Abstract:

The following report is based upon the static structural analysis of a truss bridge. A truss bridge is a bridge whose load-bearing superstructure is composed of a truss, a structure of connected elements usually forming triangular units. The connected elements (typically straight) may be stressed from tension, compression, or sometimes both in response to dynamic loads. Our job would be to analyze these characteristics. The software therefore used is ANSYS 18. The bridge's model was designed using Solid Works 19. The report will thereby brief upon the complete procedure / calculations performed. We will start by an introduction, discussing the types of various truss bridges and specifying the one we selected. After that the model's design would be discussed and finally the analysis report would be presented.

Following would be the projects outline:

- Designing a CAD model for the truss bridge.
- Importing the model onto ANSYS workbench.
- Performing the static structural analysis.
- Providing the final results based upon calculations.

Contents

List of Figures

Chapter 1: Introduction

1.1 Introduction

Truss bridge is a type of bridge whose main element is a truss which is a structure of connected elements that form triangular units. Truss is used because it is a very rigid structure and it transfers the load from a single point to a much wider area. Truss bridges appeared very early in the history of modern bridges and are economic to construct because they use materials more efficiently.

For analysis purpose trusses are assumed to be pin jointed at the pints where the straight components meet. This assumption thereby leads to the conclusion that the members of trusses will act only in tension or compression. The following figure (Fig-1) shows the internal members of a truss bridge:

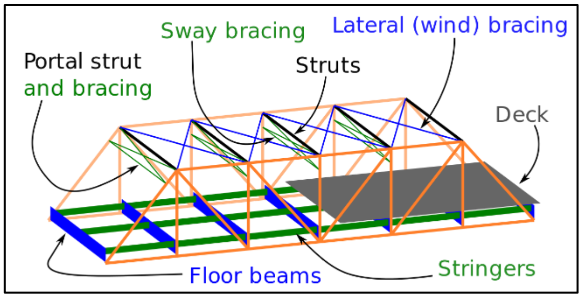


Figure 1-Internal members of a truss bridge

1.2 Types of Truss Bridge

This section would discuss the common types of truss bridges:

Allan truss:

The Allan truss, designed by Percy Allan, is partly based on the Howe truss. It has three truss style connected together. The truss can carry heavier loads and use relatively less material. Its design is shown in Fig-2:

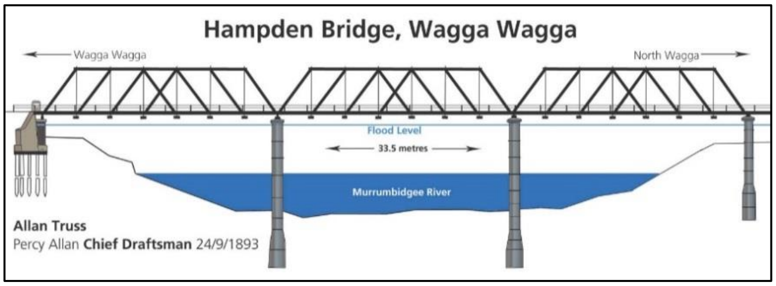


Figure 2-Allan Truss

Bailey truss:

The Bailey truss was designed by the British in 1940-1941 for military uses during World War II. The prefabricated and standardized truss elements may be easily combined in various configurations to adapt to the needs at the site.

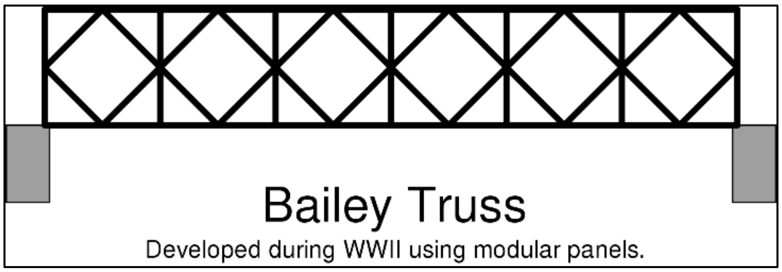


Figure 3-Bailey Truss

Baltimore truss:

A Baltimore truss has additional bracing in the lower section of the truss to prevent buckling in the compression members and to control deflection. It is mainly used for rail bridges, showing off a simple and very strong design.

Figure 4-Baltimore Truss

BollmanTruss:

The design employs wrought iron tension members and cast iron compression members. The use of multiple independent tension elements reduces the likelihood of catastrophic failure.

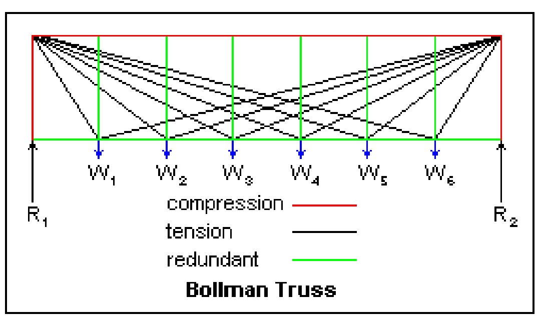


Figure 5-Bollman Truss

Howe Truss:

Fig-6 shows the Howe truss; It has vertical members and diagonals that slope up towards the center, the diagonals are under compression under balanced loading.

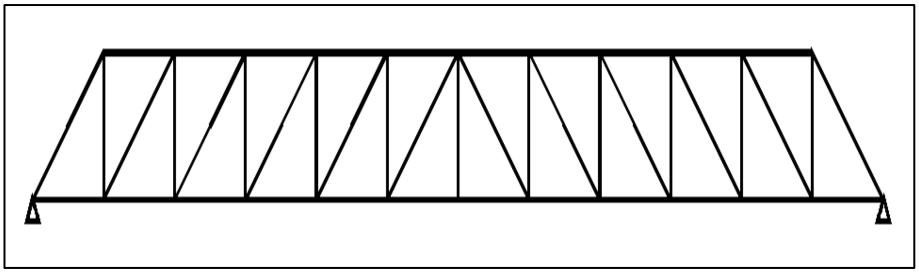


Figure 6-Howe Truss

K-truss:

The K-truss is named after the *K* formed in each panel by the vertical member and two oblique members.

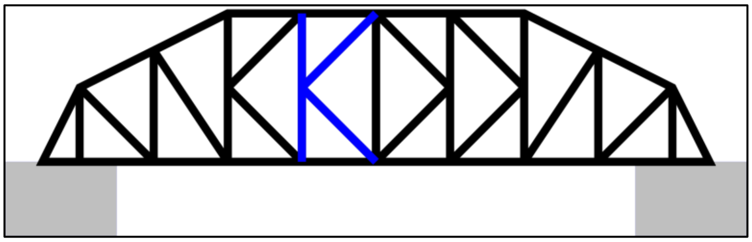


Figure 7-K-truss

King Post Truss:

One of the simplest truss styles to implement, the king post consists of two angled supports leaning into a common vertical support.

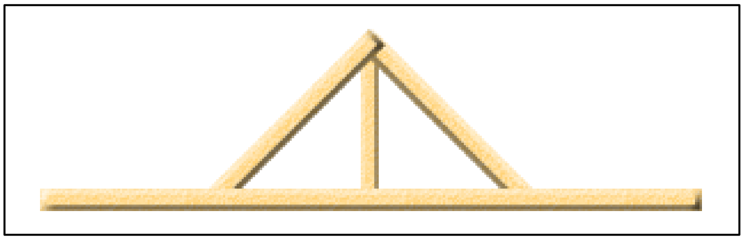


Figure 8-King post Truss

Lattice Truss:

This type of bridge uses a substantial number of lightweight elements, easing the task of construction. Truss elements are usually of wood, iron, or steel.

