Homework 6

Date : 16 November 2021

Anshul Yadav

Abstract

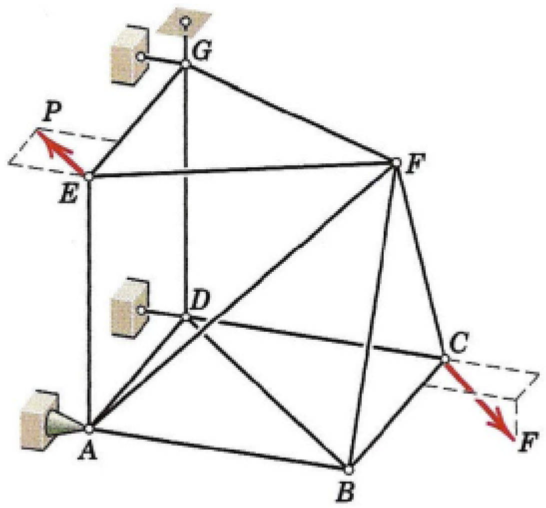
In this problem I analyse the truss system using modal analysis. The system only consists of circular cross-section steel beams connected by ball and socket joints. A finite element modal was made using beam 180 elements. Each truss member was represented by a single element. Since we are provided with an area of the rod, but not the shape of the cross-section, I presumed that the cross-sections of all truss members are circular.

The Method:

The Problem:

Consider this section of a space truss, and ignore external forces P and F. Assume that points A, D, and G are fixed in all translational DOF. The coordinates of *A* through *G* are listed below in meters.

A: 1, 0, 0
B: 1, 1.5, 0
C: 0, 1.5, 0
D: 0, 0, 0
E: 1, 0, 1.5
F: 0.5, 1.5, 1.5
G: 0, 0, 1.5



Assume the space truss is made of steel with modulus 200 GPa and Poisson's ratio 0.3. Let the cross-sectional area of the members be 0.01 m^2. Use a mass density of 7860 kg/m^3. Instead of determining whether or not this is a mechanism by counting joints, reactions, and members, solve for the natural modes of vibration. Rigid body modes should present themselves as zero energy modes ($\omega \sim 0$ Hz).
How many rigid body modes are present?
List the first 6 natural frequencies and plot the first 2 elastic mode shapes (non-zero freq. modes).
Add the minimal number of additional boundary conditions to remove all rigid body motion, and then re-compute the first 5 natural frequencies.

<u>Your report should include:</u>
Problem statement: your interpretation of the question with all inputs

- Mesh information: comment on element type, element count
- Boundary Conditions: comment on symmetry, fixed displacements/ rotations, and include a figure showing BCs
- Loading: Forces, moments, pressures, not applicable for modal analysis
- Solution: Table of natural frequencies, mode shape figures
- Discussion: Comment on results. Do they make sense?
- Appendix: Hand calculations / Sanity checks

Analytical Solution:

Since a ball and socket joint does not transmit any force/moment, we can essentially consider the member at that end as being simply supported. This is because no translation occurs at the joint but the joint is free to rotate. Thus all truss members can be modelled as simply supported beams on both sides.

We can simplify the problem using symmetry. Beams AD, BC, EG have the same length and have the same boundary conditions. Beams AB, CD, AE, DG, also have the length and the same boundary condition. Beams EF, GF, BF, CF have the same length and boundary conditions.

I calculate the first 6 natural frequencies for the following beams : AD, AE, BD, EF, AF using equation 1 (ref 1). r is the radius of gyration(in inches), (L is length in inches), Km is 1 for steel, C_n is the frequency coefficient .

$f_n = C_n * 10000 * K_m * r/L^2$(eq1)

Radius of gyration for all rods is :

$r = \sqrt{I/A} = \sqrt{(\pi * R^4/4)/(\pi * R^2)} = R/2 = 0.056419/2 = 0.028209\, m = 2.82\, cm$ (eq2)

C_f	AE	AD	BD	EF	AF
31.73	101.044	227.348	69.95	90.94	47.86
126.93	404.206	909.464	279.835	363.786	191.47
285.6	910.539	2046.35	629.645	818.539	430.81
507.73	1616.86	3637.93	1119.36	1455.17	765.879
793.33	2526.34	5684.27	1749.01	2273.71	1196.69

Table 1: First 5 frequencies for the 5 beams

Table 1 contains the first 5 natural frequencies for the 5 beams. Using this information we can predict that the modal frequency for the truss system should be on the same magnitude as individual members. This analysis is crude and should be taken with a grain of salt.

