Sudan University of Science and Technology

College of Engineering

School of Civil Engineering

Department of Structure

Analysis and Design of Superstructure Soba Bridge

A Research Submitted in Partial fulfillment for the Requirements of the degree of B. Sc. (Honors) in Civil Engineering

Prepared by:

1. Amal Galal Abdelateef Abdelgader Alnazeer

2. Mai Mustafa Amen Mustafa Frahat Omer

3. Wefag Mohammed Mohammed Ahmed Alawd

Supervisor:

Professor : Abdel Rahman El Zubair

October 2015

الأيـــة

قال تعالى :

بســـــــــــــــم الله الــــــــــــــــــرحمن الـــــــــــــــــرحيم

﴿وَالسَّـــمَاء بَنَيْنَاهَـــا بِأَيْـــدٍ وَإِنَّـــا لَمُوسِـــعُونَ﴾

،،،صدق الله العظيم ،،،

سورة الذاريات: [الذاريات: 47]

Dedication

We dedicate this humble effort. …

To who suckled me the love and tenderness…

To symbol of love and healing balm…

To the heart as pure in white…

…My mother's beloved…

❖❖❖❖❖❖

To who taught me tender without waiting…

To whom I carry his name proudly…

To big heart…

…My dear father…

❖❖❖❖❖❖

To the pure hearts and sinless souls…

To basils my life…

…My sisters & brothers…

❖❖❖❖❖❖

To all family & friends & teachers…

Project Group

Acknowledgement

We would like to thank:

❖ *Our wonderful supervisor:*

ProfessorAbdel Rahman El zubair

He was our Inspiration for doing our project. We will always remember him and appreciate very much his support.

❖*Special thanks are for their help:*

Eng. Faiga Mohammed Mohammed Ahmed Alawd

❖*Special thanks are for their help:*

Eng. Ahmed Abdel Rahman &WaledHussien

❖ To civil Engineering departments.

❖ To others provided much time and effort to see this project to completion.

Abstract

This thesis presents the structural analysis and design of the Soba Bridge [girder bridge] which connects Soba East to Soba West . The structure comprises twelve internal spans of 40.9m and two deck span of 13.2m (each deck span comprises five pre-stress girders) ,We Take the single span and one deck for analysis due to symmetrically. The analysis by finite element method consider the girder as frame element and pier as solid element by **MIDAS 2006** program .The design of superstructure according to American standard.

The study included a general introduction of bridge , types of Loads were applied and how to apply them according to the American standard **[AASHTO LRFD 2010]** , Analysis soft ware **MIDAS 2006** program , The modeling of Soba Bridge with the extract result of the analysis and design research , discussion these results then view a summary of this results and provide a general recommendation.

التجريد

هذا المشروع يقوم بالتحليل والتصميم الانشائي لكبري سوبا وهو عباره عن كبري ذو عارضات ، والذي يربط بين سوبا شرق وسوبا غرب .ويتكون من 12بحر بطول 40.9 متر وارضيتين بعرض 13.2متر(والارضيه تحتوي علي خمسة عارضات سابقة الاجهاد ذو النوع لاحق الشد) , تم تحليل بحر واحد وارضية واحدة تبعا للتماثل بواسطة البرنامج تم نمزجه العارضات كهياكل والدعامات الوسطيه كأعضاء مصمتة وصمم الانشاء العلوي للجسر حسب مواصفات دليل التصميم الامريكي .

الدراسه شملت مقدمة عامة عن الجسور والاحمال المطبقة علي الجسور حسب المدونة الأمريكية وبرنامج التحليل , واحتوت ايضا علي نمزجة جسر سوبا مع استخراج نتائج التحليل والتصميم وتم مناقشة النتائج المتحصل عليها ومن ثم تم الحصول علي خلاصة البحث والتقدم ببعض التوصيات.

Contents

List of Table

List of Figure

List of symbols

B_i	=	effective width
Y_{bg}	=	centeroid of bottom girder
Y_{tg}	=	centered of top girder
I	=	second moment of area
S_{ag}	=	modulus of elasticity
Y_{bc}	=	centered of bottom composite area
Y_{tc}	=	centered of top composite area
η_D	=	a factor relating to Ductility
η_R	=	a factor relating to Redundancy
η_{Ia}	=	factor relating to operational
K. L	=	number of lane
n_c	=	Transformed factor:
γ_c	=	concrete density
A_g	=	Area of girder
f_{pu}	=	Tensile strength
f_{pe}	=	yield strength
A_Φ	=	section area of strand
E_p	=	Elastic modulus of prestress tendon
F_t	=	tensile force

f_f = minimum prestress force

ΦM_n = Cross sectional bending strength

A_g = gross area of section

E_{ci} = modulus of elasticity of concrete at transfer

E_p = modulus of elasticity of prestressing tendons

e_m = average prestressing steel eccentricity at mid span

f_{pbt} = stresss in prestressing steel

β_1 = stress block factor a depth of the equivalent stress block

Δ_x = Timing displacement of dead load

V_n = nominal shear strength

P_r = factored axial resistance

S_c = section modulus for the extreme fiber of the composite section

M_g = mid span moment due to member self-weight

e = eccentricity of strand

A_{ps} = total area of strand

Δf_{pt} = prestress losses

Δf_{PES} = Initial losses {elastic strain }

f_{pi} = Initial stress reinforced strand

fi = Initial prestress forces

Δf_{pes} = elastic strain losses

Δf_{plT} = Timing losses

F_{pi} = Effective stress in transverse stage

F_{ti} = Tension stress at top flange

F_{bi} = Compressive stress at bottom flange

F_{pf} = Stress at gird rafter all losses

U = Fatigue limit state

M_{dg} = Dead load moment of girder

M_{ds} = Dead load moment of slab

M_{Dc} = Dead load moment of composite

Δ = all Permissible displacement for traffic

C = depth of compressive stress

V_c = Nominal strength of concrete for shear

b_f = Flange width of steel girder

S = Effective length of slab

M. D. L = Moment of dead load

M. L. L = Moment of live load

I = Impact factor

φ = resistance factor

D_C = dead load of structural components

D_W = dead load of wearing surfaces

M_u	=	Maximum positive & negative moment
A_s	=	area of slab
M_{cr}	=	the cracking moment
S_e	=	Effective length of the slab
S_{max}	=	Ultimate distance between reinforcement steel.
f_r	=	the modulus of rupture

Abbreviation

AASHTO	=	American Association of State Highway and Transportation Officials.
LRFD	=	Load Resistance Factor Design.
BS	=	British Standard.
MIDAS	=	Modeling, Integrated Design & Analysis Software.

Chapter 1
Introduction

1.1 Introduction

Soba Bridge will connect the East Soba area with the West Soba area over the Blue Nile. The structure will comprise twelve internal spans of 40.900 m between pier centerlines , and two 40. 150 m end span providing an overall length of 571.100 m between abutments bearing centerlines . The abutments two piers on each side will be situated wholly on the banks of the river , there mainder of the piers within the river . The bridge carries two carriage ways 10.5m wide and two foot was 1. 7 m wide each deck span comprises five precast post tensioned concrete beams .The overall width of the deck is 26.5 m.

The deck will be constructed using precast post tensioned concrete beams within situ concrete deck slab, All spans will simply supported but adjacent spans will be connected by link slabs across be connected by link slab across deck surface between expansion joints . The abutment and piers will be in situ reinforced concrete supported on bored and cast in place piles. Soba Bridge project will ease movement for transporting for all the Products , raw materials and all the vehicles. This will solve major congestion problem in 60 th street . The bridge will create a portal to a major over lapping point for the east area of the Nile. Soba Bridge will attract the investors for the availability of unpopulated

lands and will with stand. a major development in populating the area and the growth of the state in a major view.

1.2 Objectives

1. To know the history, types, and General components of the bridges.
2. To learn how to calculate the loads on bridges in general.
3. To understand the behavior of the pre-stress girder {precast-post tension} bridges and it′s advantages.
4. To learn how to analyze bridges using finite element based programs [MIDAS program].
5. To analyze Soba Bridge using MIDAS program.
6. To design Soba Bridge according to AASHTO (manual design)

1.3 Methodology of research

1. Studying references (Books), previews studies and research to know the bridge [Components, type, Analyze and design].
2. Collection the necessary data about the bridge from the ministry of urban planning for obtaining the bridge specification, and visiting the Soba Bridge site for acknowledging the environment and the topography of the area.
3. Studying the finite element method and MIDAS program.
4. Developing of Soba Bridge by finite element model using MIDAS program.
5. Defining the different types of loads acting on the Soba Bridge.
6. Calculating the loads accordance to AASHTO LRFD.
7. Analyzing the Soba Bridge using the MIDAS program.
8. Designing the Superstructure of Soba Bridge accordance to AASHTO LRFD.
9. Discussion of the results obtained from the program and drawing conclusions and recommendations.

1.4 Research Outline

1. Chapter one includes a general introduction, the objectives , the methodology of research and the outlines of thesis.
2. Chapter two contains the literature review of the bridges generally and loading on bridges.
3. Chapter three includes general introduction about pre-stress concrete girder bridge.
4. Chapter four present general introduction of finite element method and the computer program MIDAS.
5. Chapter five present the analysis of superstructure and pier and discussion of results.
6. Chapter six present the design of super structure and discussion of results.
7. Chapter seven present the conclusions and recommendations .

Chapter 2
Literature review

2.1 Definition of bridge

Abridge is a structure built to span a valley, road, body of water, or other physical obstacle , for the purpose of providing passage over the obstacle. Designs of bridges vary depending on the nature of the terrain and the function of the bridge and where it is constructed.

2.2 Importance of bridge

Bridges have always been an important part of our environment. They have been major subjects of literature and art, both ancient and modern. Wars have been fought over bridges and in many cases the capture of strategic structure has had a pronounced effect on the final outcome of the war, bridges have been the center of village or city life.

2.3 Historical review

The history of development of bridge construction is closely linked with the history of human civilization. Bridges began with the “trial and error process". People have always been interested in transporting themselves and their goods from one place to another , so the rivers, mountains and valley are considered as a basic problem facing the people in their transportation and movement from one place to another, so in the beginning they thought to pass that obstruction and move away. They used a rope or swimming and finally they reached to use a bridge that was mode of simple materials like rock , stone , timber and other materials was available at that time.

The first bridges were made by nature itself as simple as a log fallen across a stream or stones in the river. The suspension bridge in nature by swinging vine, utilized by animals and people to pass from one tree to another over a stream the first bridges made by humans were probably spans of cut wooden logs or planks and eventually stones, using a simple support and crossbeam arrangement .The construction of long span bridges came from a natural scene of an edge cantilever on the mountains

The development of bridge engineering was first attended by the Romans. They were considered the only ones considering bridge engineering as a science . They build their bridges mainly from wood ,bricks , and concrete. The roman used cofferdams. They drove the piles into river bed the intended site of the piers, and lined the rams of piles with clay to make them watertight. Then the interior of the cofferdam could be pumped out and concrete poured in to form the pier. In case of deep water a concrete blocks where dropped to form an artificial floors. Looking back about 2000 years ago, in light of the modern knowledge of bridge construction, it is quite surprising to criticize work of ancient Roman science of bridge construction and engineers. The Roman built exclusively semi-circular arches. The Roman engineers, despite of their skill in using cofferdams, did not succeed completely in mastering the bottom of the river. However, the roman built bridges of monumental strength and striking beauty. Their engineer’s triumphs long out lasted their own empires.

2.4 Classification of bridges

May be classified in many ways, as below:-

1. According to function as high way , railway, aqueduct, viaduct etc .

2. According to the material of construction of superstructure as timber , masonry, iron, steel, reinforced concrete, prestresssed concrete etc.

3. According to the form or type of superstructure as slab, beam, truss , arch or suspension bridge.

4. According to the interior span relations as simple, continuous or cantilever bridge.

5. According to the position of the bridge floor relative to the superstructure , as deck , though , half-through or suspended bridge .

6. According to the method of connections of the different parts of the superstructure , particularly for steel construction pin connection etc.

7. According to the method of clearance for navigation as high-level, movable-bascule, movable-swing or transporter bridge.

8. According to span length as culvert (less then 8m), miner bridge (8to30m), major bridge (above 30m) or long span bridge (above120m).

9. According to degree of redundancy as determinate or in determinate bridge.

10. According to the anticipated type of service and duration of use as, permanent, temporary, military (pontoon, bailey) bridge.

2.5 Types of bridges

2.5.1 Slab on stringer

Also know beam bridge, beam bridge is the simplest and most in expensive common kind of bridge. Beam bridge consists of a concrete slab resting on a set of horizontal beams, which are connected together by diaphragms supported at each end by piers to form a frame shown in figure (2-1). It is used to span short distances 250 feet , this doesn't mean beam bridges aren’t used to cross great distances it only means creating continuous span.

Figure 2-1: Slab on stringers

2.5.2 One way slab

For a very short span less than 30 ft a one way concrete slab or circular voids in the slab are sometimes used to reduce the dead load supported on either end by small abutments is an economical structure. Shown in figure (2-2).

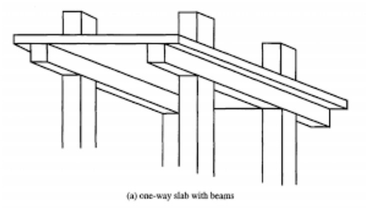


Figure 2-2: One way slab

2.5.3 Arch bridge

Arch bridges are one of the oldest types of bridges and have great natural strength. Instead of pushing straight down, the weight of an arch bridge is carried outward along the curve of the arch to the supports at each end shown in figure (2-3). Arch bridges must be made of materials that are strong under compression. Modern arch bridges span between 200-800feet. The most common material used to construction is stone, concrete and steel.

Figure 2-3: Arch bridge

2.5.4 Truss bridges

Truss bar members are theoretically considered to be connected with pins at their ends to form triangles shown in figure (2-4). Each member resists an axial force, either in compression or tension. The main difference is that trusses are less heavy than beams.

Figure 2-4: Truss bridge

2.5.5 Cable stayed bridges

The girders are supported by highly strengthened cables which stem directly from the tower, the cables are attached at different heights along the tower, running parallel to one other, which alone bear the load shown in figure (2-5). Used for medium length spans between 500 and 2,800 ft. Cable stayed bridge have gained great popularity in recent years because of their great beauty, economy and faster to build.

Figure 2-5: Cable stayed bridge

2.5.6 Suspension bridges

Suspension bridge as one of the consummate marvels of civil engineering. Suspension bridge suspends the road way from huge main cables, which extend from one end of the bridge to the other. These cables rest on top of high towers and are secured at each end by anchorages shown in figure (2-6). It is can span distances from 2,000 to 7,000 feet, it is more expensive and longer than any other kind of bridge.

Figure 2-6: Suspension bridges

2.5.7 Box girder bridges

Box girders are a particular form of plate girder , with two webs joined by common top and bottom flanges, which resists not only bending and shear but also torsion effectively, shown in figure (2-7). This makes the box girder the ideal choice for bridges with any significant curve in them. It is able to span greater distances and are often used for longer spans. If using a concrete top flange then the box girder is called open top, box girders with a steel top flange called an orthotropic deck.

Figure 2-7: Box girder bridge

2.5.8 Fixed or movable bridges

Most bridges are fixed bridges, meaning they have no moving parts and stay in one place until they fail or are demolished. The movable bridges are designed to move out of the way of boats or other kinds of traffic, which would otherwise be too tall to fit. These are generally electrically powered shown in figure (2-8).

Figure 2-8: Movable bridges

2.5.9 Temporary bridges

Bridges made from modular basic components that can be moved by medium or light machinery shown in figure (2-9). They are usually used in military engineering or in circumstances when fixed bridges are repaired.

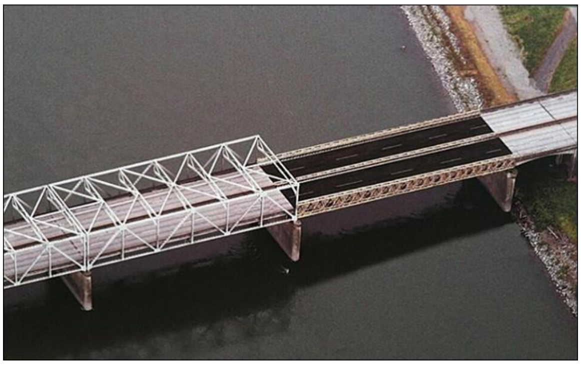


Figure 2-9: Temporary bridges

2.5.10 Floating bridges

They are cost effective solution for crossing large bodies of the water with unusual depth and very soft bottom where conventional piers are impractical shown in figure (2-10).

Figure 2-10: Floating bridge

2.5.11 Rail road bridges

The construction of rail road bridges are mainly based on existing rail road and routs because new rail road routs are not common to construct these days shown in figure (2-11). The rail road industry extends life of existing bridges as long as economically justified.

Figure 2-11:Rail road bridge

2.5.12 Cantilever bridges

They are built using cantilevers horizontal beams supported on only one end shown in figure (2-12). The cantilever bridges are constructed using much the same materials and techniques as beam bridges.

Figure 2-12: Cantilever bridge

2.6 General components of bridges

Abridge structure is divided for two main parts:

1. The superstructure comprises all the components of a bridge above the supports; it provides horizontal spans between supports.

2. The substructure consists of all elements required to support the superstructure and overpass road way, it supports the horizontal spans, elevating above the ground surface. The basic components of bridges shown in figure (2-13).

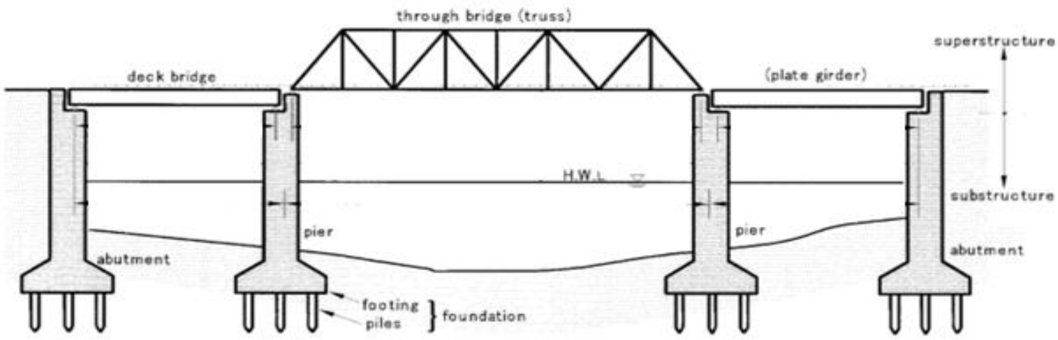


Figure 2-13: Main parts of bridge

2.6.1 Superstructure

The basic superstructure components shown in figure (2-14) consist of the following:-

2.6.1.1 Wearing surface

The wearing surface is that portion of the deck cross section ,which resists traffic wear. In some instances this is a separate layer made of bituminous material usually varies in thickness from 2 to 4 in, while in some other cases it is an integral part of concrete deck thickness of integral wearing surface is 0.5 to 2 in.

2.6.1.2 Deck

The deck is the physical extension of the road way across the obstruction to be bridged. The deck may be reinforced concrete slab or stiffened steel plate rests directly on the primary member, a small fillet or haunch can be placed between the deck slab and the top flange of the stringer. The primary function for the haunch is to adjust the geometry between the stringer and the finished deck. The main function of the deck is to distribute loads transversely along the bridge cross section.

2.6.1.3 Primary members

The primary members consist of rolled wide flange beams, primary members such as stringers (girders) or box girder can be constructed out of steel or prestresssed concrete. In some instances, the outside or fascia primary members possess a larger depth and may have cover plate welded to the bottom of them to carry heavier loads. The main function of the primary members to resist flexure and shear.

2.6.1.4 Secondary members

Secondary members are bracing between primary members, secondary members such as diaphragms used between rolled section stringers. The main function of secondary members resist cross sectional deformation of the superstructure frame, help distribute part of the vertical load between stringers and used for the stability of the structure during construction.

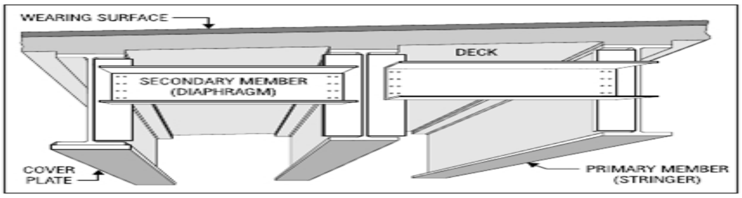


Figure 2-14: Components of Superstructure

2.6.2 Substructure

The basic substructure components shown in figure (2-15)consist of the following:-

2.6.2.1 Abutments

Abutments are earth – retaining structures, which support the superstructure and over pass road way at the beginning and end of a bridge. Like a retaining wall, the abutments resist the longitudinal forces of the earth underneath the over pass road way.

2.6.2.2 Piers

Piers are structures, which support the superstructure at intermediate points between the end supports. Piers are one of the most visible components of a high way bridge.

2.6.2.3 Bearings

Bearings are mechanical systems, which transmit the vertical and horizontal loads of the superstructure to the substructure, and accommodate movement between the superstructure and the substructure bearings allowing both rotation and longitudinal translation are called expansion bearings, and those, which allow rotation only, are called fixed bearings.

2.6.2.4 Pedestals

A pedestal is a short column on an abutment or pier under a bearing, which directly supports a superstructure primary member. Normally pedestals are designed with the different heights to obtain the required bearing elevations.

2.6.2.5 Back wall

A back wall, sometimes called the stem, it is the primary component of the abutment acting as a retaining structure at each approach.

2.6.2.6 Wing wall

A wing wall is a side wall to the abutment back wall or the stem designed to assist in confining earth behind the abutment.

2.6.2.7 Footing

Footings transfer loads from the substructure to the subsoil or piles. A footing supported by soil without piles is called a spread footing, a footing supported by piles is known as a pile cap.

2.6.2.8 Piles

When the soil under a footing cannot provide adequate support for the substructure, support is obtained through the use of piles, which extend down from footing to a stronger the soil layer or to bed rock. There is a variety of types of piles ranging from concrete, which is cast in place or precast, to steel H-sections driven to sound rock.

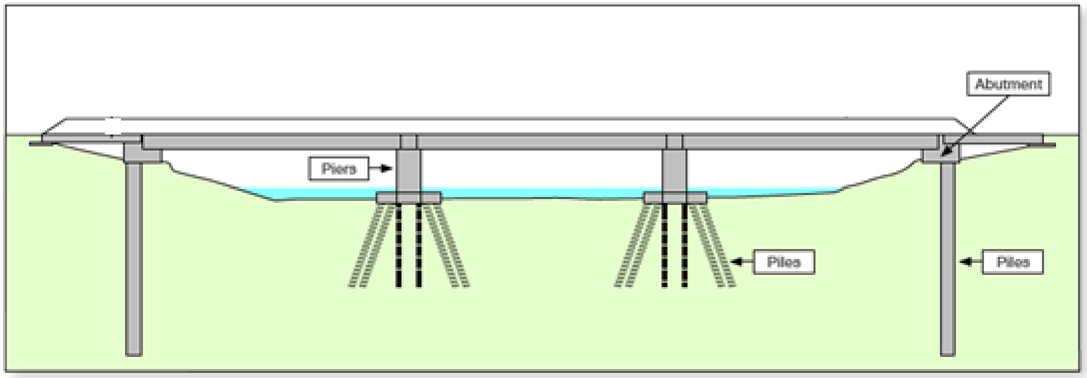


Figure 2-15: Component of Substructure

2.7 Bridge loading

The design of the bridge is based on a set of loading conditions which the component or element must with stand. The engineer must consider all the loads that are expected to be applied to the bridge during its service life. Design loads include:-

1. The self-weight of the bridge.
2. The weight of pedestrians and kerbs.
3. Vehicle weight.
4. Centrifugal force due to braking.
5. Impact loads (dynamic vertical loads) influenced by such factors as road roughness and vehicle suspension characteristics.
6. Longitudinal forces.
7. Loads due to deformation such as support settlement, environmental
8. Loads such as rain and snow and thermal loads and axial deformation.
9. Earth pressure.
10. Stream loads.
11. Wind loads.

12. Earthquake loads.

2.7.1 Load definition according to AASHTO LRFD

2.7.1.1 Gravity dead loads

Loads may be divided into two broad categories applied in a downward direction:

2.7.1.1.1 Permanent loads

The permanent loads remain on the bridge for an extended period, usually for the entire service life.
Permanent loads include the following:-

- Dead load of structural and nonstructural components (DC):

Structural components refer to self weigh to fall elements are part of the load resistance system. Nonstructural attachments refer to such items as curbs, parapets, barrier rails, signs, illuminators and rails, the weight of such items can be estimated by using the unit weight of material combined with the geometry.

- Dead load of wearing surface and utilities (DW):

Dead load of wearing surface is estimated by taking the unit weight times the thickness of the surface and take in consider future overlays.

-Dead load of earth fill (EV):

Dead load of earth fill must be considered for buried structures as culverts, it is determined by multiplying the unit weight times the depth of materials.

-Earth loads:

Earth loads include the earth pressure load (EH), earth surcharge Load (ES) and down – drag (DD). Earth loads are more difficult due to the greater variability involved, where variability's are greater, higher load factors are used for maximum load effects and lower factors are used for minimum load effects.

2.7.1.1.2 Transient loads

Transient loads apply on the loads change with time, several directions and locations. Transient loads typically include gravity loads due to vehicular, railway and pedestrian traffic as well as lateral loads such as those due to environmental loads include water flow, ice flow, wind flow and earthquakes. In addition loads due to deformation such as support settlement, thermal loads and axial deformation due to creep and shrinkage.

2.7.1.2 Live Load

2.7.1.2.1 Gravity loads

2.7.1.2.1.1 Vehicular Live Load

There are two ways for loading Vehicular on roadways of bridge:

2.7.1.2.1.1.1 Equivalent lane loads:

Consist Equivalent lane load from two parts as shown in figure (2-16):

Uniform distributed load per feet for Equivalent lane

Concentrated load at mid lane

2.7.1.2.1.1.2 Standard trucks

There are two types for truck

1. H-loading truck

Represent light truck loads, truck consist from two axes, there are three trucks H-10, H-15, and H-20,as shown in figure(2-17)

2. H_S-loading truck

Represent heavy truck loads, truck consist from two axes and trailer, there are two trucks H_S-15, and H_S-20,as shown in figure(2-18).

-Vehicular live loading on the roadways of bridges or incidental structures, consist of a combination of the:

❖Design truck:

The weights and spacing's of axles and wheels for the design truck shall be as specified in figure (2-17) and figure (2-18) .

❖Design tandem:

The design tandem shall consist of a pair of 25.0 kip axles spaced 4.0 ft a part. The transverse spacing of wheels shall be taken as 6.0 ft.

❖Design lane load:

The design lane load shall consist of a load of 0.64KLF uniformly distributed in the longitudinal direction. Transversely, the design lane load shall be assumed to be uniformly distributed over a 10.0 ft width as shown in figure (2-19). The force effects from the design lane load shall not be subject to a dynamic load allowance.

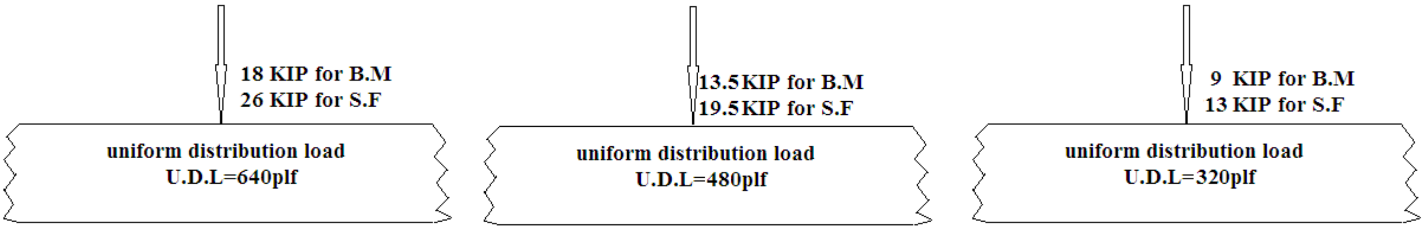


Figure 2-16: Equivalent lane loads

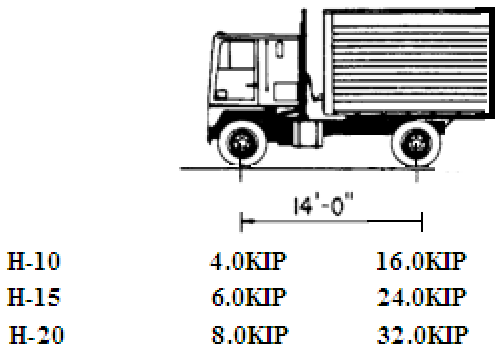


Figure 2-17: Standard trucks ; H-loading

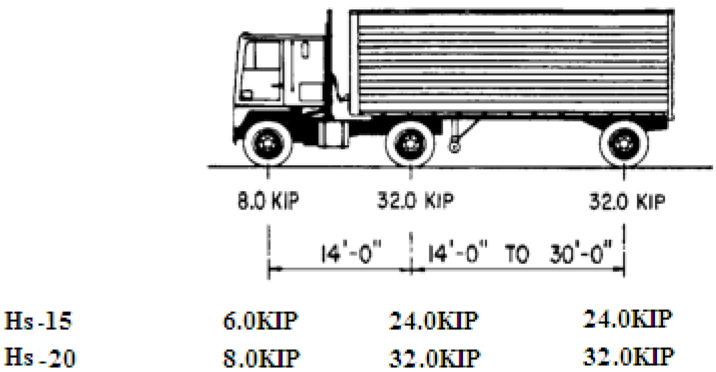


Figure 2-18: Standard trucks ; Hs-loading

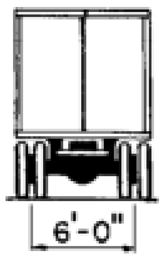


Figure 2-19: Design lane load

2.7.1.2.1.2 Pedestrian loads

A pedestrian load of 0.075 KSF shall be applied to all sidewalks wider than2.0 ft and considered simultaneously with the vehicular design live load in the vehicle lane. Where vehicles can mount the side walk , sidewalk pedestrian load shall not be considered concurrently as shown table (2-1) .

