LIGHTWEIGHT TANDEM BICYCLE FRAME

Imperial College London

Dyson School of Design Engineering

MEng Design Engineering

Clara Arcos Jiménez

CID: 01348932

21/03/2019

Lightweight Tandem Bicycle Frame: Table of Contents

1. Introduction

The task at hand was to design a lightweight unisex tandem bicycle frame by using the finite element method. The main design requirements were that the frame needed to be unisex and lightweight. However, there were two other requirements to aim for; the natural frequencies should be larger than 30 Hz, and its effective life should be at least 10 years. The final design was to be made from aluminium and improved through iteration.

2. Methods

2.1. Modelling assumptions

In order to guide us through the modelling of the bicycle frame, a number of dimensions were given in the project brief.

In terms of the overall frame dimensions, the length should be between 2 and 3 metres, and the diameter of the wheels was given as 26 inches (660.4 millimetres). The seat joints should be 800 millimetres above the ground.

The crank shells have an external diameter of 70 millimetres, a thickness of 10 millimetres, and a length of 100 millimetres. The fork shell has the same diameter and thickness as the crank shells, but a length of 200 millimetres. Other structural members could have up to 40 millimetres external diameter, which was chosen in increments of 2 millimetres. Their thickness was chosen in increments of 0.5 millimetres. Fillets of radius between 5 to 10 millimetres were used to model welds.

The material to be used was Aluminium Alloy (7075-T6). However, all three frames were also analysed with Magnesium Alloy for testing and comparison purposes.

2.2. Boundary conditions

The assumptions made in terms of boundary conditions were that the rear wheel bearing was a fixed hinge, as this would allow the wheel to rotate, and that the inner surface of the fork shell was fixed.

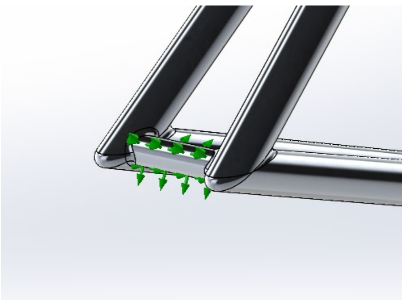

Figure 1: Fixed hinge at the rear wheel bearing.

Figure 2: Fixed inner surface of the fork shell.

2.3. Loads

The weight of each adult riding the bicycle was assumed to be 100 kilograms, which would make for a force of 981 Newtons. Additionally, the push of each pedal was modelled as an oscillating load, which was at a forward distance of 200 millimetres and a sideway distance of 100 millimetres from the centre of the circumference of the crank shell. This is shown in *Figure 3* and *Figure 4* for more clarity.

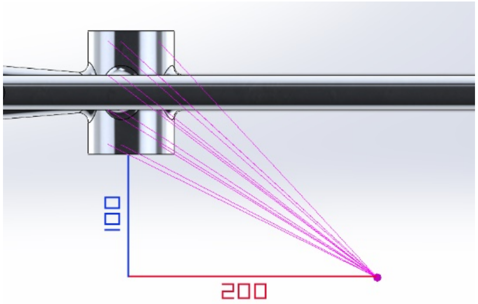

Figure 3: Modelling of a pedal push as an oscillating load (mm).

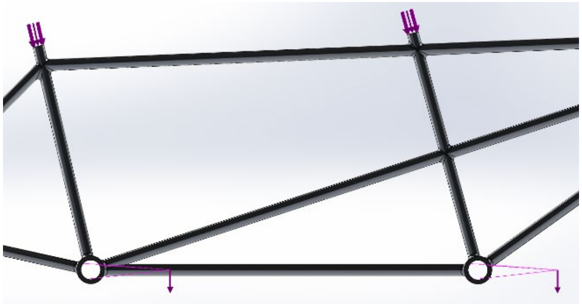

Figure 4: All loads applied. Top arrows refer to the weight of the adults, bottom arrows refer to each pedal push.

