2.6.3 / Finite Element Analysis

To allow for bending calculations to be carried out, the blade was simplified to a cantilever beam. This assumption does not accurately reflect the effect of the aerofoil shape, the different chord lengths and twist angles, or the internal webbed structure of the blade. Thus, FEA was implemented to generate more accurate reflections on the stresses and displacements of the blade that considers the shape and the internal features. This was carried out using Autodesk Fusion 360 [h].

Table 32: Justification of modelling simplifications.

Reason for simplifications		
The ideal model that would accurately reflect the blade, would be a model of the full 60 m length blade with the internal features included as shown in Figure 48. However, it was found that this model would not mesh or solve in the FEA software. Thus, model simplifications were required.		
Simplification	Justification	Limitation
The internal features were removed and the blade was modelled as a solid body as shown in Figure 49.	Enable us to obtain results that reflect the stresses and displacements present in the full 60 m length blade.	There will be differences in the stress distributions and stress concentrations predicted by a model of a solid blade, compared to a model of a blade with internal features. This can affect our results.
The blade model was sliced into 2 parts as shown in Figures 32 and 33. These parts included the internal webbed structures and were able to mesh and solve on Fusion 360. A simplified solid model of these 2 parts were created as shown in Figures 34 and 35. The distribution and the maximum values of stress and deflection were compared between the solid version and internal structure version of the blades.	This was used to analyse the differences between the results of the solid models vs the models with internal features. These differences can then be then taken into consideration when analysing the results from the solid model of the full length of the blade.	-

Tables 33 and 35 outline the model simplification, input boundary conditions and meshing conditions. Tables 34a, 34b, and 35 outline the results for the spliced component models and the full 60 m blade model respectively.

Table 33: *Model simplifications and boundary conditions used for FEA analysis*

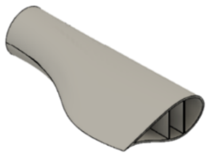
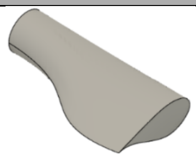
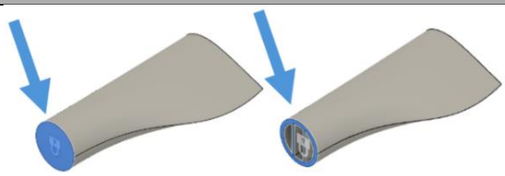
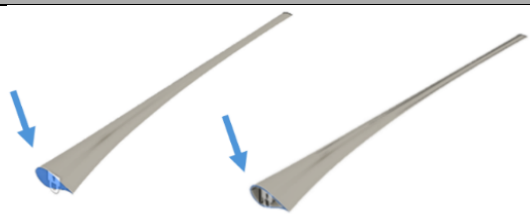
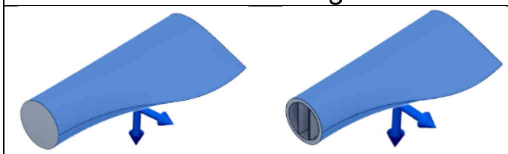
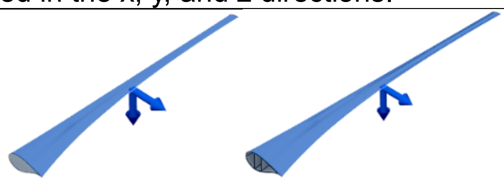
Model	
Model: Static stress analysis	Justification: Suitable for analysing the deformation and stress in the model.
 Figure 32: Variation 1 - simplified CAD model of a portion of the wind turbine blade with the internal webs.	 Figure 33: Variation 2 - simplified CAD model of a portion of the wind turbine blade with the internal webs.
Model Simplification	
 Figure 34: Variation 3 - simplified CAD model of a portion of the wind turbine blade modelled as a solid component.	 Figure 35: Variation 4 - simplified CAD model of a portion of the wind turbine blade modelled as a solid component.
Mesh Settings	
Settings: • Automeshing. • Tetrahedral solid elements. • Element size 5% - 10% of model size. • Curved mesh elements	Justification: • Literature suggests automeshing is suitable for complex models [18]. • Used for meshing volume structures (3D solid CAD models) [18]. • Reccomended by autodesk for solid elements [19]. • Reccomended by autodesk for accurate representation. [19]
Boundary Conditions	
 Figure 36: arrow indicating the face used to apply a fixed boundary condition to the two blade models (variations 1 & 3).	 Figure 37: arrow indicating the face used to apply a fixed boundary condition to the two blade models (variations 2 & 4).
The faces selected in Figures 36 and 37 were fixed in the x, y, and z directions.	
 Figure 38: uniformly distributed loads applied to the two blade models (variations 1 & 3).	 Figure 39: uniformly distributed loads applied to the two blade models (variations 2 & 4).
Using values from the calculations stage, the resultant flapwise UDL of 9.77 kN/m, and the edgewise UDL of 16.22 kN/m, were applied to the model as uniformly distributed loads in the vertical and horizontal directions respectively, as shown in Figures 38 and 39.	

