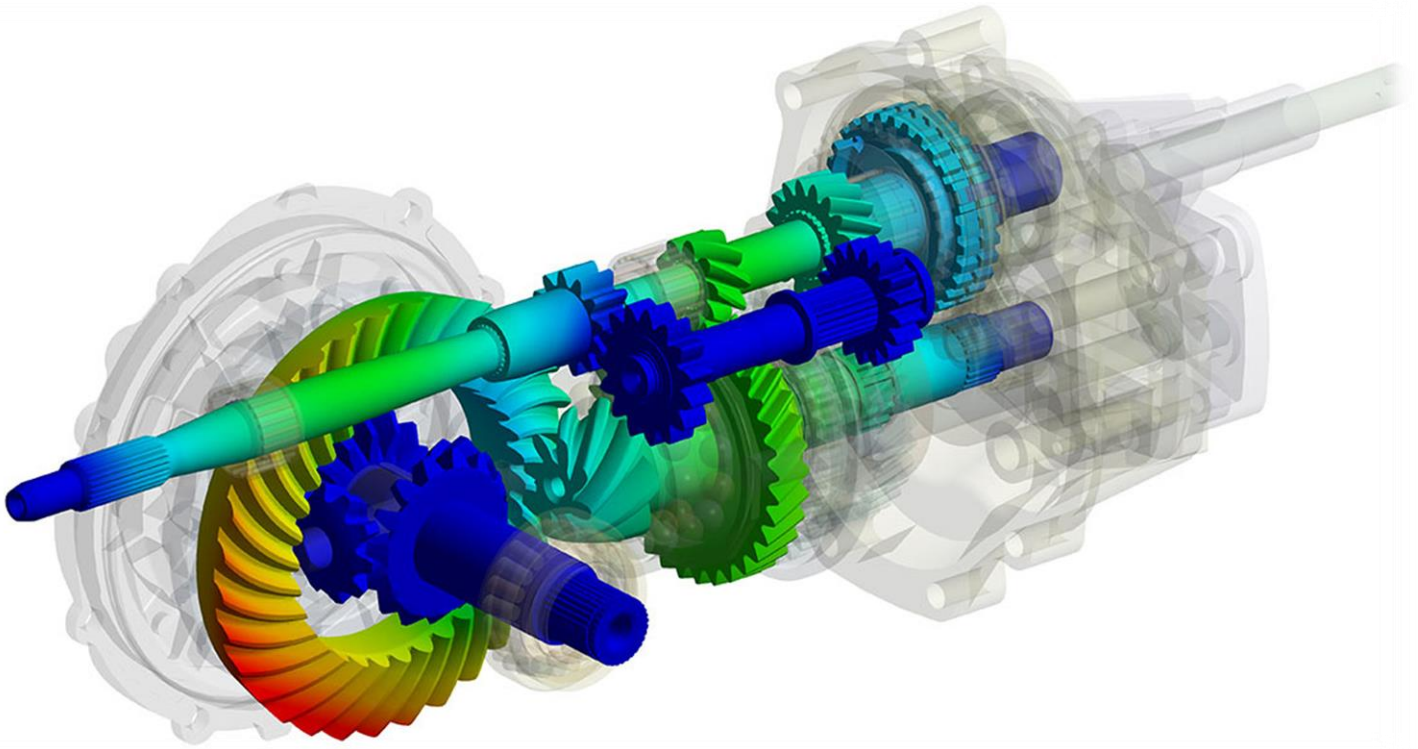




# Analysis of a Helicopter Rotor

---

Advanced Computational  
Methods (M6X00408)

---

# 1. Index

|                                        |    |
|----------------------------------------|----|
| 1. Index.....                          | 1  |
| 2. Abstract.....                       | 3  |
| 3. Introduction .....                  | 4  |
| 4. Investigation and preparation ..... | 5  |
| 5. Forces .....                        | 9  |
| 5.1. Thrust force.....                 | 9  |
| 5.2. Centrifugal force .....           | 10 |
| 6. FEA Analysis .....                  | 11 |
| 7. Analysis of results .....           | 22 |
| 8. Conclusion.....                     | 23 |
| 9. Works Cited.....                    | 24 |

---

## 2. Abstract

This report looks to analyse a helicopter tail rotor blade with the use of SolidWorks FEA analysis package. There are many factors to consider, most of which are analysed whereas some are not. The general forces of centrifugal, thrust to counteract the torque generated by the main rotor, frequency to ensure that the natural frequency of the body in operation is avoided, and finally the fatigue analysis. The latter was not analysed, simply due to the fact that the chosen material did not have the full mechanical properties available within SolidWorks, and therefore it was not possible to generate an S/N curve.

---

### 3. Introduction

A helicopter tail rotor is subject to several stresses and strains, the most obvious of which is that of centrifugal force due to the high speed of rotation, for an Apache helicopter specifically this is 1,411 RPM [1]. Knowing the rate at which the rotor rotates, we need to understand why this is. The sole purpose of the tail rotor is to counteract the force in the form of torque that is created by the main rotor [2], it allows the aircraft to remain straight when in hover or travelling in a single direction, it also allows the helicopter to turn or rotate/turn right and left [2] [3] [4].

Having identified the two main forces acting on the tail rotor, one should consider frequency, this is certainly a moving part, it has the potential to vibrate and therefore we must consider this as a factor for analysis [5].

---

## 4. Investigation and preparation

Upon analysis of the CAD model, one does become aware of the fact that the available models are not a true representation of an actual helicopter, this is most prominent when evaluating the scale of the model itself [1].