It is important to note that each static simulation only considers the load of two pedal pushes and these are later combined within the fatigue analysis environment, as each side is considered to be a different loading case.

2.4. Frequency simulation parameters

In order to find the natural frequency of the frame, a frequency study was run, considering only the weight of the two people riding the bike. Fixtures and loads were imported from the static study. 'Direct sparse solver' was chosen for this simulation.

2.5. Fatigue simulation parameters

To run the fatigue study, two different static studies had to be set up first. Both studies had the same loads and fixtures, but the pedal loads were on different sides, as explained in *Section 2.3*. These static studies were run for both Aluminium Alloy and Magnesium Alloy, as well as for each of the three iterations that were modelled. They were then combined within the fatigue study environment to create an event that could be used to determine the number of loading cycles the bicycle frame would last.

When setting up the study, a new event of the type 'find cycle peaks' was created. The number of cycles was set to 1000000, which is equivalent to 10 years, as this was the minimum life required. Two different loading cases were imported from the static studies. The pre-defined S-N curves for Aluminium Alloy and Magnesium Alloy found in the Solidworks Library were used to estimate the life, with the Gerber mean stress correction, as the curves had a ratio of -1.

3. Results

3.1. Mesh refinement study results

To ensure that the results given by the software were accurate, several sanity checks related to the mesh density were carried out. Some further reading was done to find different techniques in which a mesh refinement study could be performed, and two of these proved to be effective.

The first technique to check if the mesh is fine enough consists in comparing averaged and unaveraged results in the area of interest [1]. Averaged stress results are those which are generated from averaging the reported values from all elements adjacent to a node, therefore they are known as nodal values. Unaveraged stresses, or element stresses, are those which result from averaging the stress values from each Gaussian point within the element [2]. Following this technique, the mesh will be fine and accurate enough once there is not difference between the stress reported by averaged and unaveraged results, or the difference is not significant. In *Figure 5* and *Figure 6*, it can be observed how the difference between stresses decreases as the mesh gets finer.

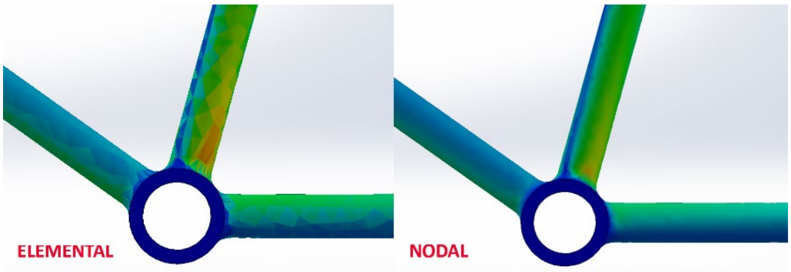

Figure 5: Differences between elemental and nodal plots for a maximum element size of 50 mm.

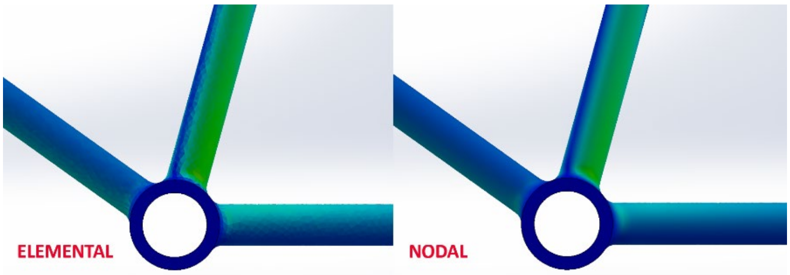

Figure 6: Differences between elemental and nodal plots for a maximum element size of 5 mm.

This method was useful for initial mesh refinement of the overall frame. The final maximum element size for the frame was 5 millimetres, with a minimum element size of 1 millimetre. The ideal solution would have been to keep decreasing the mesh size around the areas where stress concentrations happen in order to completely remove the difference between averaged and unaveraged stresses. However, due to hardware limitations, this was not possible. The final difference was found to be 8.5 MPa.