ANSYS Solution:

A finite element modal was made using beam 180 elements. Each truss member was represented by 1 element. Values of Young's Modulus and Poisson's ratio were provided in the question. No loads/pressures were applied on the system and the effect f gravity was not taken into consideration.

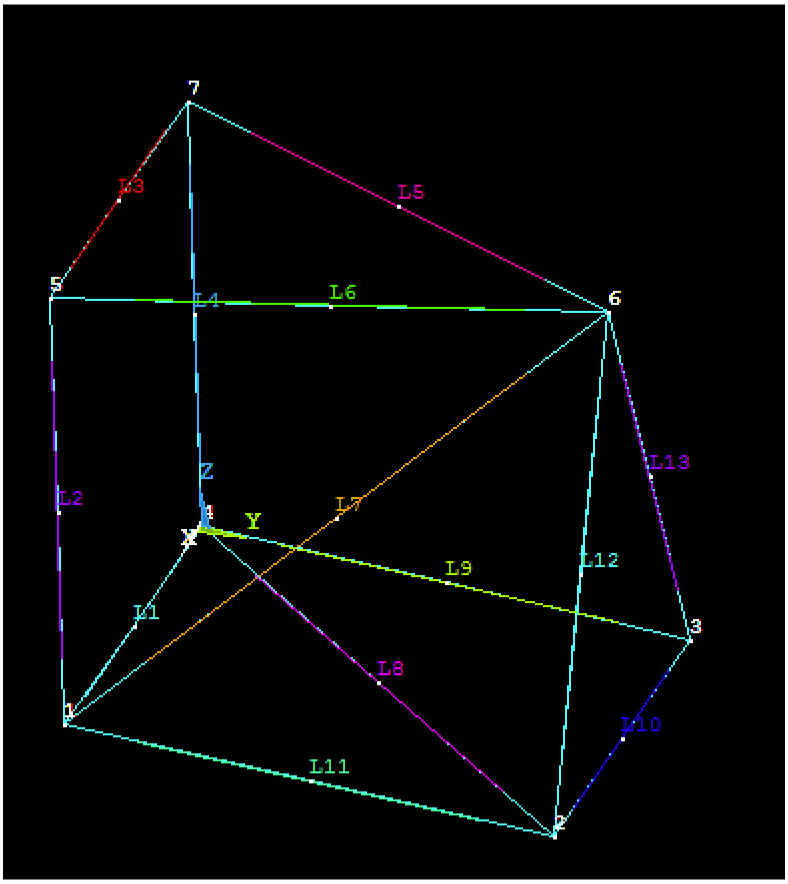


Figure 1: The modal

Figure 1 shows the modal. Key Points 1 (point A), 4(point D), 7 point(G) were constrained in all translations to obtain the rigid body modes. It was observed that there were 14 rigid body modes.

These are 13 rotational modes (one for each truss member's rotation about its own axis) and 1 translational mode.

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	0.0000	1	1	1
2	0.0000	1	2	2
3	0.0000	1	3	3
4	0.0000	1	4	4
5	0.0000	1	5	5
6	0.0000	1	6	6
7	0.0000	1	7	7
8	0.0000	1	8	8
9	0.0000	1	9	9
10	0.10428E-04	1	10	10
11	0.15619E-04	1	11	11
12	0.15831E-04	1	12	12
13	0.40310E-04	1	13	13
14	0.50010E-04	1	14	14
15	51.802	1	15	15
16	54.335	1	16	16
17	73.845	1	17	17
18	75.238	1	18	18
19	90.034	1	19	19
20	95.337	1	20	20
21	97.386	1	21	21

FIgure 2: Rigid body modes and natural frequencies

Figure 2 lists the first 14 rigid body modes. The first 6 natural frequencies are 51.802 Hz, 54.335 Hz, 73.845Hz, 75.238Hz, 90.034Hz, 95.337 Hz.

These frequencies are nowhere close to the theoretical predictions. One should bear in mind that the accuracy of ansys decreases as it tries to estimate higher modes. Natural frequencies would be more accurately computed by ansys once the rigid body modes are removed.