Table 2-1 Pedestrian load

Span length	Loading
≤ 25/ (7. 62m)	85psf (4. 1kn/m^2)
>25/ - 100/ (3. 5m)	60psf (2. 87kn/m^2)
>100/	P = (30 + $\frac{3000}{L}$)*($\frac{55-W}{50}$) ≤ 60 psf

2.7.1.2.2 Dynamic load allowance

Vehicular live loads are assigned a "dynamic load allowance "load factor of 1.75 at deck joints, 1. 15 for all other components in the fatigue and fracture limit state, and1.33 for all other components and limit state. This factor accounts for hammering when riding surface discontinuities exist, and long undulations when the settlement or resonant excitation.

2.7.1.3 Impact load

When the vehicle moving across of a bridge, a normal rate of speed produces greater stress than if the vehicle is in static condition, and for the computing the dynamic effect the AASHTO specification gives the equation for determining the impact factor:

$$I = \frac{50}{L + 125} \leq 30\%$$

2.7.1.4 Longitudinal forces

When a vehicle brakes or accelerates longitudinal forces are transmitted from it is wheels to the deck of the bridge. The magnitude of the longitudinal forces depends on the amount of acceleration or deceleration. The maximum longitudinal force from sudden braking of the vehicle, the magnitude of which is dependent on it is weight, it's velocity at the instant of braking, and the time it takes to come to a complete stop. This force takes 5% from live load without impact effect ,usually it is taken according AASHTO LRFD 4. 1 kips (18. 2KN)for H_s20 as shown figure (2-20) .

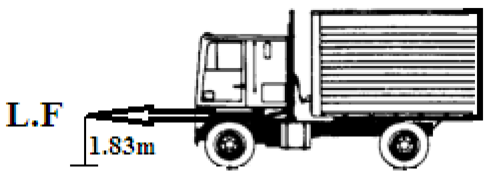


Figure 2-20: Longitudinal load

2.7.1.5 Centrifugal forces

Where a road bridge is located on a curve, the effects of centrifugal forces due to movement of vehicles should be taken into account. The centrifugal factor taken from live load without impact effect is given by equation:

$$C = \frac{6.6ws^2}{R}$$

2.7.1.6 Wind Load

2.7.1.6.1 Wind pressure on structure

All bridges structure should be designed to against the wind forces; there forces are considered to act horizontally and in a direction as to cause the maximum stresses in the member under consideration. Wind load shall be assumed to be uniformly distributed on the area exposed to the wind. The exposed are a shall be the sum of areas of all components, including floor system and railing, as seen in elevation taken perpendicular to the assumed wind direction. This direction shall be varied to determine the extreme force effect in the structure or in its components. Areas that do not contribute to the extreme force effect under consideration may be neglected in the analysis.

2.7.1.6.2 Wind pressure on vehicles

When vehicles are present, the design wind pressure shall be applied to both structure and vehicles. Wind pressure on vehicle shall be represented by an interruptible, moving force of 0. 4KLF acting normal and 6.0ft above, the road way and shall be transmitted to the structure.

Chapter 3
Introduction about Girder Bridge and MIDAS program

3.1 Girder Bridge

3.1.1 Introduction of Girder Bridge

Girder bridge is the simplest and most in expensive common kind of bridge. It is used to span short distances 250 feet, this doesn't mean beam bridges aren’t used to cross great distances it only means creating continuous span. The beam itself must be strong so that it does not bend under its own weight and the added weight of crossing traffic.

When a load pushes down on the beam, the beam's top edge is pushed together (compression) while the bottom edge is stretched (tension). Pre-stressed concrete is an ideal material for beam bridge construction ;the concrete with stands the forces of compression well and the steel rods imbedded within resist the forces of tension. Pre-stressed concrete also tends to be one of the least expensive materials in construction, but even the best materials can't compensate for the beam bridge's biggest limitation: its length.

3.1.2 Compare between I-beam and box-girder Bridge

The design, fabrication, build and works of box girders is more difficult than that of I beam, but the bridge contains any curves, the beams become subject to twisting forces, also known as torque. The added second web in a box girder adds stability a decreases’ resistance to twisting forces. This makes the box girder the ideal choice for bridges with any significant curve in them . Box girders ,being more stable are also able to span greater distances and are often used for longer spans, where I-beams would not be sufficiently strong or stable.

3.1.3 Components of Girder Bridge

Girder consists of a concrete slab resting on a set of horizontal beams, which are connected together by diaphragms supported at each end by piers to form a frame as shown in figure (3-1).

1-Deck and over pass

2-Stringer

3-Bearing

4-Pedestal

5-Footing

6-Piles

7-Underpass

8-Embankment

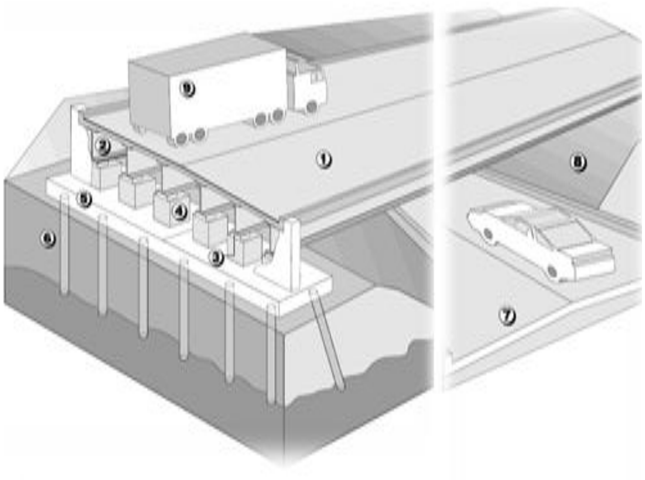


Figure 3-1: Components of girder bridge

3.1.4 Pre-stressed concrete

3.1.4.1 History

The first use of post - tensioned concrete was on the walnut lane bridge in Philadelphia in 1949 tensioned with the European magnel system. The first post tensioning in US . Building construction was in the mid- to late-1950s in buildings using the lift-slab construction method.

3.1.4.2 Definition of pre-stressing

Pre -stressing is defined as a method of applying pre-compression to control the stresses resulting due to external loads below the neutral axis of the beam tension developed due to external load which is more than the permissible limits of the plain concrete.

3.1.4.3. Compare between pre-stressed concrete &reinforced concrete

The pre-stressed concrete act slight arch notice able due to energy is stored in the unit by the action of the highly tensioned steel which places a high compression in the lower portion of the member, an upward force is there by created which in effect relieves the beam of having to carry its own weight. The ordinary concrete under own weight, the bottom of the beam will develop hairline cracks as shown in figure (3-2)

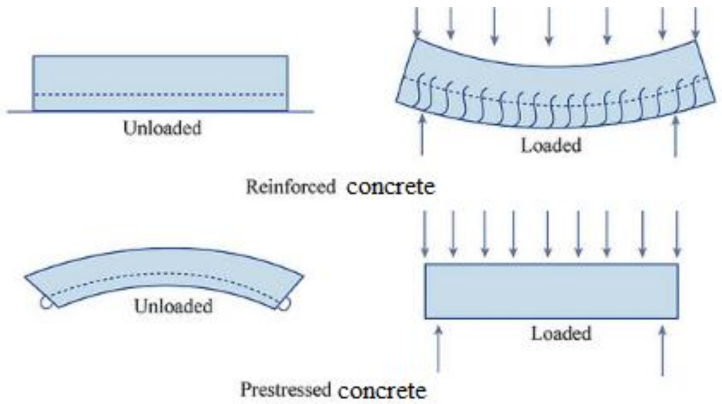


Figure 3-2: Compare between pre-stressed concrete and reinforced concrete

3.1.4.4 Materials for pre-stressed

<u>3.1.4.4.1 Concrete</u>

Pre-stress concrete requires concrete, which has a high compressive strength reasonably early age with comparatively higher tensile strength than ordinary concrete. The concrete for members shall be air – entrained concrete composed of Portland cement, fine and coarse aggregates, admixtures and water.

3.1.4.4.2 Steel

►High strength steel contains:

0. 7 to 0. 8% carbons

0. 6 % manganese

0. 1 % silica

►Forms of pre-stressing steel:

3.1.4.4.3 Wire

Pre-stressing wire is a single unit made of steel.

3.1.4.4 .4 Strands

Two, three or seven wires to form a pre-stressing strand.

3.1.4.4.5 Tendon

A group of strands or wires to form a pre-stressing tendon.

3.1.4.4.6 Cable

A group of tendons form a pre-stressing cable.

3.1.4.4.7 Bars

A tendon can be made up of a single steel bar. The diameter of a bar is much larger than that of a wire.

3.1.4.5 Types of pre-stressing systems

3.1.4.5.1 Pre-tensioning system

In pre-tensioning systems, the tendons are first tensioned between rigid anchor-blocks cast on the ground or in a column or unit – mould type's pre - tensioning bed, prior to the casting of concrete in the mould.

The tendons comprising individual wires or strands are stretched with constant eccentricity or a variable eccentricity with tendon anchorage at one end and jacks at the other. With the forms in place, the concrete is cast around the stressed tendon as shown in figure 3-3.

.

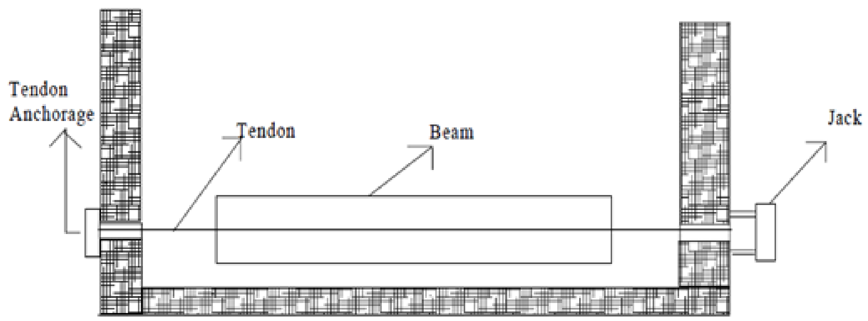


Figure 3-3: Pre-tensioning system

3.1.4.5.2 Post-tensioned system

In post-tensioning concrete unit are first cast by incorporating ducts or grooves to house the tendons. When the concrete attains sufficient strength, the high – tensile wires are tensioned by means of jack bearing on the end of the face of the member and anchored by wedge or nuts. The post-tensioning concrete divide to bonded post-tensioned concrete when there is adequate bond between the pre - stressing tendon and concrete. Also UN bonded post-tensioned concrete when there is no bond between the pre - stressing tendon and concrete

❖Post-tensioned beam construction: (see figure 3-4)

Stage1: Cable ducts and reinforcement are positioned in the beam mould. The ducts are usually raised towards the neutral axis at the ends to Reduce the eccentricity of the stressing force.

Stage2: Concrete is cast into the beam mould and allowed to cure to the required Initial strength.

Stage 3: Tendons are threaded through the cable ducts and tensioned to about 70% of their ultimate strength. The diagram above indicates jacking from both ends of the beam.

Stage 4:

A) Wedges are inserted into the end anchorages,
B) The tensioning force on the tendons is released. Grout is then pumped into the ducts to protect the tendons.

Stage 5: Loss of pre-stress due to elastic deformation of the concrete and relaxation of the steel need to be considered. Further loss of prestress will also occur due to shrinkage and creep of the concrete; as these are time related then the effects will need to be considered both at short term and long term.

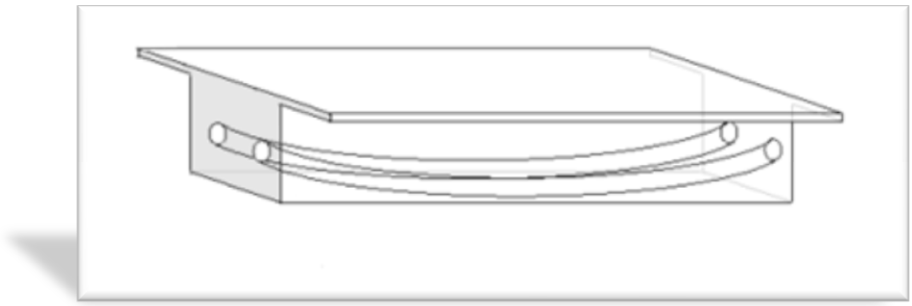


Figure 3-4: Pos-tensioning system

3.1.4.6 Advantages of pre-stressed concrete

1. The use of high strength concrete and steel in pre-stressed members results in lighter and slender members than is possible with R C members.

2. In fully pre-stressed members the member is free from tensile stress under working loads, thus whole of the section is effective.

3. Pre – stressed concrete member posse's better resistance to shear forces due to effect of compressive stresses presence or eccentric cable profile.

4. Use of high strength concrete and freedom from cracks, contribute to improve durability under aggressive environmental conditions.

5. Long span structures are possible so that saving in weight is significant & thus it will be economic.

6. Factory products are possible.

7. Pre-stressed concrete structure deflects appreciably before ultimate failure, thus giving ample warning before collapse.

8. Fatigue strength is better due to small variations in pre - stressing steel, recommended to dynamically loaded structures.

3.1.4.7 Disadvantages of pre-stressed concrete

1. The availability of experienced engineers and builders is scanty.
2. Initial equipment cost is very high.
3. Pre - stressed concrete sections are less fire resistant.

3.1.4.8 Types of losses in pre-stress

3.1.4.8.1 Pre-tensioning

1. Elastic deformation of concrete (5-10) %.
2. Relaxation of stress in steel (8-10) %.
3. Shrinkage of concrete (10-20) %.
4. Creep of concrete.

3.1.4.8.2 Post-tensioning

1. No loss due to elastic deformation if all wires are simultaneously tensioned and if the wires are successively tensioned, there will be loss of pre - stress due to elastic deformation of concrete (2-3) %.
2. Relaxation of stress in steel (8-10) %
3. Shrinkage of concrete (10-20) %
4. Creep of concrete
5. Friction (1-2) %

3.1.4.9 Stages of loading

The analysis of pre – stressed members can be different for the different stages of loading as shown in figure (3.5).
The stages of loading are as follows:

Initial: it can be subdivided into two stages:

1. During tensioning of steel.
2. At transfer of pre - stress to concrete.

Intermediate: This includes the loads during transportation of the pre–stressed member

Final: it can be subdivided into two stages:

1. at service, during operation.
2. at ultimate, during extreme events.

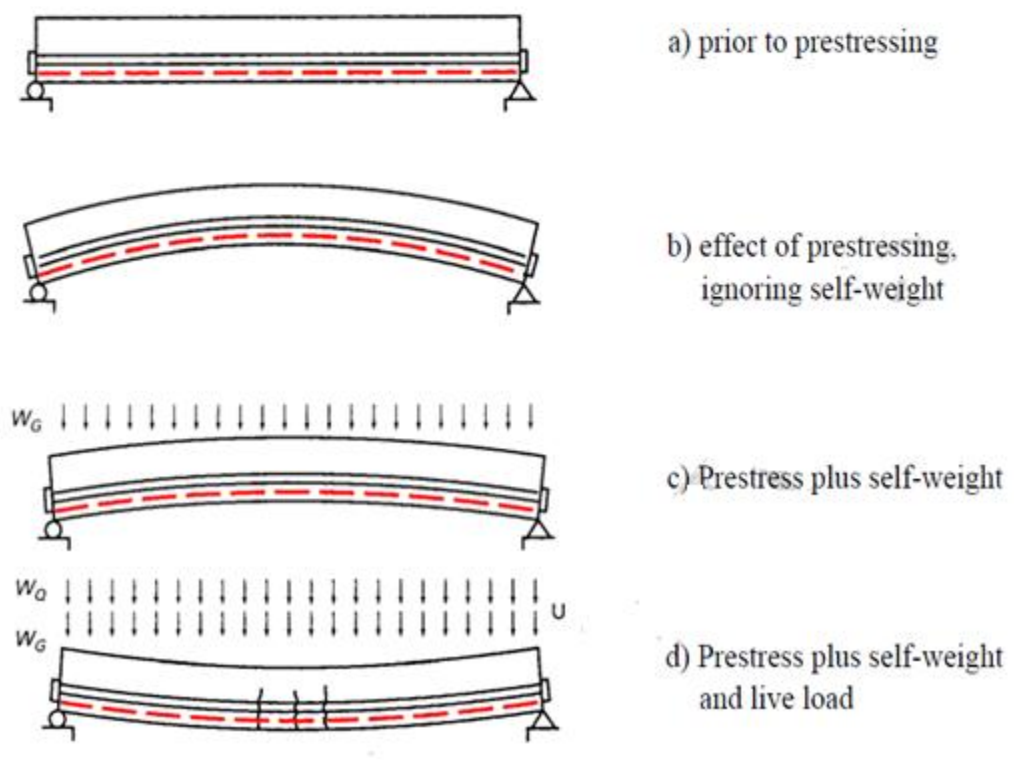


Figure 3-5: Effect of load during pre-stressing

3.2 Introduction about MIDAS

3.2.1 Finite Element Method

3.2.1.1 Introduction of Numerical Methods

There are practical engineering problems, which we cannot obtain exact solution .This inability to obtain an exact solution may be attributed to either the complex nature of governing differential equation or the difficulties that arise from dealing with the boundary condition to deal with such problem resort to numerical approximations in constant to analytical solution which show the exact behavior of a system at any point within the system. The complexity of normal integration in analysis methods led to find a new method using numerical analysis to solve the complex integration and to make the procedure of solution easier .The most important method of numerical methods is Finite Element Method.

3.2.1.2. Introduction of Finite Element Method

The finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. Applications range from deformation and stress analysis of Automotive, aircraft, building, and bridge structures to field analysis of heat flux, fluid flow, magnetic flux, seepage, and other flow problems.

With advances in computer technology and CAD system, complex problems can be modeled with relative ease. Several alternative configurations can be tested on a computer before the first prototype is built. All of this suggests that we need to keep pace with these developments by understanding the basic theory, modeling techniques, and computational aspects of the finite element method.

In this method of analysis, a complex region defining a continuum is discredited into simple geometric shapes called finite elements .The material properties and governing relationships are considered over these elements and expressed in terms of unknown values at element corners. An assembly process, duly considering the loading and constraints ,results in a set of equations .Solution of these equations gives us approximate behavior of the continuum.

3.2.1.3 Definition of Finite Element Method

Finite Element Method is Numerical Method for solving deferential equations (D. E. ^{S}) is replaced by their integral form, an application of the R-R Method or Galerkin's Method.

3.2.1.4 Procedure of Finite Element Method

1. Nodes are points on the structure at which displacements and rotations are to be found or prescribed.
2. Element is a small domain on which we can solve the boundary value problem in terms of the displacements and forces of the nodes on the element.
3. Mesh is the discrete representation of the structure geometry by elements is assumed to be connected at the nodes only.
4. Discretization is the process of creating a mesh (discrete entities).
5. Interpolation function is a kinematically admissible displacement function defined on an element that can be used for interpolating displacement values between the nodes.
6. Evaluation of individual element properties. G. Element stiffness matrix and Nodal load vector.

7.Model is the mesh, boundary conditions, loads, and material Properties representing the actual structure.

8. Global stiffness matrix is an assembly of element stiffness matrix that relates the displacements of the nodes on the mesh to apply external.

9. Calculations of any required gradients are then completed using element matrices (e. G. Stress, strain …. Etc)

3.2.1.5 Modeling Considerations

An establishment of appropriate finite element model for an actual practical problem depends to a large degree on the following factors: -

1.Understanding of the physical problem includes a qualitative knowledge of the structural response to be predicted, knowledge of the basic principles of mechanics and good understanding of the finite element procedures available for analysis.

2. Discretization of the domain into finite elements is the first step in the finite element method. This is equivalent to replacing the domain having an infinite number of degrees of freedom by a system having finite number of degrees of freedom.

3.The shape, size, number and configuration of elements have to be chosen carefully so that the original body or domain is simulated as closely as possible without increasing the computational effort needed for the solution.

4.They various considerations taken in the discretization process are Type of elements, Size of elements, Location of nodes , Number of elements, Simplifications afforded by the physical configuration of the body , Finite representation of in finite bodies , Node numbering scheme, Automatic node generation. After meshing of the body it is necessary to add them at serial properties, external loads, and apply the boundary conditions. Before start of the problem, only parameters of the calculation regime should be added to the input file.

3.2.1.6 Finite Element Program Packages

The general applicability of the finite element method makes it power full and universal tool for a wide range of problems .Hence, a number of computer program packages have been developed for the solution of variety of structural and solid mechanics problems .Among more widely used packages are ANSYS, NASTRAN, ADINA, LS-DYNA, MARC,SAP, COSMOS, ABAQUS, and NISA. Each finite element program package consists from three parts:

1. Programs for preparation and control of the initial data.
2. Programs for solution of the finite element problem.
3. Programs for processing of the results.

3. 2.2 MIDAS program

3.2.2.1 Definition

MIDAS is a program for structural analysis and design in the civil engineering domains. The program has been developed so that structural analysis and design can be accurately completed within the shortest possible time. It is the ultimate Integrated Civil Engineering Solution for designing bridges and general civil structures .It retains construction stage analysis capabilities for prestresssed / post-tensioned concrete, suspension, cable stayed, specialty and conventional bridges and heat of hydration.

3.2.2.2 Uses of MIDAS program

1. 3D FED package.
2. Innovative user interface.
3. Optimal solutions for all kind of bridges.
4. Post tension bridge design.
5. Cable bridge design.
6. Nonlinear analysis.

❖**In this project we are using MIDAS for:**

Post tension bridge design.

3.2.2.3 Types of Analysis in MIDAS Program

1. Moving load Analysis
2. Construction stage Analysis

3.2.2.4 Element types in MIDAS Program

See table (4-1)

Table 3-1 Element types in MIDAS Program

1. D – Element:	2. 2-D Element:	3. 3-D Element:
Truss	plate -3Node, 4 Node	Solid – 4 Node , 6 Node , 8Node
Tension only	General / Tapered Beam	
Hook	Plane stress	
Cable	Plan strain	
Compression only	Ax symmetric	
Gab		

❖**The element used is:**

Tapered Beam & 8Node Solid

3.2.2.5 Main Window of MIDAS Program

Main window of MIDAS program show in Figure (4-1) consist from the following:

Main Menu: The command sand short cut keys for all the functions necessary to run .The main Menu consist from File ,Edit ,View , Model, Load, Analysis, Results, Design, Mode, Query, Window and Help.

Tree Menu: The tree menu under the Menu tab is an outline of the entire data entry procedure from modeling to analysis to design.

Icon Menu: The icon menu provides access to frequently used functions, Which can be selectively organized to construct customized toolbars.

Context Menu: The context menu is prompted with just one click of the Right mouse button in modeling window will display modeling options, Thus reflecting the work environment.

Task Pane: The task pane is a feature for easy entry of analysis data. It displays the work procedure for advanced analysis functions and provided descriptions on input items.

Table Menu: Table Windows display all types of data entry, analysis and design results in the Spread Sheet format.

Message Window: Message Window is plays all types of information necessary for modeling, warnings and error messages.

Status Bar: Status Bar presents matters related to all kinds of coordinate systems, unit systems conversion, select filtering, fast query, elements nap control, etc. ,

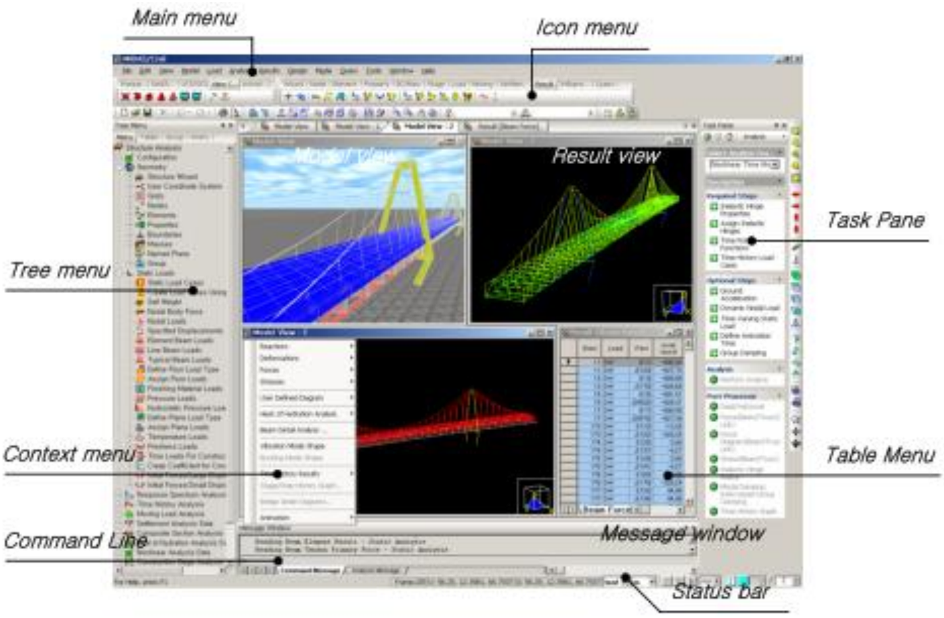


Figure 3-6: Main Window of MIDAS Program

3.2.2.6 Advantages and Features of MIDAS/Civil

MIDAS program has been developed in Visual C++, an object oriented programming language, in the windows environment. The program is remarkably fast and can be easily mastered for practical applications. By using the elaborately designed GUI (Graphic User Inter face) and the up-to-date Graphic Display functions, a structural model can be verified at each step off formation and the results can be directly set in to document formats .During the development process, MIDAS/Civil has been verified through numerous examples .Each of the functions has been verified by comparing the results with the critical values and output from other similar programs. The program has been applied to over 5,000 projects and the reliability and effectiveness have been established. Representative examples are in the Verification Manual. The latest theories form the bases for the finite element algorithm that determines the accuracy of analysis results .Excellent results are achieved compared to other similar programs.

Chapter 4
Analysis of single span of Soba Bridge

4.1 Description of Soba Bridge

4. 1. 1 Location of Soba Bridge

Extends from West Soba to East Soba through Blue Nile as shown in figure (4-1).

Figure 4-1: Location of Soba Bridge

4. 1. 2 Description of road way of Soba Bridge

The bridge carries two carriage ways10.5m wide and two foot ways 1.7m wide; each deck span comprises five precast beam to form 6 lanes as shown in figure (4-2).

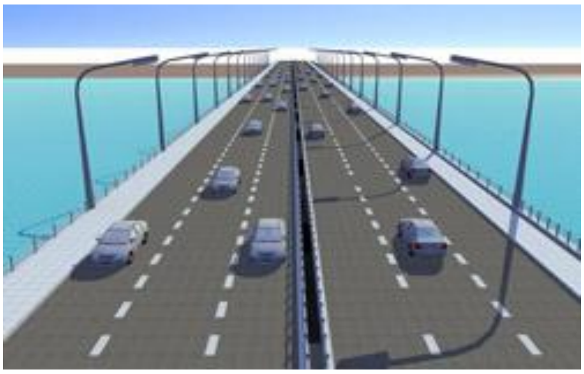


Figure 4-2: Soba Bridge

4. 1. 3 General shape of Soba Bridge

Soba Bridge consist from ten of precast post – tensioned concrete beams within situ concrete deck slab, two abutments, thirteen piers for each one pier consist four columns and foundation is pile cape and piles As Shown in figure (4-3) and (4-4).

4. 1. 4 General Information of Soba Bridge

4.1.4.1 Bridge Type

Pre-stressed Concrete Girder Bridge (Post-Tension).

4.1.4.2 Bridge length

The overall length of Soba bridge is equal to 571.100 m in the form twelve internal spans of 40.900m between pier centerline and two 40.150 m end span.

4.1.4.3 Bridge width

The width of the Soba Bridge equal to 26.5m.

4.1.4.4 Bridge height

The maximum height in Soba Bridge equal to 50.5m.

