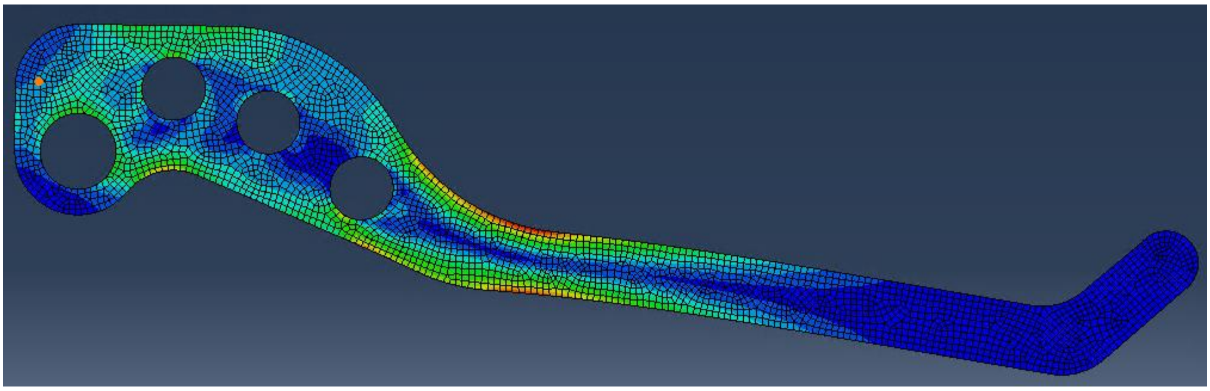
EMS602U

Solid mechanics and finite elements

Report on FEM lab

Student ID: 210478226

Name: Shannen Venethethan



Abstract:

This study explores the use finite element analysis (FEA) to design a lightweight motorbike brake lever while maintaining high structural integrity. Reduction in weight correlated to increased actuation performance from lower inertia and lower carbon footprint from lower fuel consumption. A 2-dimensional planar stress model was analysed in Abaqus [7] with 6061 T-6 aluminium being used as the material. Linear and Quadratic elements were tested, and quadratic elements are more efficient. Local meshing was also implemented. The maximum Von Mises stress of 154.7 MPa simulated is well below the yield strength of the material selected at 260 MPa giving a safety factor of 1.68 and a maximum deformation of 2.524mm. This study shows that significant weight reduction can be made without compromising on safety. Areas of improvement would be to run the study in 3 dimension and alternative materials.

Introduction:

This study will explore how using finite element analysis (FEA) using the finite element method (FEM) can be used to make a lighter weight motorcycle brake lever [1]. This was chosen as an area of focus as weight saving can mean faster actuation as there is less inertia which can improve the safety and performance of the braking system. Making a component lighter weight has significant fuel saving implication which is useful in motorsport where every gram counts and environmental conservation by lowering fuel consumption consequently lowering carbon emissions.

Finite element analysis (FEA) is a substantial tool when designing such a part, as simulation of the object can be made, and the design can be improved such that it can withstand the forces that are going to be applied to it in the real world without the cost of prototyping on the final material which can be significantly more expensive. In this study a 2-dimensional planar stress model will be analysed.

The Initial design of the handle is shown below:

Fig.1 example of motorbike apparatus [2]

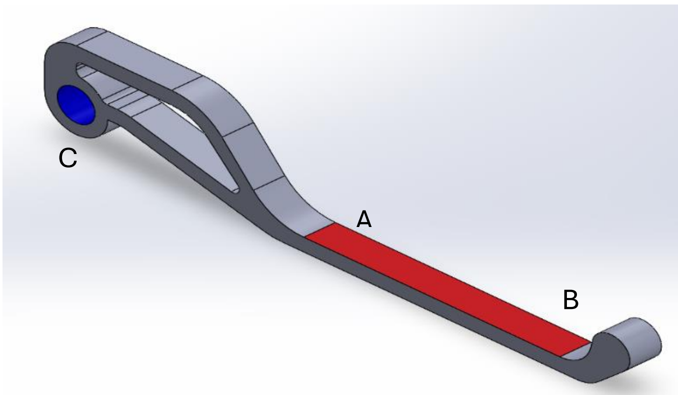


Fig.2 Initial design modelled in SolidWorks [3]

From [4] it can be found that the maximum force that can be exerted by a human hand on average is around 600N which in an emergency under the fight or flight [5] response might increase so a safety factor of more than 1.5 will be achieved by the end of this study. This load will be applied as pressure on the part from point A to B. Point C is the pivot therefore it will be a fixed support. For this study simulation will be ran using 6061 aluminium below are the relevant specifications for the material:

Material	Youngs Modulus (MPa)	Poisson ratio	Yield strength (Elastic) (MPa)
6061-T6 Aluminium	69000	0.33	260

Table 1. Material properties from Granta EduPack [6].