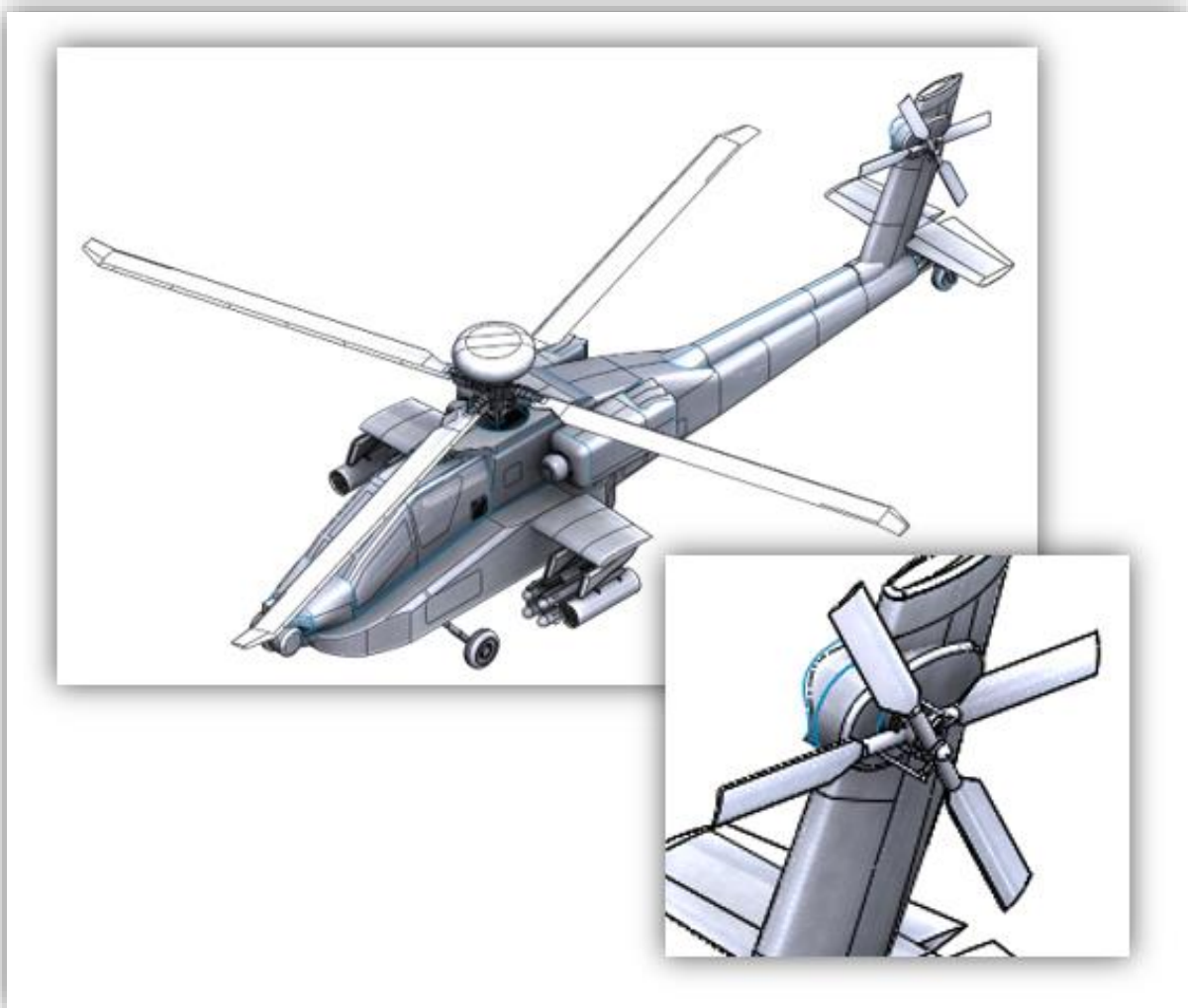


*Figure 1 - Helicopter model isometric view (SolidWorks, Kilbane, 2019)*

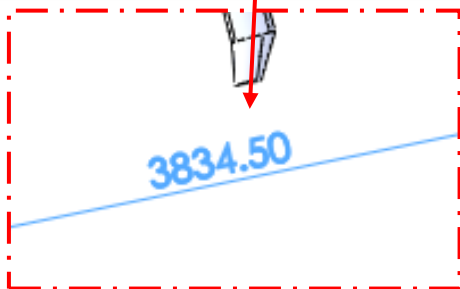
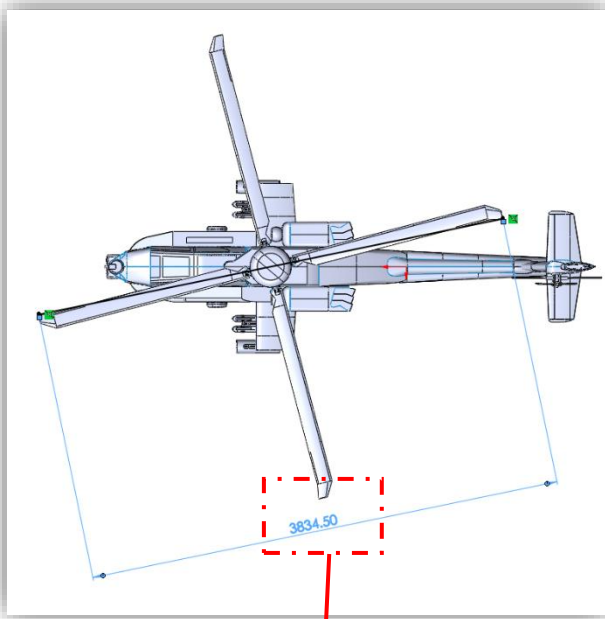


Figure 2 - Helicopter model plan view  
(SolidWorks, Kilbane, 2019)

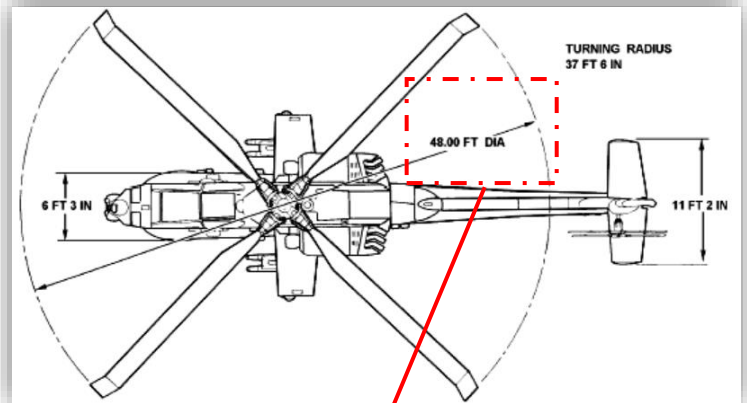


Figure 3 - Apache dimensions (Inetres.com, 1999)

$$48 \times 12 \times 25.4 = 14630.4 \text{ mm}$$

To find conversion rate;

$$\frac{\text{Actual size}}{\text{Model size}} = \text{Conversion scale}$$

$$\frac{14630.4}{3834.5} = 3.815464859$$

∴ The supplied model needs scaling to 3.815464859 of its original size in order to obtain realistic results.

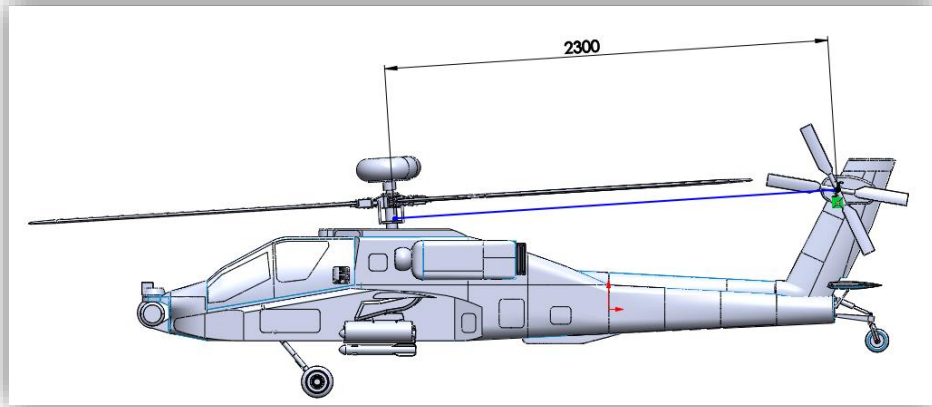


Figure 4 - Helicopter model side view (SolidWorks, Kilbane 2019)

$$2300 \times 3.815464859 = 8.776 \text{ m}$$

This is important as we require it to calculate the force required by the tail rotor to counteract the torque created by the main rotor [2]. Now that the conversion rate is known, we need to apply the same for the tail rotor blade that is to be analysed, but first, we need to ensure that the model is adequate for the analysis.

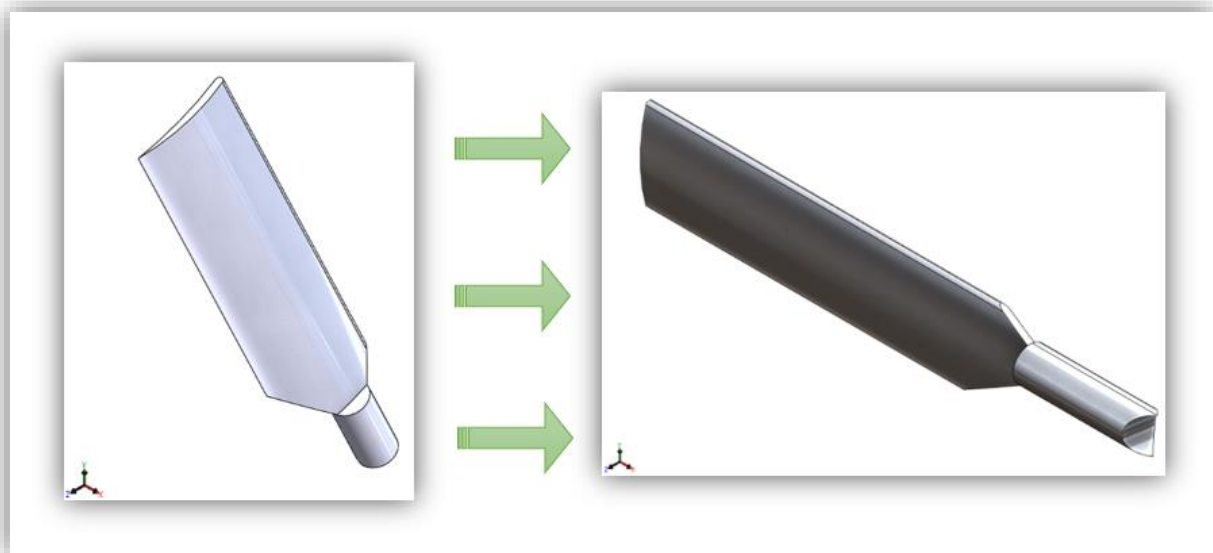


Figure 5 - Rotor blade models (SolidWorks, Kilbane, 2019)

---

The modifications have been made to the model:

1. The scale has been changed.
2. The Shaft has been extended to the true centre of the rotor providing an accurate length, this is important for accurately analysing the centrifugal force as it provides an accurate mass and a true radius for rotation.
3. When applying the loads in FEA, it is necessary to specify a direction, due to the complexity of this shape it is good to orientate to a common position as shown. This is not however essential as basic trigonometry may be used, but this makes life easier.

The material is another important factor, in reality the rotor blade would be a composite structure [6], often consisting of a metallic or carbon fibre skin, with a honeycomb and foam centre [7]. Due to the complexity of such a product, a decision has been made to choose a tough material that is also light, therefore aerospace grade 7075-T6 aluminium [8] has been chosen. It has a density of  $2,810 \text{ kg/m}^3$  [8] [9], in comparison, Titanium Ti-6Al has a density of  $4,429 \text{ kg/m}^3$  [9], meaning that the smaller the density value, the lower the centrifugal force [10].