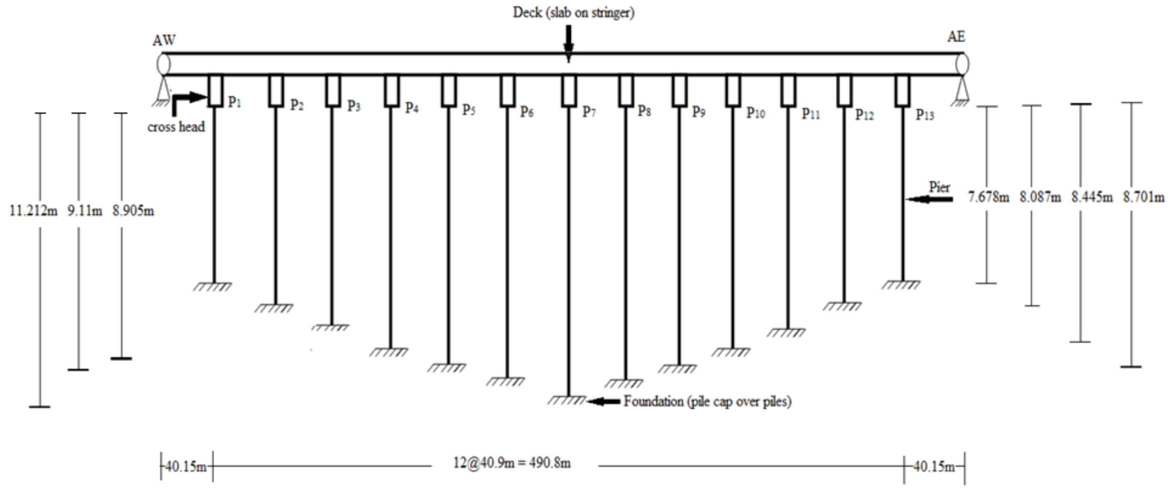


Figure 4-3: Elevation of Soba Bridge

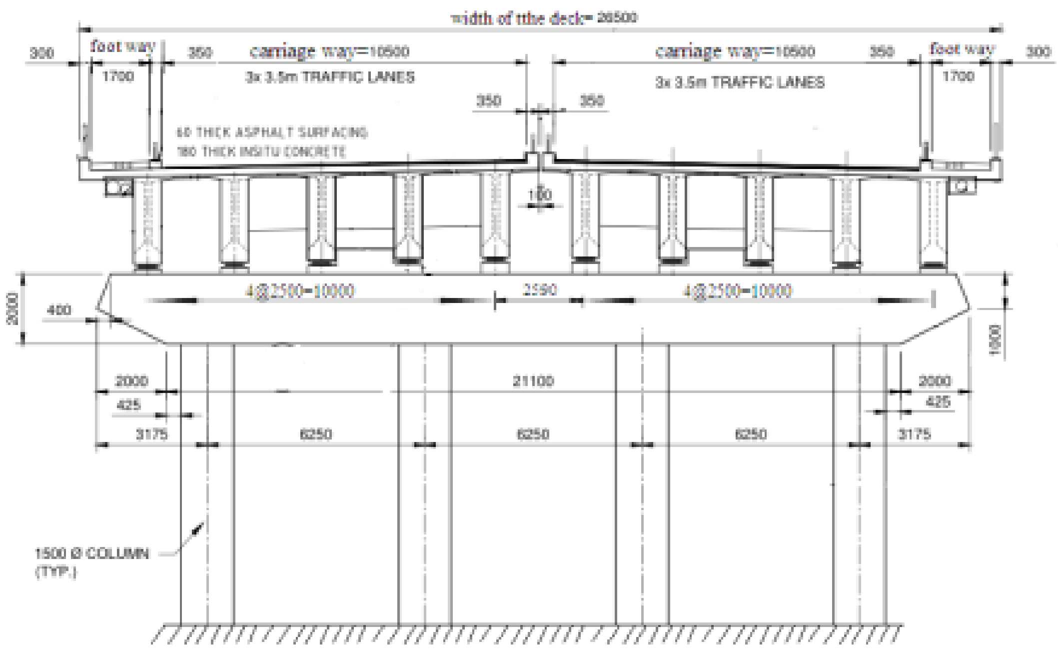


Figure 4-4: Side view of Soba Bridge

4. 2 Superstructure of Soba Bridge

As shown in figure (4-5).

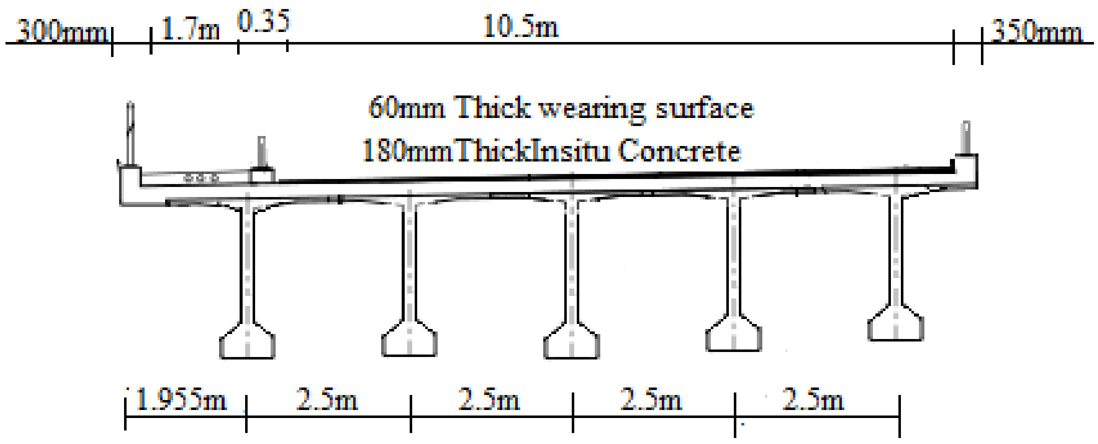


Figure 4-5: Side View of Superstructure of Soba Bridge

4.2.1 Analysis of Slab of Soba Bridge

- Flange width of steel girder (b_f) = 800mm
- Effective length of slab (S) = 2.5- 0.8 /2 = **2.1mm**
- Dead load of slab (D. L) = 0.18×25 + 0.06×22.5 = **5. 85 KN/m^2**

➢ Moment of Dead Load (D.L.M):

Moment of Dead load per 1ft for Continuous Slab according to (Clouse 5.3.1.1) due to AASHTO from this expression:-

$$D.\ L.\ M = \frac{\omega L^2}{10} = \frac{5.85 \times 2.1^2}{10} = \pm\ \mathbf{2.58\ KN.\ m/m}$$

➢ Moment of Live Load (L.L.M):

The Main Reinforcement Perpendicular to the direction of motion, so Moment of Live Load per 1 ft according to(AASHTO Clouse 5.4.2.6) from this expression:-

L.L.M = $\emptyset \, \frac{S+2}{32} \, P$

For Continuous Slab $\emptyset$ = 0.8 and For Truck Load 93KIP (803.52 KN)

L.L.M = $0.8 * \frac{2.1+2}{32} * 321.41$ = **33KN.m/m**

- Moment of Impact Load (I.L.M):

Impact Factor according to (AASHTO Clouse 5.4.3.7) from this expression:-

I.F = $\frac{50}{L+125} = \frac{50}{2.1+125}$

Impact factor (I.F) =0.39 =39% >30% Take I.F = 30%

I.L.M = 0.3 L.L.M = 0.3 * 33 = **9.9KN.m/m**

- Total Moment (T.M)

T.M = D.L.M + L.L.M + I.M.L

T.M = 2.58+ 33 + 9.9 = **45.48KN.m/m**

4.2.2 Analysis of girder bridge using manual

4. 2. 2.1 Girder Section

The internal section of girder depended in analysis of girders , as shown in figure (4-6).

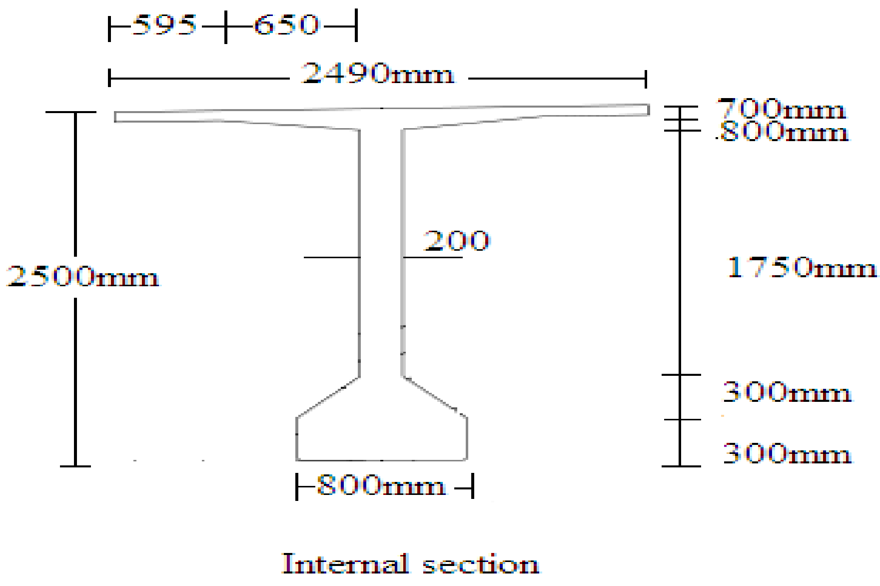


Figure 4-6: Section of girder

4. 2. 2.2 Section Properties

4. 2.2. 2.1 Properties of cross section of girder

4.2.2.2.1.1Centroid of cross section:

See table (4-1) and shown in figure (4-7).

$$Y_{bg} = \frac{\sum A\,y}{\sum A} = \frac{1208.113*10^6}{982300} = \underline{\underline{\mathbf{1229.\ 88mm}}}$$

$$Y_{tg} = 2500 - 1229.\ 88 = \underline{\underline{\mathbf{1270.\ 12mm}}}$$

Table 4-1 Centered of cross section

Component	**Area (mm^2)**	**Y Mm**	**A * Y (mm^3)**
Top flange	174300	2465	$429. 65*10^6$
Web	426000	1365	$581. 45*10^6$
Bottom	240000	150	$36*10^6$
Top triangular	52000	240. 333	$124. 973*10^6$
Bottom triangular	90000	400	$36*10^6$

4.2.2.2.1.2Second moment of Area:

See table (4-2).

Table 4-2 Second moment of area

Component	**Area (mm^2)**	**Y Mm**	**I_0 (mm^4)**	**$I_x=I_0+AY^2$ (mm^4)**
Top flange	174300	1235. 12	$71.173*10^6$	$2.66*10^{11}$
Web	426000	135. 12	$1.611*10^{11}$	$1.689*10^{11}$
Bottom	240000	1079. 88	$1.8*10^9$	$2.817*10^{11}$
Top triangular	52000	1173. 453	$9.244*10^6$	$7.161*10^{10}$
Bottom triangular	90000	829. 88	$450*10^6$	$6.243*10^{10}$
∑	982300			$8.506*10^{11}$

4.2.2.2.1.3Section modulus:

$$S_{tg} = \frac{Ig}{ytg} = \frac{8.506*10^{\wedge}11}{1270.12} = \underline{\underline{\mathbf{669.701*10^6 mm^3}}}$$

$$S_{tg} = \frac{Ig}{ybg} = \frac{8.506*10^{\wedge}11}{1229.88} = \underline{\underline{\mathbf{691.612*10^6 mm^3}}}$$

4. 2.2. 2.2 properties of cross section of compound girder:

4. 2.2. 2.2.1 Centered of cross section:

see table (4-3) and shown in figure (4-7).

Table 4-3 Centered of cross section

Component	**Area (mm^2)**	**Y Mm**	**A * Y (mm^3)**
Deck slab	386141. 58	25900	$1000.107*10^6$
I-Section	982300	1229. 88	$1208.111*10^6$
∑	1385500		$2208.218*10^6$

$Y_{bc} = \frac{2208.218*10^6}{1385500}$ = **1593. 806mm**

Y_{tc} = 2680 – 1593. 806 = **1086. 194mm**

4. 2.2. 2.2.2 Second moment of Area:

see table (4-4)

Table 4-4 Second moment of area

Component	**Area (mm^2)**	**Y Mm**	**I_0 (mm^4)**	**$I_x=I_0+AY^2$ (mm^4)**
Top flange	174300	871. 194	$71.173*10^6$	$1.324*10^{11}$
Web	426000	228. 806	$1.611*10^{11}$	$1.834*10^{11}$
Bottom	240000	1443. 806	$1.8*10^9$	$5.021*10^{11}$
Top triangular	52000	809. 527	$9.244*10^6$	$3.409*10^{10}$
Bottom triangular	90000	1193. 806	$450*10^6$	$1.287*10^{11}$
Deck slab	386141. 58	996. 194	$1042.582*10^6$	$3.843*10^{11}$
∑	1385500			$1.365*10^{12}$

4. 2.2. 2.2.3 Section modulus:

$S_{tc} = \frac{Ic}{ytc} = \frac{1.365*10^{12}}{1086.194} = \underline{\underline{\mathbf{1256.682*10^6 mm^3}}}$

At the top of girder:

$S_{ic} = \frac{Ic}{ytc-ts} = \frac{1.365*10^{12}}{1086.194-180} = \underline{\underline{\mathbf{1506.\ 3*10^6\ mm}}}$

At the bottom of girder:

$S_{bc} = \frac{Ic}{ybc} = \frac{1.365*10^{12}}{1593.806} = \underline{\underline{\mathbf{856.\ 44*10^6 mm^3}}}$

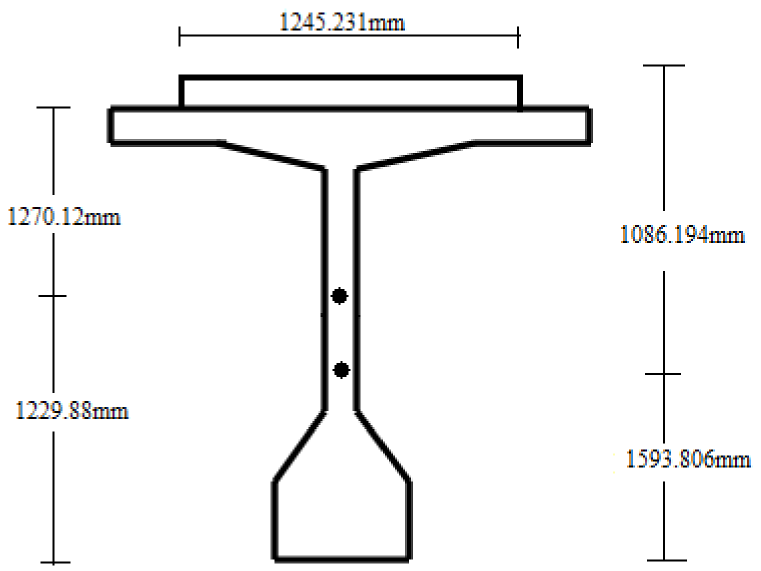


Figure 4-7: Centered of Cross Section and Compound Section

4. 2. 2.3 Material Data

- Wearing surface density γ_w =22.5 KN/m^3

- Concrete density γ_c =25 KN/m^3

- Concrete compressive strength of girder at 28 days f_{cg} =45MPa

- Concrete modulus of elasticity of girder at 28 days E_{cg} =32.5GPa

- Concrete compressive strength of slab at 28 days f_{cs} =30MPa

- Concrete modulus of elasticity of top slab at 28 days E_{cs}=28GPa

- Reinforcement yield strength f_y =420MPa

- Reinforcement modulus of elasticity E_s =200GPa

4. 2.2. 4 Pre-stress Data

- Ultimate strength of pre-stressing steel f_{pu} =1860MPa

- Yield strength of pre-stressing steel f_{py}=0.9f_{pu} f_{py}=1674MPa

- Elastic modulus of pre-stress tendon E_{ps}=195 GPa

- Anchorage slip dx=6mm

- Curvature coefficient mu mu= 0.3(1/rad)

- Wobble friction coefficient K=3.3*10^6(1/mm)

- Area of one strand_7 wire A_{ps} =140mm^2

- Duct diameter d_d =80 mm

- Total number of tendons N_t = 5

- Stress in tendon before transfer (75% of ultimate strength) = 1395 MPa

- Assumed initial loss due to elastic shortening = 9.2 % (18.6 ksi)

Therefore: Stress in tendon after transfer = 183.90 ksi

4. 2.2. 5 Tendon profile of pre – stressed concrete
As shown in figure (4-8) and (4-9) and Table (4-5)

Reinforced:

Grade 420,Type 2,high yield deformed bar,
yield strength f_Y=420N/mm^2

Pre - stressing:

15. 2mm 7 wire strand, $f_{pu} = 1860N/mm^2$

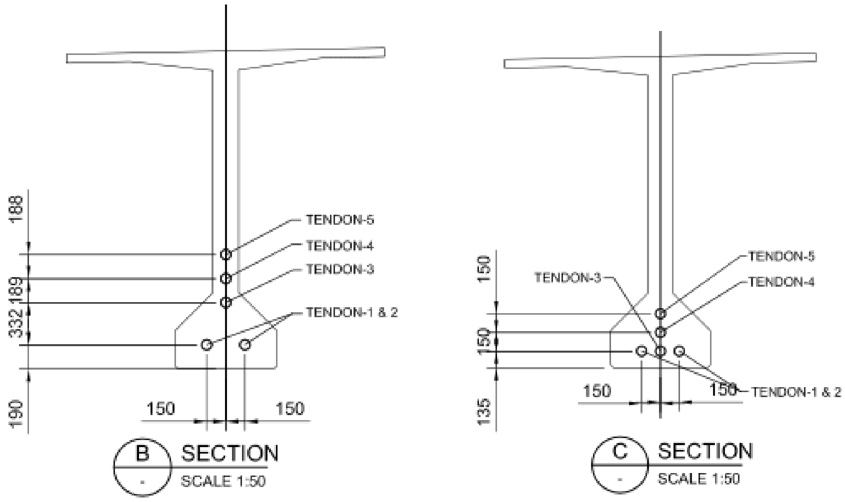


Figure 4-8: Cross section of compound girder

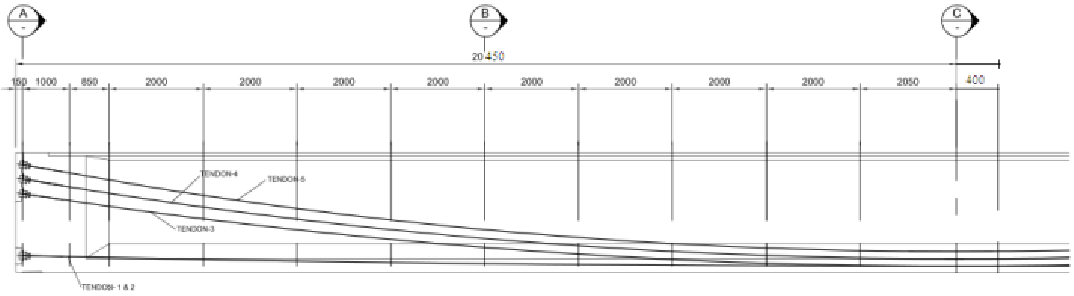


Figure 4-9: Shape of Tendons Profile

Table 4-5 Element type in MIDAS program

High of tendon above bottom of beam (mm)	Distance from end of beam(mm)													
	0	150	1150	2000	4000	6000	8000	10000	12000	14000	16000	18000	20050	20450
Tendon 1&2	353.15	350	329	312	275	242	214	190	170	155	144	137	135	134.61
Tendon 3	1671.75	1650	1505	1385	1123	892	692	522	384	275	198	151	135	131.89
Tendon 4	1973.85	1950	1791	1658	1371	1117	897	711	558	439	354	303	285	281.49
Tendon 5	2276.1	2250	2076	1932	1619	1342	1102	899	733	603	510	454	435	431.3

4. 2.2. 6 Loading on Girders

Calculation of the loads on Exterior girder with side walk, Exterior girder without side walk and interior girder, as show in figure (4-10).

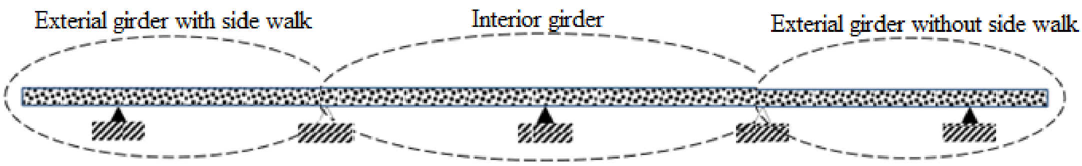


Figure 4-10: Soba Bridge Girders

4.2.2.6.1 Dead Load

- Self weight

Automatically calculated within the program, but from code:-

$\gamma_c = 2240+2.29f_c^{\backslash}$ (from table 3.5.1-1- densities)

$\gamma_c = 2240+2.29*45 = 2343.05\ kg/m^3 = 23.431KN/m$

$W_g = \gamma_c * A_g = 23.431 * 982300 * 10^{-6} = 23.016KN/m$

The self weight of girder is 23.016 KN/m, show in figure (4-11).

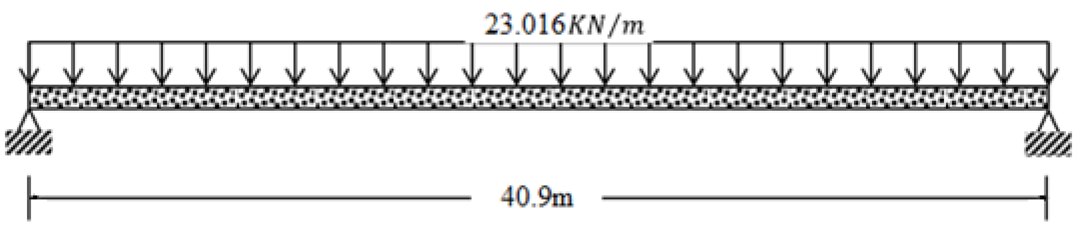


Figure 4-11: load due to self weight

- Deck slab:-

The Loads = Density of Concrete × Thickness of slab

❖ Exterior with side walk girder: - As show in figure (4-12-a).

The Loads = [25 * (0.18 + 0.15)] *1.955 + [25 * 0.18] * $\frac{2.5}{2}$

R= 8.23*1.955 + 4.5* $\frac{2.5}{2}$ = 21.715KN/m

❖Interior girder: - As show in figure (4-12-b).

The Loads = [25 * 0.18] *1.245 + 4.5* $\frac{2.5}{2}$

R= 4.5*1.245 + 4.5* $\frac{2.5}{2}$ = 11.23KN/m

❖Exterior with without walk girder: - As show in figure (4-12-c).

The Loads = [25 * 0.18] *2.5

R= 4.5*2.5 = 11.25KN/m

- Future Wearing Surface:-

The Loads = Density of Wearing Surface ×Thickness of Wearing Surface

❖Exterior girder with side walk girder: - As show in figure (4-13-a).

The Loads = [22.5 *0.06] * 2.1* $\frac{1.050}{2.5}$

R=1.35 * 2.1* $\frac{1.050}{2.5}$ =1.191KN/m

❖Interior girder:- As show in figure (4-13-b).

The Loads = [22.5 *0.06] * $\frac{2.5}{2}$ + [22.5 *0.06]*0.895

$R=1.35 * \frac{2.5}{2} + 1.35*0.895 = 2.9KN/m$

❖Exterior with without side walk girder: - As show in figure (4-13-c).

The Loads = [22.5 *0.06] * 2.5

R=1.35 * 2.5 =3.375KN/m

- Barriers + Utilities:-

❖Exterior with side walk girder:- As show in figure (4-14-a)

$R=6.67 + 3.15 + 1.8* \frac{2.325}{2.5} =11.5KN/m$

❖Interior girder:- As show in figure (4-14-b).

R=1.8 KN/m

❖ Exterior with without side walk girder: - As show in figure (4-14-c).

$R=1.8* \frac{0.175}{2.5} =0.126KN/m$

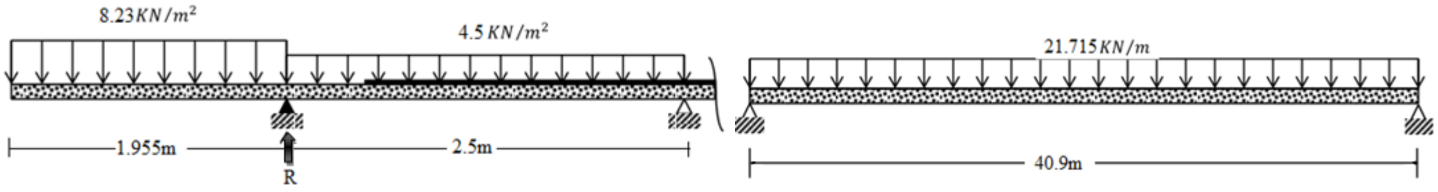
- Pedestrian**:** As show in figure (4-15).

Length of side walk = 40.9m (136.333') >30.5m (100')

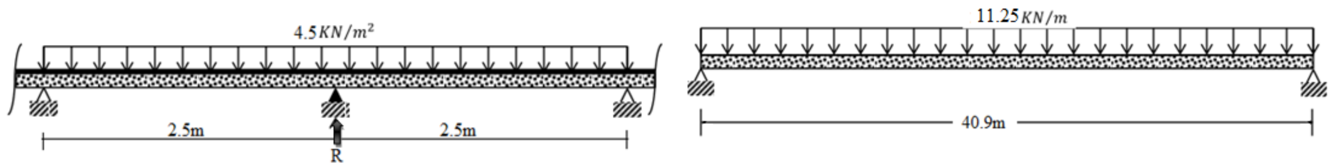
$\therefore P = (30+ \frac{3000}{L})*(\frac{55-W}{50}) \leq 60 \text{ PSF}$

$P = (30+ \frac{3000}{L})*(\frac{55-W}{50}) = (30+ \frac{3000}{136.333})*(\frac{55-5.67}{50}) =51.5 \text{ PSF} = 2.5 \text{ KN/m}^2$

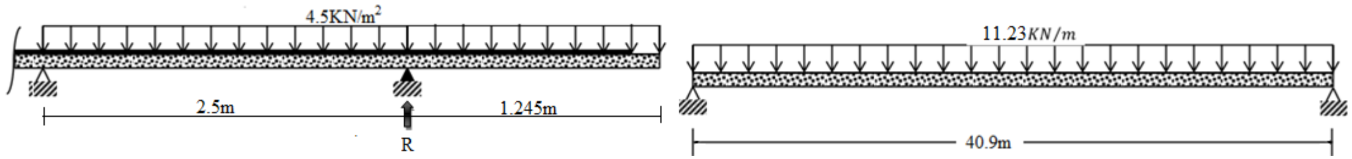
$R= 2.5 * 1.655 + 2.5 * 0.045 * \frac{0.0225+2.455}{2.5} = 4.25KN/m$



a: load due to deck slab for Exterior girder with side walk

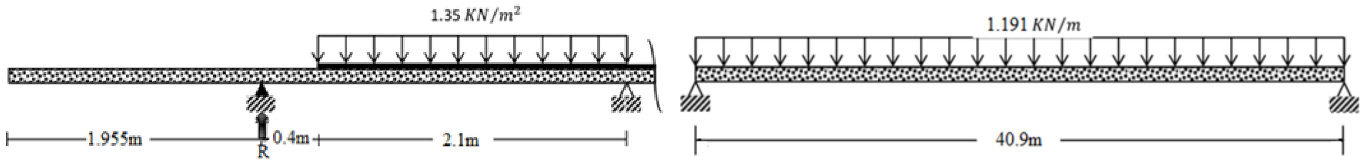


b: load due to deck slab for Interior girder

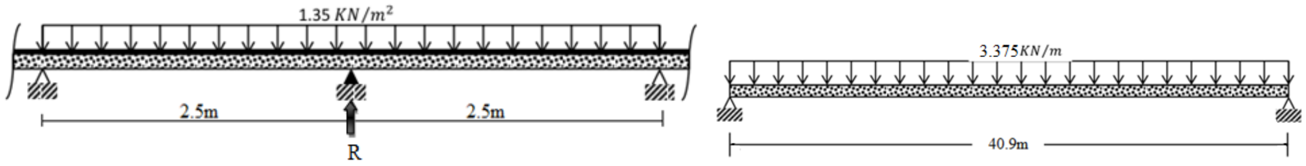


c: load due to deck slab for Exterior girder without side walk

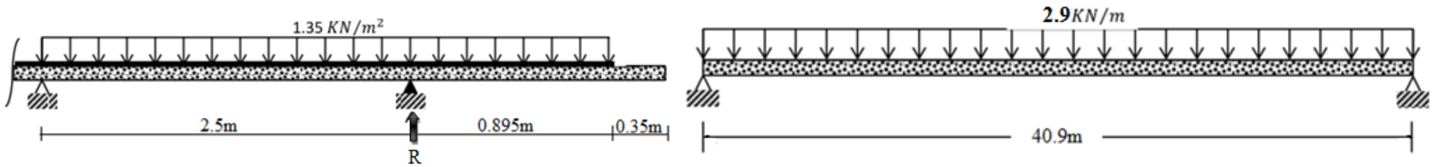
Figure 4-12: load due to deck slab



a: load due to wearing surface for Exterior girder with side walk

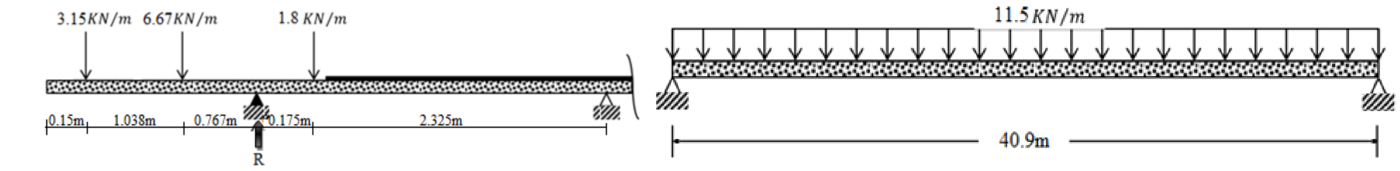


b: load due to wearing surface for Interior girder

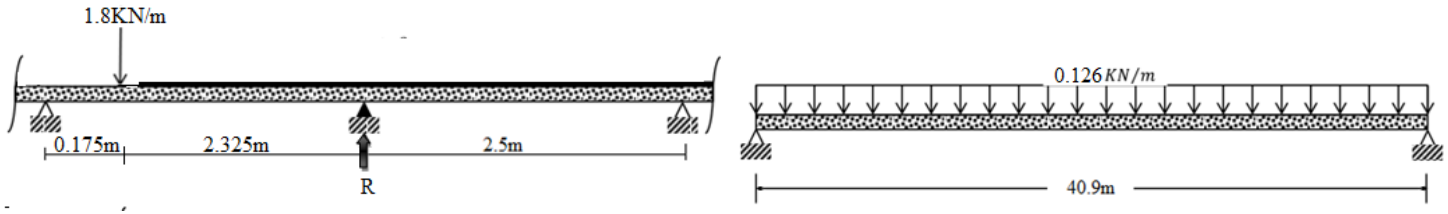


c: load due to wearing surface for Exterior girder without side walk

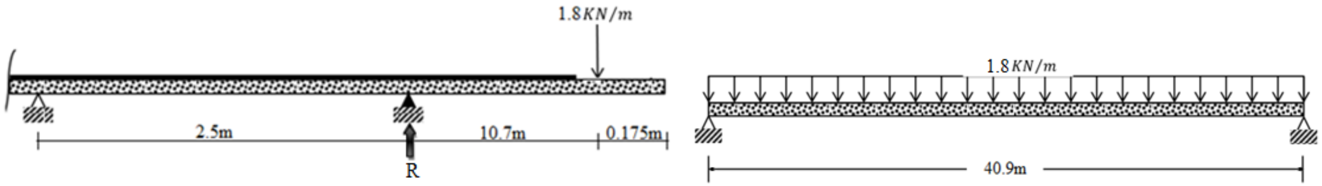
Figure 4-13: load due to wearing surface



a: load due to Barrier and Utilities for Exterior girder with side walk



b: load due to Barrier and Utilities for Interior girder



c: load due to Barrier and Utilities for Exterior girder without side walk

Figure 4-14: load due to Barrier

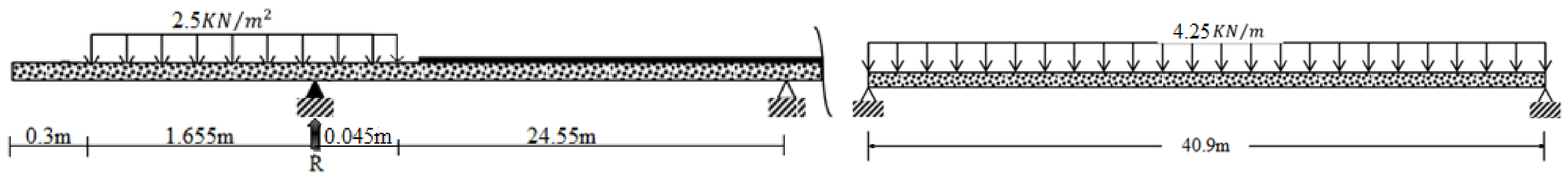


Figure 4-15: load due to Pedestrian

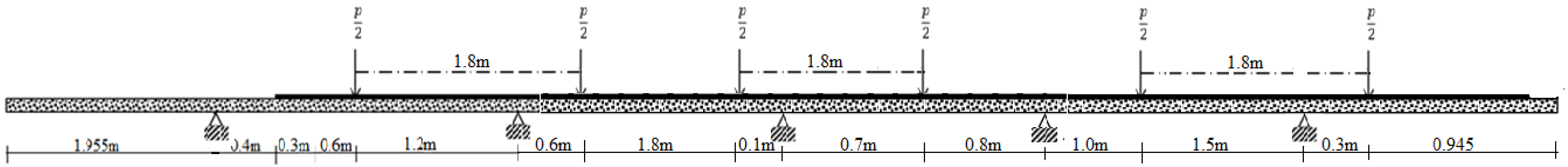


Figure 4-16: Live load distribution on bridge width

4.2.2.6.2 Truck load

Show in figure (4-16)