For further refinement around stress concentration areas, a convergence study was carried out. This is a technique based on plotting the number of elements in an area against the stress concentrated in it. The finer the mesh gets, the smaller the difference in stress will be. Therefore, the ideal number of elements will be found when the mesh reaches a converged value. Several tests were performed and recorded in *Figure 7*.

Figure 7: Number of elements plotted against stress.

As it can be observed from *Figure 7*, the difference between the last two tests is only 0.1 MPa, which suggests that this mesh size should be used for localized refinement in areas with stress concentrations. However, when this size was tested it took a long time to run, and a slightly higher size was chosen to save time. The final mesh sizes used throughout the model are 5 millimetres for the general mesh and 2 millimetres for localized mesh refining. *Table 1* contains the used and proposed mesh sizes, based on the different tests run.

	Proposed 'ideal' mesh size	**Used mesh size**
General bicycle frame	3 mm	5 mm
Localized refining	<1 mm	2 mm

Table 1: Comparison of used mesh sizes and proposed mesh sizes based on tests.

Areas that were in need of localized mesh refinement were usually were pipes met. These locations had higher concentrations of stress and were the most problematic when it came to fatigue analysis. Some of these spots are illustrated in *Figure 8*.

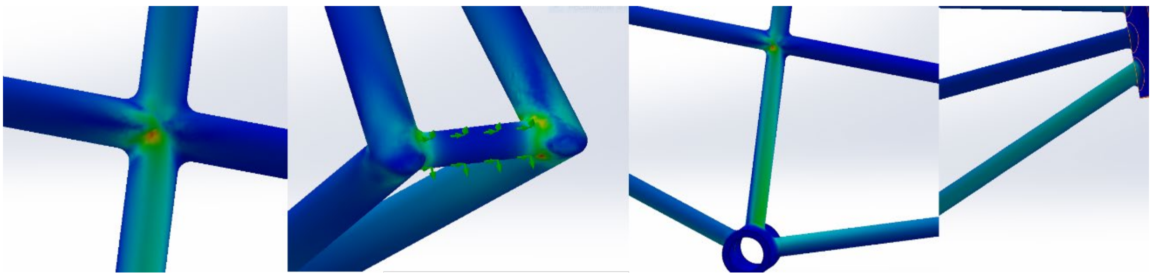

Figure 8: Locations with stress concentration considered for refinement.

3.2. Frequency simulation results

3.2.1. First iteration

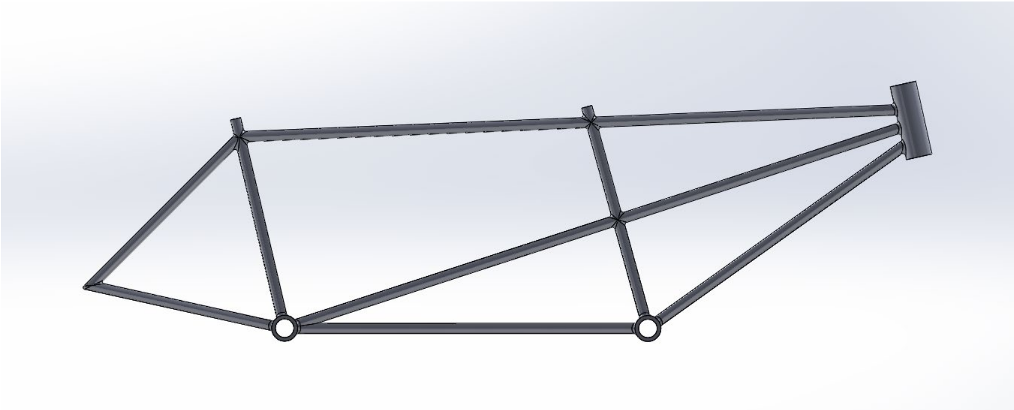

Figure 9: First model.