---

## 5. Forces

### 5.1. Thrust force

A force is acting on the tail rotor blade due to the torque created by the main rotor, the force may be calculated by dividing the torque by the distance between centres of the rotors. Calculating the torque may be done by dividing the power output of the engine by the angular velocity of the main rotor blades. The selected engine (T700-701D) has a power output of 1,491kW [11], the Apache helicopter has two engines [12], making a total output of 2,982 kW. Also, the engine will experience some loss of efficiency due to gearing and the due to the power required to operate the tail rotor itself, this loss is estimated to be ~8% under normal working conditions [2], making a total available engine power of 2,743 kW. The main rotor speed is 289 rpm, this must be converted to radians per second.

$$F_{rotor} = \frac{\tau}{d}$$

$$\omega = \frac{v \cdot \pi}{30}$$

$$\tau = \frac{p}{\omega}$$

$F_{rotor}$  = Force on the rotor

$\tau$  = Torque

$\omega$  = radians per second

$F_{blade}$  = Force on the blade

$$\omega = \frac{289 \cdot \pi}{30} = 30.264 \text{ rad/sec}$$

$$\tau = \frac{2,743}{30.264} = 90.65 \text{ kNm}$$

$$F_{rotor} = \frac{90.65 \times 10^3}{8.776} = 10.329 \text{ kN}$$

The force is applicable to the entire tail rotor, therefore the force actin on a single blade is:

$$F_{blade} = \frac{10.318 \times 10^3}{4} = 2.5795 \text{ kN}$$

---

## 5.2. Centrifugal force

When a body of mass is fixed at an apex point or centre of rotation, and is then rotated about that centre, a natural force is acting in a directions from the centre outwards, this is known as a centrifugal force [10].

$$F_{centrifugal} = \frac{m \cdot v^2}{r} \quad \text{and} \quad F_{centrifugal} = m \cdot \omega^2 \cdot r \quad [10]$$

$$\begin{aligned} F_{centr} &= \text{centrifugal force} \\ m &= \text{mass of blade} \\ r &= \text{radius from base of blade to CofG (centre of gravity)} \end{aligned}$$

$$rpm = 1,411 \text{ rpm}$$

$$m = 15.567 \text{ kg}$$

$$r = 0.533 \text{ m}$$

$$\omega = \frac{1,411 \cdot \pi}{30} = 147.76 \text{ rad/sec}$$

It is now possible to calculate the centrifugal force:

$$F = 15.567 \times 147.76^2 \times 0.633 = 215.141 \text{ kN}$$

---

## 6. FEA Analysis

Having identified the two forces acting on the body, we need to consider a practical way of analysis that would best represent the forces in a realistic scenario. It would be easier to carry out the analysis in two separate studies, but this does not offer a true reflection of the blade in operation, therefore the CofG (Centre of Gravity) has been selected as a point to apply the forces. First stage is to measure the mass properties of the body, this provides the CofG, the coordinates of which are required in order to apply the forces.

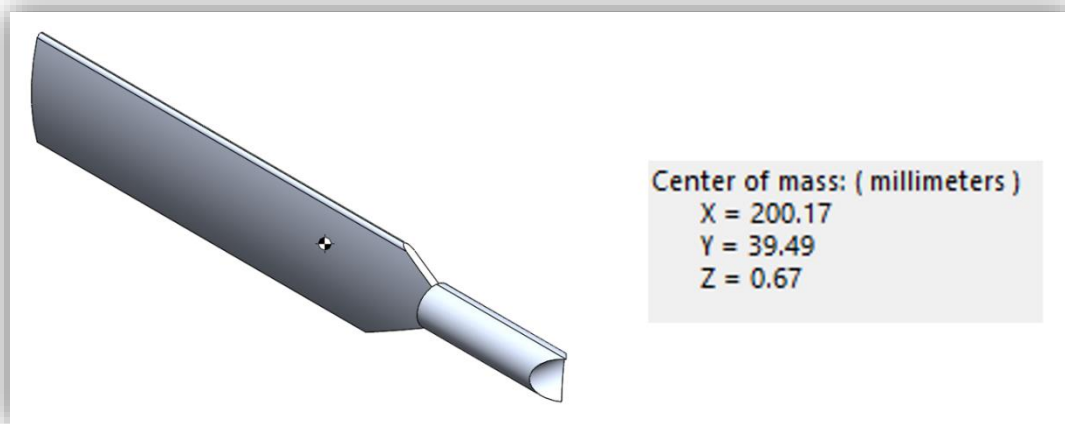


Figure 6 - CofG of modified rotor blade (SolidWorks, Kilbane, 2019)

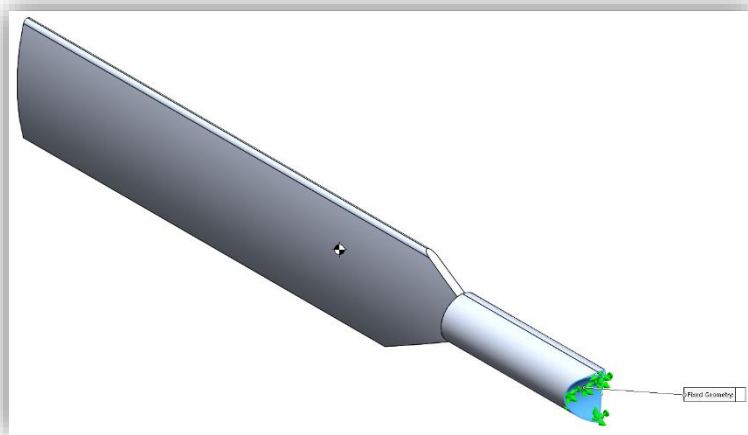


Figure 7 - Fixing face of modified rotor blade (SolidWorks, Kilbane, 2019)

The geometry is fixed at one end using the standard fixed geometry command, this is because the blade would be fully attached to the spindle.

Using the remote load/mass command within SolidWorks, the CofG coordinates are used to determine the exact location of where the load will act.

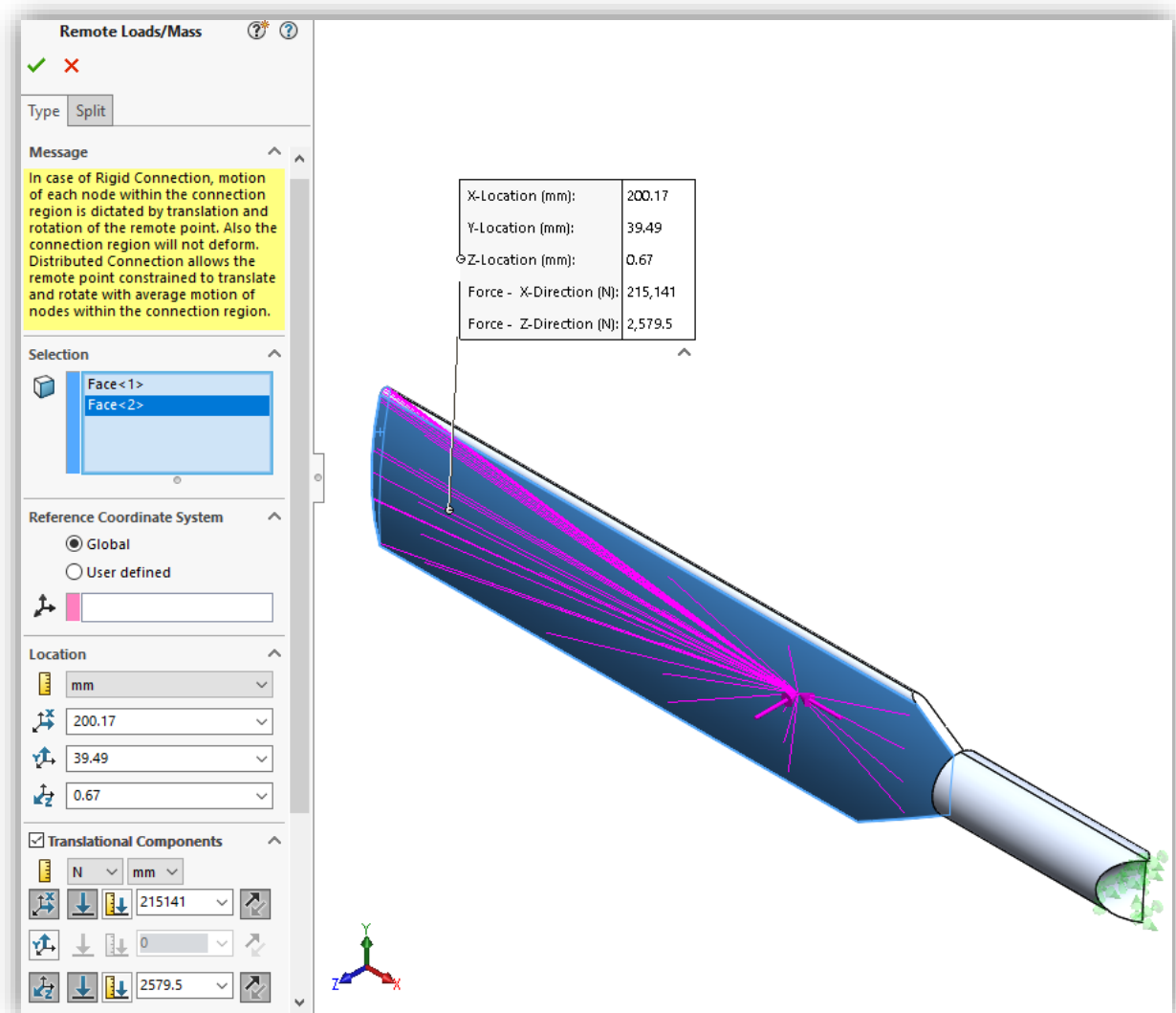


Figure 8 - Applying forces to modified rotor blade (SolidWorks, Kilbane, 2019)