Loads on wheels:-

Use weight of truck ω = 93Kip or 803.52KN

Load on forward centerline = 0.2 * 803.52 = 160.704KN

Load on backward centerline = 0.8 * 803.52 = 642.816KN

Position of wheels for give maximum moment show in figure (4-17)

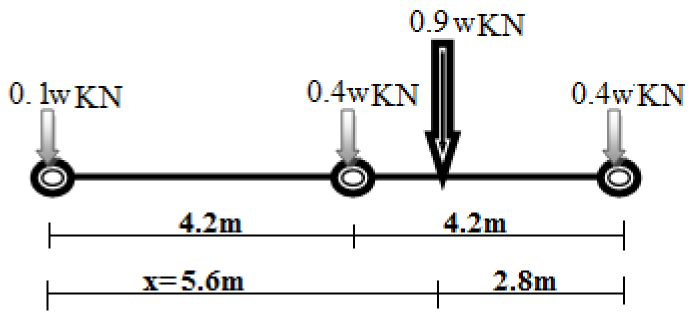


Figure 4-17: position of wheels

$$x = \frac{0.1w*8.4+0.4w*4.2}{0.9w} = 5.6m$$

Exterior girder with side walk:-

R = $\frac{P}{2} * \frac{1.2}{2.5}$ = 0.24P

Force due to forward axis=0.24*160.7 = 38.57KN

Force due to forward axis=0.24*642.816 = 154.276KN

❖ Maximum Moment:- show in figure (4-18)

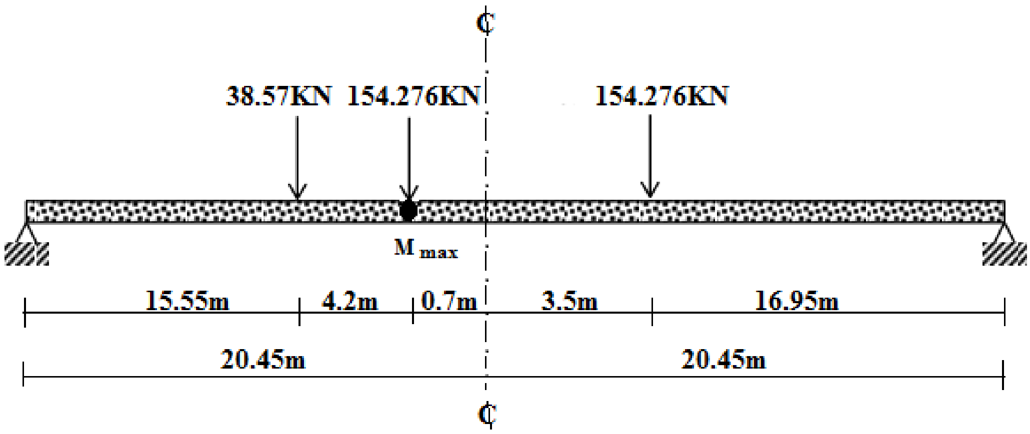


Figure 4-18: Maximum Moment

$$M_{max} = \left(\frac{38.57*25.35+154.276*21.15+154.276*16.95}{40.9} * 19.75\right) - 38.57*4.9$$

M_{max} =3148.505KN.m

❖ Maximum Shear force:- show in figure (4-19)

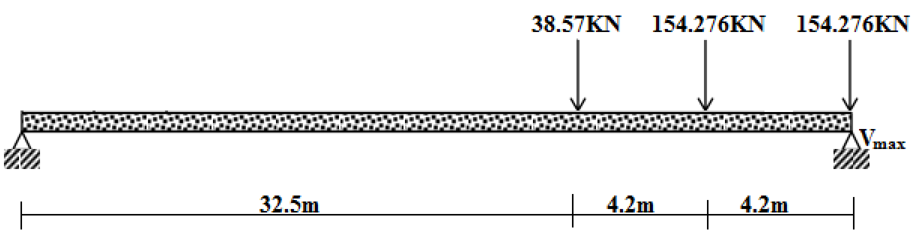


Figure 4-19: Maximum Shear force

$$V_{max} = \frac{38.57*32.5+154.276*36.7}{40.9} + 154.276 = 323.36KN$$

Exterior girder without side walk:-

$R = \frac{P}{2} * \frac{1.0}{2.5} + \frac{P}{2} = 0.7P$

Force due to forward axis=0.7*160.7 = 112.493KN

Force due to forward axis=0.7*642.816 = 449.97KN

❖ Maximum Moment:- Show in figure (4-20)

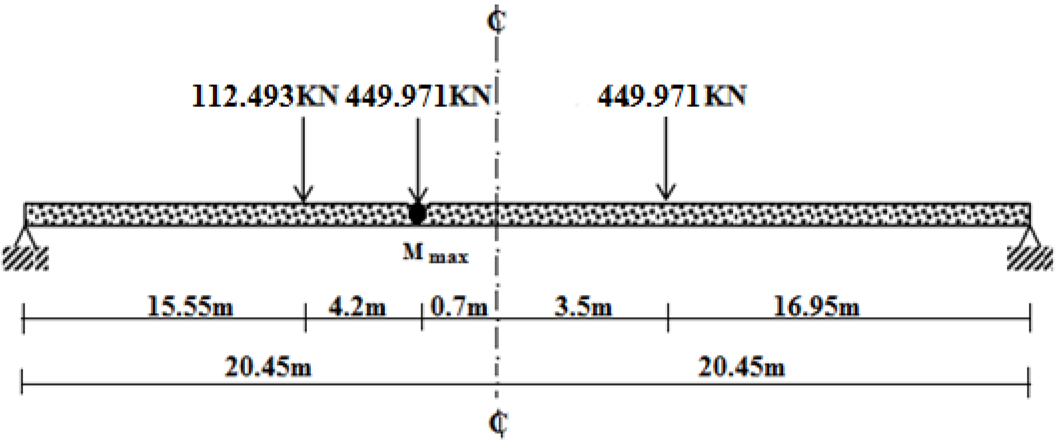


Figure 4-20: Maximum Moment

$M_{max} = \left(\frac{112.493*25.35+449.971*21.15+449.971*16.95}{40.9} * 19.75\right) - 449.971*4.9$

M_{max} = 9970.234KN.m

❖ Maximum shear force:-show in figure (4-21)

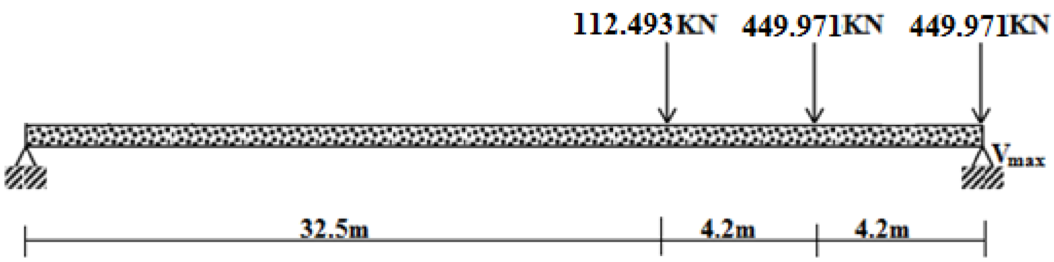


Figure 4-21: Maximum Shear force

$V_{max} = \frac{112.493*32.5+449.971*36.7}{40.9}$ +449.971= 1023.963KN

Interior girder:-

The maximum truck loads distribution on interior girder show in figure (4-22)

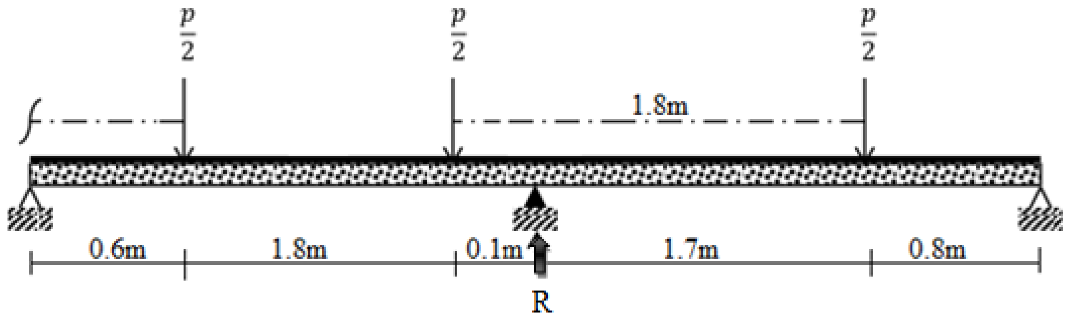


Figure 4-22: Maximum Truck loads distribution on interior girder

R = $\frac{P}{2} * \frac{2.4}{2.5} + \frac{P}{2} * \frac{0.6}{2.5} + \frac{P}{2} * \frac{0.8}{2.5}$= 0.76P

Force due to forward axis=0.76*160.7 = 112.135KN

Force due to forward axis=0.76*642.816 = 488.54KN

❖ Maximum Moment:- show in figure (4-23)

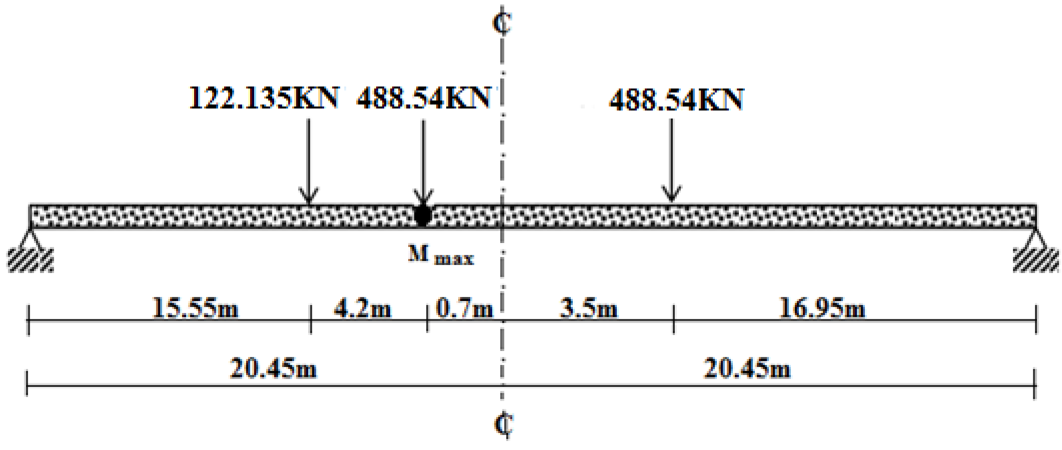


Figure 4-23: Maximum Moment

$M_{max} = \left(\frac{122.135*25.35+488.54*21.15+488.54*16.95}{40.9} * 19.75\right) - 122.135*4.9$

M_{max} =9183.107KN.m

❖ Maximum shear force:- show in figure (4-24)

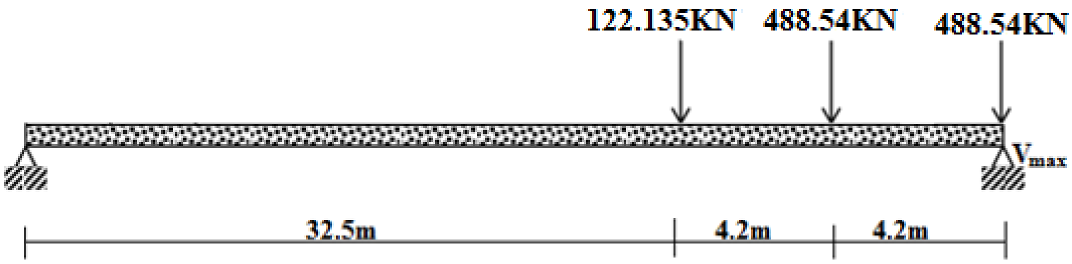


Figure 4-24: Maximum Shear force

$$V_{max} = \frac{122.135*32.5+488.54*36.7}{40.9} +488.54= 943.124KN$$

4.2.2.7 Result of Analysis

Table (4-6) illustrate Result of Analysis for dead load , truck load and impact load .

4.2.2.8 Load Combinations

See Table (4-7)

Table 4-6 Results of Analysis

Type of girder	***Exterior girder with side walk***			***Exterior girder without side walk:-***			***interior girder***		
Type of load	Distributed load w KN/m	Shear at support V_{max} KN	Moment at 0.5L M_{max} KN.m	Distributed load w KN/m	Shear at support V_{max} KN	Moment at 0.5L M_{max} KN.m	Distributed load w KN/m	Shear at support V_{max} KN	Moment at 0.5L M_{max} KN.m
S.Wofgirder	23.016	470.677	4812.67	23.016	470.677	4812.67	23.016	470.674	4812.67
S.W of slab	21.715	444.072	4540.639	11.228	229.613	2347.789	11.25	230.0625	2352.39
Bituminous	1.164	23.804	243.394	2.896	59.223	605.557	3.375	69.02	705.72
Barriers	11.17	228.427	2335.661	1.8	36.81	376.382	0.126	2.577	26.347
Pedestrian load	4.249	4.249	86.892	-	-	-	-	-	-
Live Load	-	323.357	3148.505	-	943.124	9183.107	-	1023.963	9970.234
ImpactLoad	-	97	944.6	-	282.94	2754.93	-	307.2	2991.1

➢ **Strength I Limit State:-**

U = 1.25DC + 1.50DW + 1.75(LL + IM)

Table 4-7 load combination

Girder	**Combination 1**	
	Shear force(KN)	**Moment(KN.m)**
Exterior girder with side walk	2207.74	22291.3
Exterior girder without side walk	3077.1	31220.95
Interior girder	3312.2	32730.2

➢ From table (4-7) illustrate load Combination find the Interior girder subject to Maximum moment and maximum shear fore then we depend in the design and generalization on other girders

4.2.2.9 Analysis of Soba Bridge by MIDAS Program

1. File opining and preferences setting:

● File / New Project

● File / Save Bridge of Soba

● Tools / Unit system:- Length >**m** And Force >**KN**

2. Materials: (see figure 4-25 and 4-26)

The following materials are defined:

1. Deck
2. Precast beams
3. Tendon
4. Cross beams

● In the Tree Menu: Geometry / Properties / Material

● Properties dialog box > Material tab >Click Add

● Name >**Deck**

● Type of design >**concrete**

● Standard >**BS (R. C)**

● DB = **C30**

● Click Apply

Name >**Precast beams**

● Type of design >**concrete**

● Standard >**BS (R. C)**

● DB = **C45**

● Click Apply

● Name >**Tendon**

● Type of design >**User defined**

● Standard >**None**

● Modulus of Elasticity >**1. 9500e+005**

● Poisson's Ratio >**0. 3**

● Thermal coefficient >**0. 0**

● Weight density >**0. 0**

● Click Apply

● Name >**Cross beam**

● Type of design >**concrete**

● Standard >**BS (R. C)**

● DB = **C30**

● Click OK

● Click Close

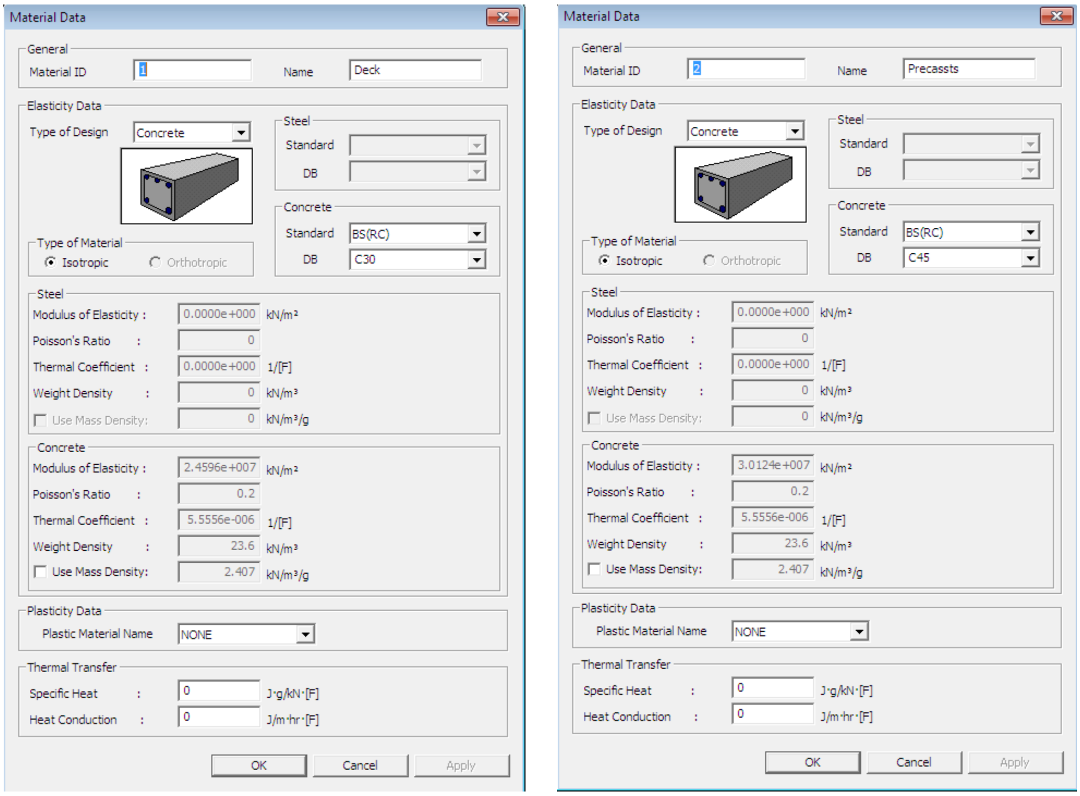


Figure 4-25: Material of Deck and Precast beam

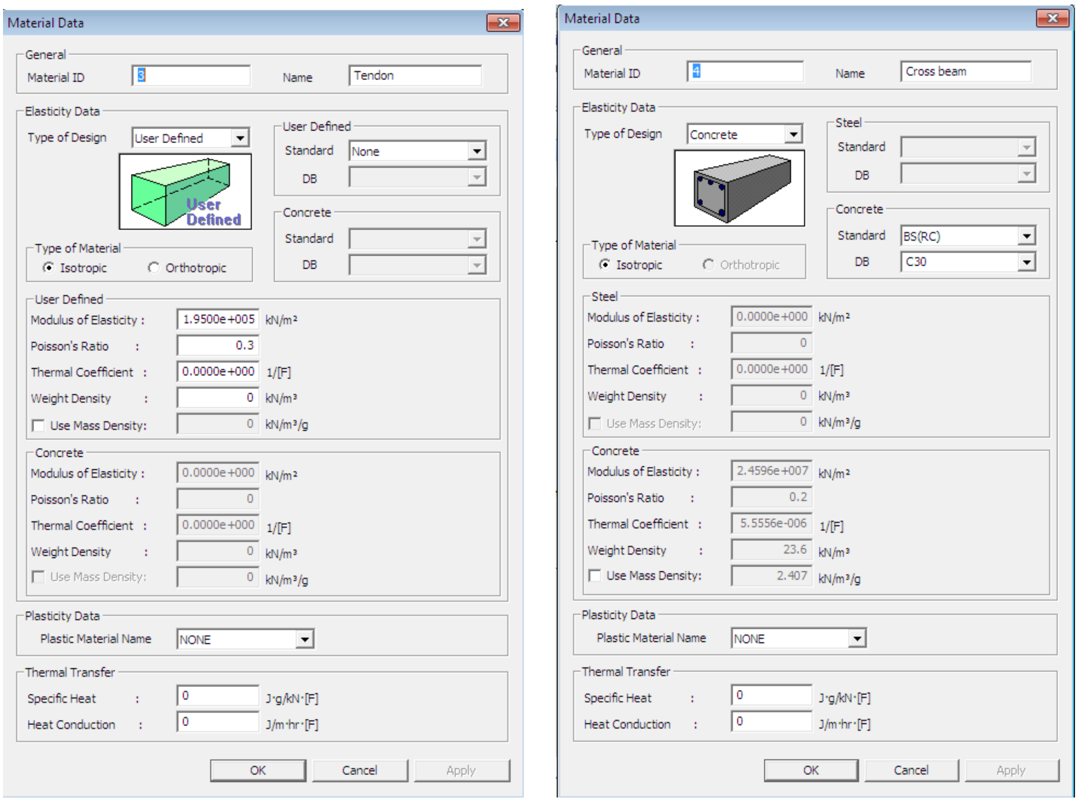


Figure 4-26: Material of Tendon and Cross beam

3.Time dependent material properties:

● In the Tree Menu: Geometry / Properties / Time Dependent Material (Creep/ Shrinkage) (See figure 4-27 and 4-28)

● Time Dependent Material (Creep/Shrinkage) dialog box >Click Add

● Name >**CEB – FIP** ● Code >**CEB – FIP**

● Compressive strength of concrete at the age of 28 days >**45KN/m^2**

● Relative Humidity of ambient environment (40 - 90) >**70%**

● Notational size of member >**0. 254m**

(This is a provisional value that will be replaced later after calculation By the program)

- Type of cement >**Normal or rapid hardening cement (N, R)**
- Age of concrete at the beginning of shrinkage >**3days**
- Click Show results

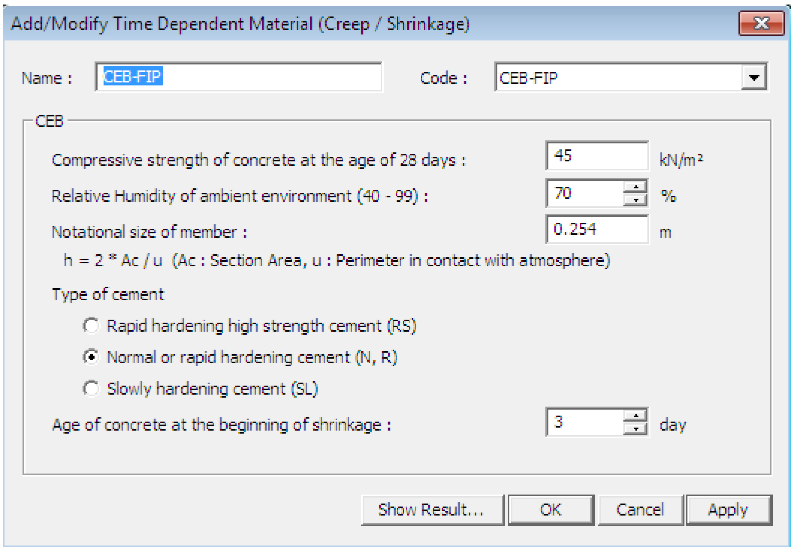


Figure 4-27: Input data of time dependent material properties (Creep – Shrinkage)

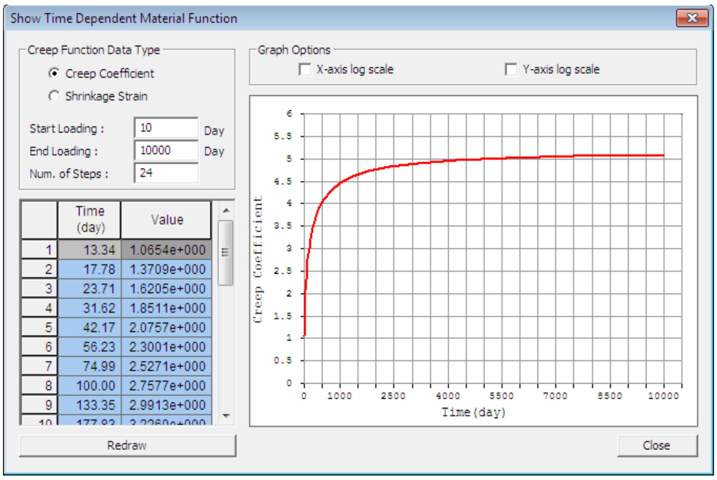


Figure 4-28: Time dependent material properties (Creep – Shrinkage)

● In the Tree Menu: Geometry / Properties / Time Dependent Material (Comp. Strength) (See figure 4-29)

● Time Dependent Material (Comp. Strength) dialog box >Click Add

● Name >**C45** ● Type >**Code**

● Development of Strength > Code >**CEB – FIP**

● Concrete Compressive Strength at 28 days >**45KN/m^2**

● Cement Type(s) >**N, R = 0. 25**

● Click Redraw Graph

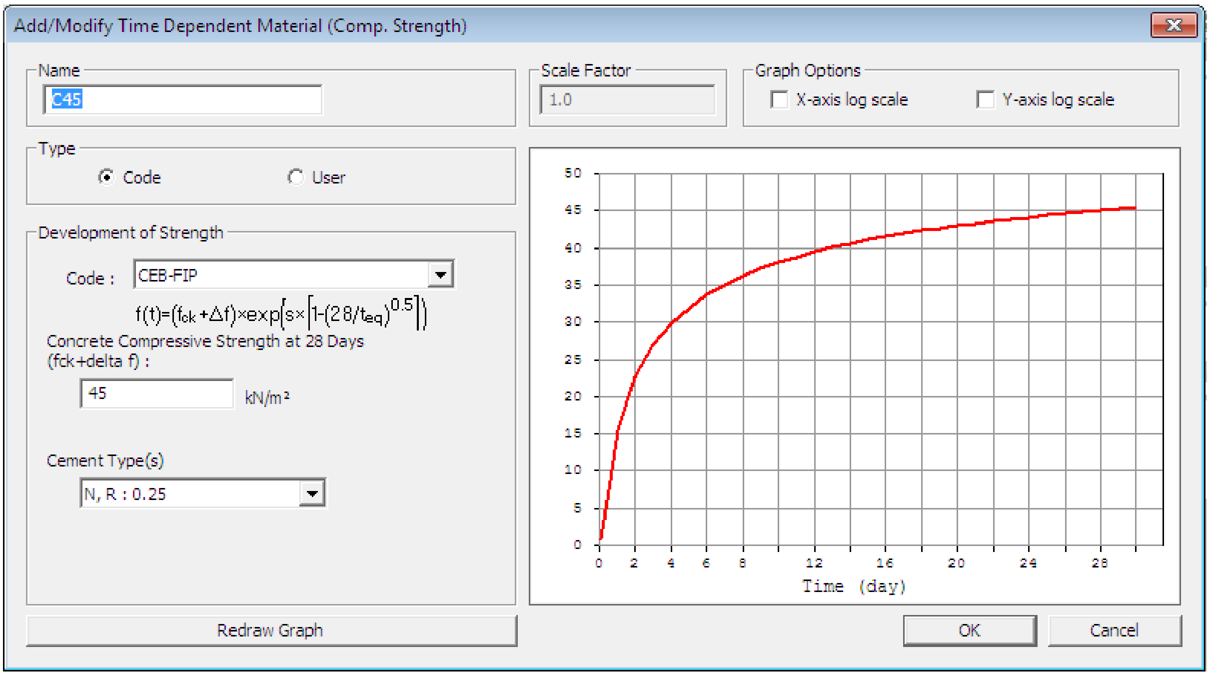


Figure 4-29:Time Dependent Material (Comp. Strength)

● Click OK ● Click Close

● In the Tree Menu: Geometry / Properties / Time Dependent Material Link (See figure 5-30).