The first model was mostly made from 28 millimetres external diameter pipes, with just 1 millimetre thickness. The pipes connected to the rear bearing had a 26 millimetres external diameter. Fillets of 5 millimetres radius were used for the bearing, combined with 8 millimetres radius used for every other weld. This design did not meet the natural frequency requirement of 30 Hz for any of the materials tested.

Aluminium Alloy	Magnesium Alloy
26.051	27.035
43.784	42.054
66.55	67.435
100.58	99.769
130.75	129.25

Table 2: List of resonant frequencies for the first model. (Hz)

3.2.2. Second iteration

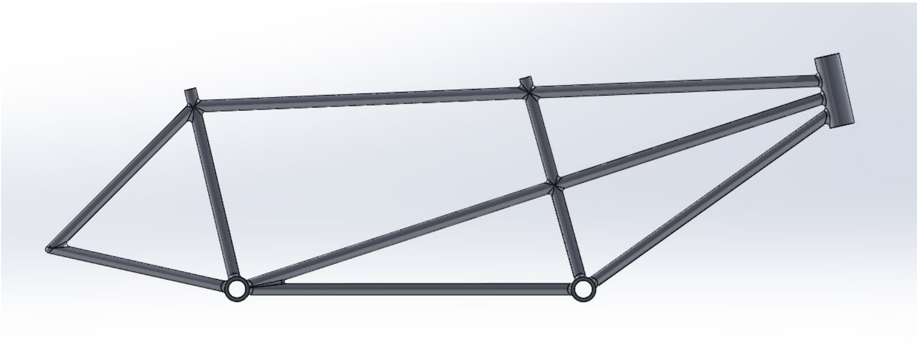

Figure 10: Second model.

The second model was made from pipes with 30 millimetres external diameter and 2 millimetres thickness. The rear pipes had a 28 millimetres external diameter. Previous fillets were replaced for 10 millimetres radius fillets. This design met the natural frequency requirement when tested with both materials.

Aluminium Alloy	Magnesium Alloy
34.909	35.904
54.297	54.216
91.745	93.728
121.55	122.99
149.03	151.25

Table 3: List of resonant frequencies for the second model. (Hz)

3.2.3. Third iteration

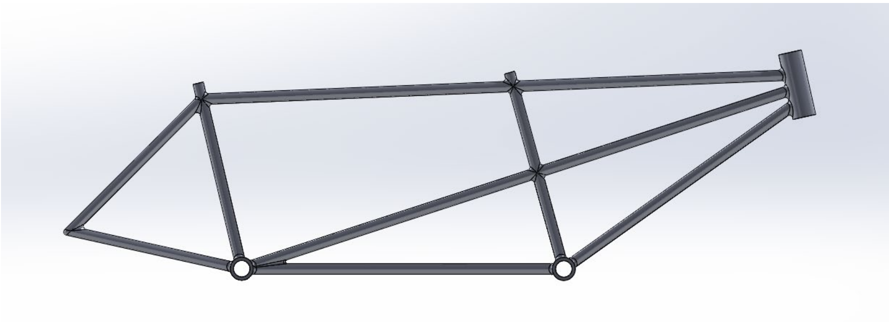

Figure 11: Third model.

The third model was constructed with pipes of 32 millimetres external diameter and 5 millimetres thickness. The rear pipes had a 30 millimetres external diameter. Fillets were kept at a radius of 10 millimetres. This third and final design met the resonant frequency requirement of 30 Hz.

Aluminium Alloy	Magnesium Alloy
36.348	37.165
54.545	54.861
98.251	100.18
121.7	123.37
156.65	149.28

Table 4: List of resonant frequencies for the final model. (Hz)

3.3. Fatigue simulation results

3.3.1. First iteration

The first design did not meet the life requirements, as it had some areas with very big stress concentrations that would cause it to fail only after 7000 loading cycles. This was the case for both Aluminium Alloy and Magnesium Alloy.