Figure 9-Lattice Truss

Lenticular Truss:

A lenticular truss bridge includes a lens-shape truss, with trusses between an upper arch that curves up and then down to end points, and a lower arch that curves down and then up to meet at the same end points.

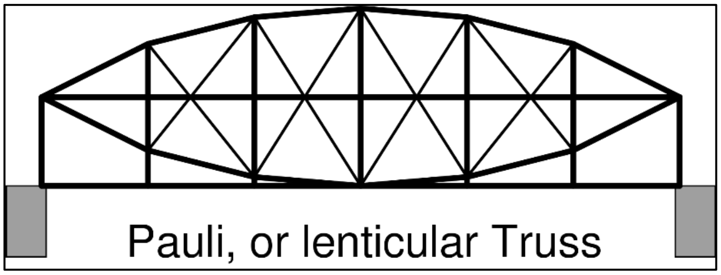


Figure 10-Lenticular Truss

Parker Truss:

A Parker truss bridge is a Pratt truss design with a polygonal upper chord.

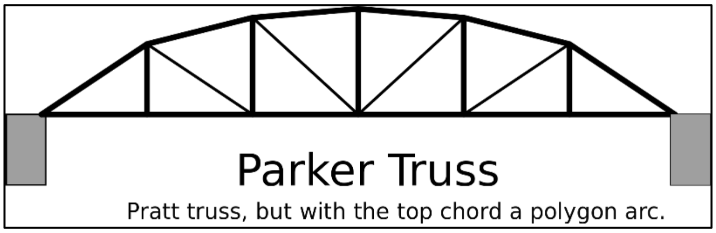


Figure 11-Parker Truss

Queen Post Truss:

The queen post truss, is similar to a king post truss in that the outer supports are angled towards the center of the structure.

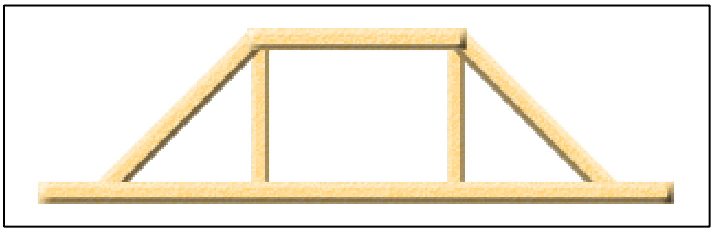


Figure 12-Queen Post Truss

1.3 Proposed Bridge

Platform of following bridge analysis was proposed

Figure 13 Proposed Bridge

We would devise the best model after **Ansys Structural and Comparative Analysis.**

Chapter 2: CAD Model

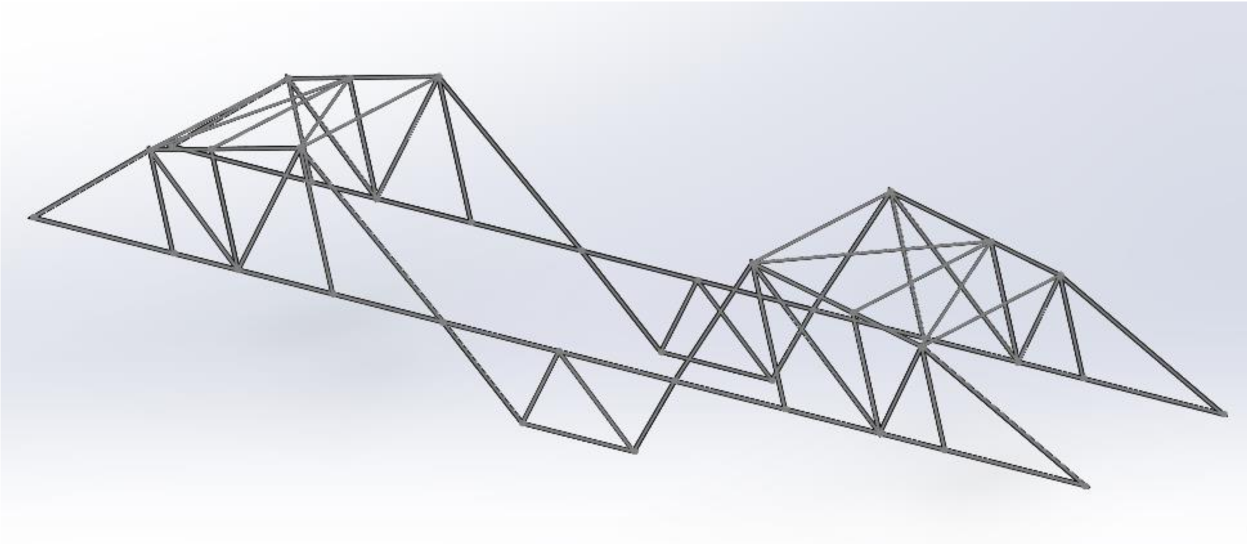
2.1 Model by Weldments

Size: 50m length, 5.5m width and 45m height.