For simplicity, the force created by the torque of the main rotor is applied directly to the CofG of the body. In reality, this load would be distributed across the entirety of the front face, but in order to run this simulation consistently and as a single study it has been decided to take that uniquely distributed load and apply it to a single point. The results would not be too dissimilar than if it were applied across the entire face. Furthermore, in a static testing scenario it is rather common to test using this logic, a CofG is determined and the test load is applied directly to that point, therefore this method applied may be considered as acceptable for this analysis.

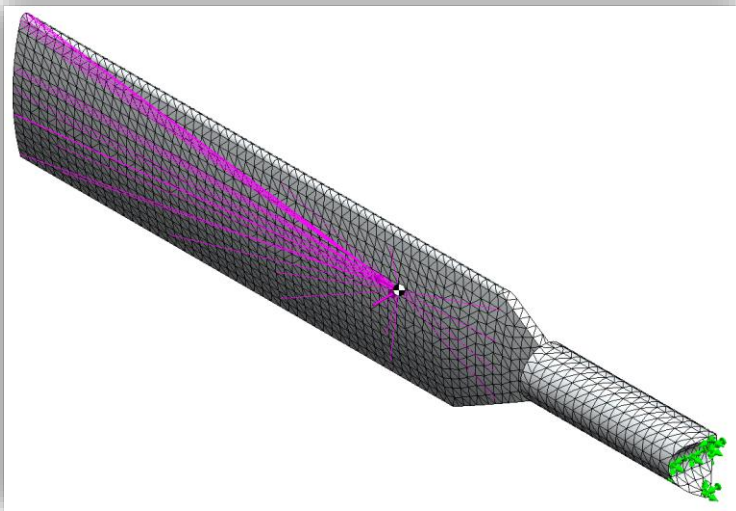


Figure 10 - 17.7mm mesh applied (SolidWorks, Kilbane, 2019)

The finite element mesh applied to the body is very important, it is used to calculate the results of the analysis. SolidWorks will automatically default to its own suggested mesh size, in this instance it is 17.7mm, this may be considered a good starting point for the analysis.

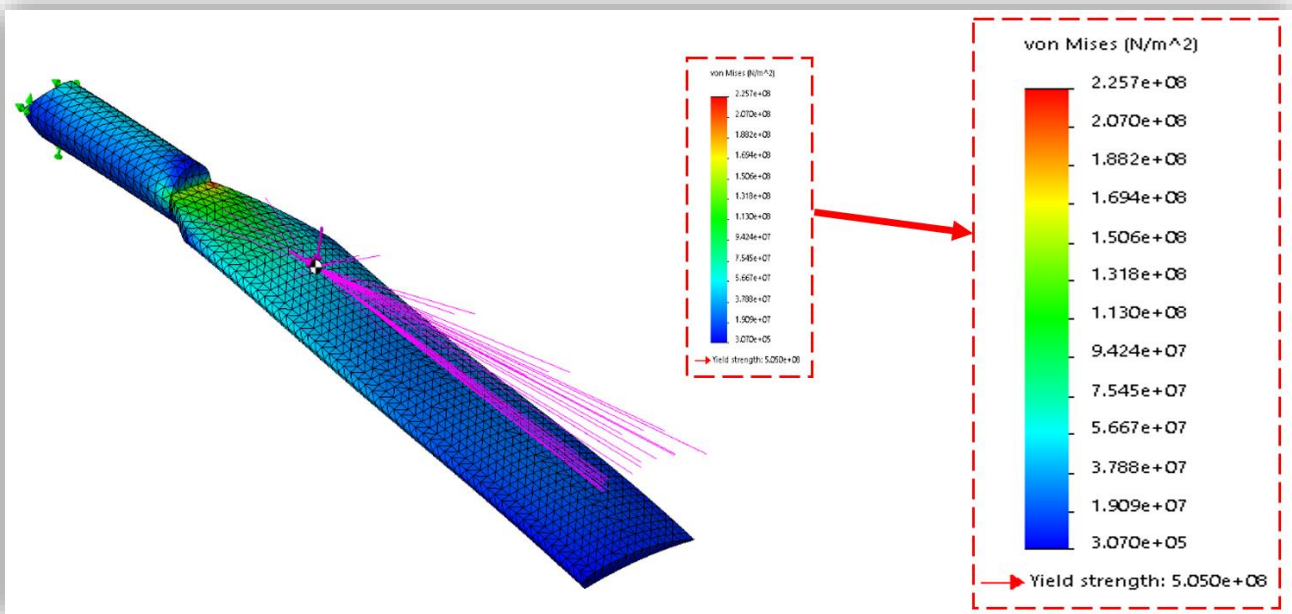


Figure 9 - Results of first analysis (SolidWorks, Kilbane, 2019)

The analysis shows that the material has not exceeded its yield strength, however, it is evident that there is a stress concentration where the blade meets the cylindrical shaft. This stress concentration is not overly concerning at first glance, but it is a good indication that further investigation may be warranted to establish the implications in finer detail.

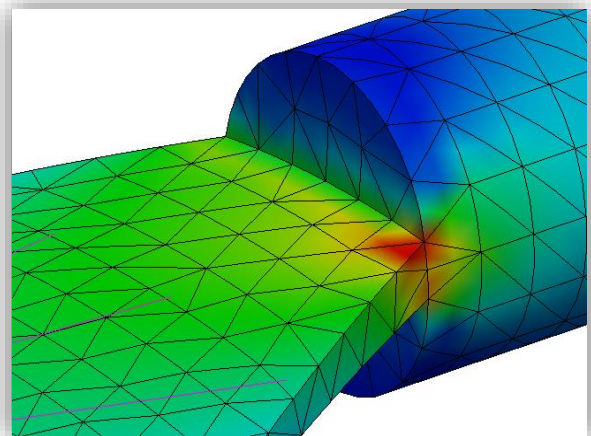


Figure 11 - Stress concentration of first analysis (SolidWorks, Kilbane, 2019)

Mesh refinement is required now, but in order to apply some form of logic to this process, we must first understand the mechanics of what is happening to the material under load. Consider a cross sectional view of a cantilever beam, fixed with a rigid connection at one end and a force is applied at the extremity of the other side in a downward direction.

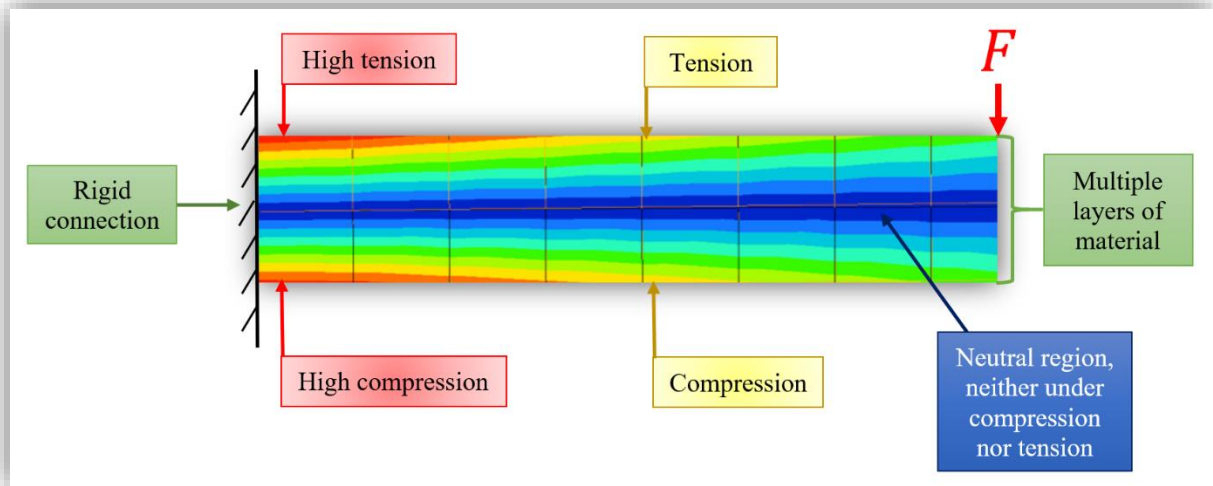


Figure 12 - Cross sectional view of layers under stress (Brancheau, 2015 & Kilbane, 2019)

The layers are coloured in this instance to display the stresses, red being highly stressed, though to blue which is rather neutral or under little stress [13]. In order to apply enough mesh, three layers should be applied through the material thickness, one outer layer will be in compression, the other in tension, with the middle layer being the neutral region.

