.1 kg and a displacement of 100mm

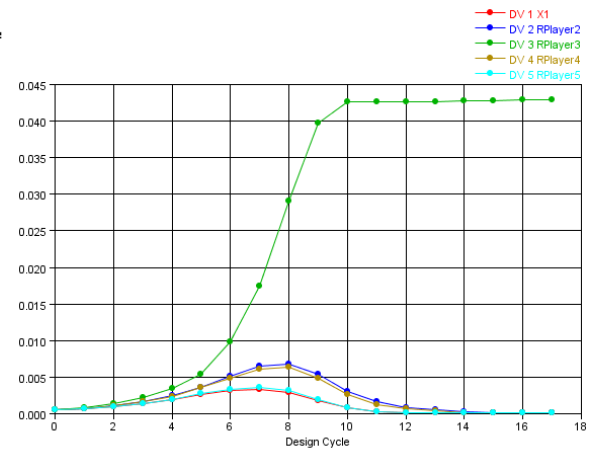
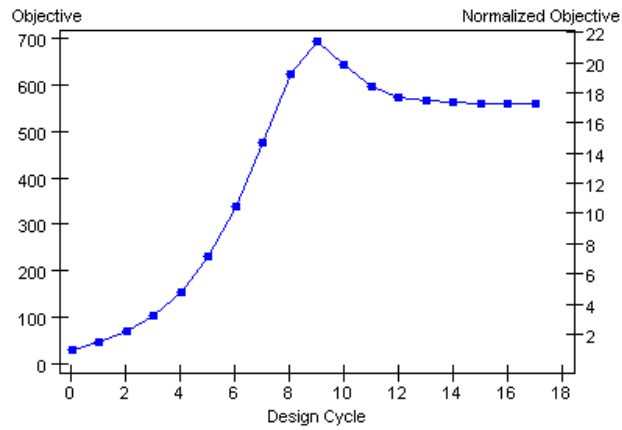


Figure 6 – Layer build up optimised

Figure 7 – optimisation of CFRP's

4.4 CFRPs optimised for manufacturing

Optimisation is creating an ideal plate with no restriction to the manufacturing process, however; this is not always optimal. The model weighs 575.5kg without violating the constraints, shown in figure 9.

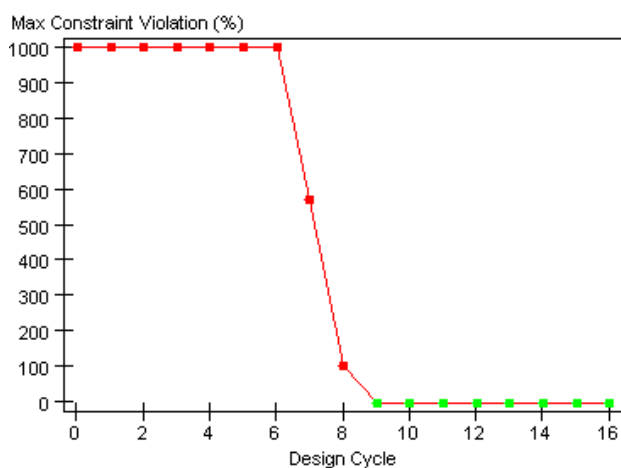
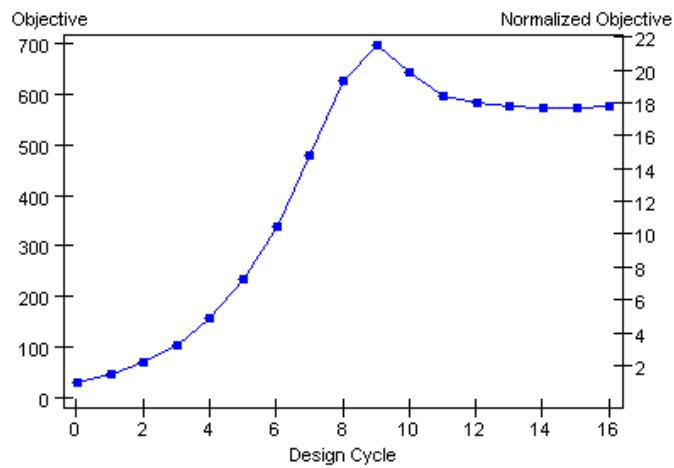