Solid Works part makin

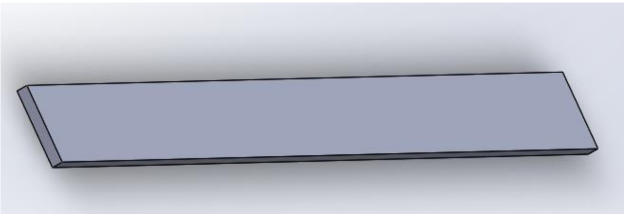
Trusses

1. Make 3D sketch
2. Apply Welments



Platform

1. 2D sketch
2. Extrude



Rod Support

1. 2D sketch
2. Extrude

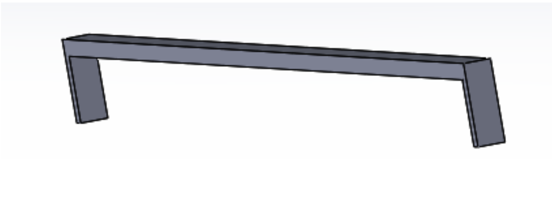


Figure 14 Parts for Weldment Model

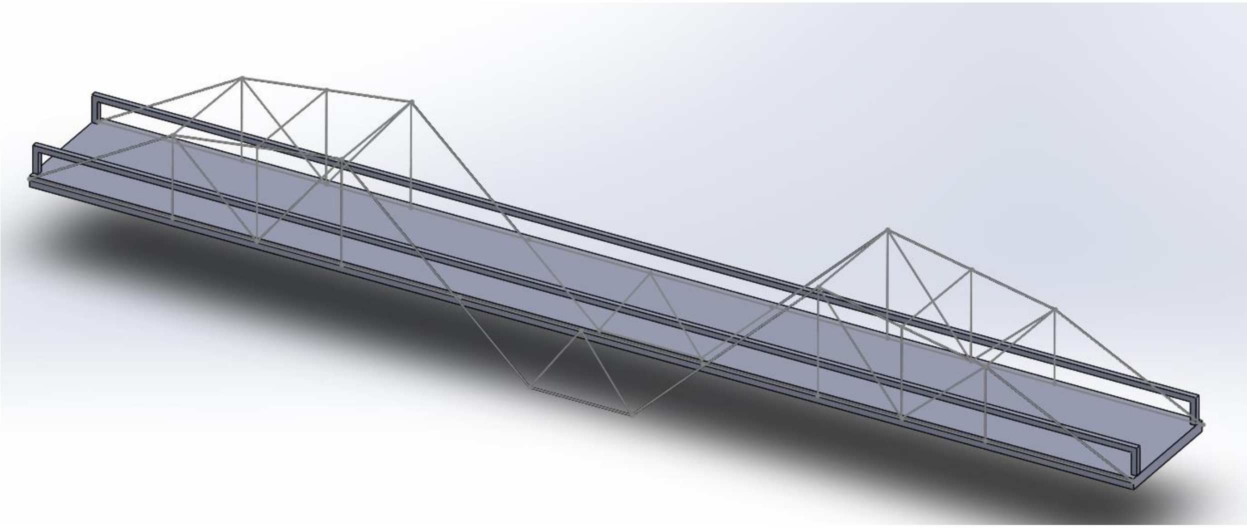


Figure 15 Initial Model Weldments

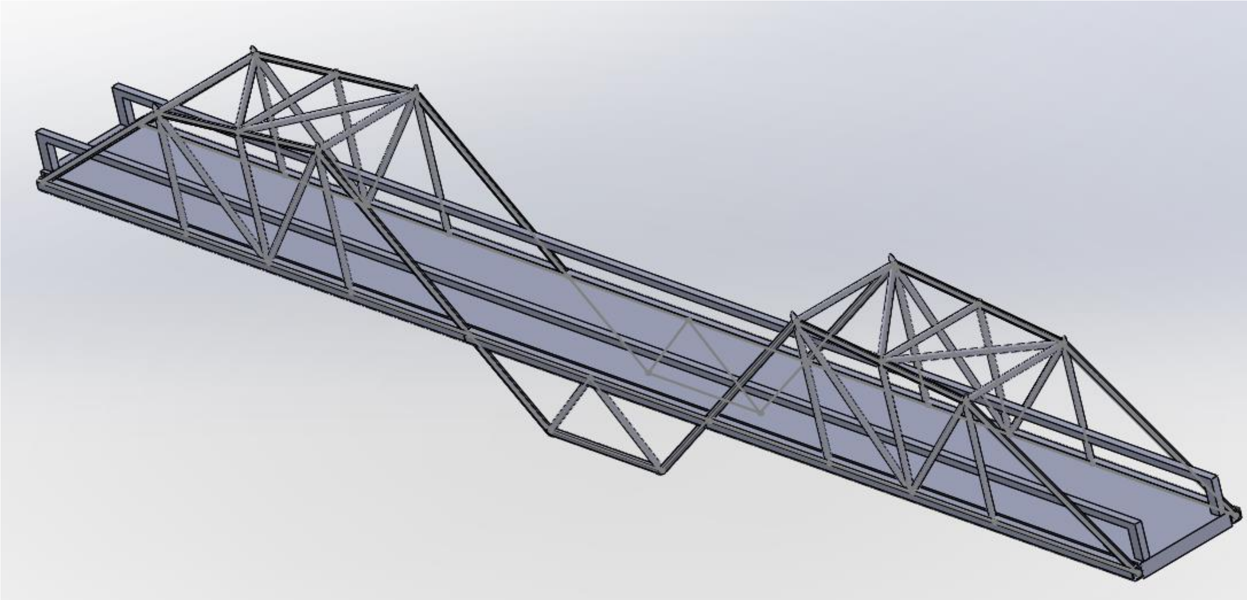


Figure 16 Weldment Model Modified by Scalling of trusses

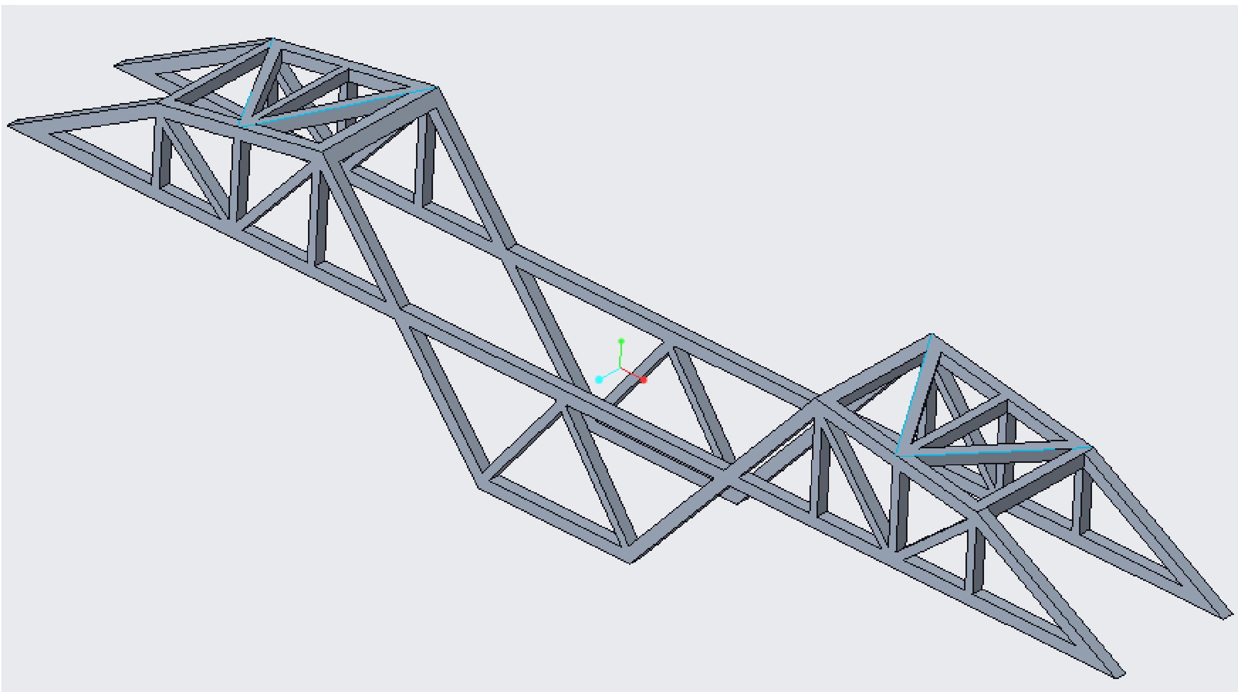
In weldment model the connecting trusses are hollow from inside.