The simulated/tested geometry of the part is shown below in 2 steps:

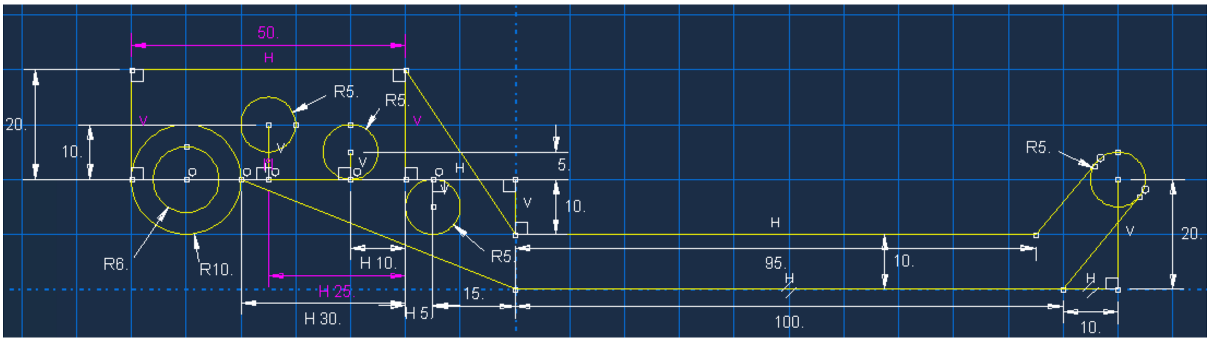


Fig. 3 Part sketch step 1 in Abaqus CAE

The second steps involve using the auto trim tool to remove the unnecessary lines and adding filets where shown:

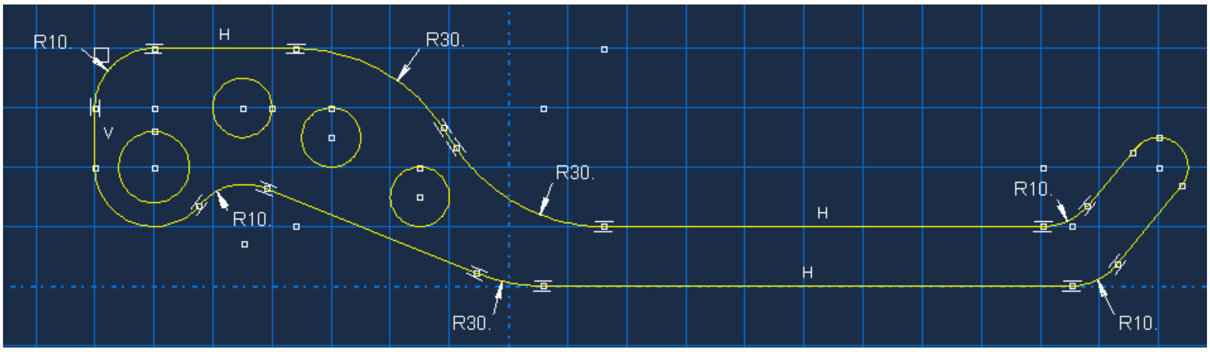


Fig. 4 Part sketch step 2 in Abaqus CAE

Results:

The problem presented will be analysed using Abaqus [7] as the FEA software. The part will first be sketched using the inbuilt tools in the parts section of the software. The properties of the two materials we will be testing then added using the create material section. After which a section will be created and then the section will be assigned to the part; this is where the thickness of the model will be input in this case 10mm. An instance is then created in the assembly section of the software. A step is also then created (static, general) as although all part created come with an initial step it cannot be used for analysis. The load and constrains are then added to the part. As discussed previously and shown in Fig.2 A will have a Displacement/Rotation Boundary condition and will be locked in the U1 and U2 directions, the third dimension is not considered as there is no force that would cause it to be displaced in that direction. A pressure load will then be added to between B and C this will be equivalent to 600N over the 10mm x 74.3mm area therefore 0.808 MPa.