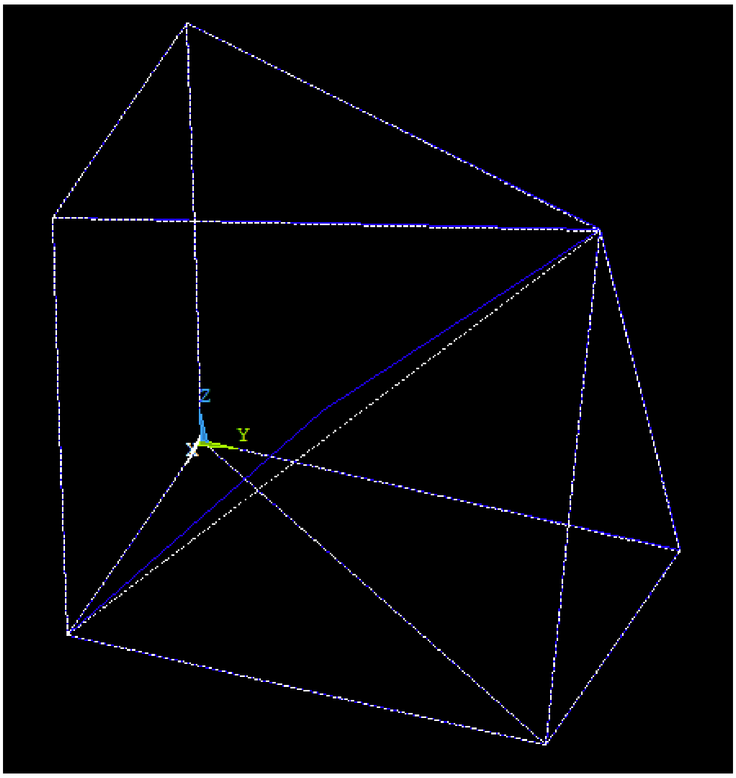


Figure 2: Modal shape for 1 natural frequency

Figure 2 shows the mode shape for the first computed natural frequency (51.802 Hz). It seems that only the beam AF undergoes vibration, so the computed natural frequency must be close to its natural frequency.

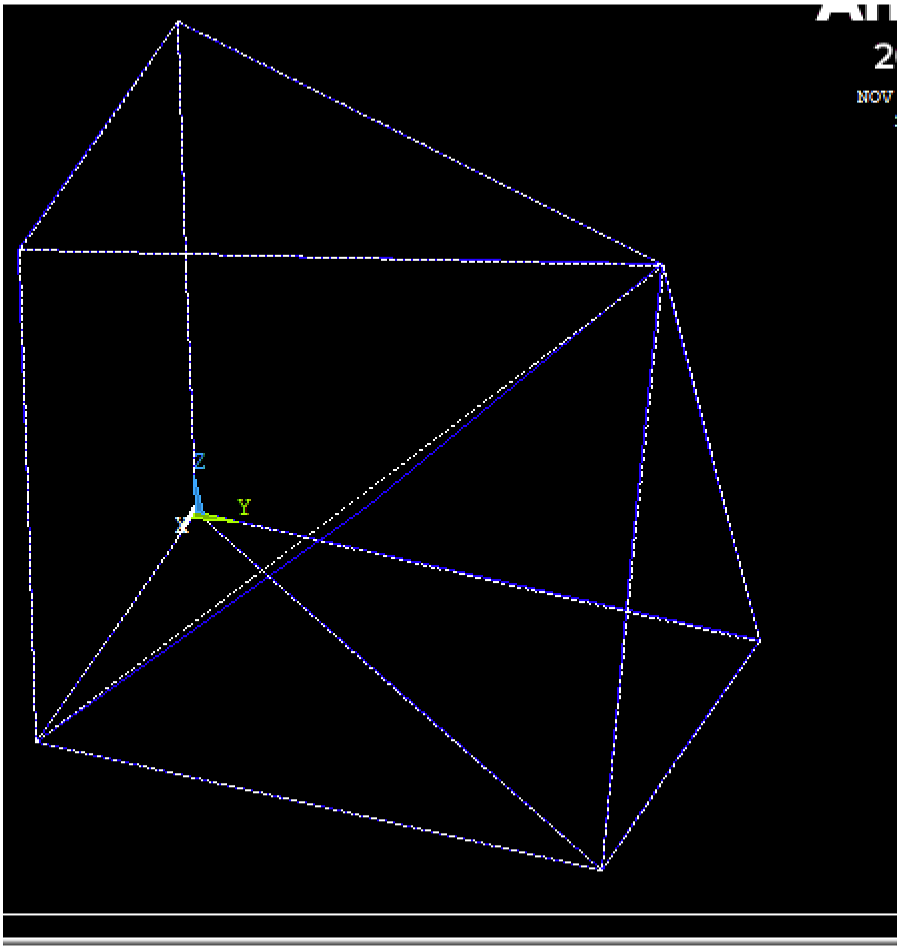


Figure 3: Mode shape for 2 natural frequency

Figure 3 shows the mode shape for the second computed natural frequency (54.335 Hz). This mode shape is also characterised by vibration in member AF but other members have some deformation.