Table 34a: Results 1 - stress analysis of solid and internally structured blades

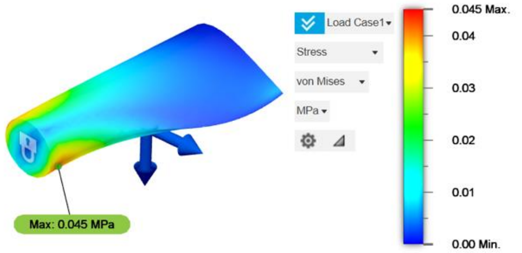
<table>
<tr><th colspan="2">Results 1</th></tr>
<tr><th colspan="2">Effect of simplification on stress</th></tr>
<tr><td>

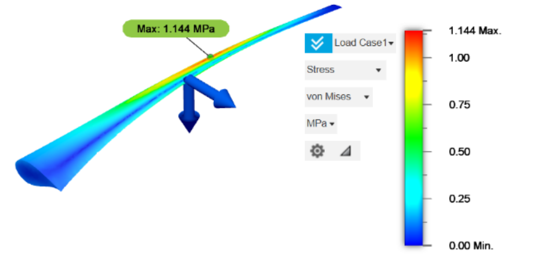
Figure 40: Autodesk Fusion360 FEA stress results for variation 3, a solid model of a section of the blade.</td><td>

Figure 41: Autodesk Fusion360 FEA stress results for variation 4, a solid model of a section of the blade.</td></tr>
<tr><td>

Figure 42: Autodesk Fusion360 FEA stress results for variation 1, a model of a section of the blade including the internal features.</td><td>

Figure 43: Autodesk Fusion360 FEA stress results for variation 2, a model of a section of the blade including the internal features.</td></tr>
<tr><td>The results from Figures 40 and 42 show that the maximum stress in variation 1 (0.124 MPa), is approximately 2.75 times greater than in variation 3 (0.045 MPa). This indicated that the solid model underestimates the stress in the blades.</td><td>The results from Figures 41 and 43 show that the maximum stress in variation 2 (2.098 MPa), is approximately 1.8 times greater than in variation 3 (1.144 MPa). This indicated that the solid model underestimates the stress in the blades.</td></tr>
<tr><td colspan="2">The results from both models show that the general location of maximum stress is the same (root, centre, or tip), but the specific locations differ slightly. This is likely because the model with the internal features will distribute the stress differently.</td></tr>
<tr><td colspan="2">As both model results show that the solid model gives an underestimation, when analysing the stress on the full 60 m blade model, this underestimation should be taken into account. To do so, the upper value of 2.75 will be used. The maximum stress predicted by the solid 60 m blade model should be multiplied by 2.75 to estimate the likely stress if the model were to have internal features. This new value should then be used to inform the design and material selection of the blade.</td></tr>
<tr><td colspan="2">Figures 42 and 43 also show that the maximum stress on the blades, occurs on the outer structure rather than on the internal web structure. Based on the colour gradient, the internal web structure tends to experience a lower amount of stress. This indicates that an internal web structure is suitable to uses to decrease the weight of the blade as long as the outer surface is reinforced to account for the higher stresses.</td></tr>
</table>

Table 34b: Results 2 - flapwise displacement analysis of solid and internally structured blades

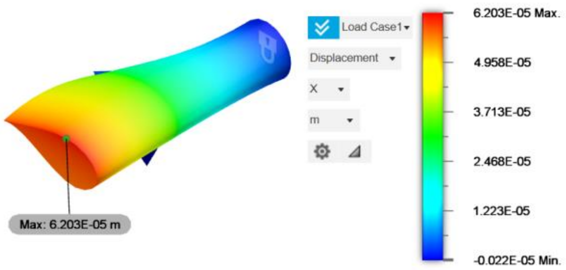
<table>
<tr><th colspan="2">Results 2
Effect of simplification on flapwise displacement</th></tr>
<tr><td>