The mesh section is where the changes will be done during this study. As a trial run however, the preset settings for element type and seed global size are kept. The part is then meshed. A new job is then created and run. The result is then visualized. An example from one of the tests is shown below

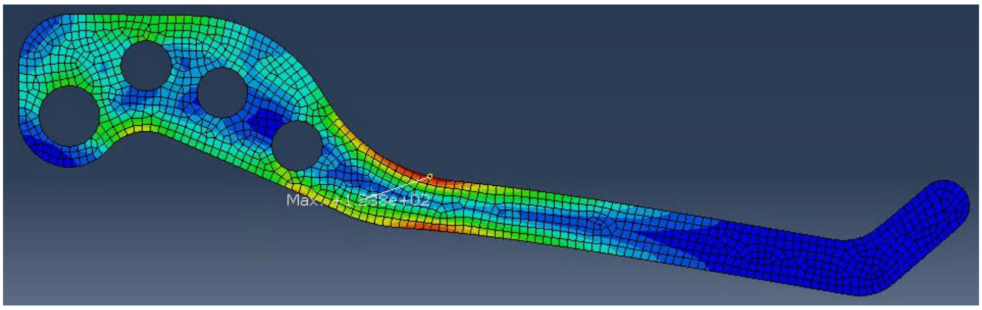


Fig. 5 Visualization of Von Mises stress for linear and global seed size of 1.6

Linear vs Quadratic:

The part was run in linear and quadratic configurations at different mesh sizes starting with 3.2 down to 0.2 the stress was then recorded and plot. Presented below are the plots.