- Time Dependent Material Type >Creep/Shrinkage>**CEB – FIP**
- Time Dependent Material Type > (Comp. Strength) >**C45**
- Select material to Assign > Material > 2: Precast beams
- Click >
- Click Add / Modify
- Click Close

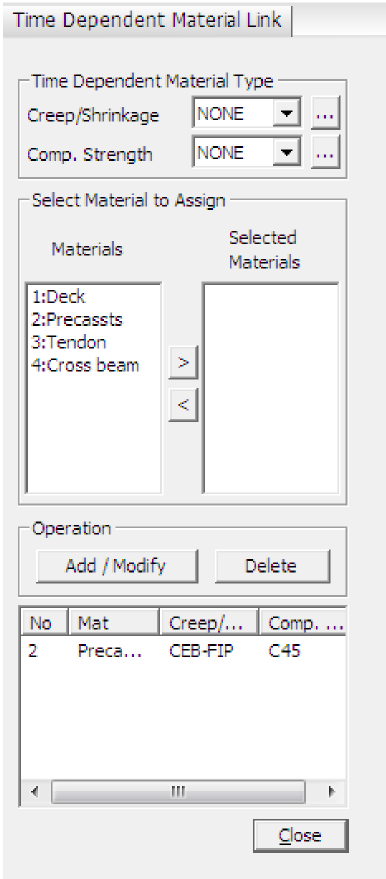


Figure 4-30: Dependent material link

4. Section Properties:

The following sections are defined:

1. Girder
2. Cross beam

● In the Tree Menu: Geometry / Properties / Section

● Properties dialog box >Section tab >Click Add

● Click **composite** tab]● Section ID >**1**

● Name >**Girder** (See figure 4-31) and (See Table 4-8)

● Section Type >**Composite-I** ● Slab Width >**13. 2m**

● Girder >Num>**5**

● CTC >**2. 5m** (center – to – center beam spacing)

● Slab > B. C = **2. 5m** ● Slab > t. C = **0. 18m**

● Slab > H. H = **0**

● Girder >**Symmetry (on)** ● Girder >**JL1 (on)**

Table 4-8 dimension of the section

HL1	**0. 07**
HL2	0. 08
HL2-1	0
HL3	1. 75
HL4	0. 3
HL5	0. 3
BL1	0. 1
BL2	1. 245
BL2-1	0. 65
BL4	0. 4

Girder Section Geometry Data:

● E_{gd}/E_{sb} = **1. 1607** ● D_{gd}/D_{sb} = **0**

● Click OK

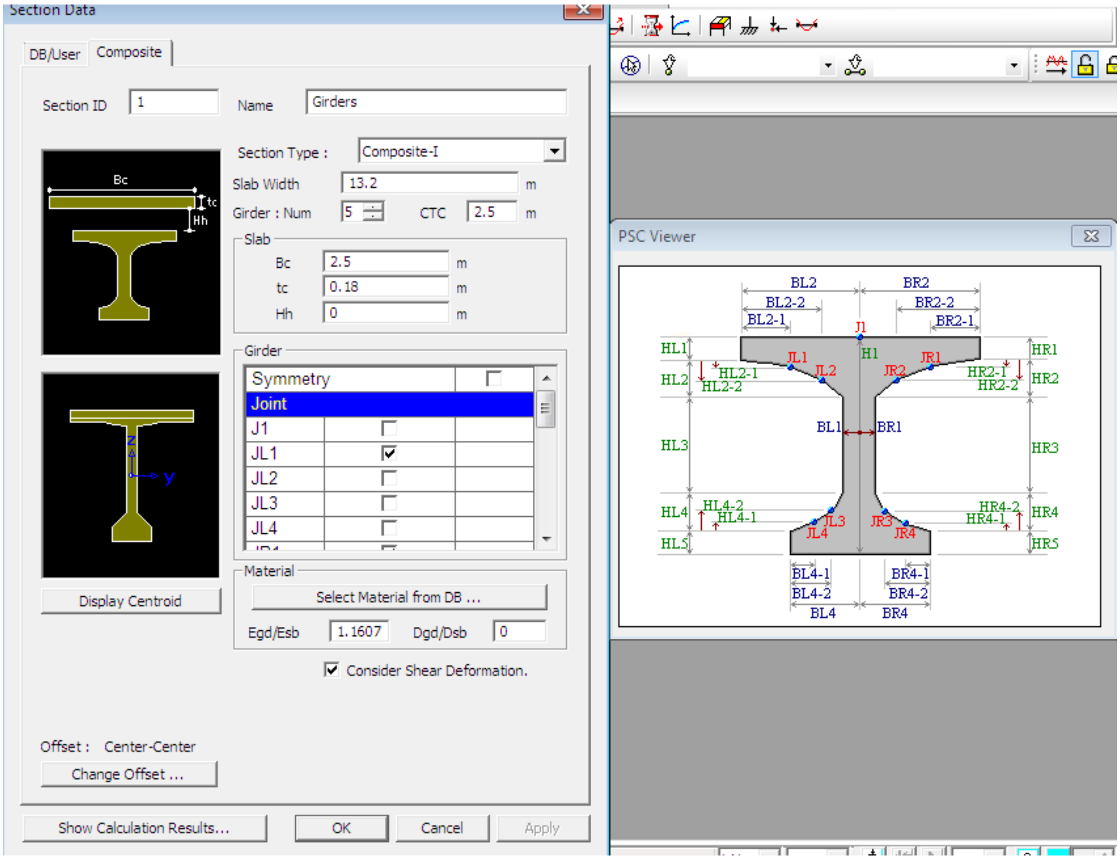


Figure 4-31: Section of girder

● Properties dialog box >Section tab > Click Add

●Click **DB/ User tab** ● Section ID >**2**

● Name >**Cross beam** (See figure 5-32) ● Click **User**

● Select **solid rectangle**● H =**1. 495m**● B = **1. 7m**

● Click OK ● Click Close

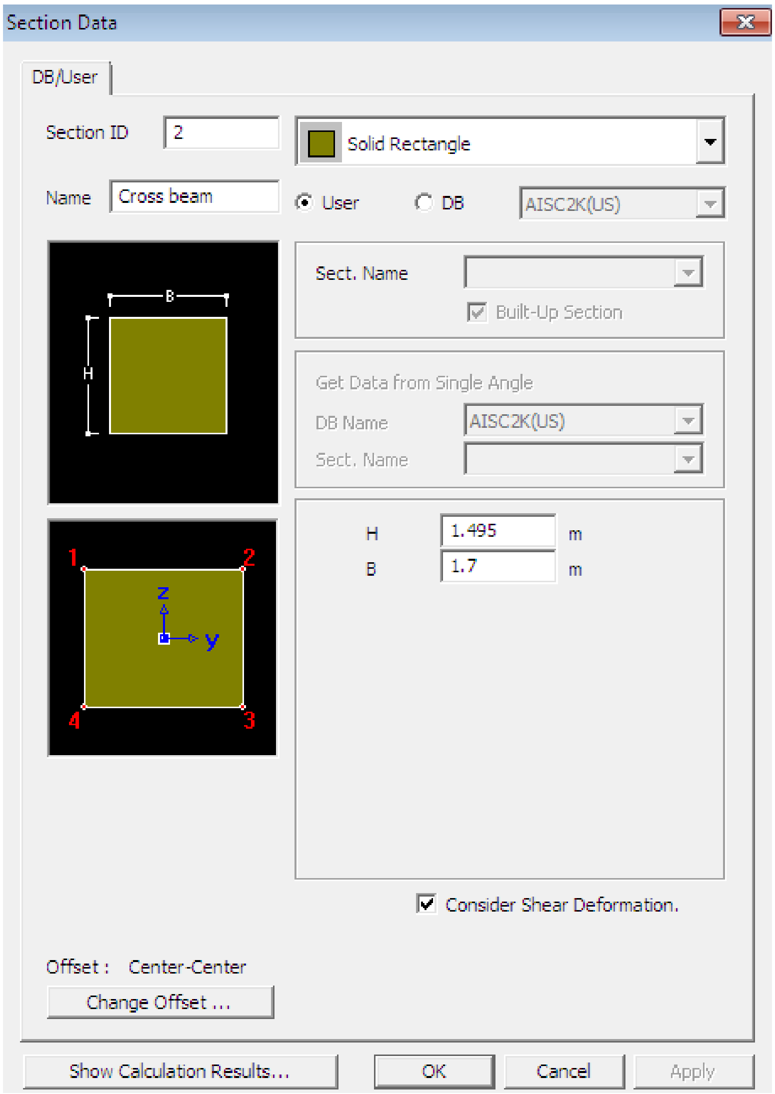


Figure 4-32: Section of Cross beam

5. Structural Modeling Using Nodes and Elements: (See from figure 4-33 to figure 4-36)

- 45Nodes

- 48 Element

● In the Tree Menu: Geometry / Nodes /  Create

● Coordinates (x, y, z) >**0, 0, 0**

● Copy > Number of times >**0**

- Copy > Distances > (dx, dy, dz) >**0, 0, 0**
- Click Apply
- Click Close

Precast beams:

- In the Tree Menu: Geometry / Nodes / Translate nodes
- Mode >**Copy**
- Translation> Equal distance > dx, dy, dz>**2, 0, 0**
- Number of times >**1**
- Click Apply

Select nodes 1 & 2

- In the Tree Menu: Geometry / Nodes / Translate nodes
- Mode >**Copy**
- Translation> Equal distance > dx, dy, dz>**0, 2. 5, 0**
- Number of times >**4**
- Click Apply

Select all nodes

- In the Tree Menu: Geometry / Nodes / Translate nodes
- Mode >**Copy**
- Translation> Equal distance > dx, dy, dz>**38. 9, 0, 0**
- Number of times >**1**
- Click Apply

Select the nodes at the second column from the left

- In the Tree Menu: Geometry / Nodes / Translate nodes
- Mode >**Copy**

- Translation> Equal distance > dx, dy, dz>**6. 15, 0, 0**
- Number of times >**5**
- Click Apply

Cross beam:

- In the Tree Menu: Geometry / Elements / Extrude
- Select Window > Nodes 1&2
- Extrude Type >**Nodes**>**Line Element**
- Element Attribute > Element Type >**Beam**
- Material >**4: Cross beam**
- Section>**7: Cross beam**
- Generation Type>**Translate**
- Translation > dx, dy, dz>**0, 2. 5, 0**
- Number of times >**4**
- Click Apply
- In the Tree Menu: Geometry / Elements / Mirror
- Mode >**Copy**
- Reflection >z-x plane x > 11. 348m
- Copy > Distances > (dx, dy, dz) >**0, 0, 0**
- Click Apply
- Click Close

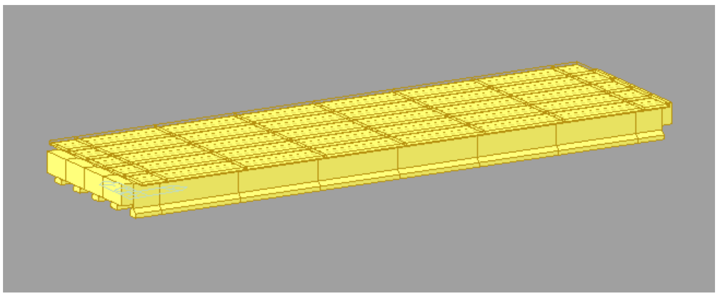


Figure 4-33: 3-D of Super Structure Modeling

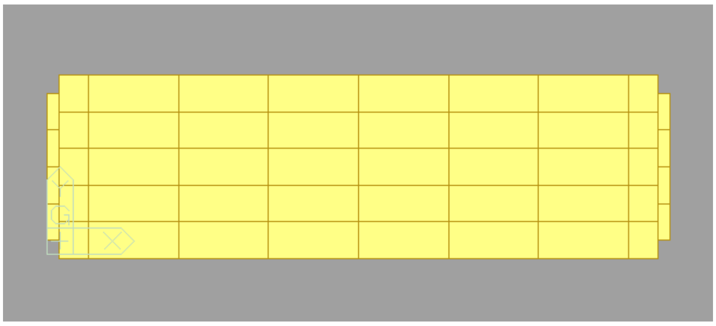


Figure 4-34: plan of Super Structure Modeling

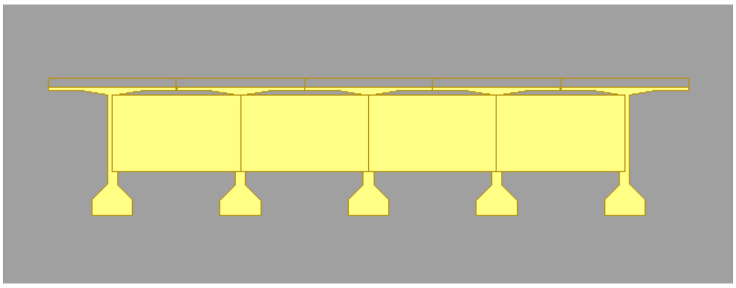


Figure 4-35: side view of Super Structure Modeling

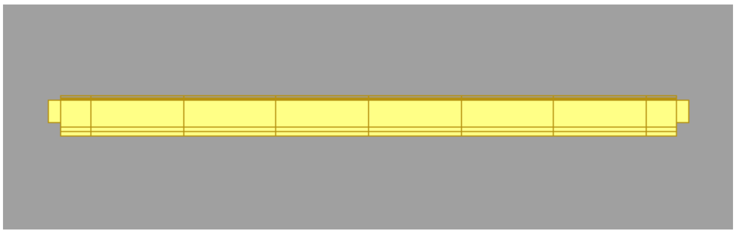


Figure 4-36: Elevation of Super Structure Modeling

6. Change element dependent Material:

● In the Tree Menu: Geometry / Properties / Change Element Dependent Material Property

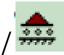
Select all nodes

● Element Dependent Material >**Notational Size of Member**

● Select **Auto Calculate**

● Click Apply ● Click Close

7. Structure Support Conditions: (See figure 4.37)

● In the Tree Menu: Geometry / Boundaries /  Support

At ends of Girders (D_x, D_y, D_z, R_x, R_z).

At symmetrical edge (D_x, D_y, R_x, R_z).

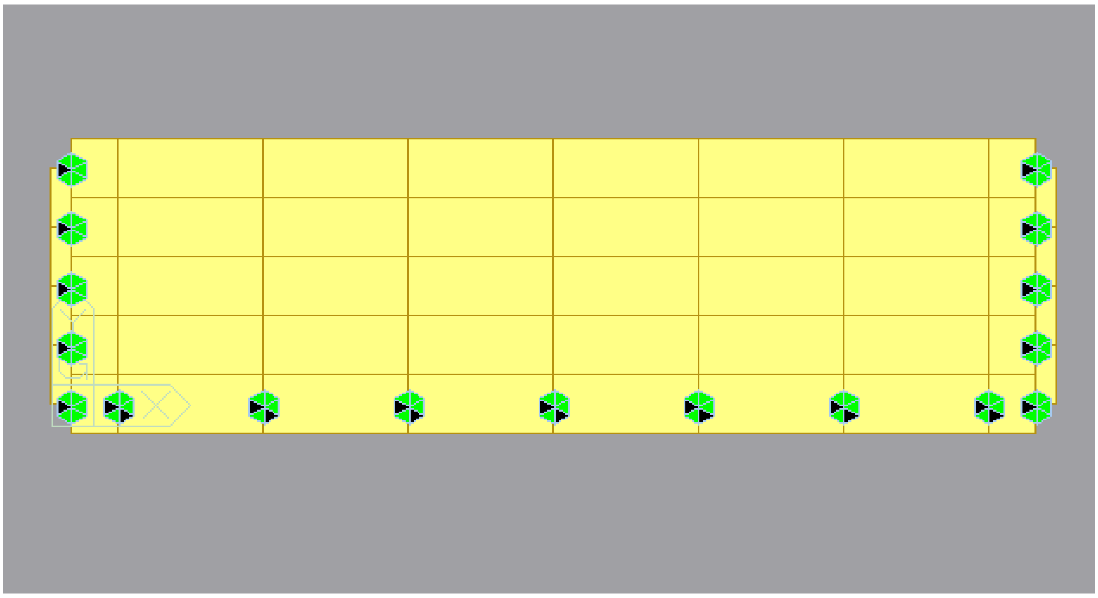


Figure 4-37: Boundary condition

8. Loading data:

The following items are defined in this section:

1. Deck
2. Wearing surface
3. Barrier
4. PC & C/B

5. Prestress 6. Creep

7. Shrinkage 8. Side walk

Load Group:

- In the Tree Menu: Click Group tab
- Right click Load Group
- Select New…
- Name >**Deck** • Click Add
- Name >**Wearing surface** • Click Add
- Name >**Barrier** • Click Add
- Name >**PC & C/B** • Click Add
- Name >**Pre-stress** • Click Add
- Name >**Creep** • Click Add
- Name >**Shrinkage** • Click Add
- Name >**Side walk** • Click Add
- Click Close

Static Loads:

- In the Tree Menu: Click Menu tab
- Static Loads > Static Load Cases
- Name >**Deck**

- Type >**Dead Load of Component and Attachments (DC)**
- Click Add
- Name >**Wearing surface**
- Type >**Dead Load of Wearing Surfaces and Utilities (DW)**
- Click Add
- Name >**Barrier**
- Type >**Dead Load of Wearing Surfaces and Utilities (DW)**
- Click Add
- Name >**PC & C/B**
- Type >**Dead Load of Component and Attachments (DC)**
- Click Add
- Name >**Pre-stress**
- Type >**Pre-stress(PS)**
- Click Add
- Name >**Creep**
- Type >**Creep(CR)**
- Click Add
- Name >**Shrinkage**
- Type >**Shrinkage(SH)**
- Click Add
- Name >**Side walk**
- Type >**Over Load Live Load (LP)**
- Click Add
- Click Close

- Click Select Window > Elements 1 to 8
- In the Tree Menu: Static Loads > Element Beam Loads
- Load Case Name >**Deck**
- Load Group Name >**Deck**
- Direction >**Global Z**
- Projection >**No**
- Value >**Relative**
- W =**-20. 625KN/m**
- Click Apply
- Select Identity all other elements expect elements **1** to **8**.
- In the Tree Menu: Static Loads > Element Beam Loads
- Load Case Name >**Deck** (See figure 4-38)
- Load Group Name >**Deck**
- Direction >**Global Z**
- Projection >**No**
- Value >**Relative**
- W =**-11. 25KN/m**
- Click Apply

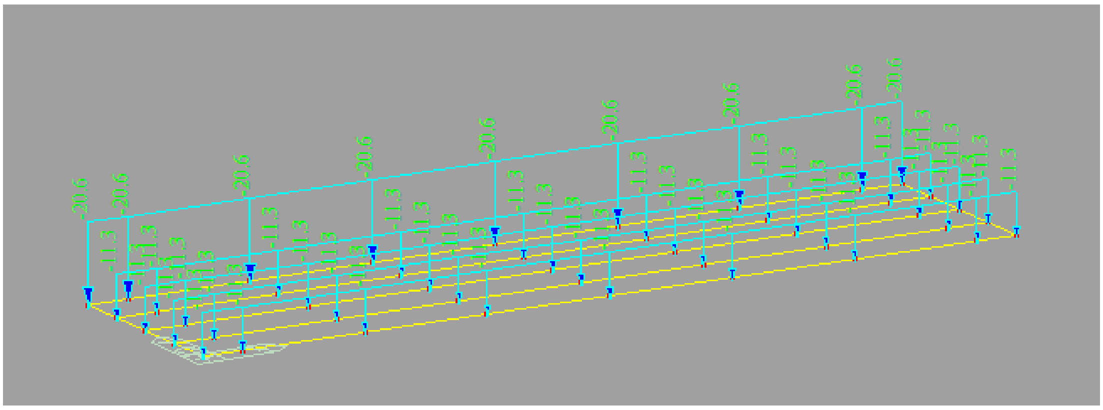


Figure 4-38: Deck slab loading

- Click Select Window > Girders
- In the Tree Menu: Static Loads > Element Beam Loads
- Load Case Name >**Wearing surface** (See figure 5-39)
- Load Group Name >**Wearing surface**
- Direction >**Global Z**
- Projection >**No**
- Value >**Relative**
- W =**-14. 18KN/m**
- Click Apply

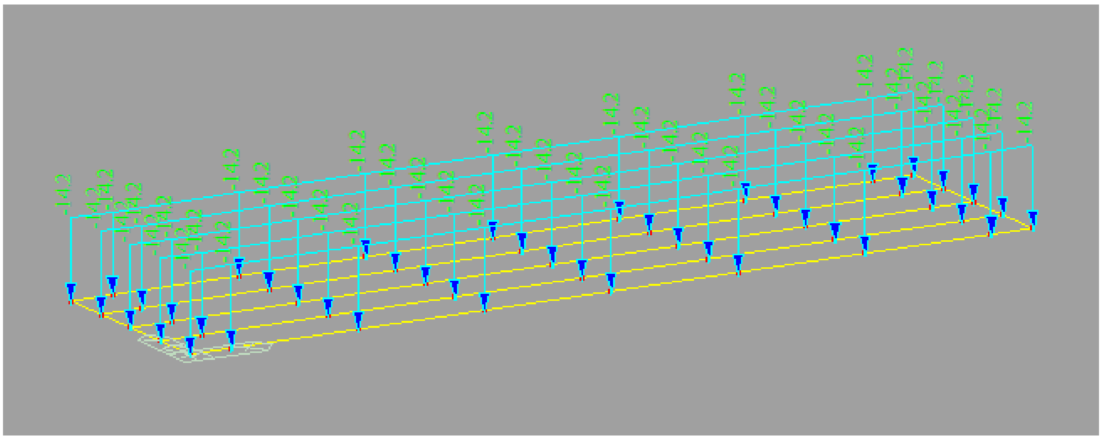


Figure 4-39:wearing surface loading

- Click Select Previous
- In the Tree Menu: Static Loads > Element Beam Loads
- Load Case Name >**Barrier** (See figure 4-40)
- Load Group Name >**Barrier**
- Direction >**Global Z**
- Projection >**No**
- Value >**Relative**
- W =**-1. 8KN/m**

- Click Apply

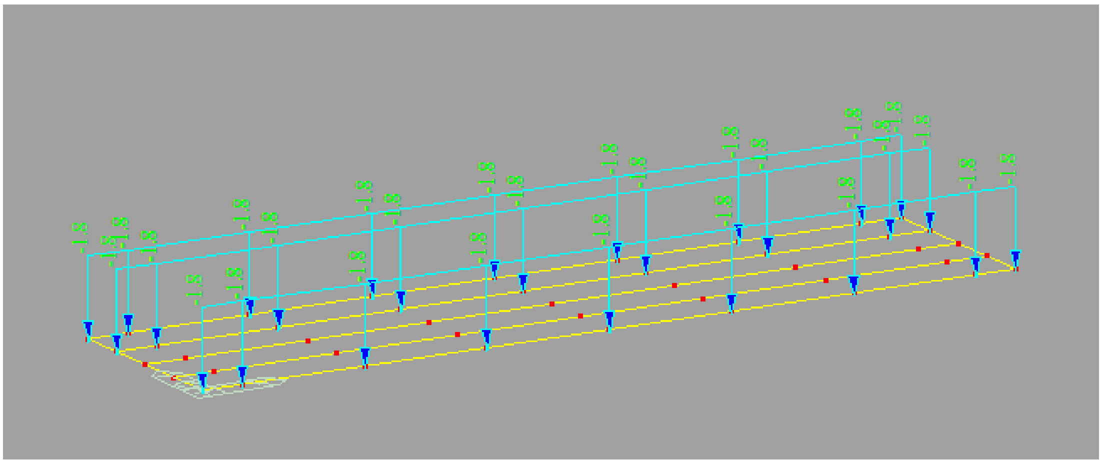


Figure 4-40: Barrier loading

- In the Tree Menu: Static Loads > Self Weight (See figure 4-41)
- Load Case Name >**PC & C/B**
- Load Group Name >**PC & C/B**
- Self Weight Factor >**Z**>**-1**
- Click Add
- Click Close

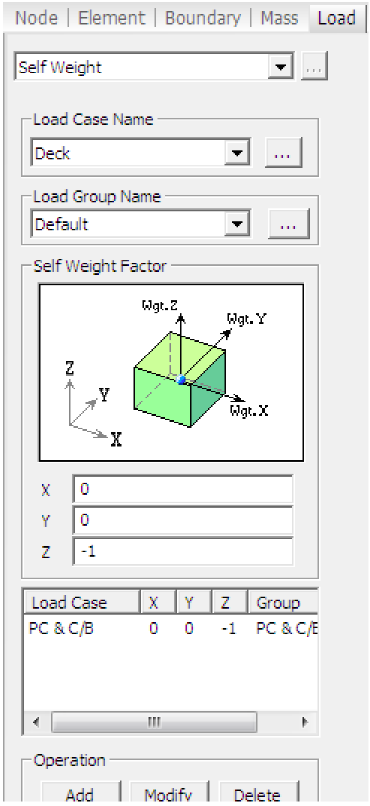


Figure 4-41: Self Weight

- Select Window > Elements **1** to **8**
- In the Tree Menu: Static Loads > Element Beam Loads
- Load Case Name >**Side walk** (See figure 4-42)
- Load Group Name >**Side walk**
- Direction >**Global Z**
- Projection >**No**
- Value >**Relative**
- W =**-4. 25KN/m**
- Click Apply

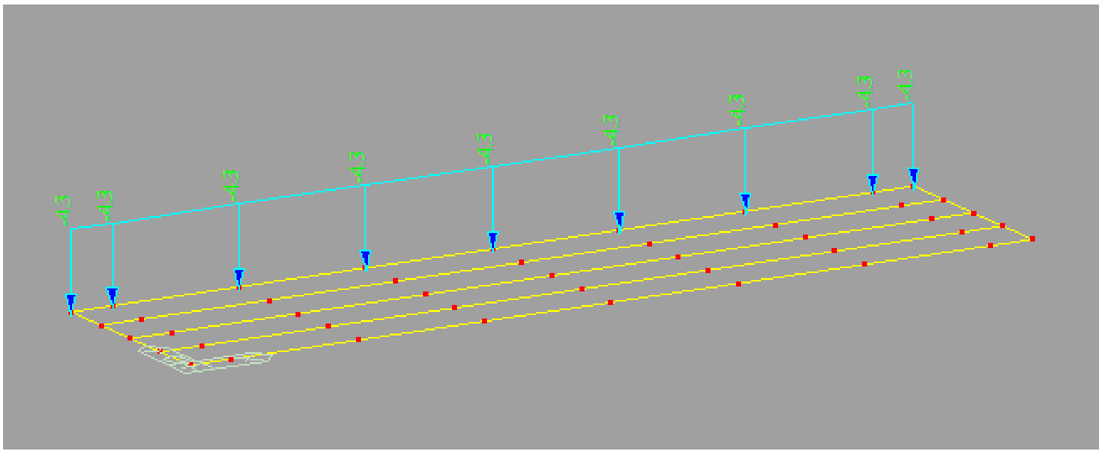


Figure 4-42: Side Walk Loading

9. Prestress Data and Loading:

● Inthe Tree Menu: Click Group tab

● Right click **Tendon Group**

● Select New…

● Name >**Tendon**● Suffix >**1 to 25**

● Click Add ● Click Close

Static Loads:

● In the Tree Menu: Click Menu tab

● Static Loads >Prestress Loads > **Tendon Property**

● Tendon Property dialog box >Click Add

● Tendon Name >**TH** (See figure 4-43)

● Tendon Type >**Internal (Post-Tension)**

● Material >**3:Tendon**

● Click --- to the right of **Total Tendon Area**

- Tendon Area dialog box >Strand Diameter>**15.2mm**
- Number of Strands >**13** • Click OK
- Duct Diameter >**0.08m**
- Select **Relaxation Coefficient**
- Relaxation Coefficient >Magura>**45**
- Ultimate Strength >**1. 86e+006KN/m^2**
- Yield Strength >**1. 674e+006KN/m^2**
- Curvature Friction Factor >**0. 3**
- Wobble Friction Factor >**0. 00331/m**
- Anchorage Slip (Draw in):