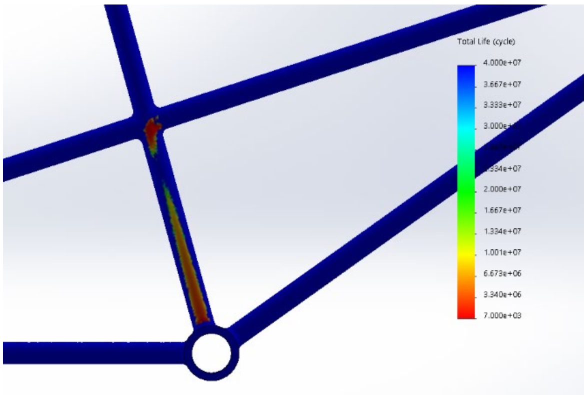

Figure 12: Fatigue life plot of the first model, showing some problematic areas.

3.3.2. Second iteration

The second iteration encountered the same problem as the first one. It seemed to perform well at first. However, very small areas would fail after 47600 loading cycles. Magnesium Alloy failed earlier than this. The second iteration still did not meet the criteria, but it was an improvement from the first one.

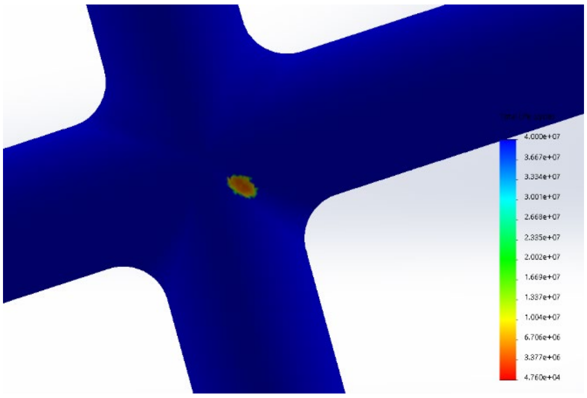

Figure 13: Fatigue life plot of the second model, showing a small problematic area.

3.3.3. Third iteration

The fatigue life criteria were met by the third and final design. According to the plot, the frame would be able to live for more than 10 years, which is equivalent to 1000000 loading cycles. Solidworks displays 40000000 cycles because it chooses a value automatically, but this does not mean that the life of the bicycle frame will be that long. Both materials passed this test.

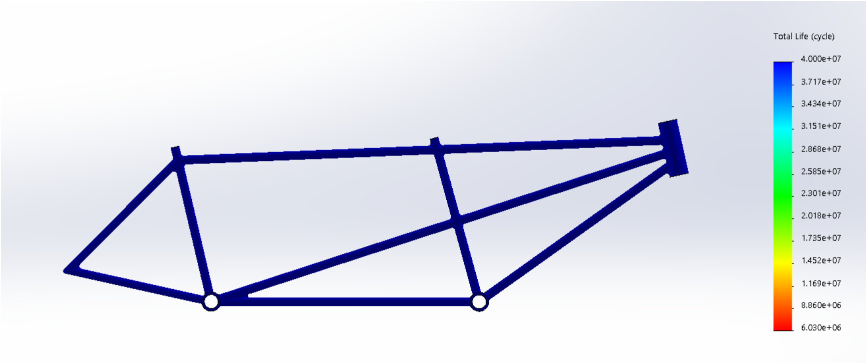

Figure 14: Fatigue life plot of the third model.

4. Discussion

4.1. Fatigue analysis

When setting up the fatigue study, a new event of the type 'find cycle peaks' was created. The reason why this option should be chosen in this case is because it allows us to base our failure analysis on multiple load cases. When using 'find cycle peaks', the software considers the combination of peaks from different fatigue loads found in different static studies to calculate the alternating stresses for each node [3]. This is particularly useful for this application, as the pushes of the pedals are considered to be two different load cases.

Gerber mean stress correction is used in this case because of the S-N curve ratio of both materials. They both have a stress ratio of -1, representing a fully reversed loading.