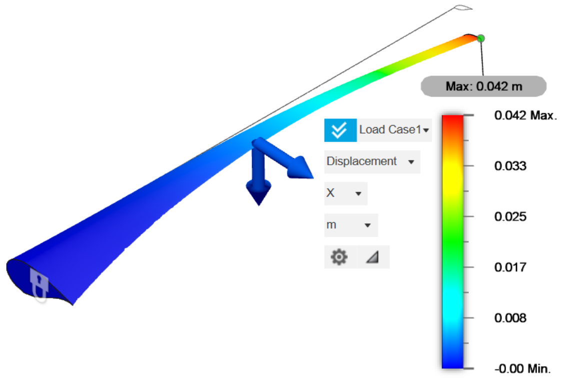
Figure 44: Autodesk Fusion360 FEA displacement results for variation 3, a solid model of a section of the blade.
</td><td>


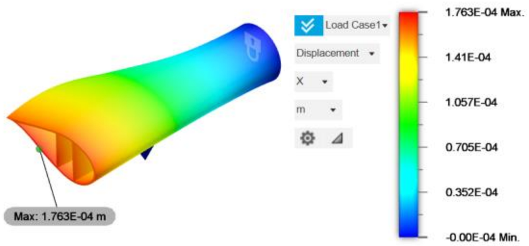
Figure 45: Autodesk Fusion360 FEA displacement results for variation 4, a solid model of a section of the blade.
</td></tr>
<tr><td>


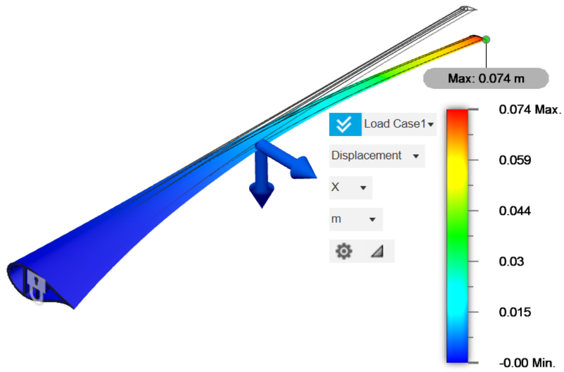
Figure 46: Autodesk Fusion360 FEA displacement results for variation 1, a model of a section of the blade including the internal features.
</td><td>


Figure 47: Autodesk Fusion360 FEA displacement results for variation 2, a model of a section of the blade including the internal features.
</td></tr>
<tr><td>The results from Figures 44 and 46 show that the maximum flapwise displacement in variation 1 (17.63 x 10^{-5} m) is approximately 2.8 times larger than in version 3 (6.203 x 10^{-5} m). This indicates that the solid model underestimates the maximum flapwise displacement of the blades.</td><td>The results from Figures 45 and 47 show that the maximum flapwise displacement in variation 2 (7.4 x 10^{-2} m) is approximately 1.8 times larger than in version 4 (4.2 x 10^{-2} m). This indicates that the solid model underestimates the maximum flapwise displacement of the blades.</td></tr>
<tr><td colspan="2">The results from both models show that the general location of maximum displacement is the same (root, centre, or tip), but the specific locations differ. This is likely because the model with the internal features will distribute the displacement differently.</td></tr>
<tr><td colspan="2">As both model results show that the solid model gives an underestimation, when analysing the displacement on the full 60 m blade model, this underestimation should be taken into account. To do so, the upper value of 2.8 will be used. The maximum displacement predicted by the solid 60 m blade model should be multiplied by 2.8 to estimate the likely displacement if the model were to have internal features. This new value should then be used to inform the design and material selection of the blade.</td></tr>
</table>

Table 35: Model simplifications and boundary conditions used for FEA analysis of the full length blade