Figure 9 - optimisation for manufacturing of CFRP's

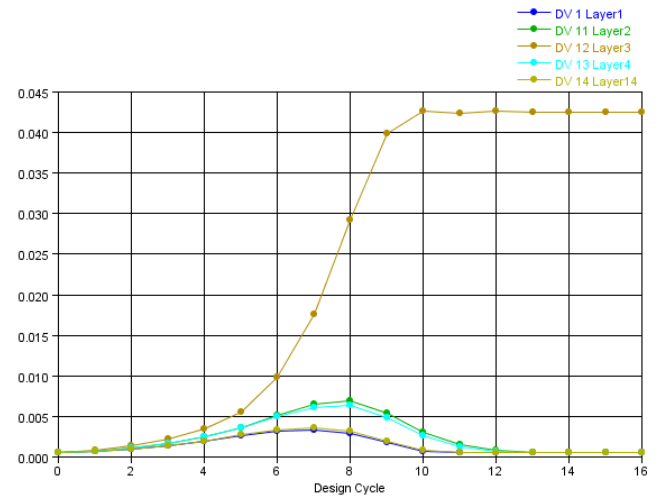


Figure 8 – optimised for manufacturing

4.5 Results Comparison

Stiffness table	Stainless steel	CFRP	CFRP optimised	Optimised for manufacturing
Mass (kg)	646.67365	32.333687	560.1	575.5
Weight	6,343.87	317.19	5,494.58	5,645.65
Displacement	5.4	5737	0.1	0.1
Stiffness	1174.791	0.055288	54945.8	56456.5

Figure 10 – results table

4.5.1 Stainless steel

Due to stainless being, P shell material are isotropic as their properties are the same in all orientations (Gu, et al., 2020). As such no optimisation can be done as the only way to reduce displacement is to add more mass to the component; stiffness remains constant as mass increases.

4.5.2 CFRP

CFRP has been layered up as per the brief, the results show that this stiffness of the unoptimised material is extremely low in comparison to the other materials. Each orientation is made up of the same number of layers. As CFRP is a Pcomp material the properties act in specific orientations (Hong, et al., 2022). In this program, the model doesn't have enough layers to resist force shown by the displacement being 5737 metres.

4.5.3 CFRP optimised

In this program the model begins by increasing all layers (seen in figure 7), increasing the overall mass, which can be observed from design cycle 0-5 of figure xx. As layers increase it is noticeable that layer 3, the layer at 0 degrees is increasing more than the other layers. From cycles 5-9 layer 3 is seen to increase exponentially; when layer 9 begins to flatten the model no longer violates the constraints (figure 7). Design cycle 9 is also when the other orientations begin decreasing in layer width. The conclusion is to be made that layer 3 has the property orientation required to increase the overall stiffness of the part (Hosoi, et al., 2015). As the other layers decline the overall mass of the propeller decreases close to its minimum value (figure 6).

4.5.4 CFRP optimised for manufacturing

Although the third program shows the optimal layer build-up of the component it isn't the most practical for manufacturing. The layers need to be of a certain thickness as it is impractical to make them from anything thinner than 5 microns and creating layers that can be between 6-9 microns thick would increase costs (figure 9). In the fourth program, discrete layer thickness has been added. At design cycle 16 the mass increases to 575.5kg instead of staying at 560kg (Figure 8). The expectation was that the mass would have decreased at this point. If the program rounded the layers downwards it would have violated the constraints, as such the program rounded layer thickness upwards.

As the results show CFRP optimised for manufacturing has the highest stiffness and is ideal for the propeller blade.

5 Prototype experimentation

To begin the prototype is created at a fraction of the size of the model, thus force applied needs to reflect this change. There are other constraints such as how the force would be applied, it would be difficult to accurately dissipate a force evenly over the area of the blade. Other issues such as density and hardness of how the pressure is applied. In the program, 2 separate pressures were applied to show differentiating pressures which would be seen in real-life use.