Begin: **0. 006m**End: **0. 006m**

- Bond Type >**Bonded** • Click OK

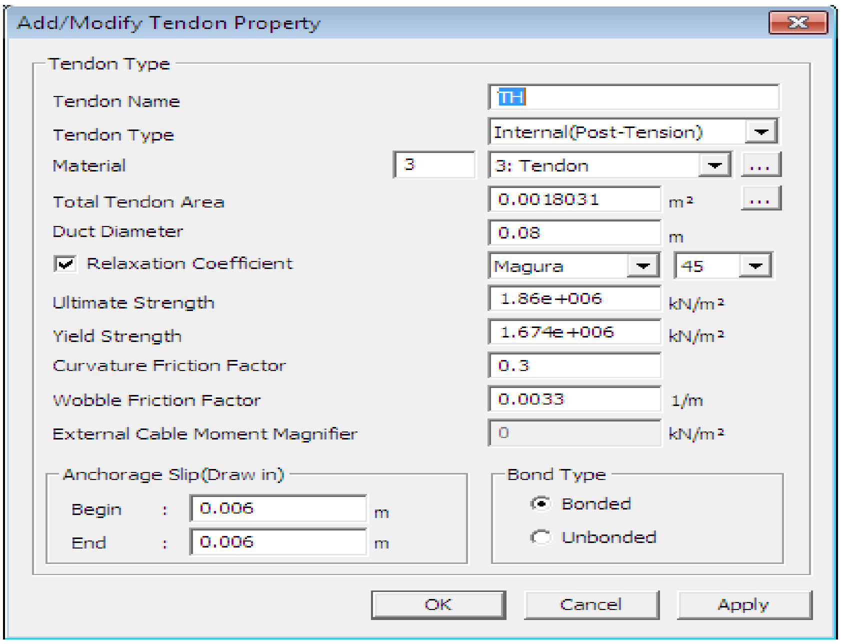


Figure 4-43: Tendon properties (TH)

● Tendon Property dialog box >Click

● Tendon Name >**TS** (See figure 4-44)

● Tendon Type >**Internal (Post-Tension)** ● Material >**3:Tendon**

● Click to the right of **Total Tendon Area** ---

● Tendon Area dialog box >Strand Diameter>**15.2mm**

● Number of Strands >**13** ● Click OK

● Duct Diameter >**0. 08m** ● Select **Relaxation Coefficient**

● Relaxation Coefficient >Magura>**45**

● Ultimate Strength >**1. 86e+006KN/m**^{2}

● Yield Strength >**1. 674e+006KN/m**^{2}

● Curvature Friction Factor >**0. 3**

● Wobble Friction Factor >**0. 00331/m**

● Anchorage Slip (Draw in):

Begin: **0. 006m**End: **0. 006m**

● Bond Type >**Bonded** ● Click OK

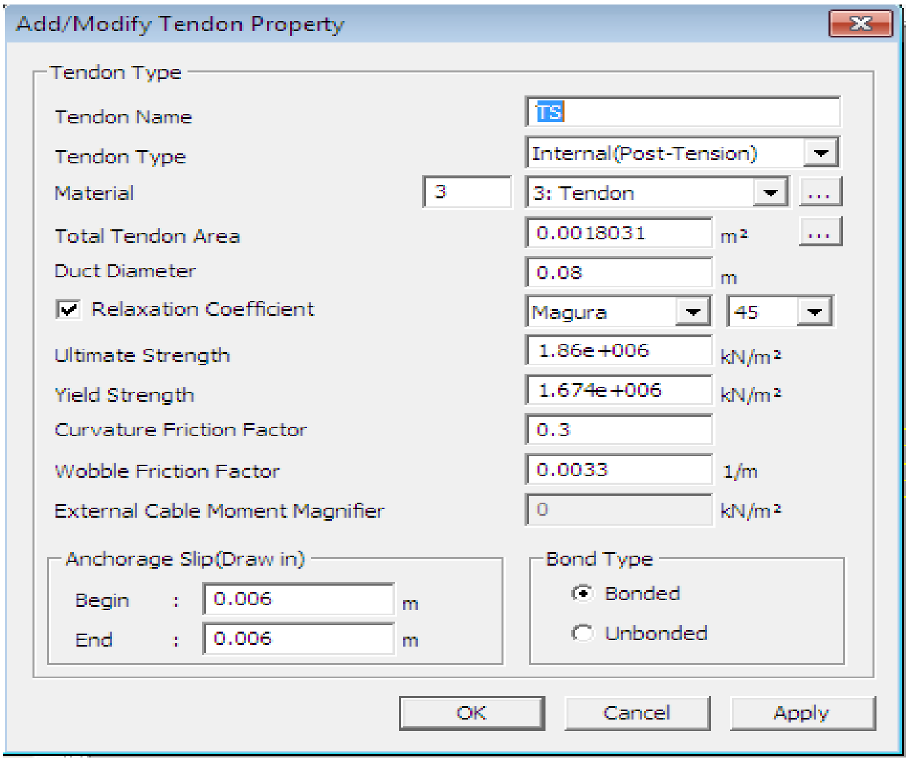


Figure 4-44: Tendon Properties (TS)

- In the Tree Menu: Static Loads >Pre-stress Loads > **Tendon Profile** (See figure 4-45 and figure 4-46)
- Tendon Profile dialog box >Click Add
- Tendon Name >**TH1** (See Table 4-9)
- Group >**Tendon1**
- Tendon Property >**TH**
- Click in Assign Elements:
- Select Window >**Elements33 to 40**
- Input Type >**3D**
- Curve Type >**Spine**
- Profile Reference Axis >**Straight**

● Enter the Following Data in the Profile Window:

Table 4-9 Profile of TH1 , TH4 , TH7 , TH10and TH13

Number	X(m)	Z(m)
1	0	0. 6736
2	0. 15	0. 6475
3	1. 15	0. 4735
4	2	0. 3295
5	4	0. 0165
6	6	-0. 2605
7	8	-0. 5005
8	10	-0. 7035
9	12	-0. 8695
10	14	-0. 9995
11	16	-1. 0925
12	18	-1. 1485
13	20. 05	-1. 1675
14	20. 45	-1. 171
15	20. 85	-1. 1675
16	22. 9	-1. 1485
17	24. 9	-1. 0925
18	26. 9	-0. 9995
19	28. 9	-0. 8695
20	30. 9	-0. 7035
21	32. 9	-0. 5005
22	34. 9	-0. 2605
23	36. 9	0. 0165
24	38. 9	0. 3295
25	39. 75	0. 4735
26	40. 75	0. 6475
27	40. 9	0. 6736

- Point of Sum > Last
- Profile insertion point > 0, 0, 0 m
- X Axis Direction > X
- Click OK
- Tendon Profile dialog box >Select **TH1**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to **TH4** (See Table 4-9)
- Group > Change to**tendon6**
- Click OK
- Tendon Profile dialog box >Select **TH4**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH7** (See Table 4-9)
- Group > Change to**tendon11**
- Click OK
- Tendon Profile dialog box >Select **TH7**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH10** (See Table 4-9)
- Group > Change to**Tendon16**
- Click OK
- Tendon Profile dialog box >Select **TH10**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify

- Tendon Name > Change to**TH13** (See Table 4-9)
- Group > Change to**Tendon21**
- Click OK
- In the Tree Menu: Static Loads >Pre-stress Loads > **Tendon**

Profile

- Tendon Profile dialog box >Click Add
- Tendon Name >**TH2** (See Table 4.10)
- Group >**Tendon2**
- Tendon Property >**TH**
- Click in Assign Elements:
- Select Window >**Elements33 to 40**
- Input Type >**3D**
- Curve Type >**Spine**
- Profile Reference Axis >**Straight**

● Enter the Following Data in the Profile Window:

Table 4-10 Profile of TH2 , TH5 , TH8 , TH11 and TH14

Number	X(m)	Z(m)
1	0	0. 3714
2	0. 15	0. 3475
3	1. 15	0. 1885
4	2	0. 0555
5	4	-0. 2315
6	6	-0. 4855
7	8	-0. 7055
8	10	-0. 8915
9	12	-1. 0445
10	14	-1. 1635
11	16	-1. 2485
12	18	-1. 2995
13	20. 05	-1. 3175
14	20. 45	-1. 321
15	20. 85	-1. 3175
16	22. 9	-1. 2995
17	24. 9	-1. 2485
18	26. 9	-1. 1635
19	28. 9	-1. 0445
20	30. 9	-0. 8915
21	32. 9	-0. 7055
22	34. 9	-0. 4855
23	36. 9	-0. 2315
24	38. 9	0. 0555
25	39. 75	0. 1885
26	40. 75	0. 3475
27	40. 9	0. 3714

- Point of Sum > Last
- Profile insertion point > 0, 0, 0 m
- X Axis Direction > X
- Click OK
- Tendon Profile dialog box >Select **TH2**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH5**(See Table 4.10)
- Group > Change to **tendon7**
- Click OK
- Tendon Profile dialog box >Select **TH5**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH8** (See Table 4.10)
- Group > Change to **tendon12**
- Click OK
- Tendon Profile dialog box >Select **TH8**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH11** (See Table 4.10)
- Group > Change to **tendon17**
- Click OK
- Tendon Profile dialog box >Select **TH11**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify

● Tendon Name > Change to **TH14** (See Table 4.10)

● Group > Change to **tendon22** ● Click OK

● In the Tree Menu: Static Loads >Prestress Loads > **Tendon**

Profile

● Tendon Profile dialog box >Click Add

● Tendon Name >**TH3** (See Table 4.11)

● Group >**Tendon3**

● Tendon Property >**TH**

● Click in Assign Elements: ● Select Window >**Elements33 to 40**

● Input Type >**3D** ● Curve Type >**Spline**

● Profile Reference Axis >**Straight**

● Enter the Following Data in the Profile Window:

Table 4-11 Profile of TH3 , TH6 , TH9 , TH12 and TH15

Number	X(m)	Z(m)
1	0	0. 0693
2	0. 15	0. 0475
3	1. 15	-0. 0975
4	2	-0. 2175
5	4	-0. 4795
6	6	-0. 7105
7	8	-0. 9105
8	10	-1. 0805
9	12	-1. 2185
10	14	-1. 3275
11	16	-1. 4045
12	18	-1. 4515
13	20. 05	-1. 4675
14	20. 45	-1. 471
15	20. 85	-1. 4675
16	22. 9	-1. 4515
17	24. 9	-1. 4045
18	26. 9	-1. 3275
19	28. 9	-1. 2185
20	30. 9	-1. 0805
21	32. 9	-0. 9105
22	34. 9	-0. 7105
23	36. 9	-0. 4795
24	38. 9	-0. 2175
25	39. 75	-0. 0975
26	40. 75	0. 0475
27	40. 9	0. 0693

- Point of Sum > Last
- Profile insertion point > 0, 0, 0 m
- X Axis Direction > X
- Click OK
- Tendon Profile dialog box >Select **TH3**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH6** (See Table 4.11)
- Group > Change to**tendon8**
- Click OK
- Tendon Profile dialog box >Select **TH6**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH9** (See Table 4.11)
- Group > Change to **tendon13**
- Click OK
- Tendon Profile dialog box >Select **TH9**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TH12** (See Table 4.11)
- Group > Change to **tendon18**
- Click OK
- Tendon Profile dialog box >Select **TH12**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify

● Tendon Name > Change to **TH15** (See Table 4.11)

● Group > Change to **tendon23** ● Click OK

● In the Tree Menu: Static Loads > Prestress Loads > **Tendon**

Profile

● Tendon Profile dialog box >Click Add

● Tendon Name >**TS1** ● Group >**Tendon4**

● Tendon Property >**TS**

● Click in Assign Elements: ● Select Window >**Elements33 to 40**

● Input Type >**3D** ● Curve Type >**Spline**

● Profile Reference Axis >**Straight**

● Enter the Following Data in the Profile Window:

Table 4-12 Profile of TS1 , TS3 , TS5 , TS7 and TS9

Number	X(m)	Y(m)	Z(m)
1	0	-0. 15	-1. 2494
2	0. 15	-0. 15	-1. 2525
3	1. 15	-0. 15	-1. 2735
4	2	-0. 15	-1. 2905
5	4	-0. 15	-1. 3275
6	6	-0. 15	-1. 3605
7	8	-0. 15	-1. 3885
8	10	-0. 15	-1. 4125
9	12	-0. 15	-1. 4325
10	14	-0. 15	-1. 4475
11	16	-0. 15	-1. 4585
12	18	-0. 15	-1. 4655
13	20. 05	-0. 15	-1. 4675
14	20. 45	-0. 15	-1. 468
15	20. 85	-0. 15	-1. 4675
16	22. 9	-0. 15	-1. 4655
17	24. 9	-0. 15	-1. 4585
18	26. 9	-0. 15	-1. 4475
19	28. 9	-0. 15	-1. 4325
20	30. 9	-0. 15	-1. 4125
21	32. 9	-0. 15	-1. 3885
22	34. 9	-0. 15	-1. 3605
23	36. 9	-0. 15	-1. 3275
24	38. 9	-0. 15	-1. 2905
25	39. 75	-0. 15	-1. 2735
26	40. 75	-0. 15	-1. 2525
27	40. 9	-0. 15	-1. 2494

- Point of Sum > Last
- Profile insertion point > 0, 0, 0 m
- X Axis Direction > X
- Click OK
- Tendon Profile dialog box >Select **TS1** (See Table 4.12)
- Distance > 0, 0, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TS2**
- y (m) > Change to **0. 15m**
- Group > Change to **tendon5**
- Click OK
- Tendon Profile dialog box >Select **TS1**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TS3**
- Group > Change to **tendon9**
- Click OK
- Tendon Profile dialog box >Select **TS3**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TS5**
- Group > Change to**tendon14**
- Click OK
- Tendon Profile dialog box >Select **TS5**

- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to **TS7**
- Group > Change to**tendon19**
- Click OK
- Tendon Profile dialog box >Select **TS7**
- Distance > 0, 2. 69, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TS9**
- Group > Change to **tendon24**
- Click OK
- Tendon Profile dialog box >Select **TS2**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TS4**
- Group > Change to**tendon10**
- Click OK
- Tendon Profile dialog box >Select **TS4**
- Distance > 0, 2. 5, 0
- Click Copy / Move
- Select Modify
- Tendon Name > Change to**TS6**
- Group > Change to**tendon15**
- Click OK
- Tendon Profile dialog box >Select **TS6**

- Distance > 0, 2. 5, 0
- Click **Copy / Move**
- Select **Modify**
- Tendon Name > Change to**TS8**
- Group > Change to**tendon20**
- Click **OK**
- Tendon Profile dialog box >Select **TS8**
- Distance > 0, 2. 5, 0
- Click **Copy / Move**
- Select **Modify**
- Tendon Name > Change to**TS10**
- Group > Change to**tendon25**
- Click **OK**

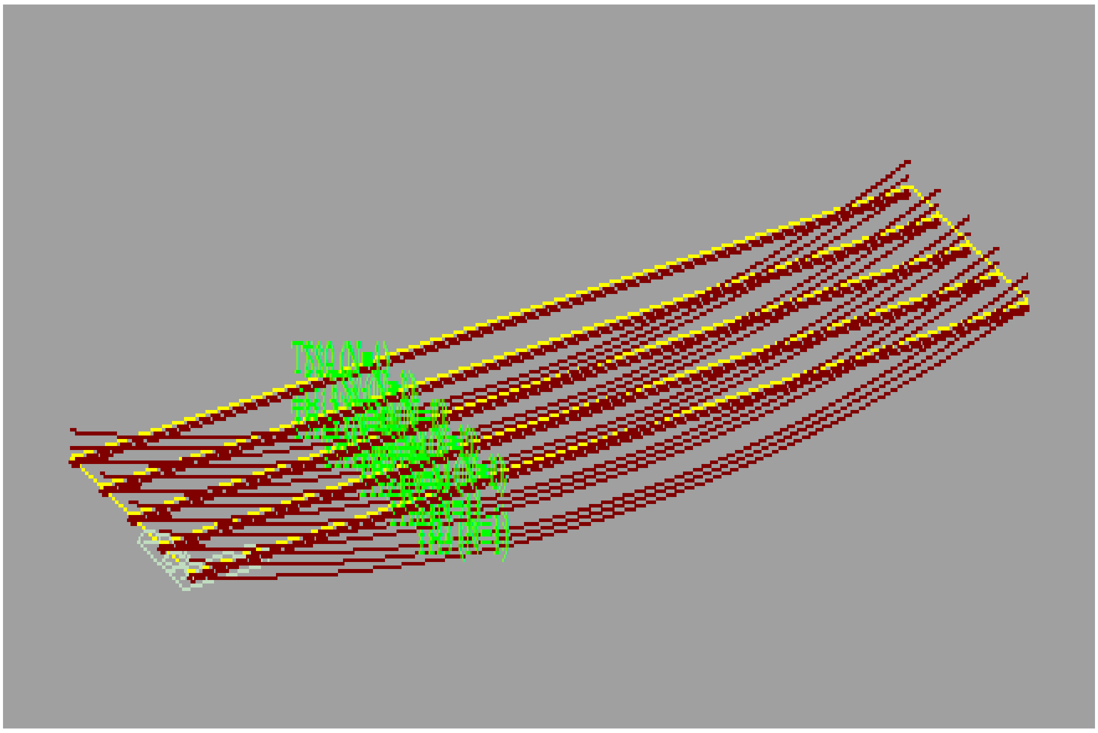


Figure 4-45:3-D of Tendons on girders

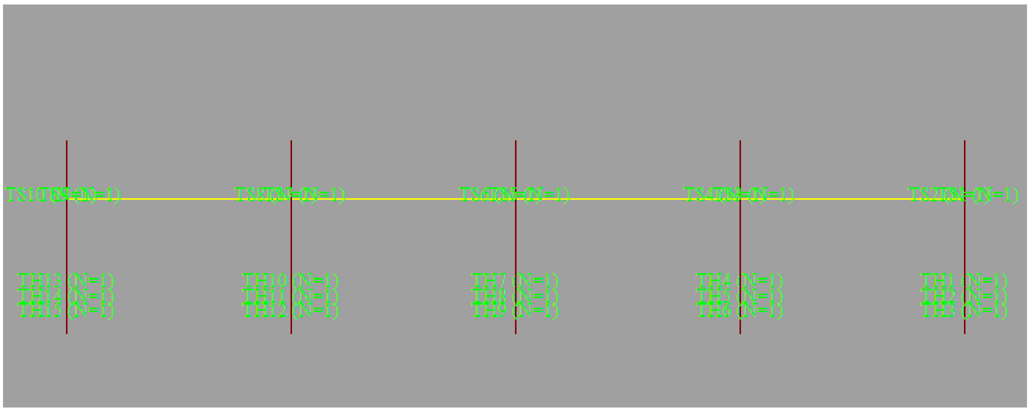


Figure 4-46:Side View of Tendons on girders

- In the Tree Menu: Click Menu tab
- Static Loads >Pre-stress Loads > **Tendon Pre-stress Loads** (See figure 4.47)
- Load Case Name >**Pre-stress**
- Load Group Name >**Pre-stress**
- Select Tendon for Loading > Tendon > Select all Tendons

(TH1…. .TH15, TS1……TS10)

- Click [>]
- Stress Value > Stress
- 1^{St} Jacking > Begin

Begin > 1. 083×10^6**KN/m^2**

End > 807. 24×10^3**KN/m^2**

- Click [Add]
- Click [Close]
- Click

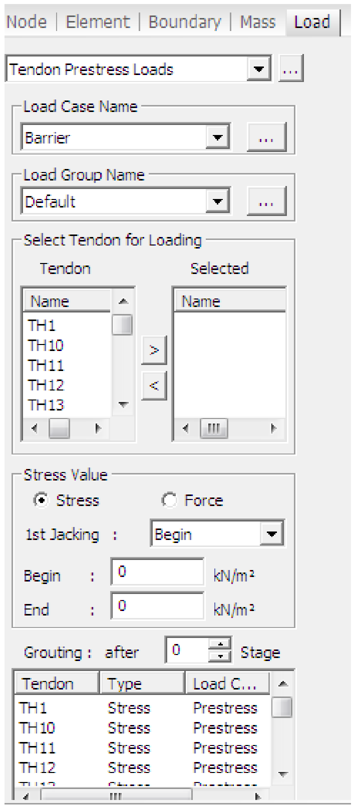


Figure 4-47:Tendon Prestress Loads Window

10. Moving Loads:

●In the Tree Menu: Click Menu tab

●Moving Loads Analysis> **Moving Load Code** (See figure 4.48)

●Select Moving Load Code dialog box>Moving Load Code>**AASHTO LRFD**

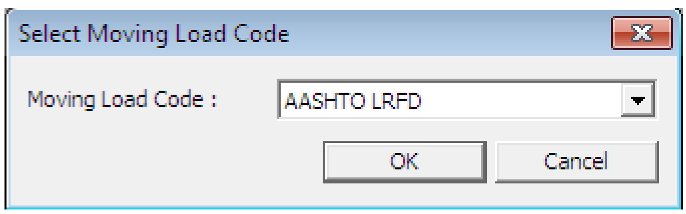


Figure 4-48:Moving Load Code

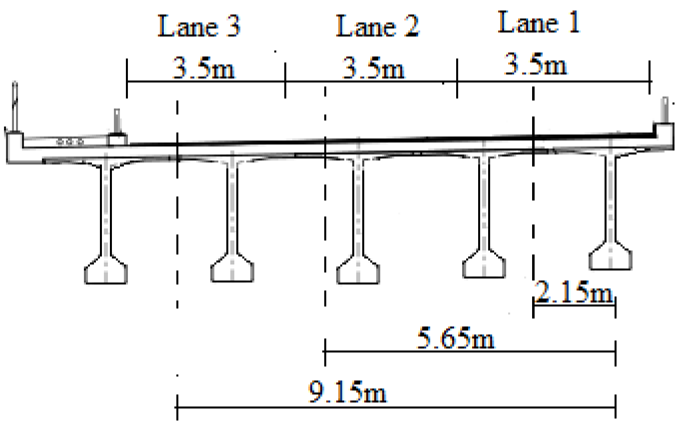


Figure 4-49:Traffic Lanes and their Eccentricities

●Moving Loads Analysis> **Traffic Line Lanes** (See figure 4.49)

● Traffic Line Lanes dialog box>Click Add

●Lane Name>**Lane 1** (See figure 4.50)

●Eccentricity>**-2. 15**

●Vehicular Load Distribution>**Lane element**

●Moving Direction>**Both** ●Selection by>**2 points**

●Click in the first box below **And** Select Window >**Nodes 1 and 12**

●Click Add ●Click OK

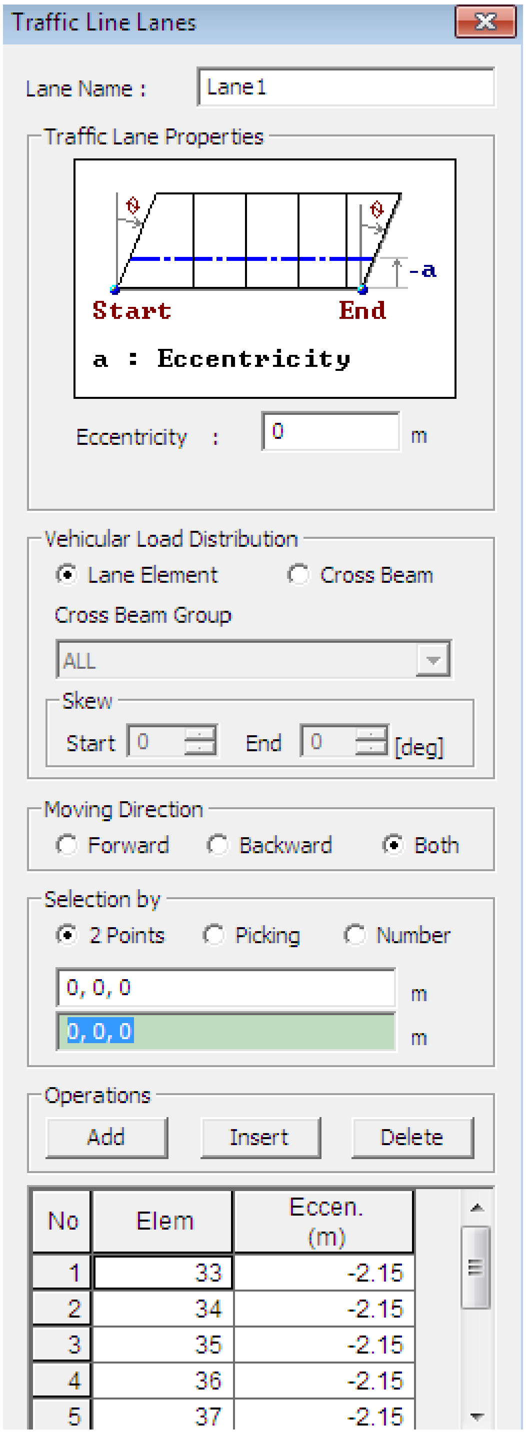


Figure 4-50:Definition of Design Traffic Line Lane1

●Traffic Line Lanes dialog box>Click Add

●Lane Name>**Lane 2** (See figure 4.51)

●Eccentricity>**-5. 65m**

● Vehicular Load Distribution>**Lane element**

●Moving Direction>**Both** ●Selection by>**2 points**

●Click in the first box below **And** Select Window  >**Nodes 1 and 12**

●Click Add ● Click OK

●Traffic Line Lanes dialog box>Click Add

●Lane Name>**Lane 3** (See figure 4.52)

●Eccentricity>**-9. 15m**

●Vehicular Load Distribution>**lane element**

●Moving Direction>**Both** ●Selection by>**2 points**

●Click in the first box below **And** Select Window  >**Nodes 1 and**12

●Click Add ●Click OK

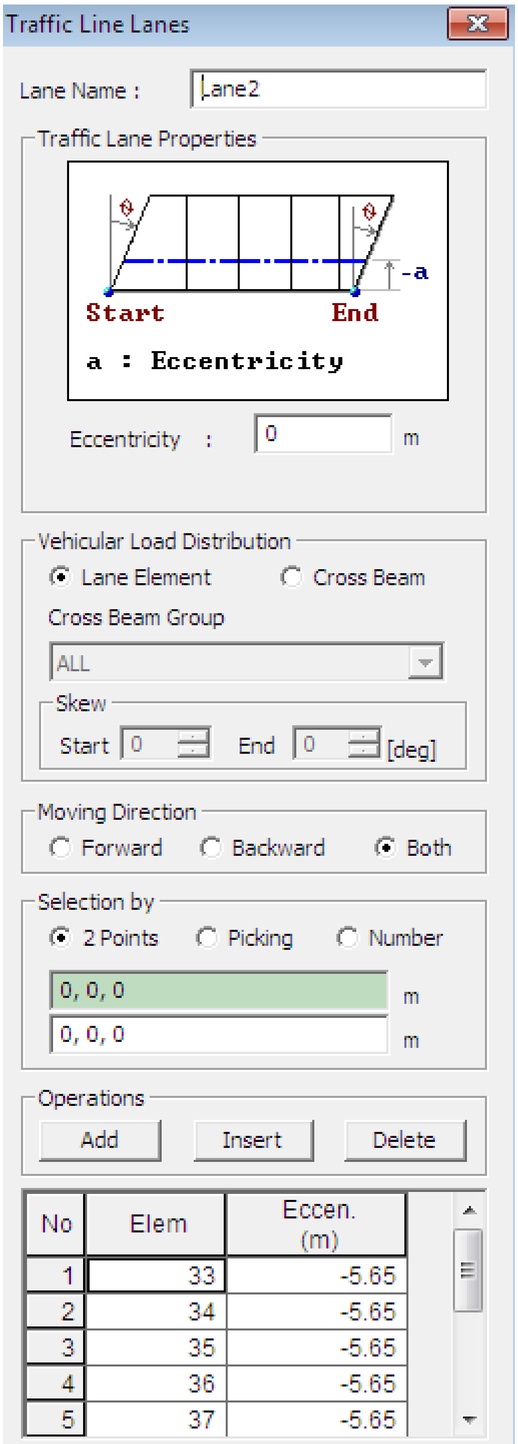


Figure 4-51 :Definition of Design Traffic Line Lane2

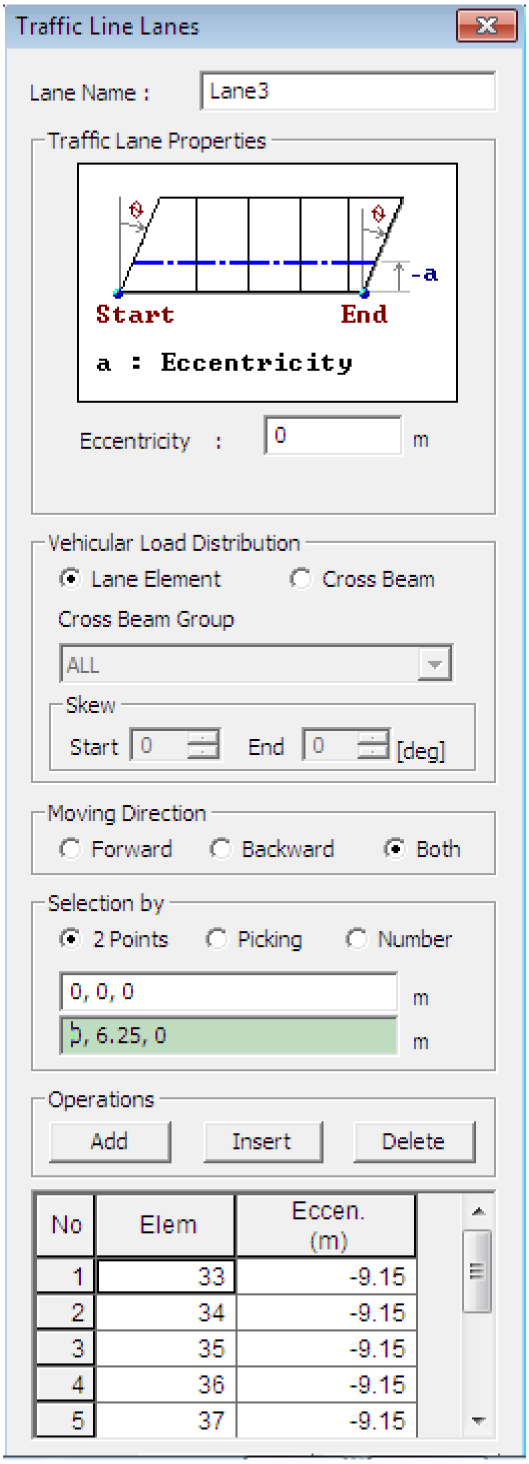


Figure 4-52: Definition of Design Traffic Line Lane3

●In the Tree Menu: Moving Load Analysis >  **Vehicles**

(See figure 4.53)