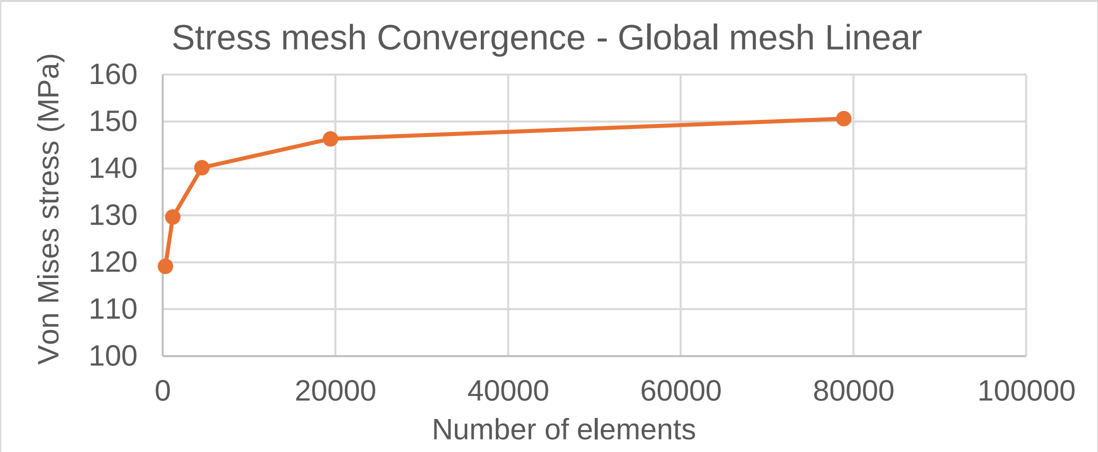


Fig. 6 Stress mesh Convergence - Global mesh Linear

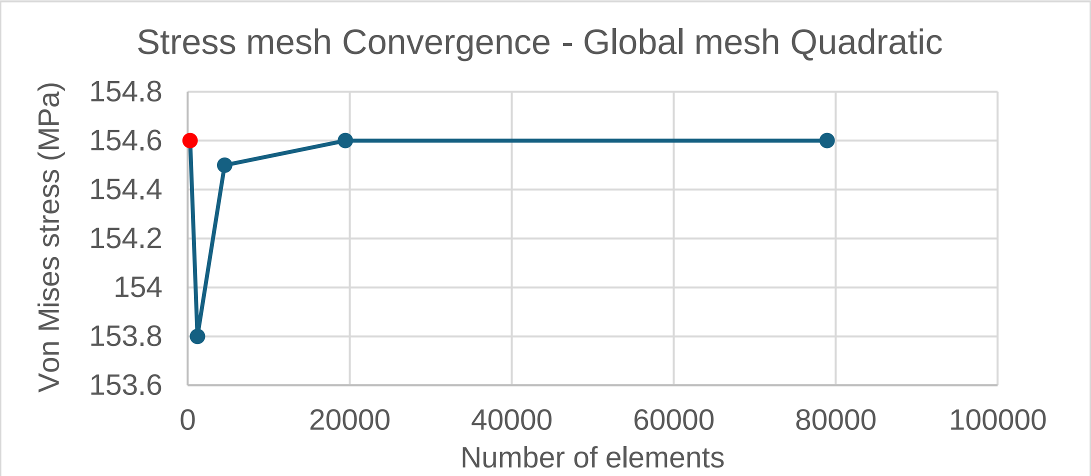


Fig. 7 Stress mesh Convergence - Global mesh Quadratic

As we can see from figure 6 and figure 7 when running convergence for both elements types, using the quadratic method converges with fewer elements than linear as even with nearly 80000 elements the linear graph does not converge towards a Von Mises stress value and a finer mesh with more elements is required. Whereas the quadratic method converges towards a value of 154.6 MPa with around 20000 elements. When simulating parts using linear will be easier to process however due to the number of elements needed to get an accurate result it would take significantly longer to get similar results to the quadratic methods. Figure 7 point in red outlier

Local mesh:

Another way of improving the accuracy of the simulated stress is by first observing where the most stress is being exerted in the part using a coarse mesh. Then using the section tool section off the parts. The rest of the part is then left as free mesh whereas the high stress portions are changed to structured mesh as shown below:

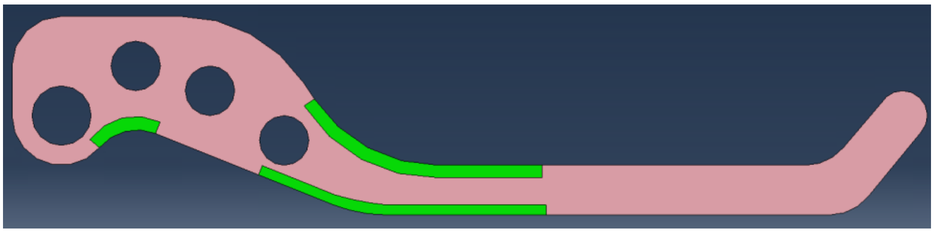


Fig. 8 Part sectioned off in free (rose) and structured (green)

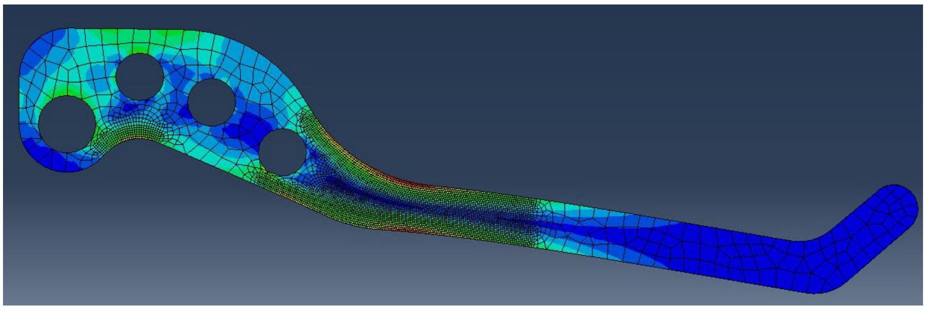


Fig. 9 Visualization of Von Mises stress for linear and global seed size of 1.6 structured mesh seed 0.4