2.2 Model by Solid Trusses

Done in Creo

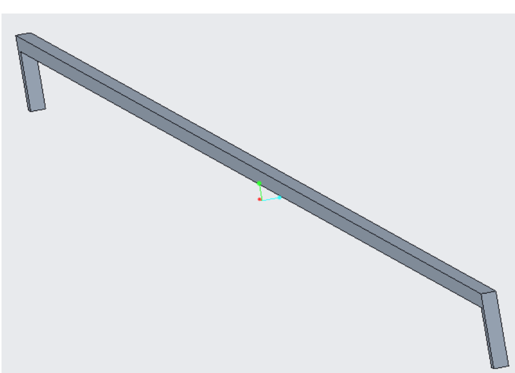
Solid Truss

1. Sketch & Extrude
2. Mirror, Sketch & Extrude



Rod

1. Sketch
2. Extrude



Plateform

1. Sketch
2. Extrude

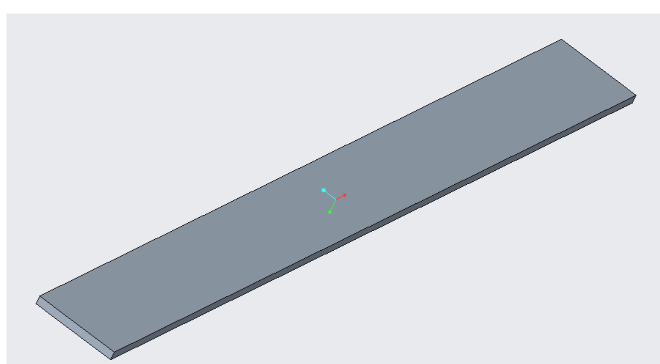


Figure 17 Solid Truss Model Parts

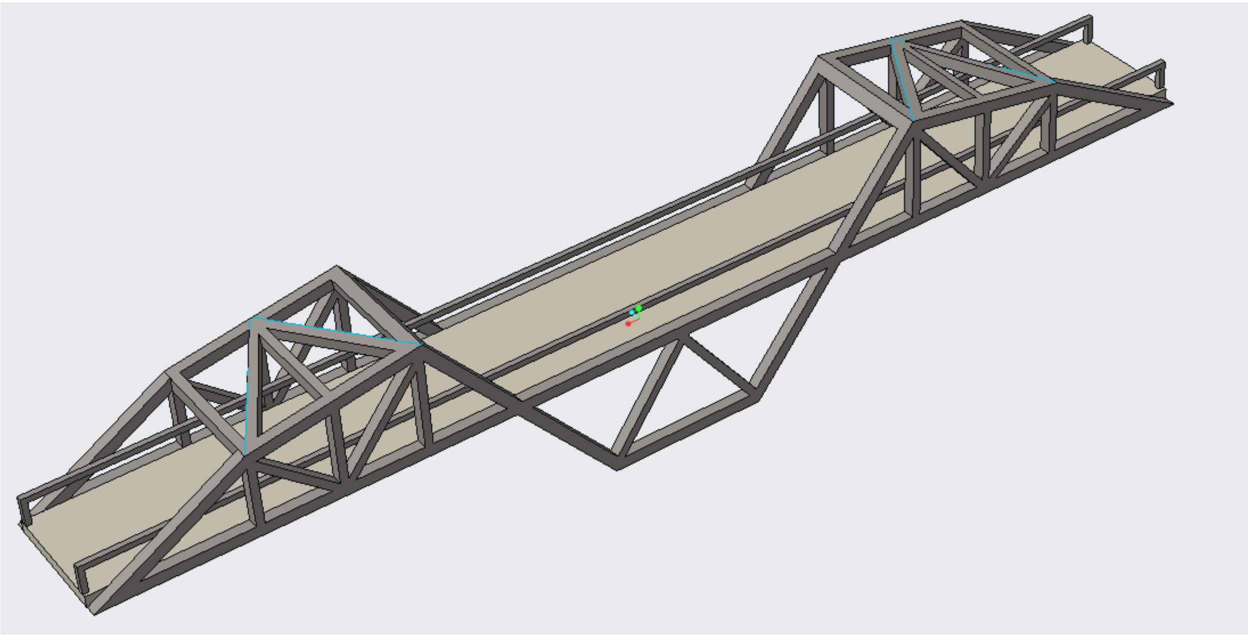


Figure 18 Solid Truss Model

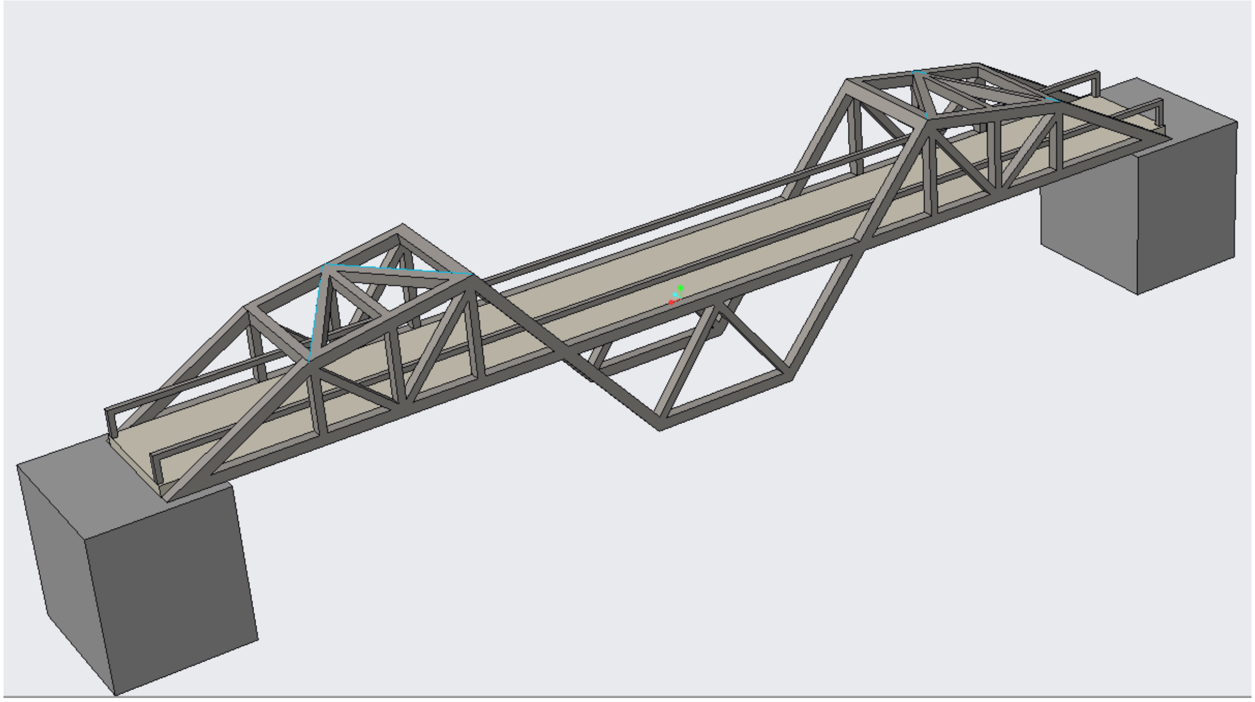


Figure 19 Solid Truss Model with Support

Chapter 3: ANSYS Static Structural Analysis

3.1 Static Structural Analysis

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads. The types of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)

Structural analysis is critical because it can determine cause and predict failure – evaluating whether or not a specific structural design will be able to withstand the external and internal stresses and forces expected for the design.