●Vehicles dialog box>Click Add standard

●Standard Name>**AASHTO LRFD Load**

●Vehicular Load Name>**HL-93TDM**

●Vehicular Load Type>**HL-93TDM**

●Dynamic Allowance: **33 %**

●Click OK

●Vehicles dialog box>Click Add standard

●Standard Name>**AASHTO LRFD Load**

●Vehicular Load Name>**HL-93TRK**

●Vehicular Load Type>**HL-93TRK**

●Dynamic Allowance: **33 %**

●Click OK

●Click Close

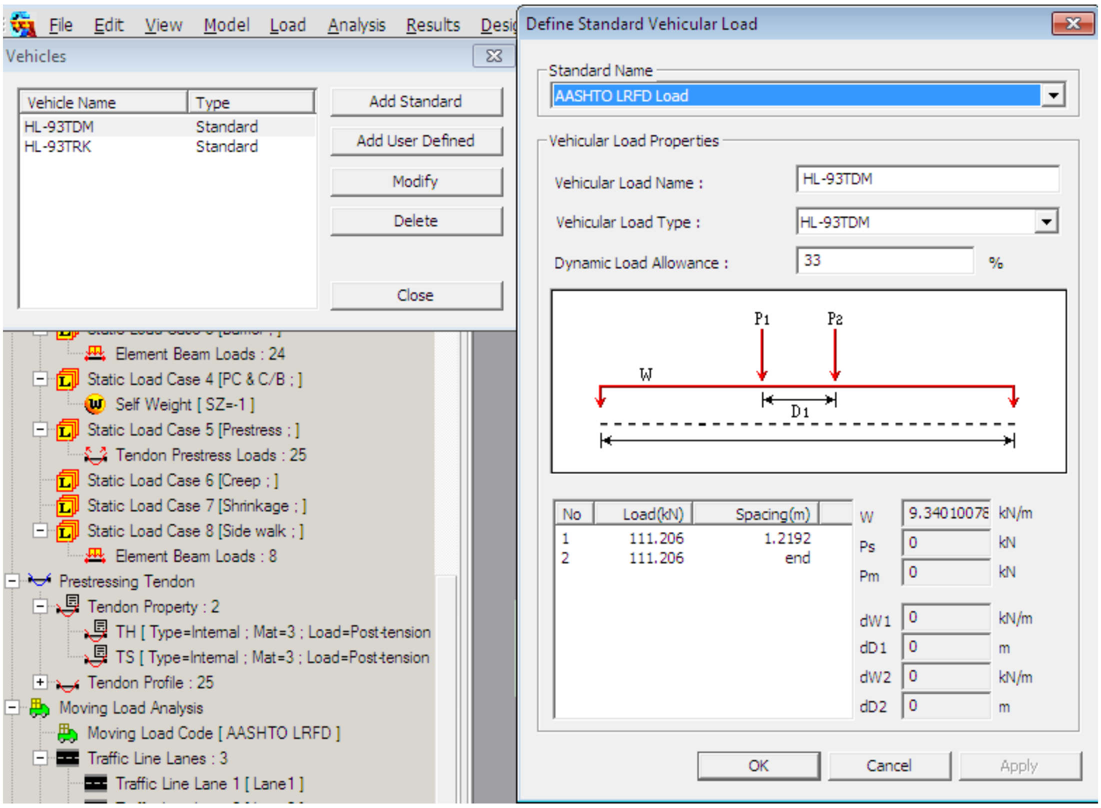


Figure 4-53: Definition of Standard Vehicular Loads

●In the Tree Menu: Moving Load Analysis > **Moving Load Cases** (See table 4-13)

●Moving Load Cases dialog box>Click Add

Table 4-13 Multiple Presence Factors

No. Of Loaded Lanes	M
1	1. 2
2	1. 00
3	0. 85
More than 3	0. 65

●Load Case Name>**MLC** (See figure 4-54)

●Enter the following data in the **Multiple Presence Factor** window:

●Sub-Load Cases>Loading Effect>**Independent**

●Click Add

●Sub-Load Case dialog box>Vehicle Class>**VL:HL-93TDM**

●Scale Factor>**1**

●Min. Number of Loaded Lanes>**1**

●Max. Number of Loaded Lanes>**3**

● Assignment Lanes>List of Lanes>Select all lanes **(Lane 1, Lane 2, Lane 3)**

●Click ->

●Click OK

●Define Moving Load Case dialog box>Click Add

●Sub-Load Case dialog box>Vehicle Class>**VL:HL-93TRK**

●Scale Factor>**1**

●Min. Number of Loaded Lanes>**1**

●Max. Number of Loaded Lanes>**3**

●Assignment Lanes>List of Lanes>Select all lanes **(Lane 1, Lane 2, Lane 3)**

●Click Add

●Click OK

●Click Close

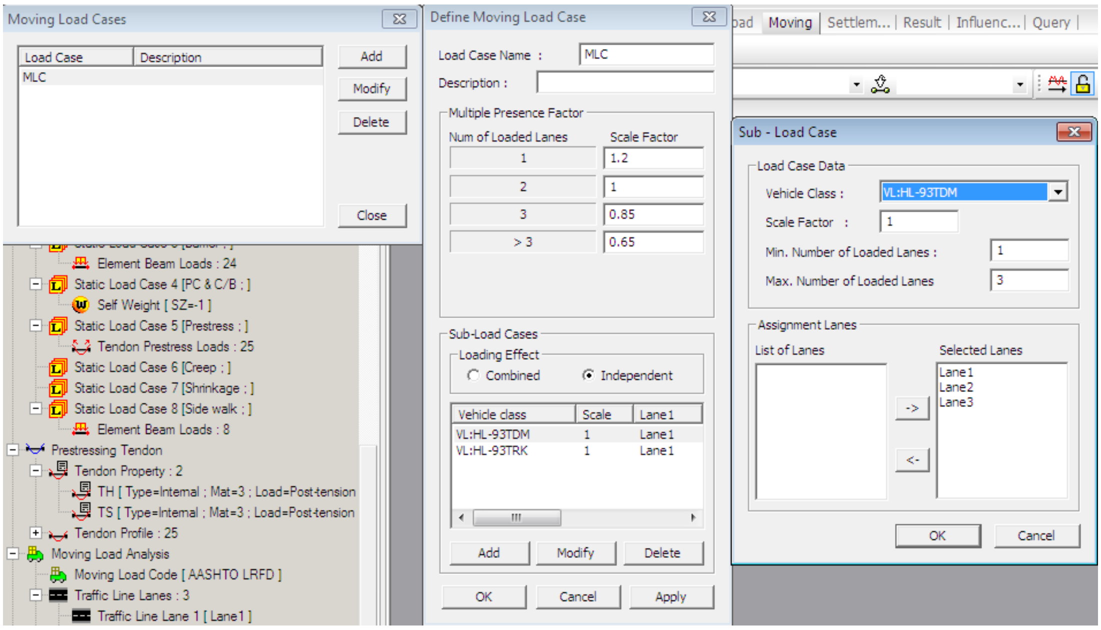


Figure 4-54: Definition of Moving Load Cases

11.Construction Stage Analysis: (See table 4-14)

Data Three stages are defined to model the bridge during construction.

Details of the construction stages are shown below:

Table 4-14 Construction Stages

Stage	Day	Description
Stage 1 (30 days)	1	Placing of precast beams and cross beams. Prestressing of strands.
	21	Pouring deck slab.
Stage 2 (30 days)	1	Composite beam & slab behavior takes place.
	1	Installation of barrier.
	6	Placing of wearing surface.
Stage 3 (10000 days)		

<u>Group:</u>

●In the Tree Menu: Click Group tab

●Right-click **Structure Group**● Select New…

●Name>**All**

●Click Add ● Click close

●Click Select All

● "Drag &Drop" All from the Tree Menu to Model View

●Right-click **Boundary Group** ●Select New…

●Name>**Supports**

●Click Add ● Click close

●Click Select All

● "Drag &Drop" Supports from the Tree Menu to Model View

●Select Boundary Type dialog box>**Support**

●Click OK

<u>Define Construction Stages:</u>

● In the Tree Menu: Click Menu tab

●Construction Stage Analysis Data > **Define Construction Stage**

●Construction Stage dialog box>Click Generate

●Stage>Name>**Stage** ●Stage>Suffix>**1to3**

●Save Result>Stage>**Additional Steps (on)** ●Click OK

●Construction Stage dialog box>Click Add

●Stage>**Stage 1** (See figure 4.55)

●Name>**Stage 1**

●Duration>**30** ●Additional Steps>Day>**21**

●Click in the Additional Steps window Add

●Click **Element** tab ●Group List>**All** ●Activation>Age>**7**

●Click Add in the Activation window

●Click **Boundary** tab ●Group List>**Supports**

●Support/Spring Position>**Deformed**

●Click in the Activation window Add

●Click **Load** tab ●Group List>**PC & C/B**

●Click Add in the Activation window

●Group List>**Pre-stress**

●Click Add in the Activation window

●Group List>**Deck** ●Active Day>**21**

●Click Add in the Activation window

●Click OK

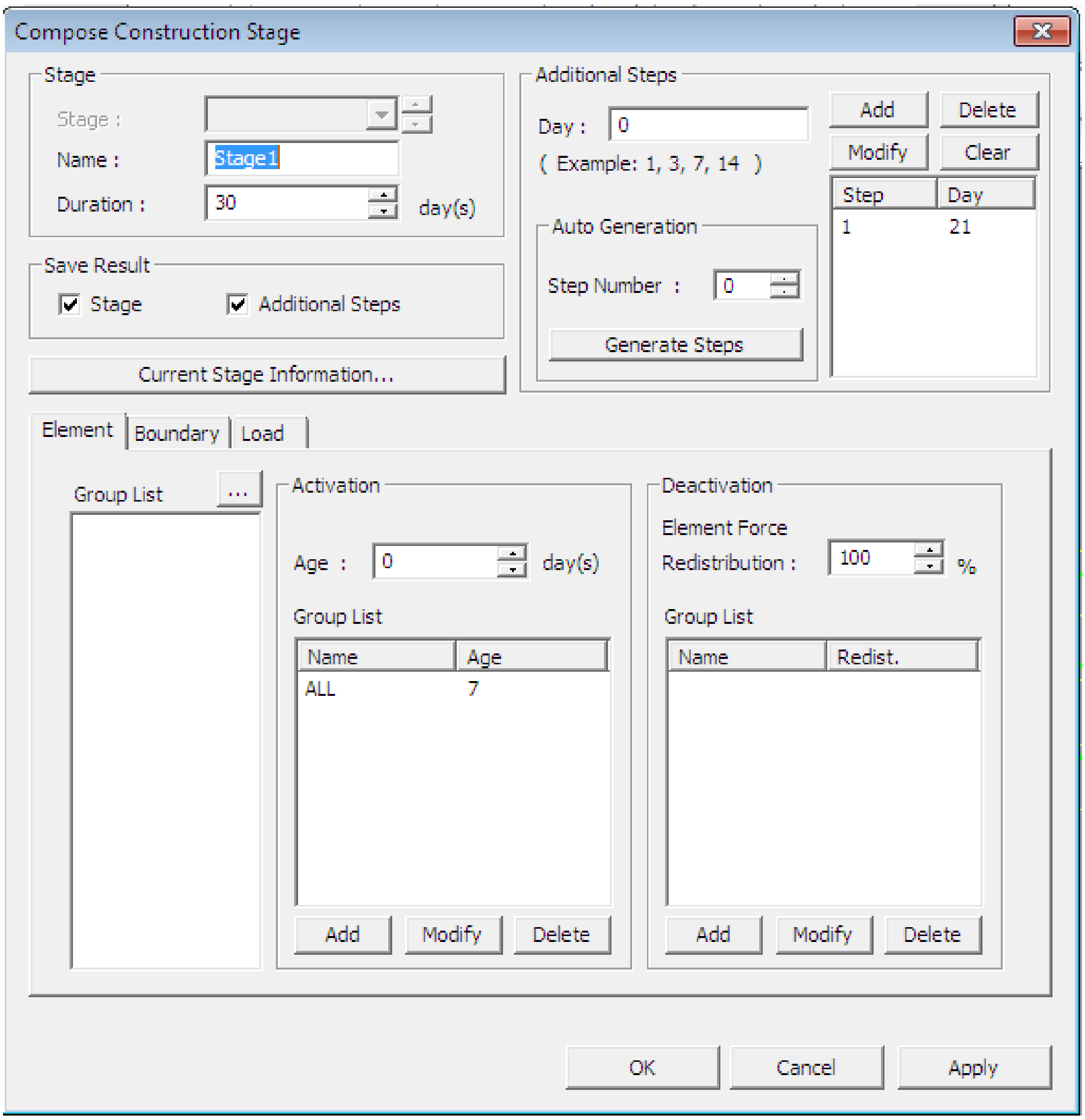


Figure 4-55: Definition of Construction Stage 1

●Construction Stage dialog box>Click

●Stage>**Stage 2** (See figure 4.56)

● Name>**Stage 2**

●Duration>**30**

●Additional Steps>Day>**6**

●Click Add in the Additional Steps window

●Click **Load** tab

●Once activated, the Element, Boundary and Load groups remain active

Unless they are specifically deactivated.

●Group List>**Barrier**

●Click Add in the Activation window

●Group List>**Wearing Surface** ●Active Day>**6**

●Click Add in the Activation window

●Click OK

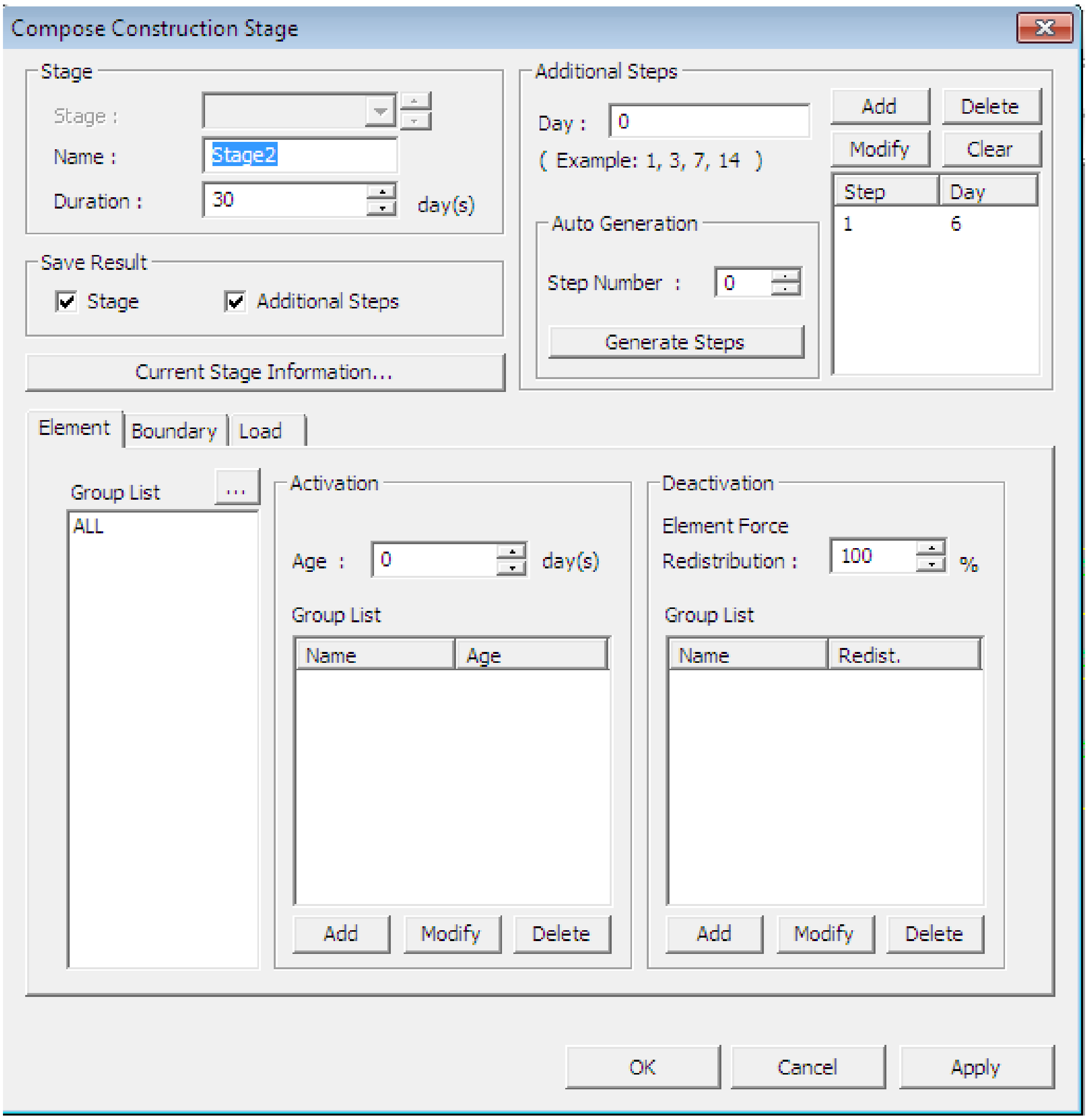


Figure 4-56:Definition of Construction Stage 2

●Construction Stage dialog box>Click

●Stage>**Stage 3** (See figure 4.57)

●Name>**Stage 3**

●Duration>**10000**

●Click OK ● Click Close

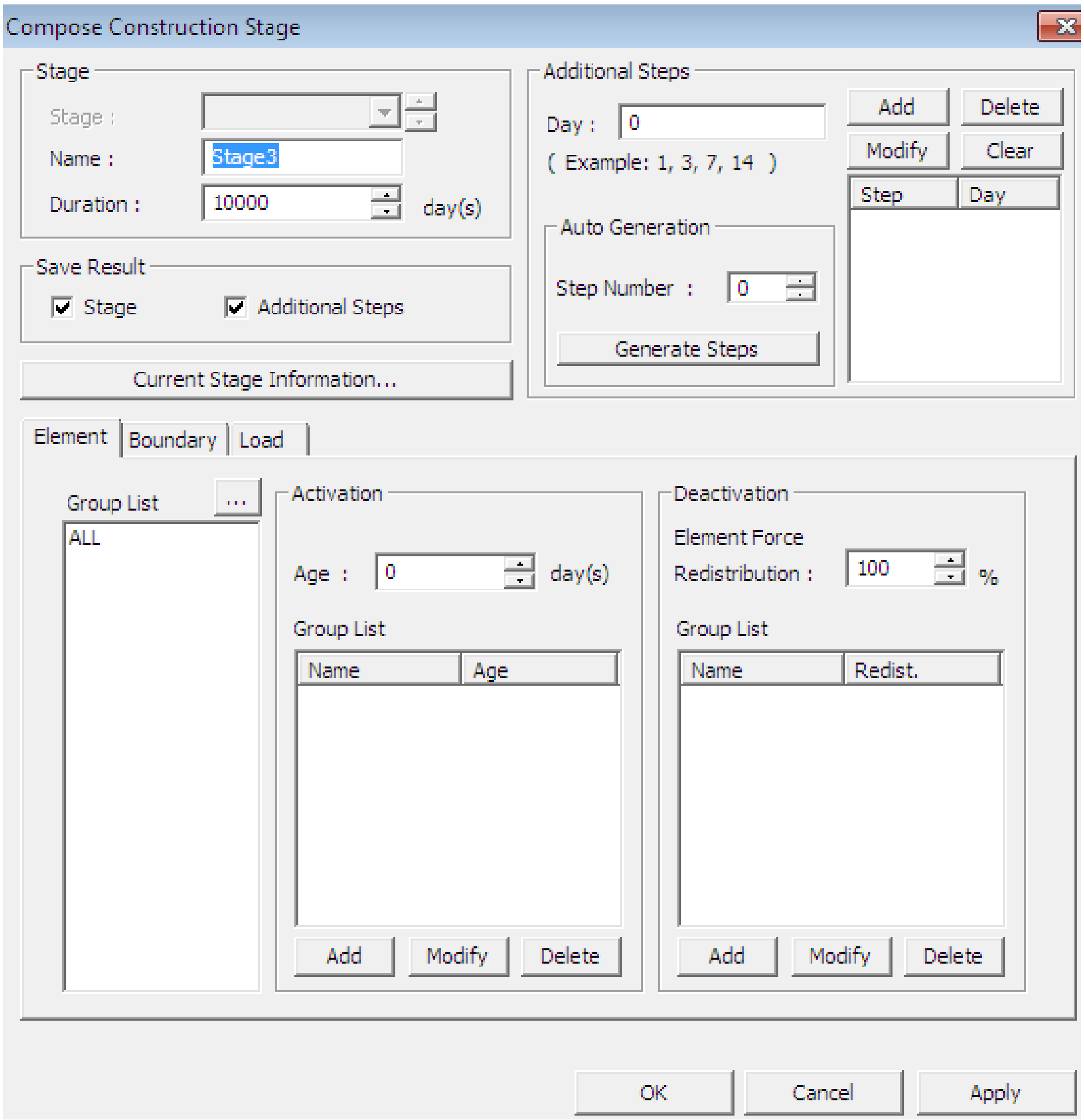


Figure 4-57:Definition of Construction Stage 3

●In the Tree Menu: Construction Stage Analysis Data > **Composite Section for Construction Stage (See figure 4.58)**

●Composite Section for Construction Stage dialog box>Click Add

● Active Stage>**Stage 1**

●Section>**1: Girders**

●Composite Type>**Normal**

●Construction Sequence>Part>**1**:

●Material Type>**Material**

●Material>**2:Precast**

●Composite Stage>**Active Stage**

●Age>**7**

●Construction Sequence>Part>**2**:

●Material Type>**Material**

●Material>**1:Deck**

●Composite Stage>**Stage 2**

●Age>**10**

●Click OK

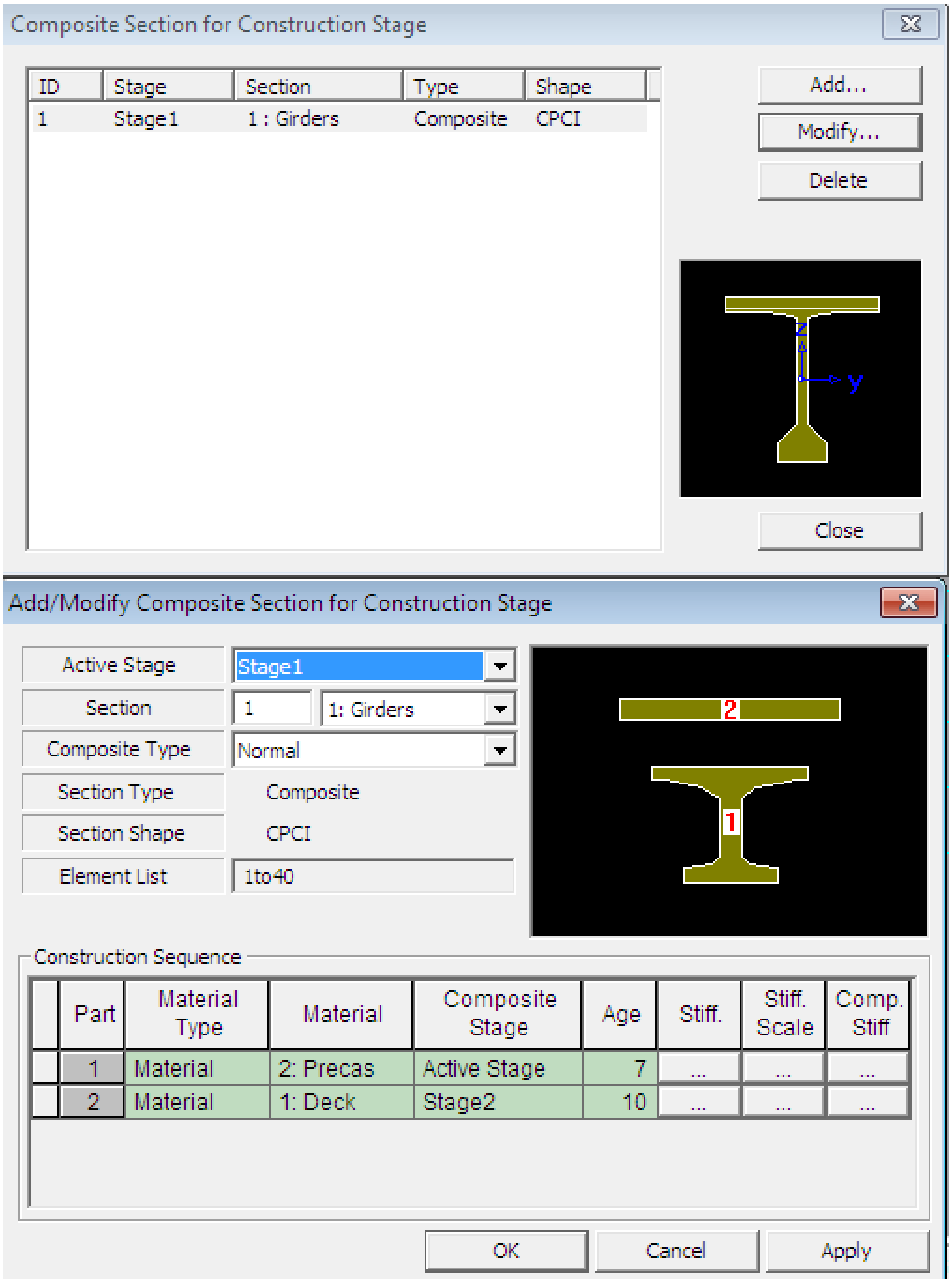


Figure 4-58:Composite girder Section during Construction Stages

●In the Main Menu: Analysis / **Construction Stage Analysis Control**

(See figure 4.59)

●Final Stage>**Last Stage**

●Analysis Option>**include time Dependent Effect** (on)

●Time Dependent Effect>**Creep & Shrinkage** (on)

●Type>**Creep & Shrinkage**

●**Auto Time Step Generation for Large Time Gap** (on)

●**Tendon Tension Loss Effect (Creep & Shrinkage)** (on)

●**Variation of Comp. Strength** (on)

●**Tendon Tension Loss (Elastic Shortening)** (on)

●Frame Output>**Calculate Output of Each part of Composite Section (on)**

●Load Cases to be Distinguished from Dead Load for CS Output:

●Load Case>**Wearing Surface**

●Click Add

●Load Case>Barrier

●Click Add

● Click OK

●Beam Section Property Changes>**Change with Tendon**

●**Save Output of Current Stage (Beam/Truss)** (on)

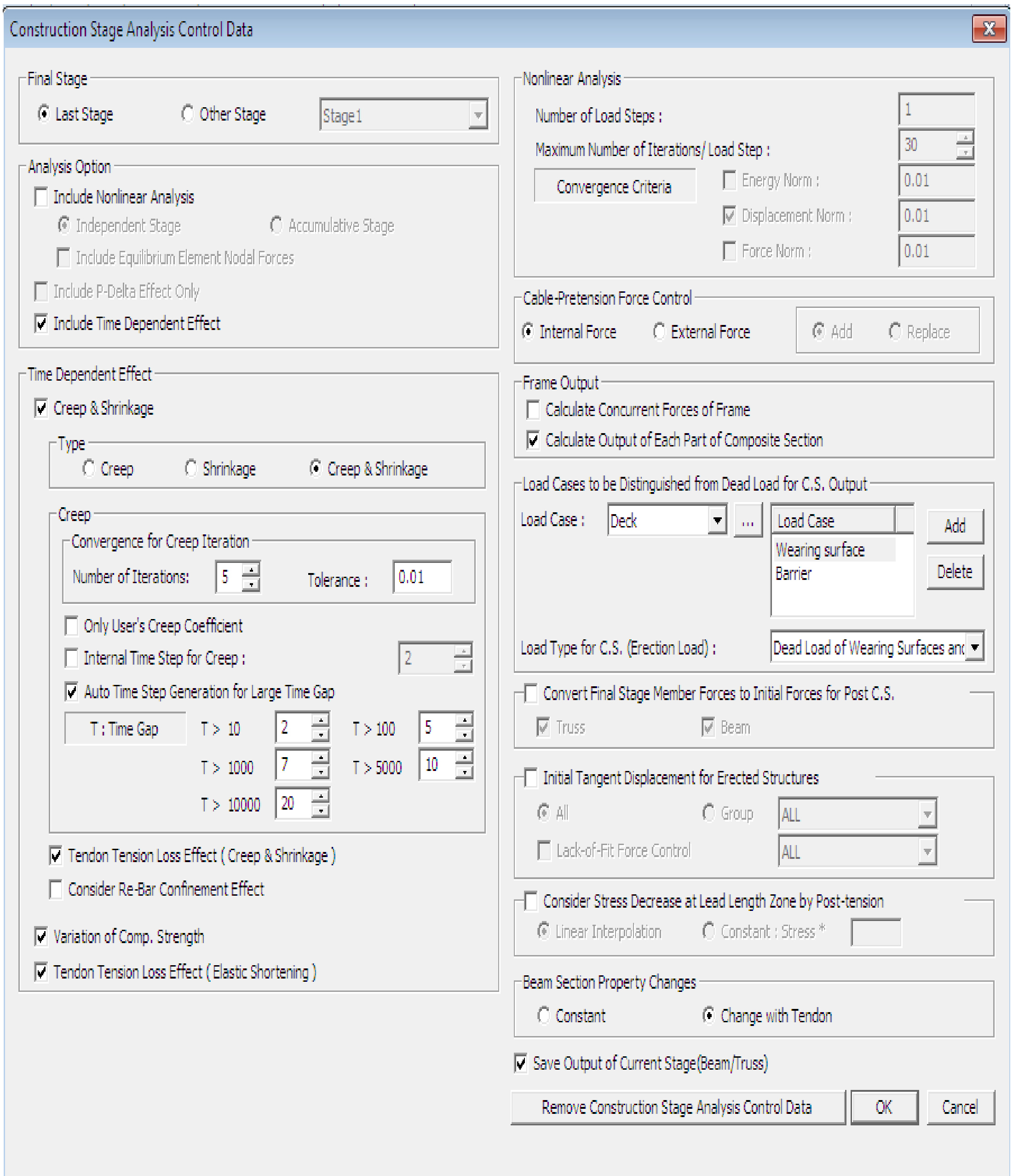


Figure 4-59:Construction Stage Analysis Control Data

12. Perform Structural Analysis

●In the Main Menu: Analysis /  Perform Analysis

13. Load Combinations:

●Select the Post Construction Stage (PostCS).