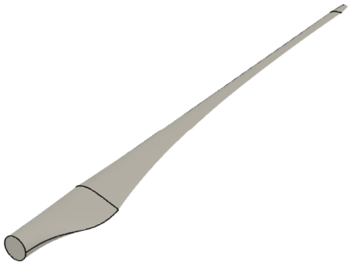
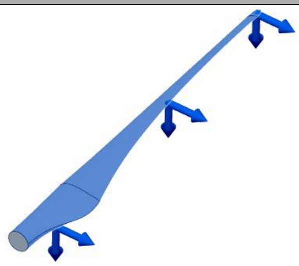
Model (Full length blade)	
Model: Static stress analysis	Justification: Suitable for analysing the deformation and stress in the model.
Original Blade Model	
 Figure 48: CAD model of the full length of the wind turbine blade.	
Model Simplification	
 Figure 49: Simplified solid CAD model of the full length of the wind turbine blade with the internal webs removed.	
Mesh Settings	
Settings: • Automeshing. • Tetrahedral solid elements. • Element size 5% - 10% of model size. • Curved mesh elements	Justification: • Literature suggests automeshing is suitable for complex models [18]. • Used for meshing volume structures (3D solid CAD models) [18]. • Reccomended by autodesk for solid elements [19]. • Reccomended by autodesk for accurate representation [19].
Boundary Conditions	
 Figure 50: Fixed constrained applied to one face of the CAD model. Arrow indicating the face.	 Figure 51: Uniformly distributed load (UDL) applied to the blade in the flapwise direction (vertical) and edgewise direction (horizontal).
The face that connects to the blade hub was fixed in all directions as shown in Figure 50.	Using the values from the calculations stage, the resultant flapwise UDL (9.77 kN/m) and the edgewise UDL (16.22 kN/m) were applied as uniformly distributed loads, as shown in Figure 51.

Table 36: Results 3 - stress and flapwise displacement analysis on a solid model of 60 m length blade

<table>
<tr><th>Results 3</th></tr>
<tr><th>Full Length Blade</th></tr>
<tr><th>Stress</th></tr>
<tr><td>

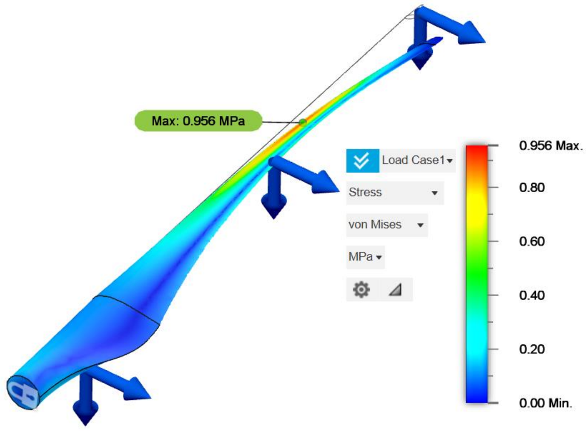


Figure 52: Autodesk Fusion360 FEA stress results for a solid model the full length of the blade.
</td></tr>
<tr><td>The maximum stress predicted by the model is 0.956 MPa, as shown by Figure 52. To take into account that the solid model may not reflect the maximum stress in a model with internal structures, this value will be multiplied by 2.75 (taken from Results 1). Thus, the maximum predicted stress is 2.629 MPa.</td></tr>
<tr><td>The yield strength of the balsa wood core is 18 MPa, and the yield strength of the epoxy E-glass outer skin is 700 MPa. Following the 2/3 yield criterion, the maximum allowable yield stress is 12 MPa for the balsa wood core, and 466.67 MPa for the epoxy E-glass outer skin. The maximum predicted stress by the model is below these two values thus validating that the design is fit for its purpose.</td></tr>
<tr><th>Displacement</th></tr>
<tr><td>

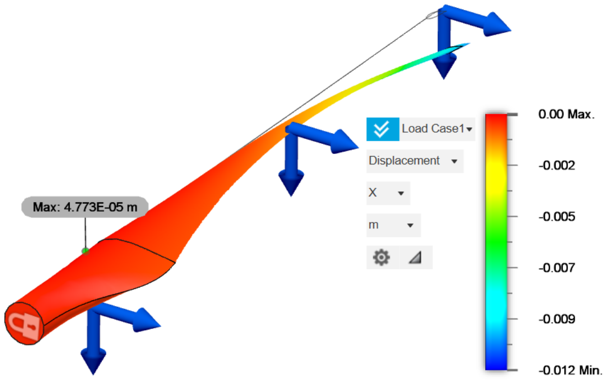


Figure 53: Autodesk Fusion360 FEA displacement results for a solid model the full length of the blade.
</td></tr>
<tr><td>The maximum displacement predicted by the model is 4.773 x 10^{-5} m, as shown by Figure 53. To take into account that the solid model may not reflect the maximum deflection in a model with internal structures, this value will be multiplied by 2.8 (taken from Results 2). Thus, the maximum predicted flapwise displacement in the blades is 9.546 x 10^{-5} m.</td></tr>
<tr><td>Literature suggests that the flapwise deflection not exceed 4.5 m (30% of the clearance between the blade tip and the turbine tower) [20]. The maximum deflection predicted by the model is significantly lower than this. Thus, validating that the blade deflection is in an acceptable range and the blade design is fit for it’s purpose.</td></tr>
</table>