To replicate the model the blade is held into a prototype propeller hub, this hub will be secured to a flat horizontal surface; the hub replicates that the stress has moved 0.4 metres in from the edge of the blade. The drawback of the positioning is it causes the weight of the blade to influence the overall stress when applying force, this fact can be considered. Once secure a hydraulic press is used to apply the stress, the point of the press would have an attachment to allow for a flat pressing motion against the blade. the hydraulic press would line up with the centre of pressure to best replicate the overall force. The hydraulic press would act like a singular node applying across the whole part. The issue with this is it could damage the blade, a soft tip with maybe a rubber surface should be implemented. The setup for this experiment could be used to test other factors such as creep and fatigue if required.

To detect the displacement of the blade, an encoder-based system can be attached linearly edges of the blade. The original idea would be to use a laser-based system on several areas; displacement would be a curvature that would be unsuitable. Ultrasonic can be used where the blade meets the hub, this would show internal stresses at this point and the microcracking in the plastic if any.

The results will be different from the program as the deflection of the blade is less gradual compared to the program; due to the force being centred at a point instead of spreading over the blade. This will lead to excess stress at the point where the blade meets the hub. The overall deflection of the experiment would be comparable but not exact as the program. The Ultrasonic will show a build-up of stress at the hub. It also identifies whether microcracking or any damage is caused to the blade. it can also gauge the creep and fatigue of the part. In real life, the blade would undergo thousands of cycles of unloading and unloading as well as sustained forces, thus testing for these is also important.

An improvement to this experiment might be to use the smaller prototype on a smaller motor, at a depth that could give comparable results to the FEA.

6 Conclusion

This report has identified the benefits of FEA in engineering and why it is a necessity in the development of parts. The reduction of costs, the time saving, and the ability to re-run and alter programs is beneficial to the design process. There must also be a balance between accuracy and time taken, there's no point running the test if the program surpasses the deadline. It also identifies the current limitations of the software and hardware which can be improved upon in the future. Optimisation techniques are used to reduce cycle time to allow for quick results whilst not compromising the accuracy of the results.

From the programs run it can be deduced that CFRP's which have been optimised with constraints, restrictions, and objectives have high stiffness value than stainless steel. It also identifies that CFRP's running parallel to the blade increases the stiffness of the blade. Whereas CFRPs which are perpendicular do not affect the blade. It acknowledges that the costs of production must be considered, not only its efficiency.

The planned experiment illustrates the benefits of the FEA as changing the model, and running the experiments every time would be costly and a waste of time. However, it also is beneficial to test the theory to have confidence in the hypothesis determined by FEA. Both FEA and experimentation are needed in modern industry to have confidence in the design.

7 Bibliography

De Borst, R. & Vermeer, P. A., 1984. Possibilities and limitations of finite elements for limit analysis. In: *Géotechnique*. Delft : ICE Virtual Library, pp. 199-210.

Engineering design and analysis, 2021. *FEA Mesh Elements & Nodes Guide | Intro to FEA*. [Online] Available at: <https://asrengineering.com/2019/09/23/fea-mesh-elements-nodes-guide-intro-to-fea/> [Accessed 04 November 2022].

Gu, J., Le, K. & Lei, S., 2020. *Mechanics of Materials*. 2nd ed. Jiangnan: Elsevier.

Hearn, E. J., 1977. *mechanics of materials*. 3rd ed. oxford: butterworth-Heinemann.

Hong, H., Jung, H. & Oh, Y., 2022. Preparation and characterization of carbon fiber reinforced plastics (CFRPs). *Composite structures*, 3(6), p. 3.

Hosoi, A., Sakuma, S., Fujita, Y. & Kawada, H., 2015. Prediction of initiation of transverse cracks in cross-ply CFRP laminates under fatigue loading by fatigue properties of unidirectional CFRP in 90° direction. *Composites Part A: Applied Science and Manufacturing*, Volume 68, pp. 398-405.

Lowe, B. & Laitinen, M., 2021. *Ceramics International*. *FEA evaluation of material stiffness changes for a polymer assisted 3D polycaprolactone/β-tricalcium phosphate scaffold in a mandibular defect reconstruction model*, 47(6), pp. 8075-8081.

National academy of engineering, 2019. *National academy of engineering*. [Online] Available at: <https://www.nae.edu/19579/19581/51314/219715/219751/RAY-W-CLOUGH-19202016>

Pradhan, K. K., 2019. *Computational Structural Mechanics*. Sneathish Chakraverty: Elsevier Inc..

Rao, S., 2010. *The finite element method in engineering*. Sixth ed. Coral Gables: Elsevier.