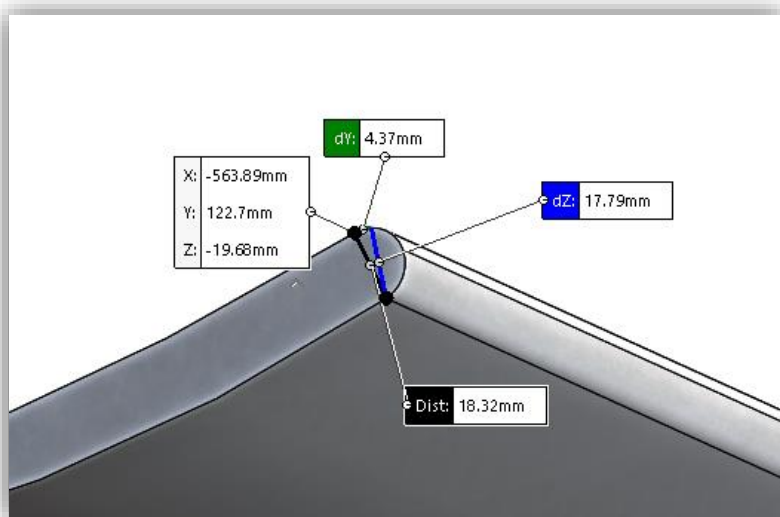


Figure 13 - Material thickness (SolidWorks, Kilbane, 2019)

The cross section of the material is  $\sim 18\text{mm}$ , although this does vary due to the geometry of the material, it is a good point to generate a mesh. Therefore, a mesh is calculated thus;  $18 \div 3 = 6$ , giving us a 6mm mesh, as discussed, one layer in tension, one in compression and one neutral.

The same process is carried out again with one deviation, of course that being the application of a 6mm mesh instead of a 17.7mm mesh, with the expected results to be far more accurate.

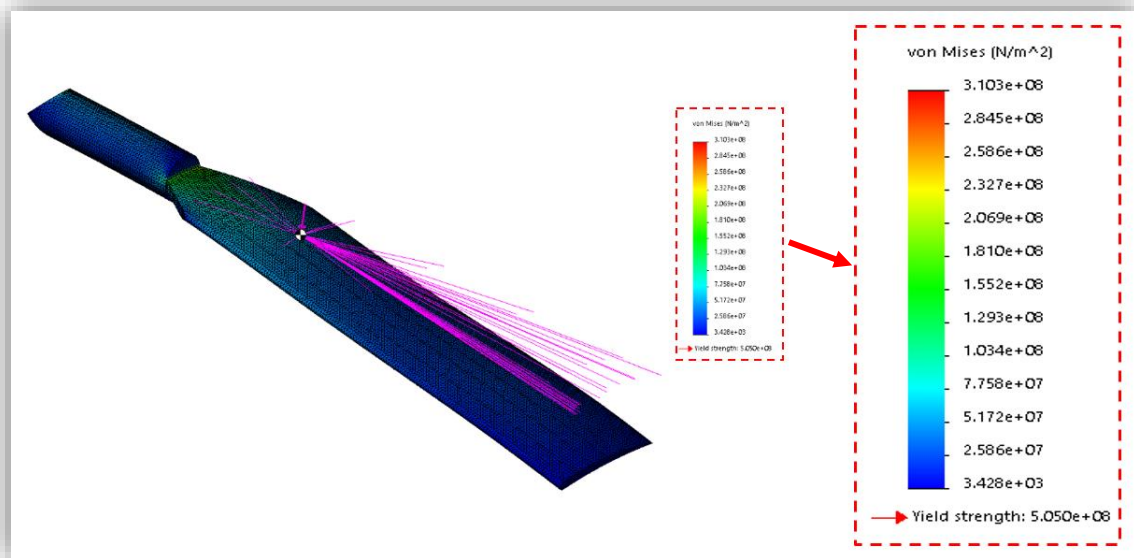


Figure 14 - Results of second analysis (SolidWorks, Kilbane, 2019)

The stress is still below the yield strength of the material, however it has increased by ~37%, this is interesting as it shows a more prominent stress concentration in the concerned region. It is evident that this is concerning, and again further analysis should be carried out to understand specifically what is causing this condition.

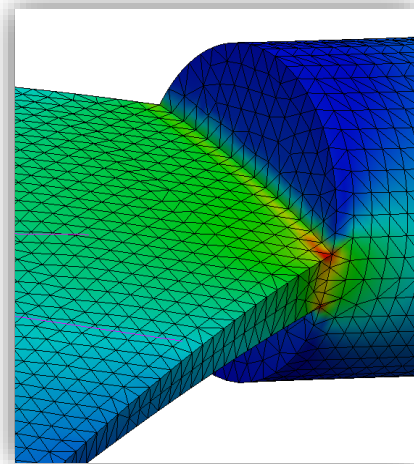


Figure 15 - Stress concentration of second analysis (SolidWorks, Kilbane, 2019)

Another analysis was run using the exact same process, but this time a 3mm mesh is applied. The results returned a stress this time of 415.5MN, this is increasingly approaching the yield strength of the material. Each time an analysis is run, the number of calculations being made increase almost exponentially, meaning the run time will also increase.

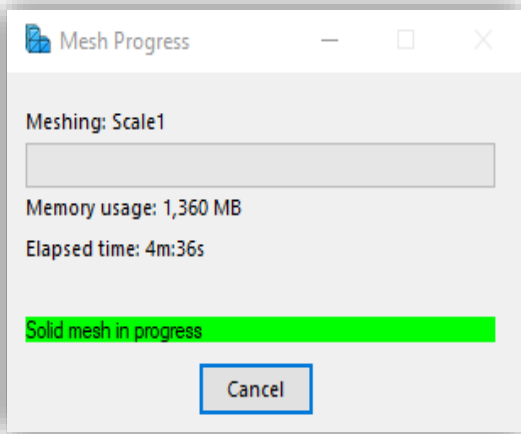


Figure 16 - Run time for creating a 1mm mesh  
(SolidWorks, Kilbane, 2019)

For the final analysis of this kind, a 1mm mesh was applied, SolidWorks took a long time to mesh so it was stopped prior to 5 minutes. Knowing the analysis takes much longer than the mesh does, there was no obvious benefit of running a finer analysis on this component.

There is something that has been neglected thus far, that is to apply a more intelligent mesh, that being a h-adaptive mesh. A refined mesh like this is not uniformly consistent throughout the body, it identifies areas of concern and refines the mesh accordingly to create a finer analysis. The analysis was run again using a 3mm mesh, with a pass of 5, meaning SolidWorks analysis and refines the mesh 5 times.