ANSYS : Removing Rigid body modes.

The rigid body modes were removed by trial and error. I tried welding some of the ends to see what happens and I found out welding the node at F removed maximum rigid body modes as many truss members had one of its endpoints at F. In general welding the node with the most truss connections reduced the most number of rigid body modes. Using this principle all key points except key points 3 (point C) and 5 (point E) were welded (Figure 4). The modal was restricted in translation at key points 1(point A), 4(point D), 7 (point G).

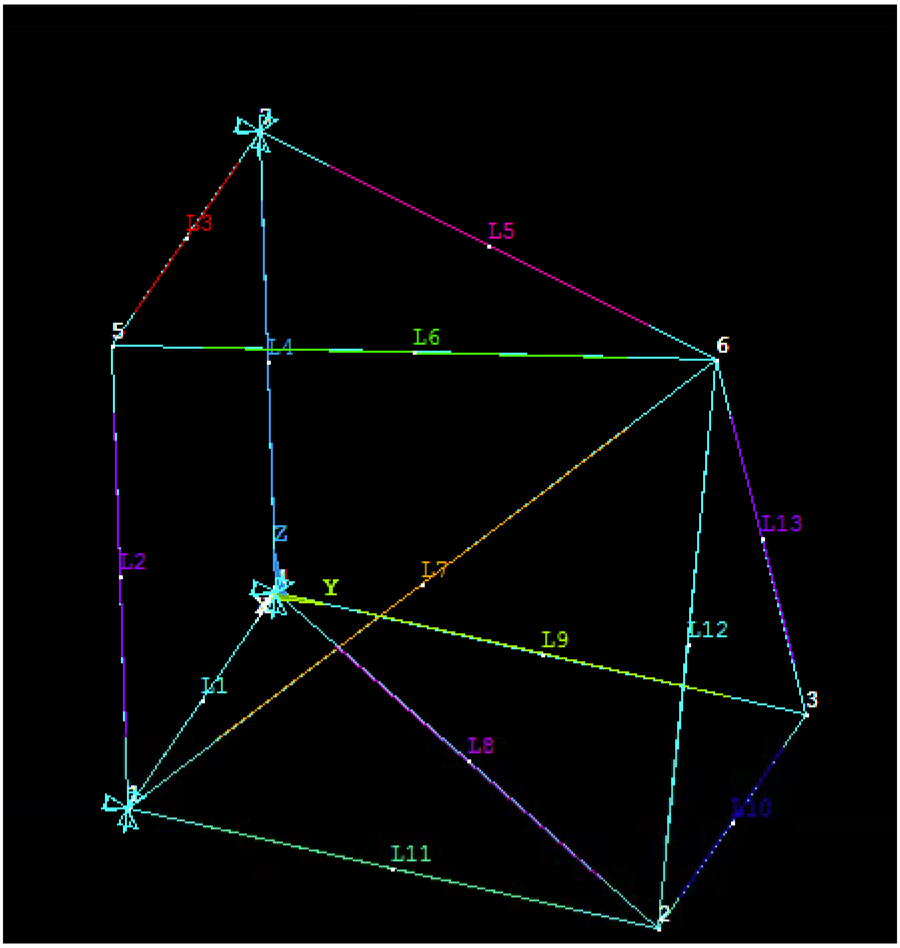


Figure 4: Rigid body modal

Modal analysis for the rigid body produced the following results.

***** INDEX OF DATA SETS ON RESULTS FILE *****

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	14.273	1	1	1
2	96.886	1	2	2
3	103.26	1	3	3
4	107.17	1	4	4
5	116.78	1	5	5

Figure 6: First 5 natural frequencies

The first 5 natural frequencies are not predicted by the theoretical results. There are a few significant reasons why that may happen.