ANSYS using Finite Element Analysis for doing Static Structural Analysis

3.2 Steps to perform

1. Open ANSYS window
2. From the Toolbox, drag a Static Structural

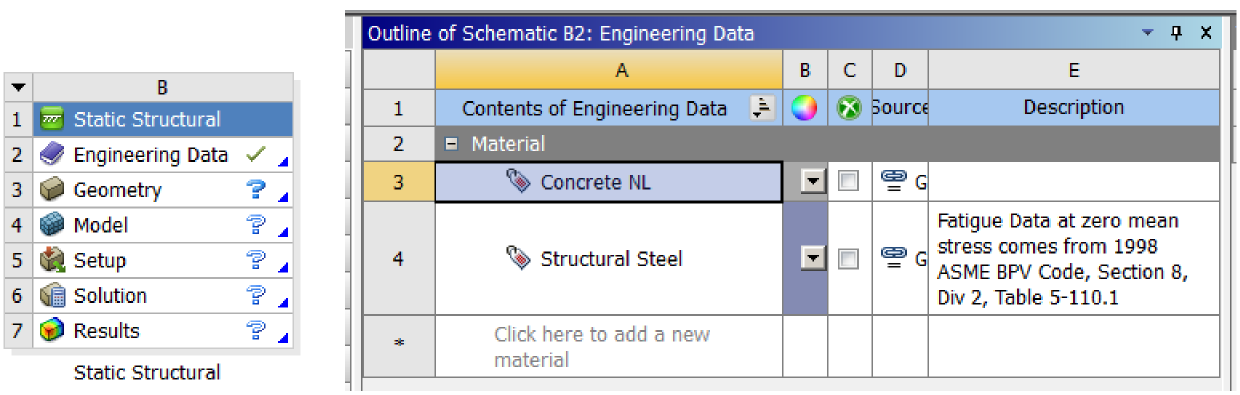


Figure 20 Step 2,3

3. In Engineering Data ensure Structural Steel and Concrete are added
4. In Geometry import the model
5. Double click on Model

Now in geometry option we can change the type of material to structural steel for all and concrete for platform (when required).

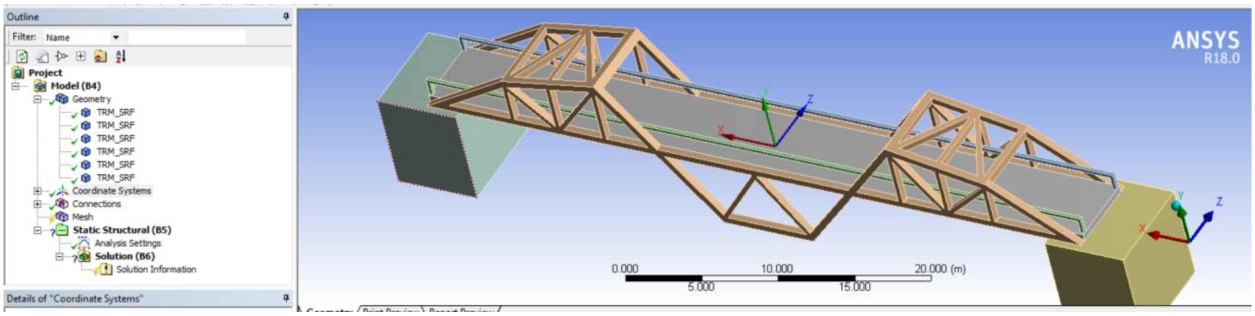


Figure 21 Step 4,5

6. Generate Mesh

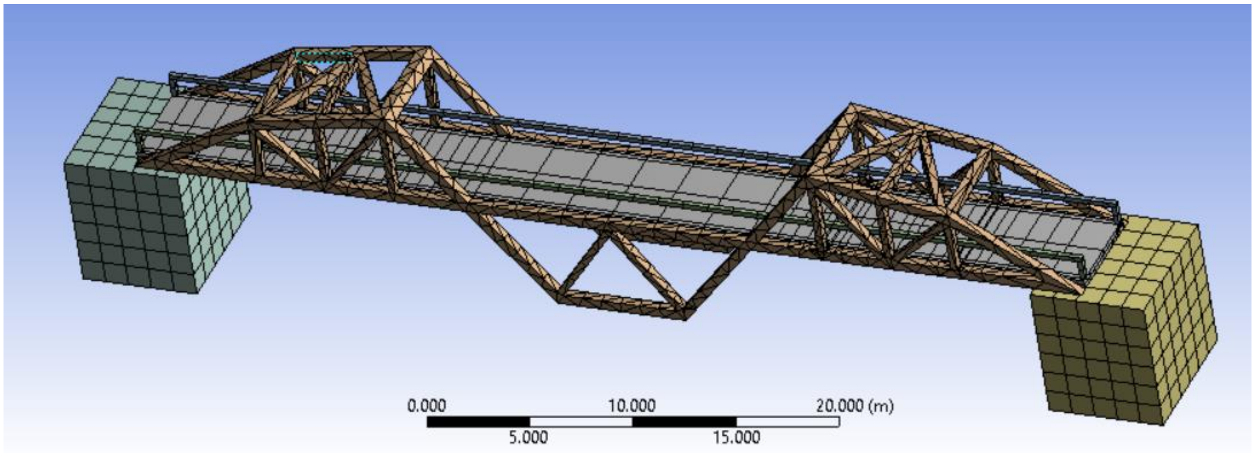


Figure 22 Step 6

We can even change size of meshing to get more accurate results

7. Add fix support at end blocks
8. Apply test force
9. Add required solutions and solve

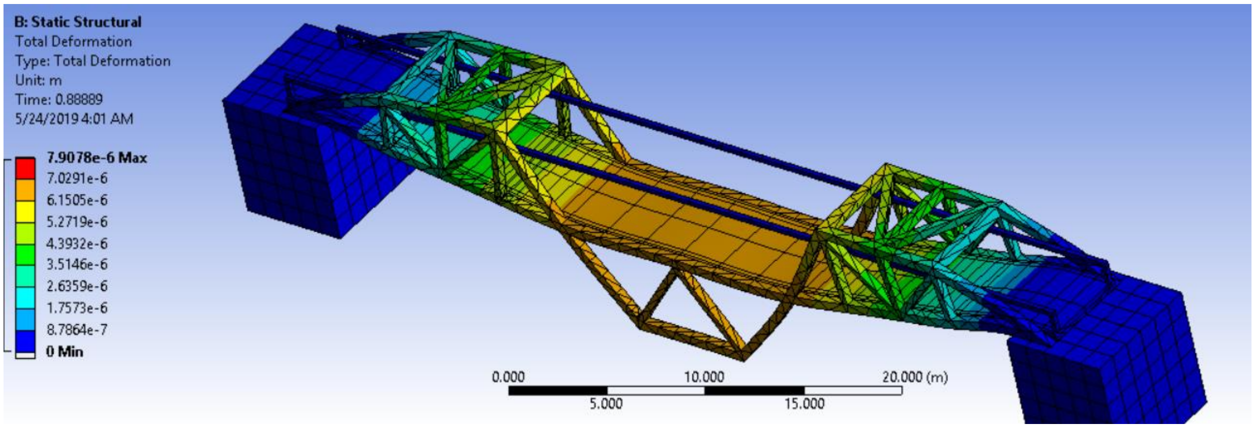


Figure 23 Total Deformation

3.3 Comparative Analysis

Different designs were tested using ANSYS to compare them with goal is to find the most appropriate design for a truss bridge. In order to provide a fair comparison following conditions were constant:

1. Same Force was applied to all of them
2. The material used for Truss is Structural Steel
3. The material used for bridge platform is concrete