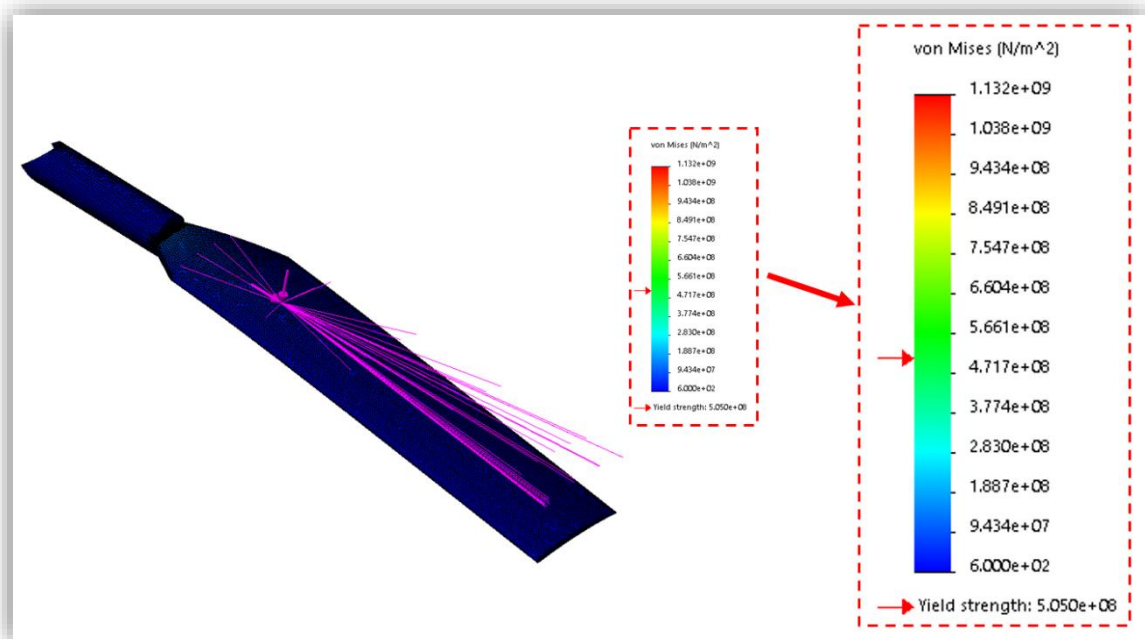


Figure 17 - H-adaptive mesh results (SolidWorks, Kilbane, 2019)

The yield strength of the material has now been exceeded significantly, however this is somewhat in contradiction to the previous results.

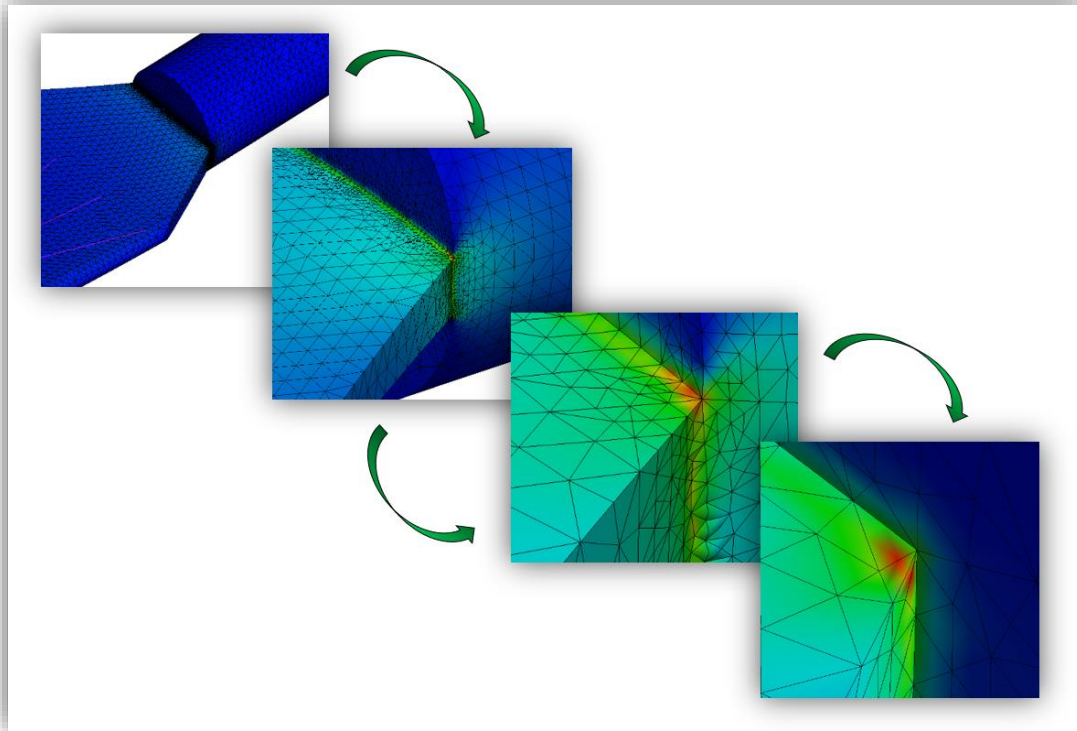


Figure 18 - Stress concentration of h-adaptive mesh results (SolidWorks, Kilbane, 2019)

The image displays the stress concentration in this single location, although it is incredibly small in form of its concentration, it may fracture. The significance of a fracture is that due to excessive movement such as vibration over time could cause the crack to propagate [10], leading to catastrophic failure.

This convergence graph shows how each loop was refined, it returned more accurate results each time. The final figure exceeding the material's yield strength by ~125%.

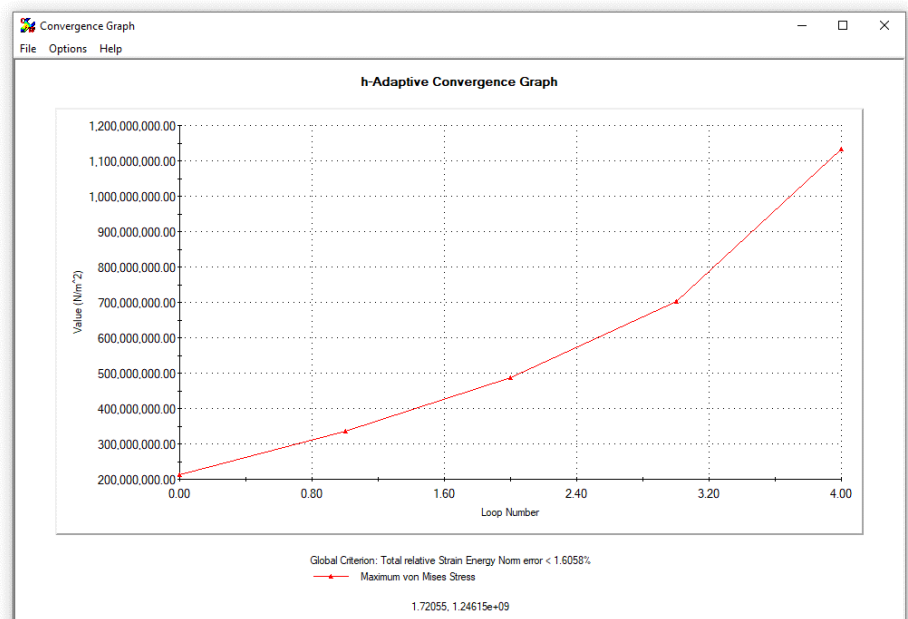


Figure 19- H-adaptive convergence graph (SolidWorks, Kilbane, 2019)

The stress has been analysed although another great consideration is that of deflection, it is important to know what form of deflection will be created in operation. However, this item is meant to generate thrust, it does not have a solid object applied to it, but air, and therefore is not *that* important. However, the maximum deflection viewed was <9.5mm, which considering the length of the item and its functionality, this is most likely acceptable, however further investigation would be required to determine for sure.

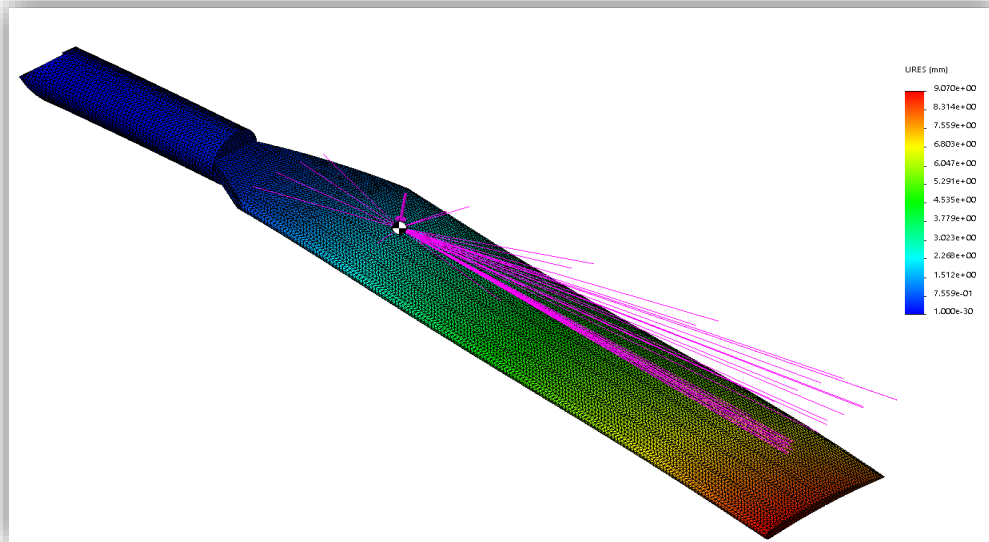


Figure 20 - Deflection results (SolidWorks, Kilbane, 2019)

All the results are displayed in the table, for comparison and to understand the significance of the mesh detail.

| Analysis | Mesh size             | $\sigma$ Max. Stress | Max. deflection | Solution time | Nodes qty. | Elements qty. | DOF qty. |
|----------|-----------------------|----------------------|-----------------|---------------|------------|---------------|----------|
| 1        | 17.7mm                | 225.7 MN             | 8.87mm          | 12 sec.       | 15818      | 8688          | 47013    |
| 2        | 6mm                   | 310.3MN              | 9.07mm          | 92 sec.       | 244082     | 157074        | 729060   |
| 2        | 3mm                   | 415.5MN              | 9.07mm          | 145 sec.      | 1629926    | 1121293       | 4878111  |
| 4        | 3mm h-adaptive 5-loop | 670.3MN              | 9.494mm         | 366 sec.      | 303722     | 197910        | 907185   |

Figure 21 - Table of results (Kilbane, 2019)

The table shows that the original default mesh of 17.7mm returned a run time of only 12 seconds, with the greatest being that of the h-adaptive 5-loop analysis. The number of nodes and elements determine the amount of calculations required, with DOF (Degrees of Freedom) being another factor.