- The theoretical calculations assumed all joints were ball and socket (hinged - hinged). Since several joints were later welded this assumption introduces error.
- The theoretical solution is calculated for individual beams. This is critically wrong as it ignores the transmission of forces/moments between truss members.
- The theoretical solution's natural frequency increases quadratically. For eg. for all members their third natural frequency is 9/4 ths of their 2nd natural frequency. This is not true of Ansys' solution where 2nd, 3rd, 4th, and 5th natural frequencies are close to each other. On closer inspection, one observes that the mode shape for ansys's natural frequency do not differ by much whereas the mode shape for the theoretical solution introduces a new mode every time its natural frequency is incremented (Figure 5,6,7).
- Lastly the ansys results before and after the removal of rigid body modes differ significantly.

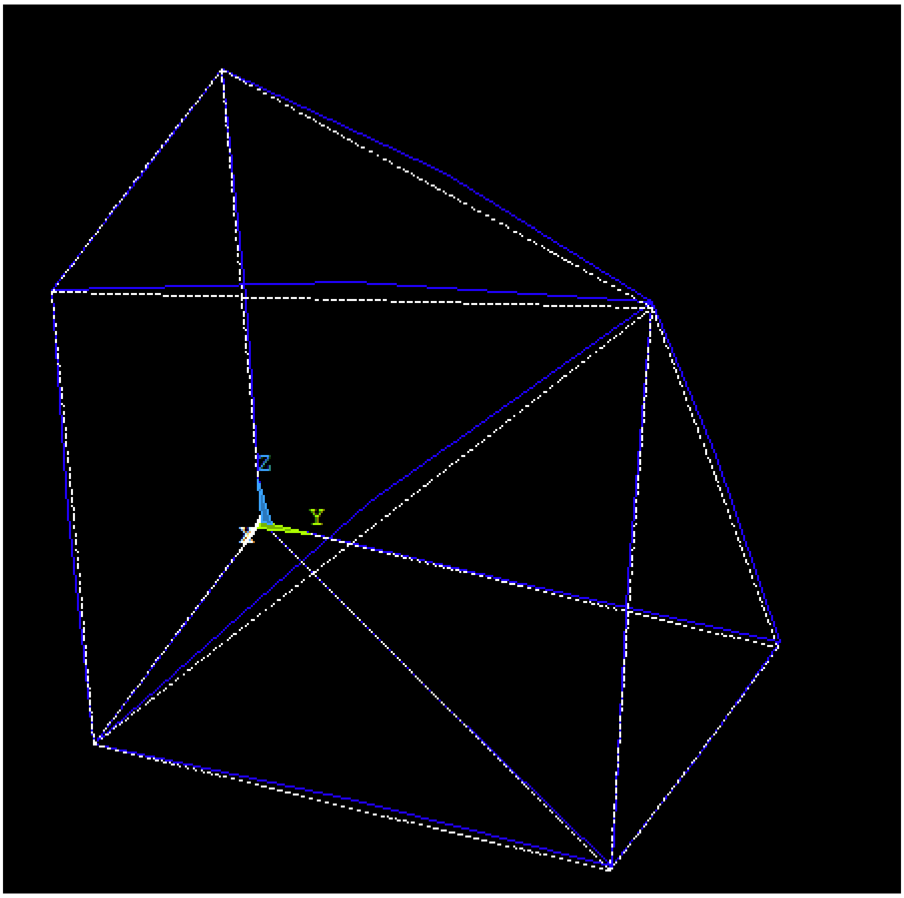


Figure 5: Mode shape for 2nd natural frequency (96.886 Hz)