3.3.1 First Comparison (Number of Railing Supports)

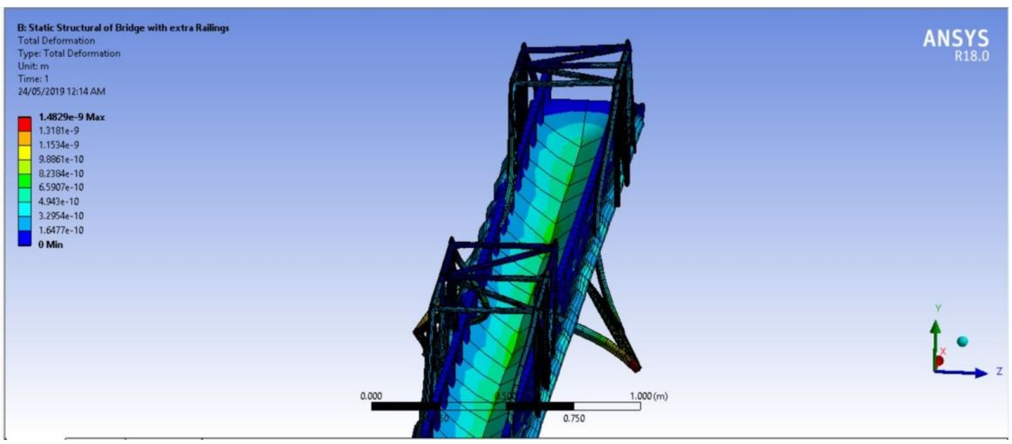


Figure 24 Evenly Spread Extension, Lowest peak extension Bridge

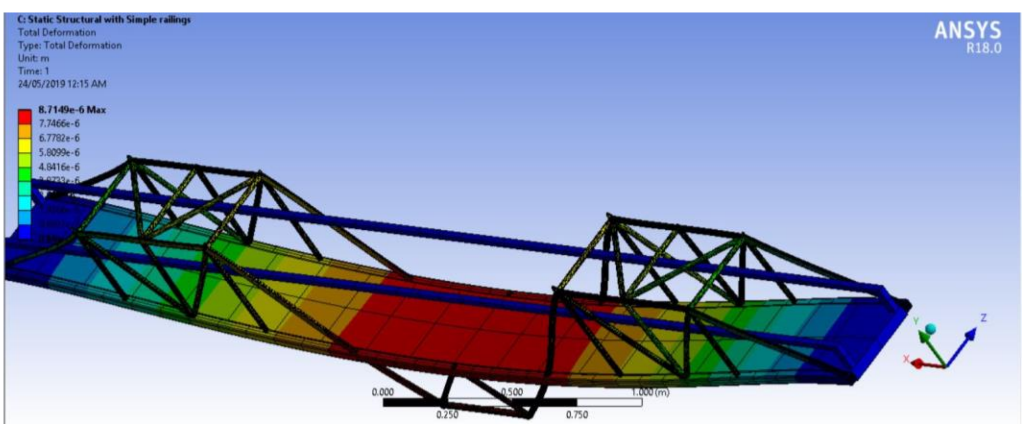


Figure 25 Evenly increasing Extension, Lower extension at ends

3.3.2 Second Comparison (Scale of Model)

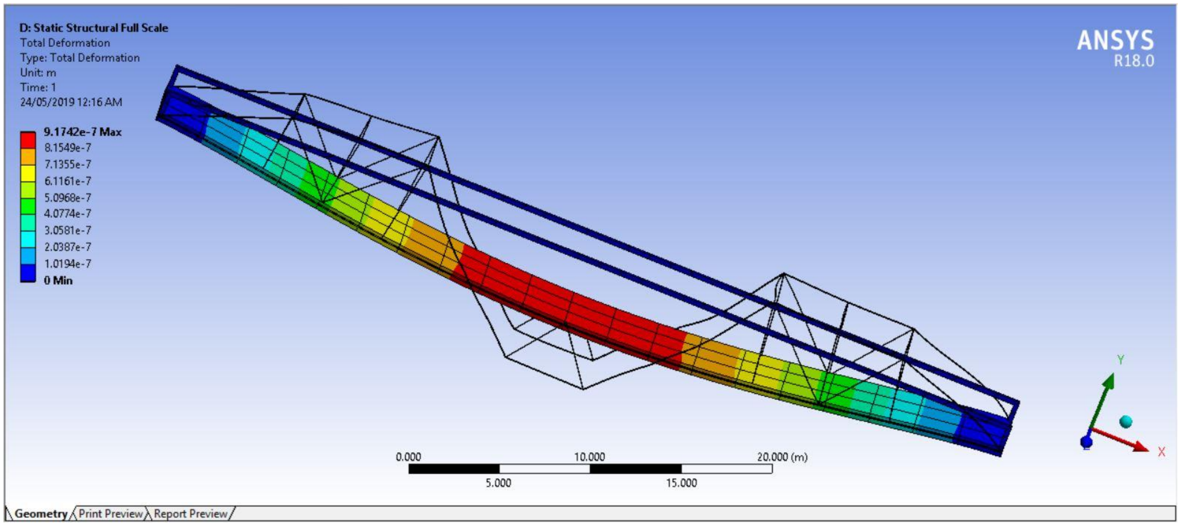


Figure 26 Full Scale Model

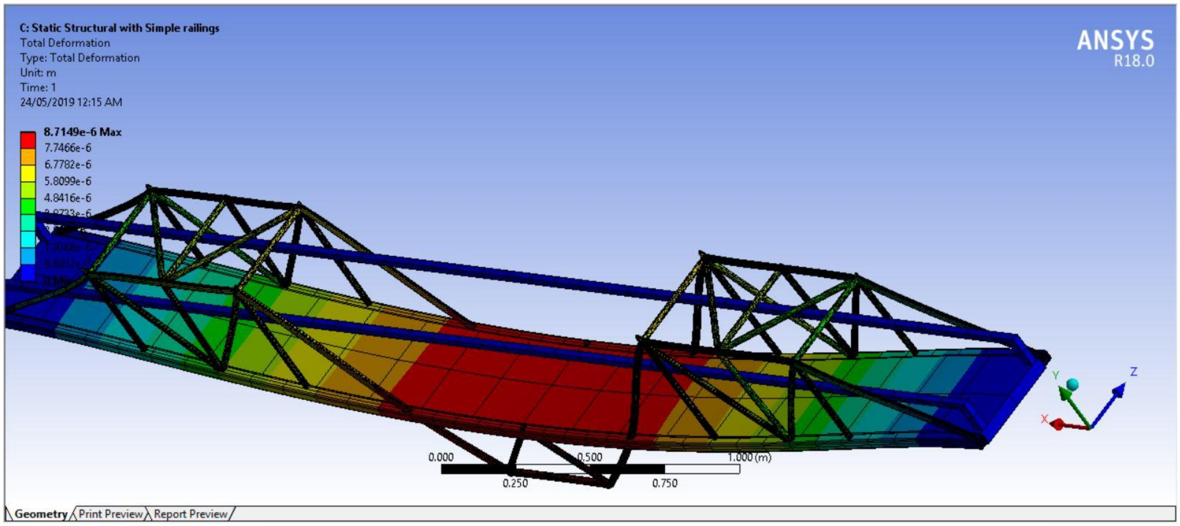


Figure 27 Smaller Scale Model

3.3.3 Third Comparison (Solid Trusses)

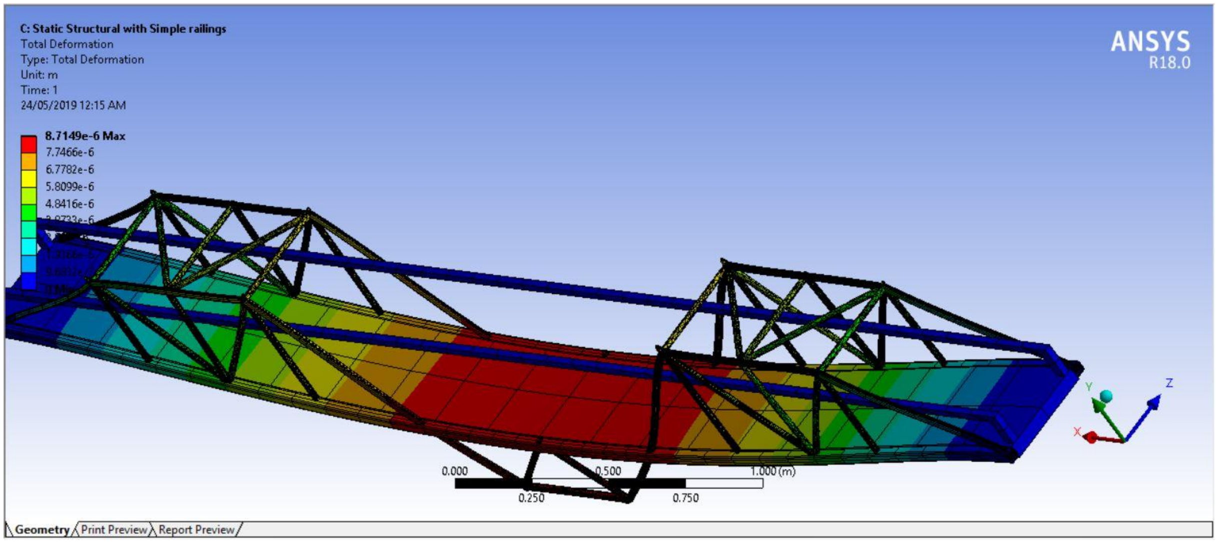


Figure 28 Less Extension in Trusses but focused extension in centre

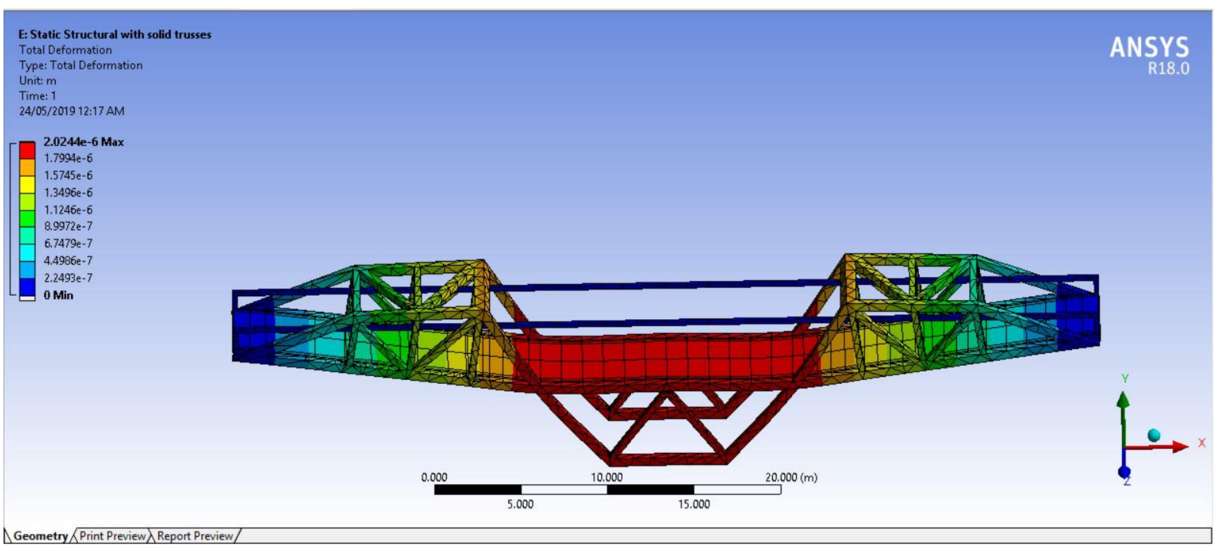


Figure 29 Extension greater in trusses but more evenly distributed

3.3.4 Fourth Comparison

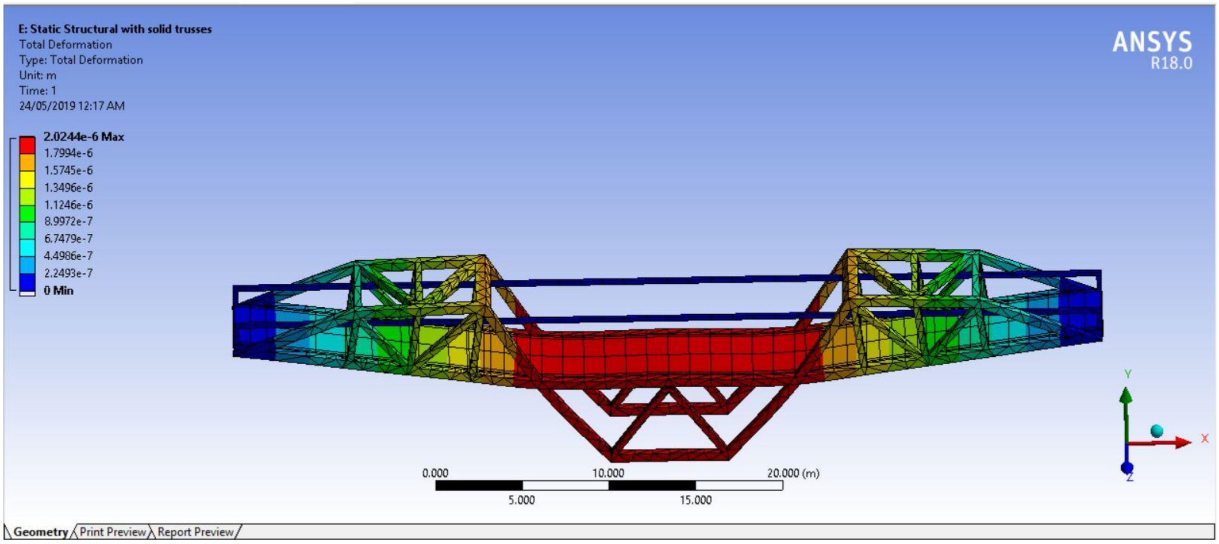


Figure 30 Greater overall extension but more evenly spread out

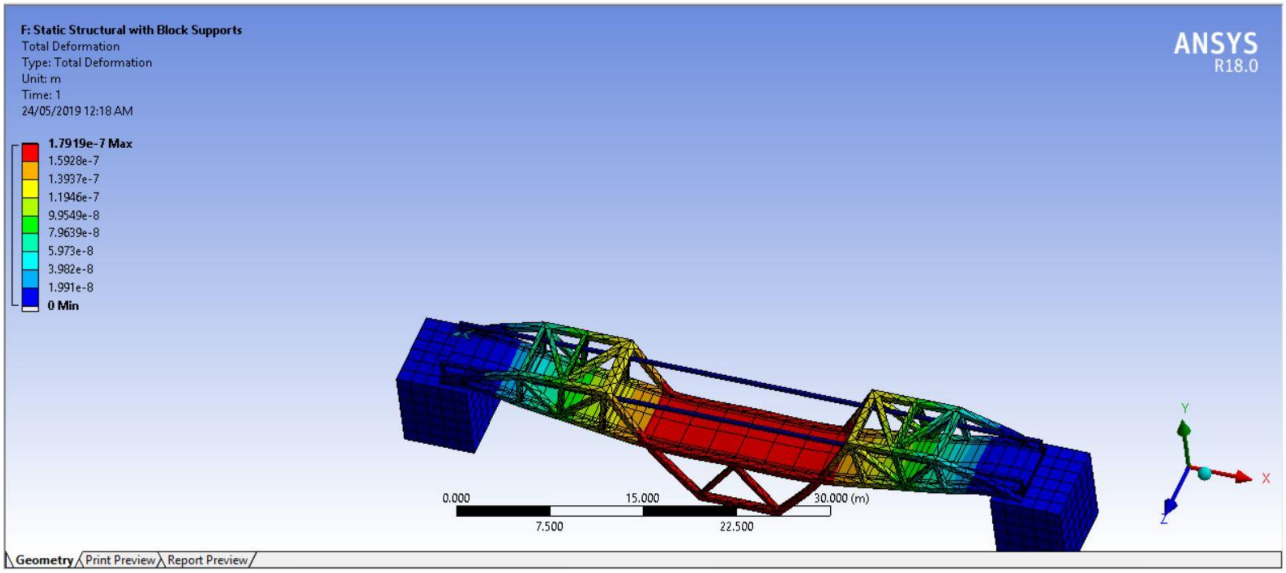


Figure 31 Decrease in total extension but concentrated in center and less at ends