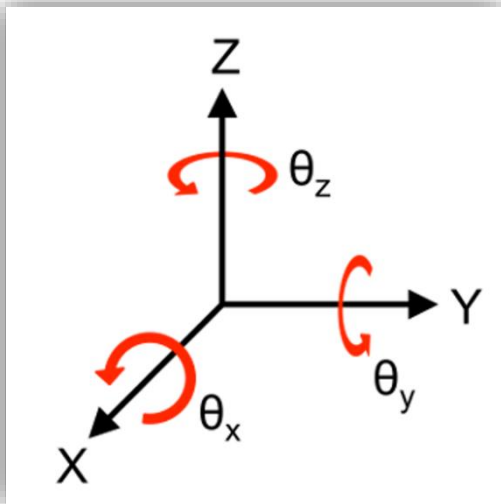


Figure 22 - Degrees of freedom diagram (mit.edu, 2017)

In reality, each object may have 6 degrees of freedom, this simply means that each object may translate in each X Y Z axis and rotate about each of those axis [13]. The significance of this in this instance is that each node may be applicable to these six degrees of freedom, with the constraint being that of its relating nodes, therefore number of DoF  $\neq$  (number of nodes  $\times$  6) as one may expect.

Another consideration is that of frequency, each object vibrates at a given frequency, and this frequency should be avoided as to prevent vibration and in some instances noise. The conversion rate of RPM to Hertz is  $1 \text{ RPM} = \frac{1}{60} \text{ Hertz}$  [14], therefore the simple equation for converting rpm to Hertz is:

$$Hz = RPM \times 0.016$$

$$\therefore Hz = 1411 \times 0.016$$

$$Hz = 23.516$$

Knowing which frequency the rotor blade now operates at, it is possible to run an analysis to understand the resonant natural frequencies. The reason for analysing the natural resonant frequencies of a body is that we can compare this to the frequencies created by the operation of the body [15]. SolidWorks defaults to five frequency modes, but in order to run a more comprehensive analysis it has been set to ten modes.

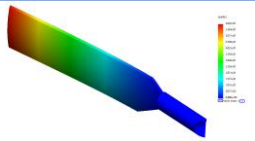
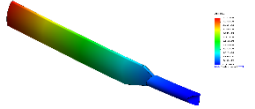
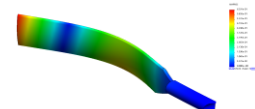
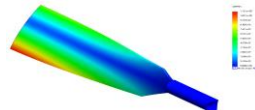
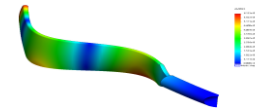
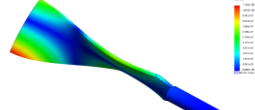
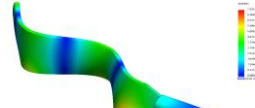
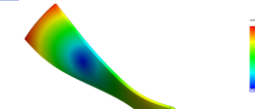
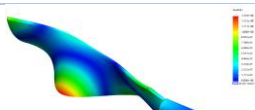
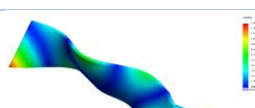
| Image                                                                               | Frequency (Hertz) | Mode Number |
|-------------------------------------------------------------------------------------|-------------------|-------------|
|    | 12.215            | 1           |
|    | 49.427            | 2           |
|    | 80.612            | 3           |
|    | 114.17            | 4           |
|    | 221.12            | 5           |
|  | 343.12            | 6           |
|  | 406.54            | 7           |
|  | 429.57            | 8           |
|  | 572.24            | 9           |
|  | 616.12            | 10          |

Figure 23 - Frequency results table (Kilbane, 2019)

Reviewing the results, we can disregard all but the first two results as the natural frequencies are so high, they surpass the operating frequency of the blade rather significantly. Furthermore, we don't really need to be concerned with the second one in reality as the operating condition will not reach that level.

There is one other obvious analysis (Fatigue) that has been intentionally omitted from this report, and that is due to constraints in the form of a lack of material mechanical properties within the SolidWorks database itself. There are links between frequency, force and fatigue, this is because fatigue is directly impacted by both and therefore the frequency and force will be a determining factor in the life expectancy of the blade.

$$\text{Goodman} \quad \sigma_a = \sigma_n \left[ 1 - \left( \frac{\sigma_m}{\sigma_{TS}} \right) \right] \quad [15]$$

$$\text{Gebber} \quad \sigma_a = \sigma_n \left[ 1 - \left( \frac{\sigma_m}{\sigma_{TS}} \right)^2 \right] \quad [15]$$

$$\text{Soderberg} \quad \sigma_a = \sigma_n \left[ 1 - \left( \frac{\sigma_m}{\sigma_y} \right) \right] \quad [15]$$

Although understood to some degree, fatigue is somewhat of a phenomenon in the sense it is no exact science. There are three well known formulas to calculate fatigue, that of Goodman, Gebber and Soderberg.

$\sigma_n$  = Fatigue strength for  $N$  cycles under zero mean stress conditions.

$\sigma_a$  = Fatigue strength for  $N$  cycles under condition mean stress  $\sigma_m$ .

$\sigma_{TS}$  = Tensile strength of the material.

$\sigma_y$  = Yield strength of the material.

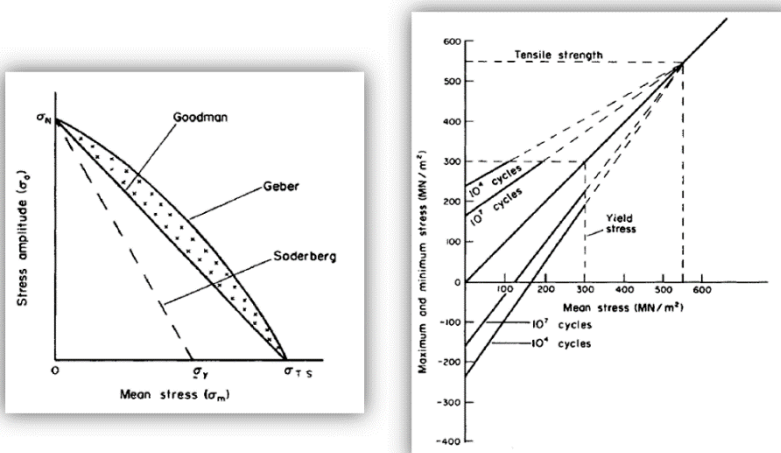


Figure 24 - Fatigue graphs (Hearn, 1999)

The image represents displays the three forms of calculations and their expected results. In reality some evidence suggests that Goodman and Gebber are most accurate, although many engineers using Soderberg as a safety factor.

## 7. Analysis of results

The only real concern for this product is the stress concentration in the very corner of where the blade meets the shaft, although it is highly concentrated in a very small area, an argument may be for this to be of little concern due to the incredible locality of the stress concentration. A counter argument to that would be as it is such a small localised issue, it can simply be resolved by adding a fillet to the region, thereby eliminating the concentration.