Fatigue analysis is an extremely important part of this design process, as it is a way to ensure that the frame which is being designed lives for as long as it should. As it happened to these three iterations, it helps the designer find which parts of the frame should be improved and strengthened to deliver a good quality product.

4.2. Frequency analysis

Conducting a frequency study is key to this bicycle frame design, as it is in contact with humans. If the frequency of the frame matched that of the human body it comes in contact with, the user could experience discomfort and physical damage. This is the reason why the fundamental frequency of this design should be kept above 30 Hz and higher if possible.

During the frequency analysis, it was found that in most cases changing the thickness and the diameter of the pipes greatly increased the resonant frequencies of the model. A fourth iteration was made as a test; this consisted of removing the diagonal pipe that joins the fork shaft and the rear crank shaft. It was observed that the frequencies decreased when the pipe was missing, and so it was decided that the design should stay as it was.

4.3. FEA and its limitations

Finite Element Analysis is limited in terms of what the software can and cannot do, or how the designer is meant to interpret the results presented to them.

One of the main confusions encountered during this task was the number of cycles in fatigue analysis. Solidworks presented results that led to believe that the bicycle frame could withstand 40000000 loading cycles because that is the value it chooses automatically. However, this seemed a bit excessive as the final design did not change much in comparison to the previous iterations that had failed the test, so it was assumed to be a wrong estimation. This led to the double-check of the S-N curve in order to find an explanation. Effectively, when looking at the S-N curve for this specific Aluminium Alloy (*Figure 15*), it is clear that the fatigue limit is above zero. Therefore, the frame will not fail if the alternating stress is below this line. Going through this process early on proved to be helpful in understanding why the software was giving results that seemed to be wrong but were actually correct when taking the theory into account.

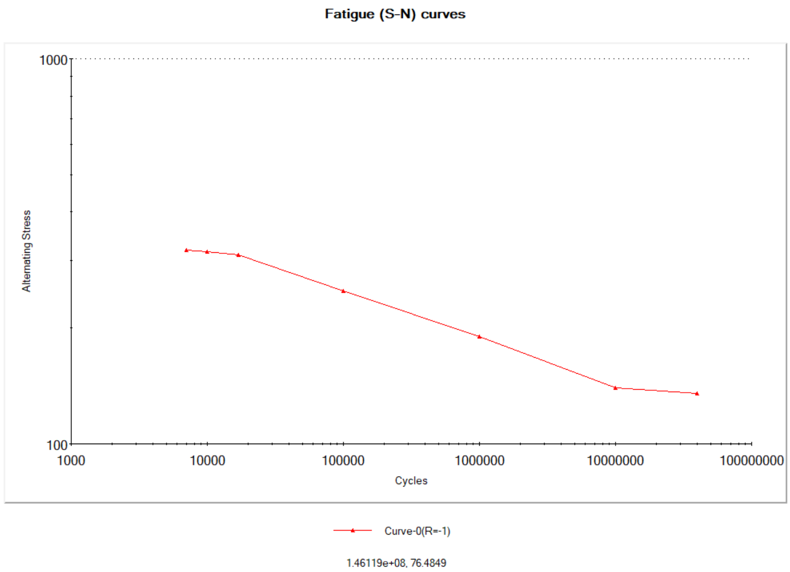

Figure 15: Aluminium Alloy (7075-T6) S-N curve.

Another issue that was found was that the CAD model cannot have very small surfaces, or it will not be meshed by the software. This accuracy may be needed for some specific small designs, so in this case the user will be limited by what the software can do.

5. References

[1] Single Fatigue Event Based on Multiple Loads. Available from: http://help.solidworks.com/2014/english/solidworks/cworks/c_single_fatigue_event_multiple_loads.htm [Accessed 20th March 2019].

[3] Elemental vs. Nodal Stress. Available from: https://forum.solidworks.com/thread/16542 [Accessed 20th March 2019].

[3] Investigating FEA Results. Available from: http://www.value-design-consulting.co.uk/investigating-fea-results.html [Accessed 20th March 2019].