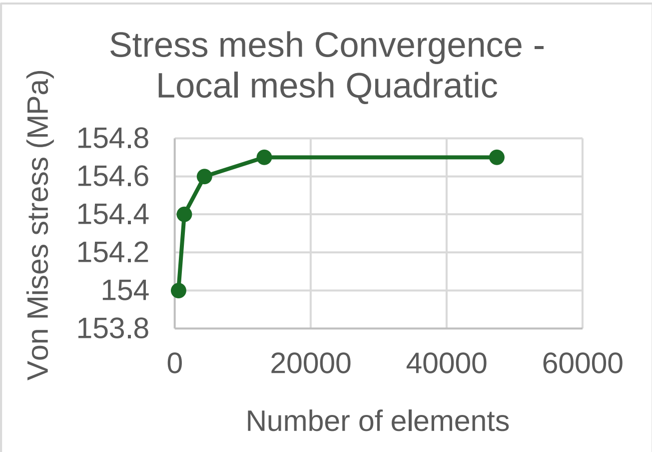


Fig. 10 Stress mesh Convergence - Local mesh Quadratic

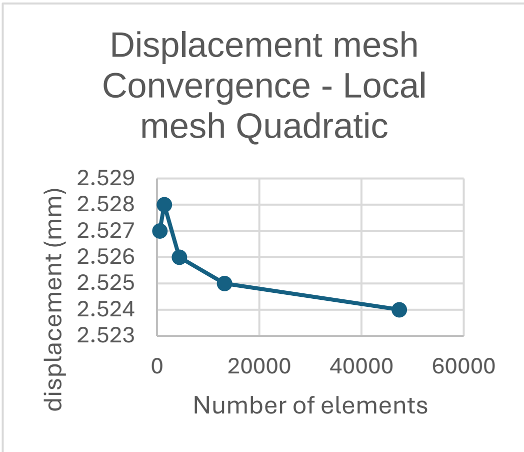


Fig. 11 Displacement mesh Convergence - Local mesh Quadratic

Using a local mesh not only reduces the amount elements needed for convergences at close to 14000 it also runs much faster with a smaller mesh size for areas of interest in this case convergence was ran with a global size of 3.2 for the free mesh and 1.6 to 0.1 for the structured mesh.

For the part tested it can be observed that the highest point of stress is 154.7 MPa and the highest deformation that is achieved is of 2.524mm. 6061 T-6 aluminium has a Yield strength

(Elastic) of 260 MPa, therefore using this material while having with M5 holes to save on weight still provides a Safety factor of 1.68. This is above target for this study.

Conclusion:

In this study the effectiveness of using finite element analysis (FEA) to simulate a potential design for a brake lever is demonstrated, as physical prototyping can be started with an approximate idea of how the part will perform reducing the cost. The part was analysed using multiple element types and mesh sizes to ensure a true value of maximum stress the part would experience would be found. It also aimed to show that adding holes to make the parts lighter would not adversely affect the structural integrity of the part by not being the area of most stress. The part being lighter weight can have the double advantage of saving fuel therefore reducing carbon emissions as well as increased performance as less inertia meaning faster actuation of the brakes.

Analysing the result of the study it can be concluded that using the quadratic mesh element is the most efficient when compared to linear elements. While linear elements are easier to compute a larger amount of element is needed for it to converge to an accurate prediction for stress. To further improve the efficiency of the simulation Local meshing was performed, where the areas of highest stress are sectioned off and higher amounts of elements simulated only in those areas Fig 8. This reduces the amounts of overall elements needed to converge to an acceptable prediction for stress while also providing the opportunity of analysing finer meshes in the high interest part faster than simulating the whole part with a finer mesh.

From the simulation ran the highest value for Von Mises stress observed was 154.7 MPa which is considerably below the Yield strength (Elastic) of 6061-T6 aluminium at 260 MPa. This is a 1.68 safety factor which is above the target of 1.5 . The maximum deformation of 2.524mm is acceptable for street motorbike use however it be below standard for a motorsport application.

Area of improvement for this study would firstly be of using the full potential of the Abaqus software and modelling the problem in 3 dimensions which would make it more complicated but more accurate to the actual real-world performance. Test could also be run using multiple materials such as carbon fibre and titanium and result compared against 6061-T6 aluminium. Linear element simulation with finer mesh could also be run to find the convergence value. Improving the areas where material has been subtracted from the part to make the item lighter but still have a safety factor of more than 1.5. Multiple platforms for finite element analysis could also be used to test the reliability of the results before prototyping the part.

References:

1. Prasetyono AP, Yudianto A, Adiyasa IW. Lightweight Design and Finite Element Analysis of Brake Lever for Motorcycle Application. ComTech: Computer, Mathematics and Engineering Applications. 2023 May 8;14(1):21-32.

2. Ditadi F. Photo by Fellipe Ditadi on Unsplash: A man standing next to a motorcycle on a dirt road. 2023. Available from: https://unsplash.com/photos/a-man-standing-next-to-a-motorcycle-on-a-dirt-road-RiBiQah4FIM. Accessed 2024 Nov 10.