Conclusion will be discussed in Chapter 5.

Chapter 4: Theoretical Calculations

Without Truss

Factor of safety n = 1.5

UTS of concrete σ_y = 40MPa

Maximum Allowable stress σ_{all}= σ_y / n = 25MPa

Find Maximum force

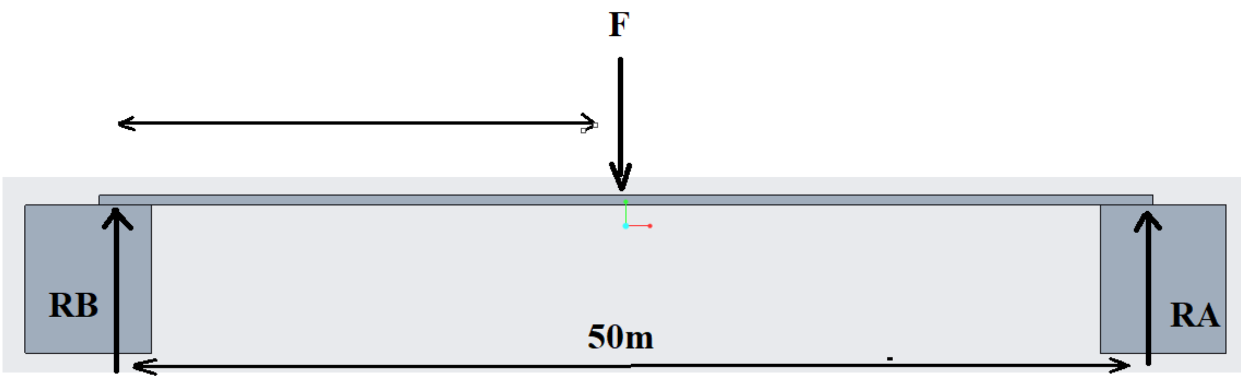


Figure 32 Solving for F without Truss

Statically determinant system

$$\sum Ma = 0$$

$$-Rb * 50 + F * 25 = 0$$

$$Rb = \frac{F}{2}$$

$$\sigma\text{all} = \frac{F}{A}$$

$$\mathrm{F} = \mathrm{A} * \sigma\text{all} = 2 * 0.5 * 50 * 25000000 = 10\text{MN}$$

We must also find Rb and ensure that check that it doesn't crosses its limit.

With Truss

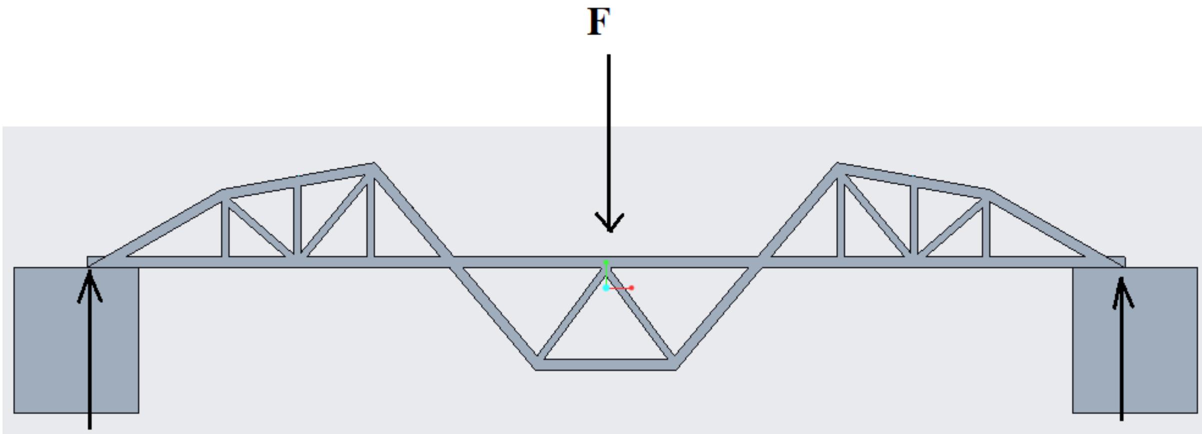


Figure 33 F with truss

F will be same as previous case but relation with Ra and Rb will be complex.

Case2:

$F = 10^6$ N

$A = 50 * 0.25 = 12.5m^2$

$\sigma = F/A = 10^6/12.25 = 0.8*10^6$ Pa $=0.8$MPa

but by ansys

$\sigma = 10^6$ Pa = 1MPa

Percentage error = 20%

Conclusion

Ansys is a very important tool for design and analysis of structure. With its help one can determine the most efficient design that can sustain maximum load without deformation. After comparing properties of our bridge designs, we have obtained the following results:

1. **Trusses with Extra Railings**

Best used for Bridges with Heavy Vehicles in all lanes as strain is evenly distributed

2. **Solid Trusses with Block Supports**

Best choice for bridges with fast traffic but for long lasting use as shape of bridge is kept linear despite forces and extension

3. **Maximum Force**

The maximum force for design of bridge depends on material used UTS and the structure of bridge.

References

1. https://en.wikipedia.org/wiki/Truss
2. https://www.youtube.com/watch?v=JtzArv2LQ7E
3. \https://en.wikipedia.org/wiki/Truss_bridge
4. https://www.sharcnet.ca/Software/Ansys/16.2.3/enus/help/wb_sim/ds_static_mechanical_analysis_type.html
5. https://en.wikipedia.org/wiki/Structural_analysis