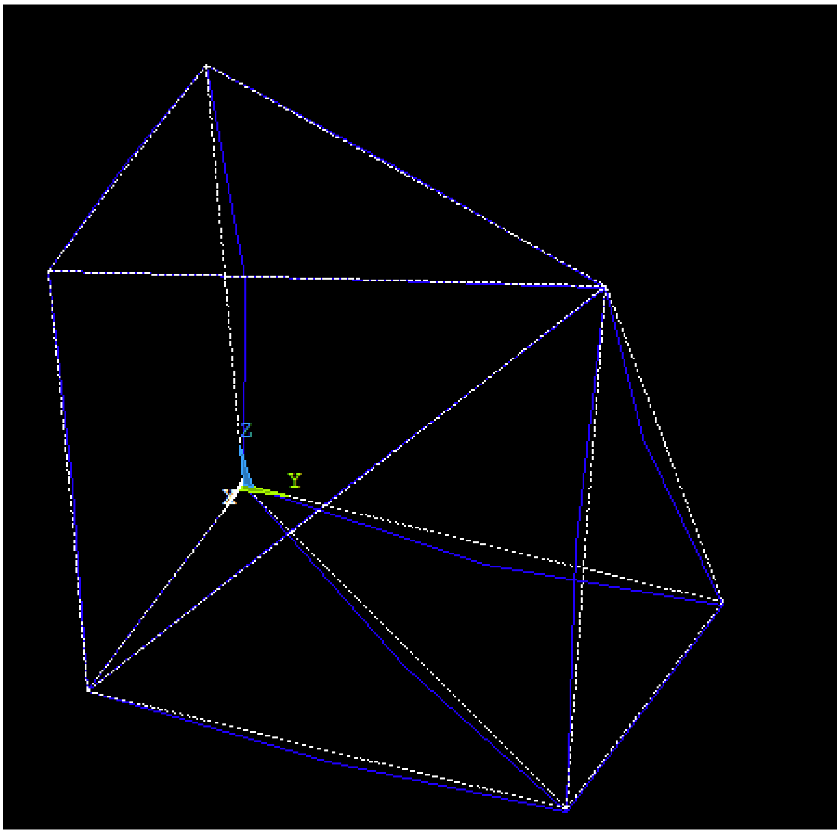


Figure 6 : Mode shape for 3rd natural frequency (103.26 Hz)

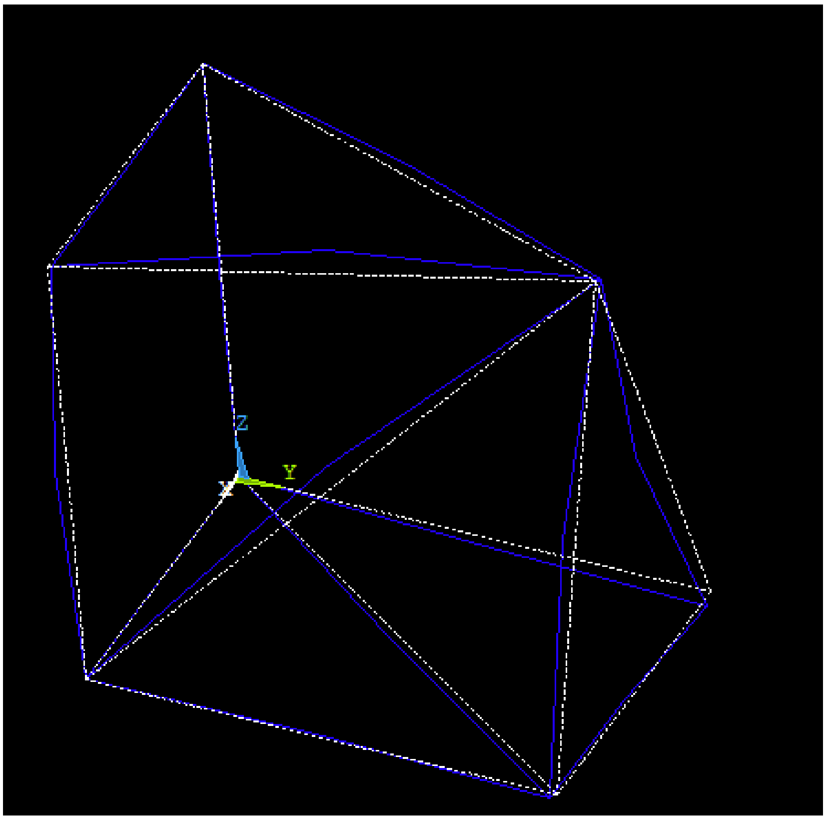


Figure 7 : Mode shape for 4th natural frequency (107.17 Hz)

Conclusion:

In conclusion, there were 14 rigid body modes that were successively removed to obtain the first 5 natural frequencies. These natural frequencies did not differ significantly from one other but were very different from the theoretical predictions. This was to be expected as the assumptions of the theoretical were not borne by ansys. That being said ANSYS results are still reliable and the truss system can be modelled using the ANSYS solution.

References:

(ref 1) Roarks Formulas For Stress and Strain - Richard G Budynas (2020), pg658-670, Table 16.2.