3. Dassault Systèmes. SolidWorks. Version 2024. Vélizy-Villacoublay, France: Dassault Systèmes; 2024.

4. Seo, N. J., & Armstrong, T. J. (2008). Investigation of Grip Force, Normal Force, Contact Area, Hand Size, and Handle Size for Cylindrical Handles. Human Factors, 50(5), 734-744. https://doi.org/10.1518/001872008X354192

5. Spierer D, Griffiths E, Sterland T. Fight or flight: Measuring and understanding human stress response in tactical situations. The Tactical Edge. 2009;147:30-40.

6. EduPack Granta. Material property database. Cambridge, UK: Granta Design; 2024

7. Dassault Systèmes. Abaqus/CAE . Version 2024 R2. Vélizy-Villacoublay, France: Dassault Systèmes; 2024.

Appendix:

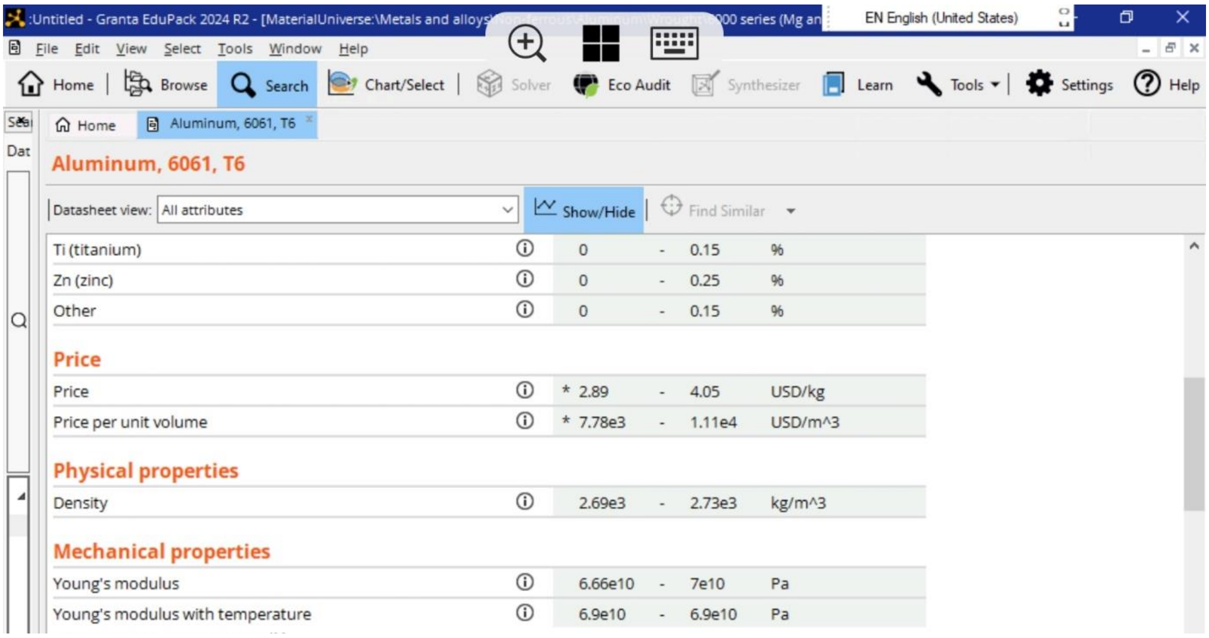


Fig. 12 EduPack Granta 6061-T6 Aluminium Youngs modulus [6]

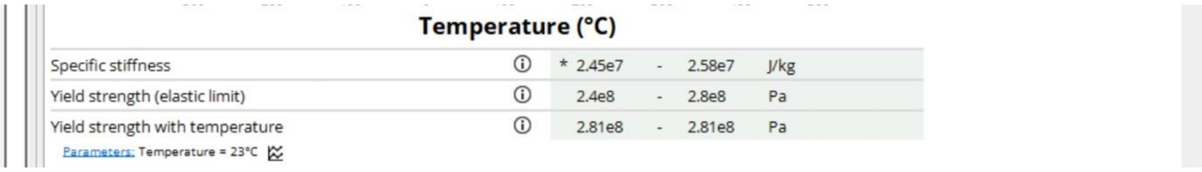


Fig. 13 EduPack Granta 6061-T6 Aluminium yield strength [6].