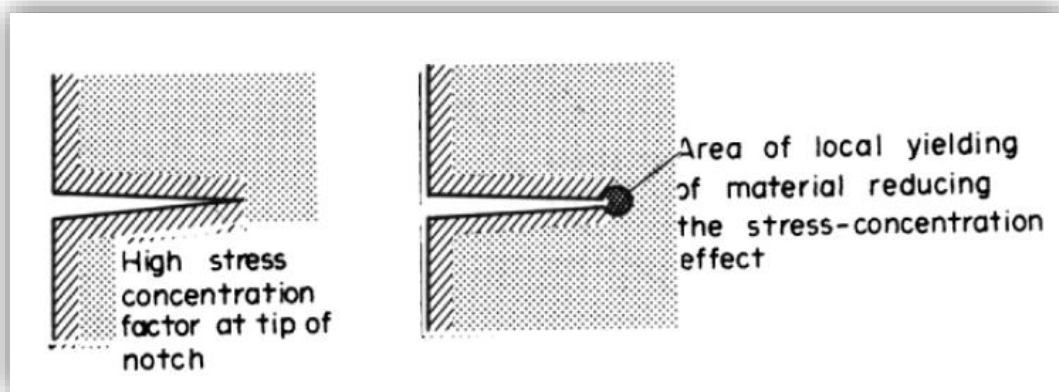


Figure 25 - Notch corner relief (Hearn, 2000)

Figure 25 shows how a relief may be added to the sharp internal corner of a notch to reduce the stress concentration [10], similarly, this helicopter rotor may be improved by adding a relief in the form of a fillet as discussed. The model has been modified now and the analysis run

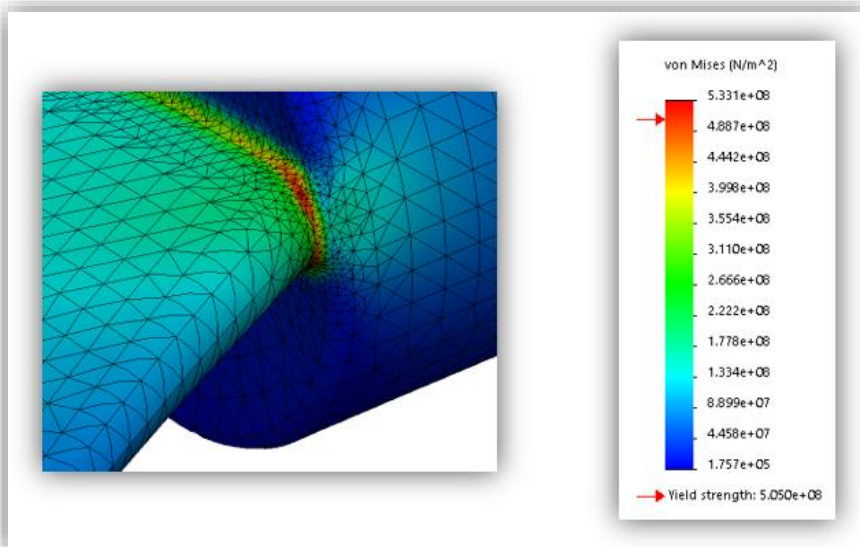


Figure 26 - Modified model to include fillets (SolidWorks, Kilbane, 2019)

again, this time the part narrowly failed the stress analysis with a small 2mm fillet, a clear correlation is evident that a small fillet may reduce the stress [10], therefore a larger fillet would satisfy the requirement.

---

## 8. Conclusion

In conclusion, the models used for analysis are not a true representation of the actual finished product, the surfaces do not align perfectly, in reality, we would not have sharp corners meeting as the original model does, it is simply poor design. If this was a simple general outline model, and it were to be analysed simply to highlight the problematic areas that are effected by stress, it is good to identify these areas prior to detailed design work being done.

FEA is an excellent tool for calculating stresses on a body, the repetitive calculations a computer can carry out would be incredibly time consuming for a human, therefore it is simply a tool, the person inputting the data and reading the results must be experienced enough to make an informed and educated decision themselves as to what is good or bad.

---

## 9. Works Cited

### Reference List

- [1] B. R. Swopes, “AH-64 Apache,” This day in aviation, 1 October 2019. [Online]. Available: <https://www.thisdayinaviation.com/tag/ah-64-apache/>. [Accessed 10 November 2019].
- [2] C. Rotaru and M. Todorov, Helicopter Flight Physics, Sofia: intechopen, 2017.
- [3] T. M. Fletcher and R. E. Brown, “Helicopter Tail Rotor Thrust and Main Rotor Wake Coupling in Crosswind Flight,” *Journal of Aircraft*, vol. 47, no. 6, pp. 2136-2148, 2010.
- [4] J. Seddon and S. Newman, Basic Helicopter Aerodynamics (Aerospace Series, Chichester: Wiley, 2011.
- [5] M. P. Norton and D. G. Zarczub, Fundamentals of noise and vibration analysis for engineers, Cambridge: Cambridge University Press, 2003.
- [6] R. Ganguli, D. Thakkar and S. R. Viswamurthy, Smart Helicopter Rotors, New York: Springer, 2016.
- [7] B. Lindenbaum, “Understanding the rotary wing,” Redback Aviation, March 2017. [Online]. Available: <http://www.redbackaviation.com/understanding-rotor-blades-the-rotary-wing/>. [Accessed 5 November 2019].
- [8] Smiths, “7075 Aluminium Alloy Data Sheet,” Smiths Metal, 2018. [Online]. Available: <https://www.smithmetal.com/pdf/aluminium/7xxx/7075.pdf>. [Accessed 5 November 2019].
- [9] SolidWorks, *SolidWorks Materials Library*, Vélizy-Villacoublay: Dessault Systèmes, 2019.
- [10] E. J. Hearn, Mechanics of Materials 1, Warwick: Elsevier, 2000.
- [11] GE Aviation, “The T700 Engine,” GE Aviation, 2019. [Online]. Available: <https://www.geaviation.com/military/engines/t700-engine>. [Accessed 7 November 2019].
- [12] J. Pike, “AH-64 Apache,” fas.org, 29 May 2000. [Online]. Available: <https://fas.org/man/dod-101/sys/ac/ah-64.htm>. [Accessed 12 November 2019].
- [13] J. E. Brancheau, Practical Aspects of Finite Element Simulation, Troy: Altair University, 2015.
- [14] R. L. Timings, Newnes Mechanical Engineer’s Pocket Book, Oxford: Elsevier, 2006.
- [15] E. J. Hearn, Mechanics of Materials 2, Oxford: Elsevier, 1999.

---

## Table of Figures

|                                                                                                 |    |
|-------------------------------------------------------------------------------------------------|----|
| Figure 1 - Helicopter model isometric view (SolidWorks, Kilbane, 2019).....                     | 5  |
| Figure 2 - Helicopter model plan view (SolidWorks, Kilbane, 2019).....                          | 6  |
| Figure 3 - Apache dimensions (Inetres.com, 1999).....                                           | 6  |
| Figure 4 - Helicopter model side view (SolidWorks, Kilbane 2019) .....                          | 7  |
| Figure 5 - Rotor blade models (SolidWorks, Kilbane, 2019).....                                  | 7  |
| Figure 6 - CofG of modified rotor blade (SolidWorks, Kilbane, 2019) .....                       | 11 |
| Figure 7 - Fixing face of modified rotor blade (SolidWorks, Kilbane, 2019) .....                | 11 |
| Figure 8 - Applying forces to modified rotor blade (SolidWorks, Kilbane, 2019) .....            | 12 |
| Figure 9 - Results of first analysis (SolidWorks, Kilbane, 2019).....                           | 13 |
| Figure 10 - 17.7mm mesh applied (SolidWorks, Kilbane, 2019).....                                | 13 |
| Figure 11 - Stress concentration of first analysis (SolidWorks, Kilbane, 2019) .....            | 13 |
| Figure 12 - Cross sectional view of layers under stress (Brancheau, 2015 & Kilbane, 2019) ..... | 14 |
| Figure 13 - Material thickness (SolidWorks, Kilbane, 2019).....                                 | 14 |
| Figure 14 - Results of second analysis (SolidWorks, Kilbane, 2019) .....                        | 15 |
| Figure 15 - Stress concentration of second analysis (SolidWorks, Kilbane, 2019).....            | 15 |
| Figure 16 - Run time for creating a 1mm mesh (SolidWorks, Kilbane, 2019).....                   | 16 |
| Figure 17 - H-adaptive mesh results (SolidWorks, Kilbane, 2019).....                            | 16 |
| Figure 18 - Stress concentration of h-adaptive mesh results (SolidWorks, Kilbane, 2019)....     | 17 |
| Figure 19- H-adaptive convergence graph (SolidWorks, Kilbane, 2019) .....                       | 17 |
| Figure 20 - Deflection results (SolidWorks, Kilbane, 2019) .....                                | 18 |
| Figure 21 - Table of results (Kilbane, 2019).....                                               | 18 |
| Figure 22 - Degrees of freedom diagram (mit.edu, 2017) .....                                    | 19 |
| Figure 23 - Frequency results table (Kilbane, 2019) .....                                       | 20 |
| Figure 24 - Fatigue graphs (Hearn, 1999).....                                                   | 21 |
| Figure 25 - Notch corner relief (Hearn, 2000).....                                              | 22 |
| Figure 26 - Modified model to include fillets (SolidWorks, Kilbane, 2019) .....                 | 22 |