●In the Tree Menu: Click Menu tab.

●Results> **Combinations** (See figure 4-60)

●Load Combinations dialog box>General tab>Click Auto generation

●Option>**Add** ●**Add Envelope** (on)

●Code Selection>**Concrete** ●Design Code>**AASHTO-LRFD02**

●Manipulation of Construction Stage Load Case>**CS Only**

●Load Modifier>**1**

●Load Factors for Permanent Loads (Yp):

●Component and Attachments>Load Factor>**Both**

●Wearing Surfaces and Utilities>Load Factor>**Both**

●Condition for Temperature, Creep, Shrinkage Factor>**Deformation Check**

●Click OK

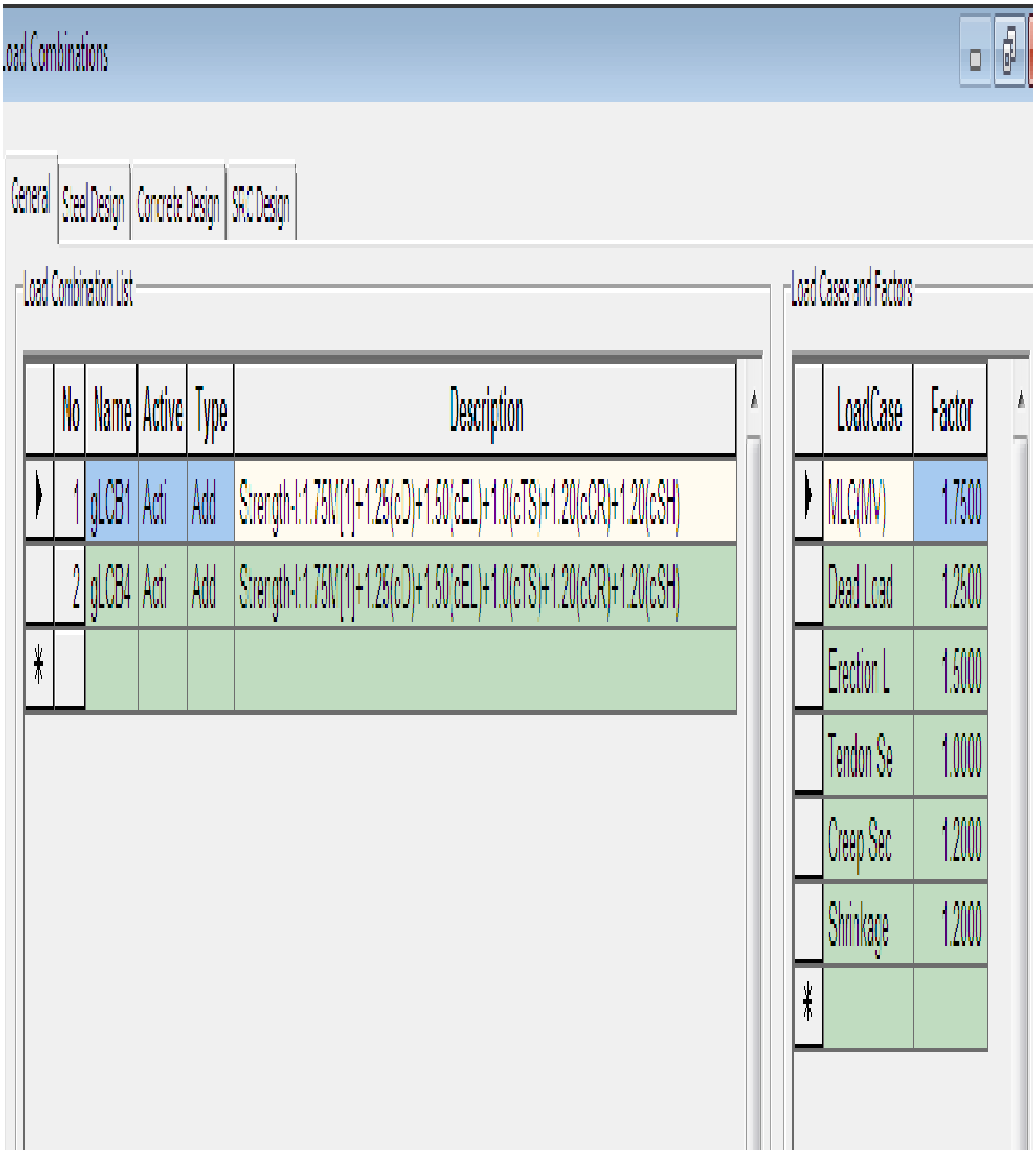


Figure 4-60: Generation of Load Combination

14. Analysis Result:- see Table(4-15)

LCB1 =1.25DC + 1.50DW + 1.75(LL + IM)

LCB3 =1.0(DC + DW) + 1.0(LL + IM) + 0.3(WS + WL)

❖ ***Bending Moment Diagrams in Precast Beams:-***

As shown in figure (4.61) for LCB1 and figure (4.62) for LBC3

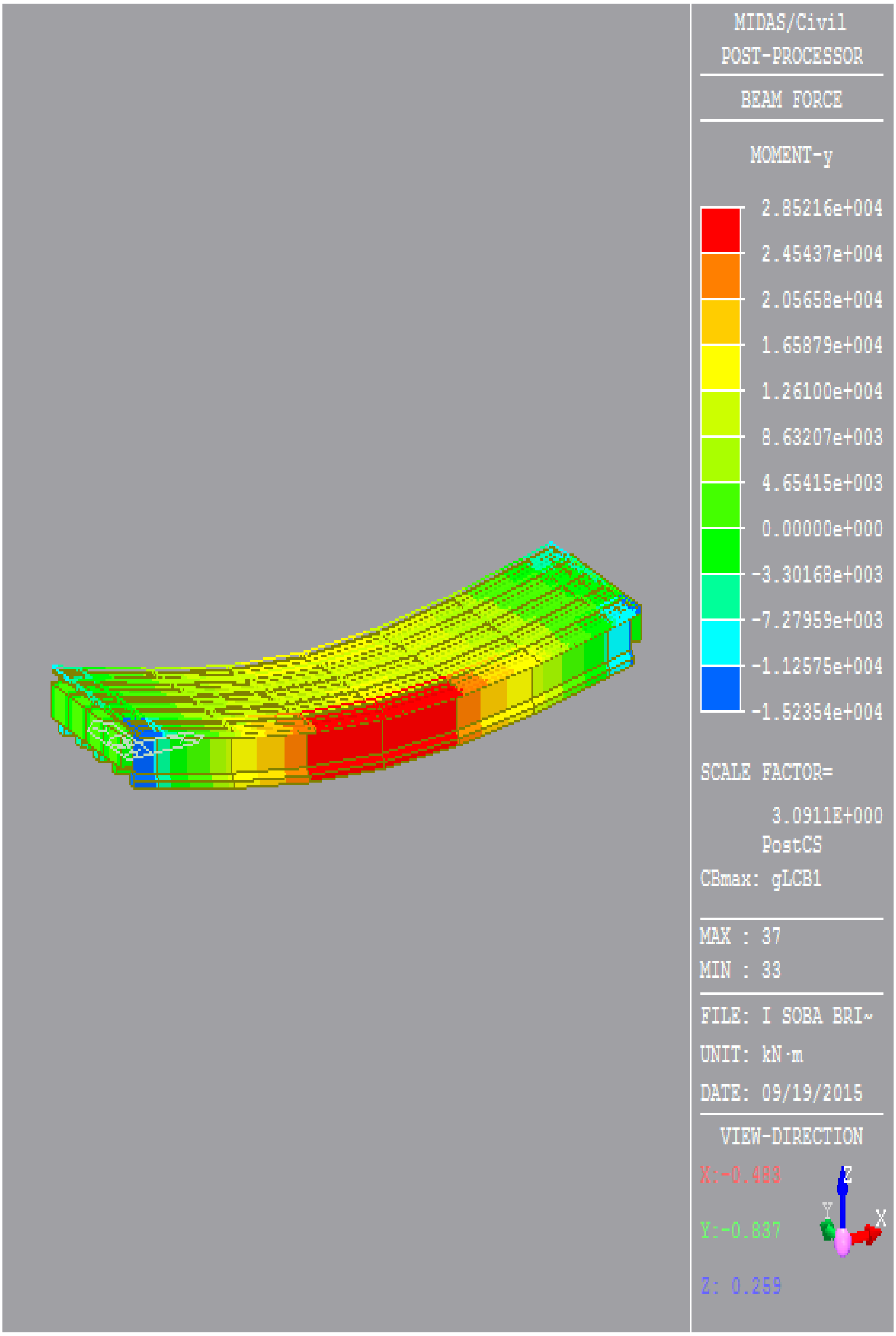


Figure 4-61: Bending Moment in the girders for LCB1

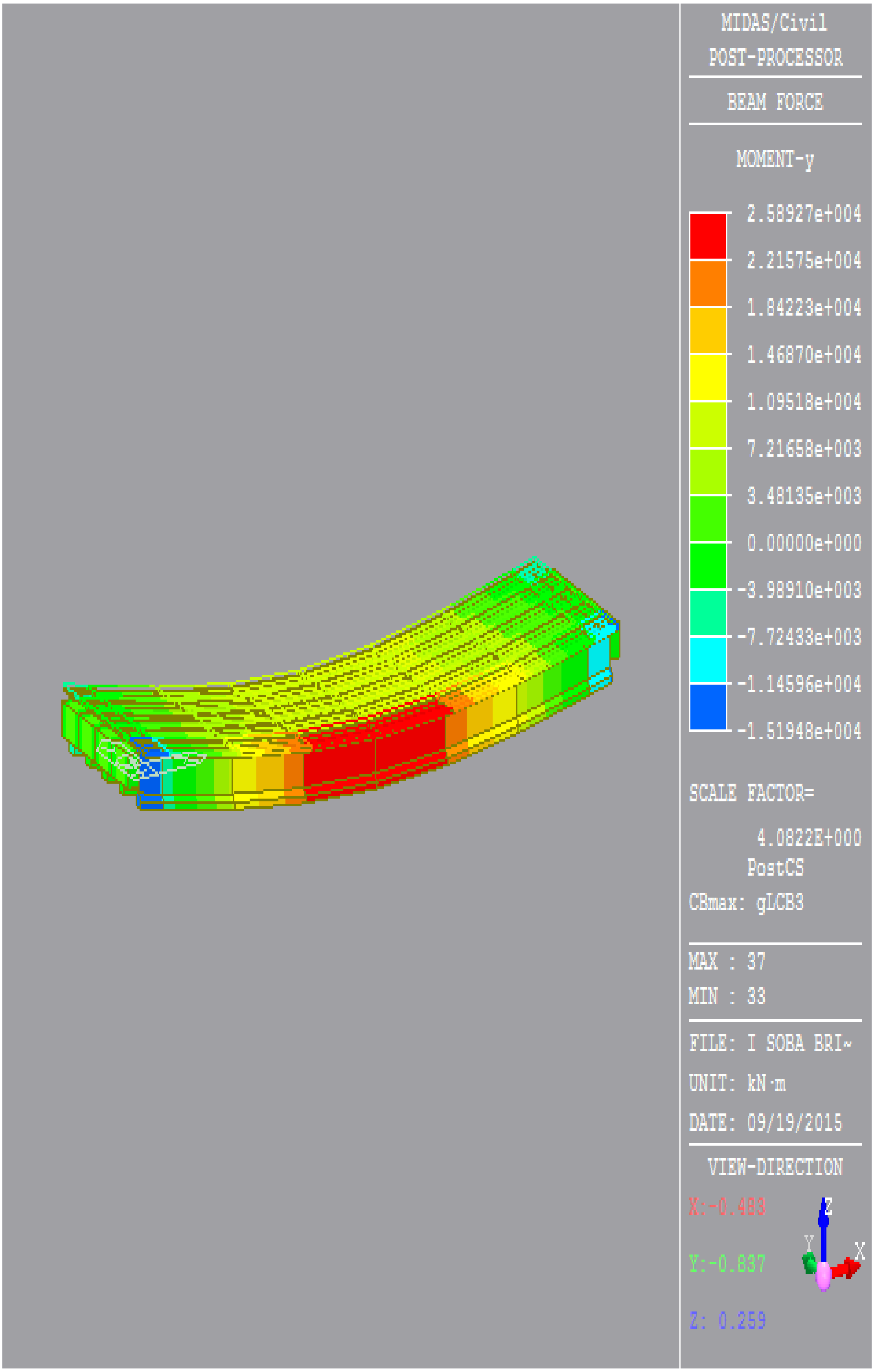


Figure 4-62: Bending Moment in the girders for LCB3

❖ ***Shear Force Diagrams in Precast Beams:***

As shown in figure (4.63) for LCB1 and figure (4.64) for LBC3

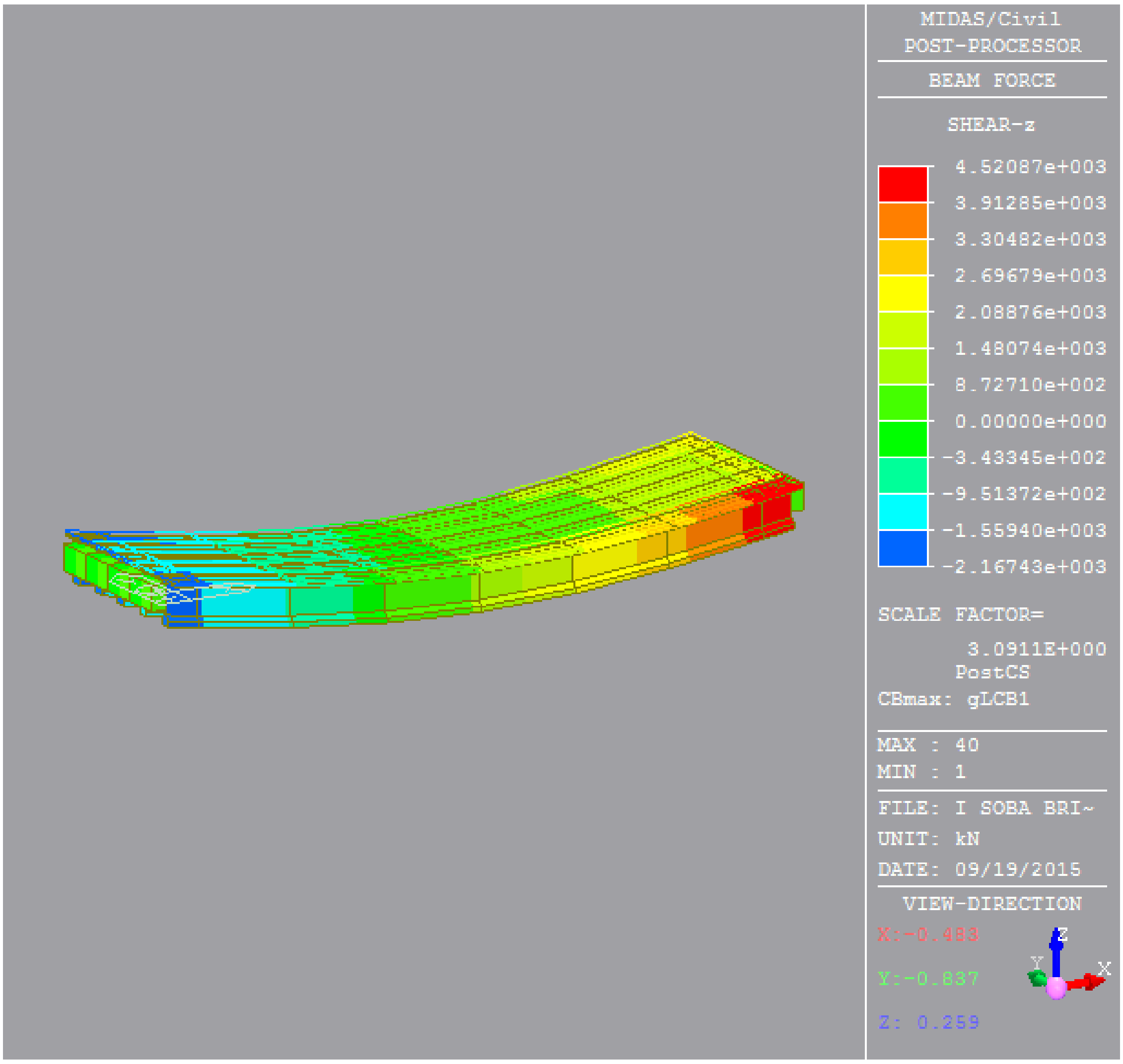


Figure 4-63: Shear Force in the girders for LCB1

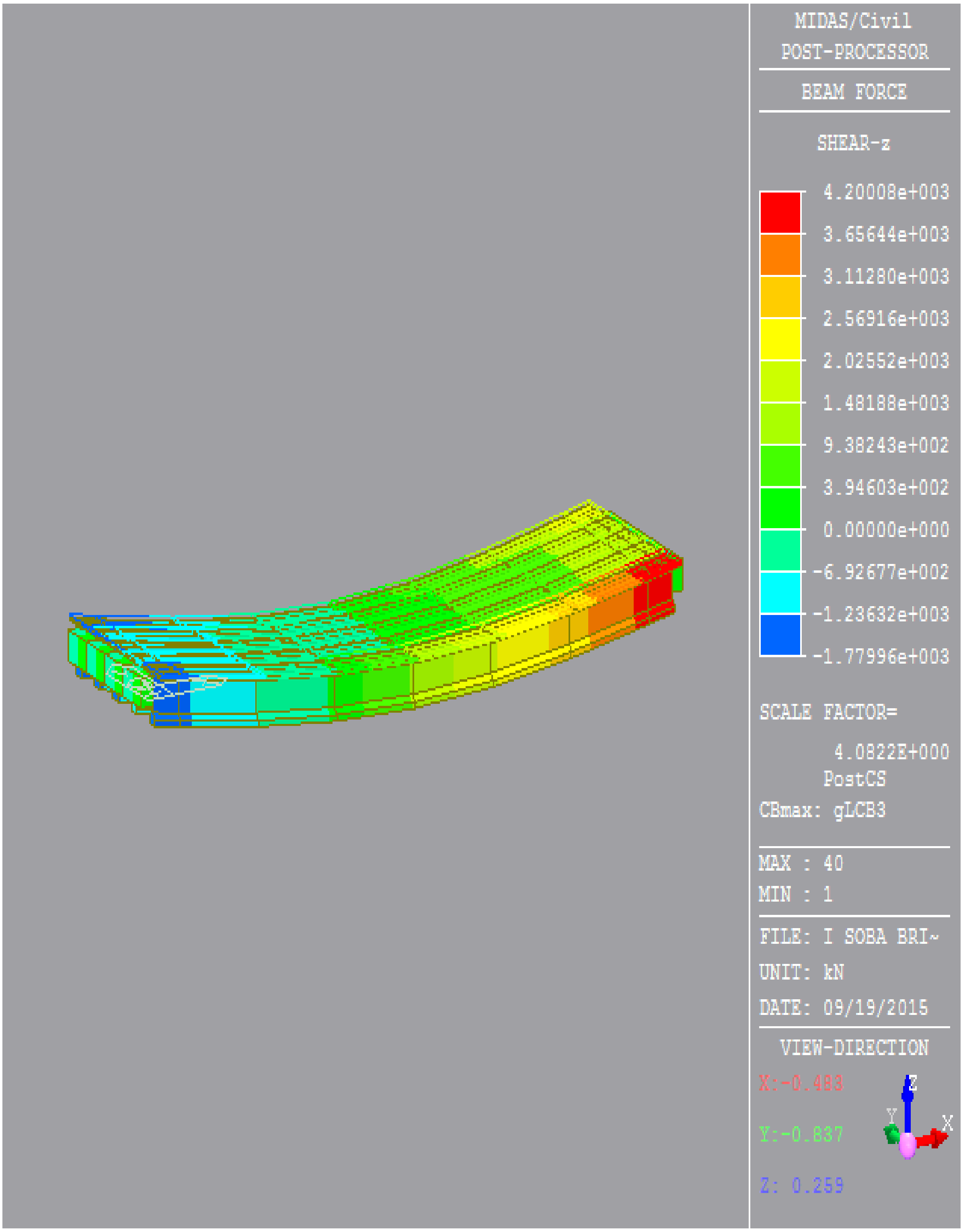


Figure 4-64: Shear Force in the girders for LCB3

❖ **Reaction:**

As shown in figure (4.65) for LCB1 (Max Combination)

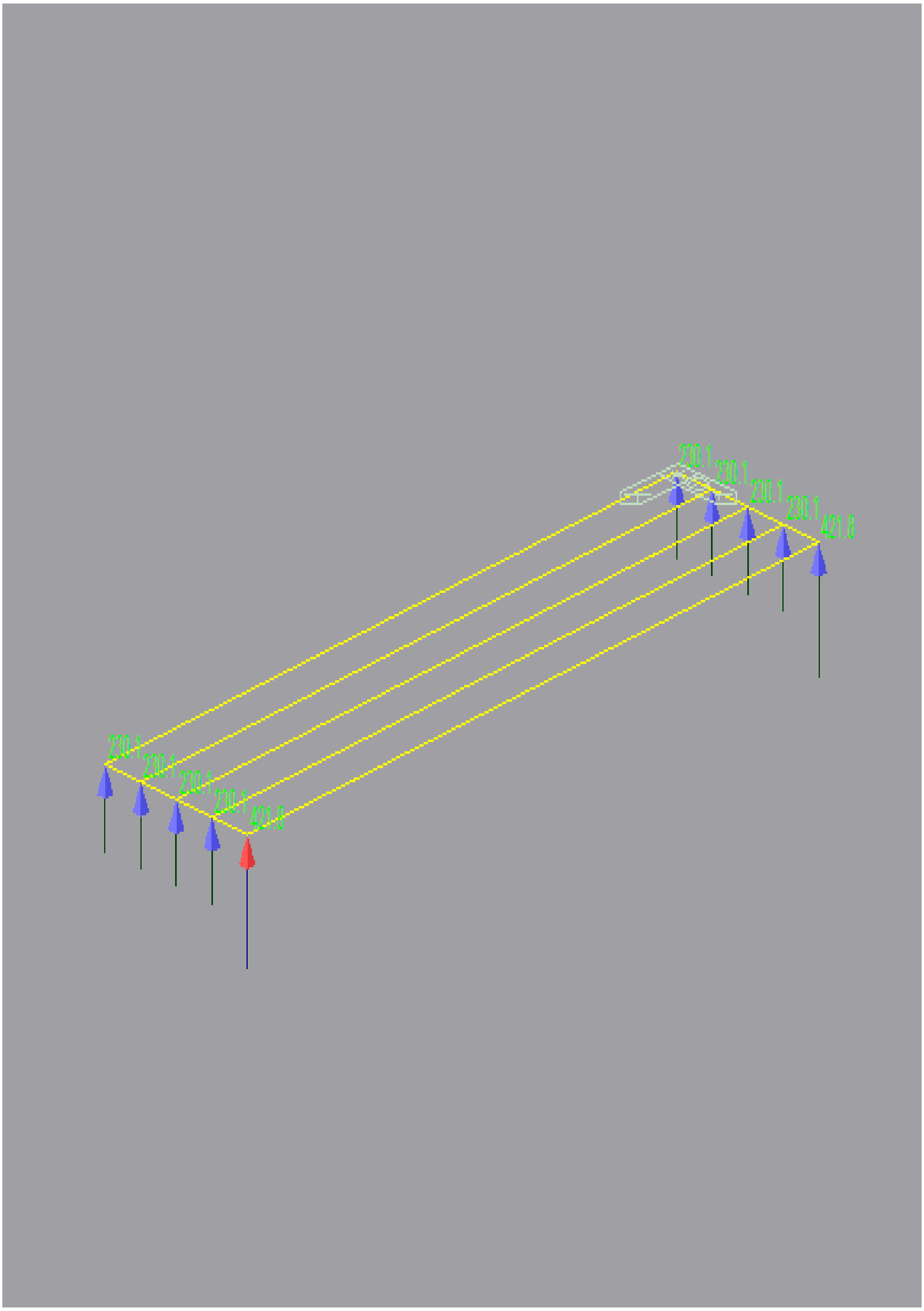


Figure 4-65: Reaction in the girders for LCB1

❖ **Deflection:-**

As shown in figure (4.66) for LCB1and figure (4.67) for LCB3

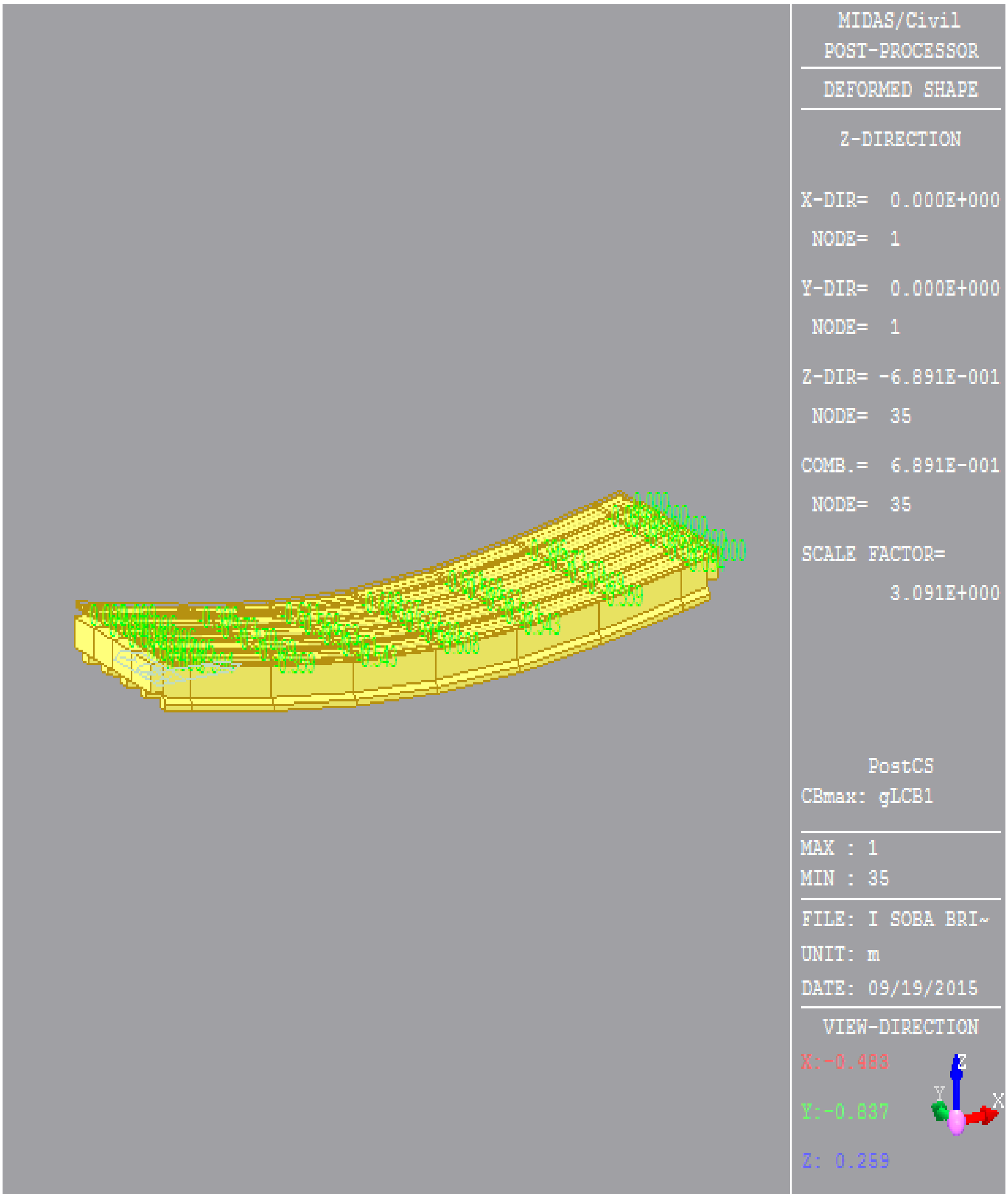


Figure 4-66: Deflection in the girders for LCB1

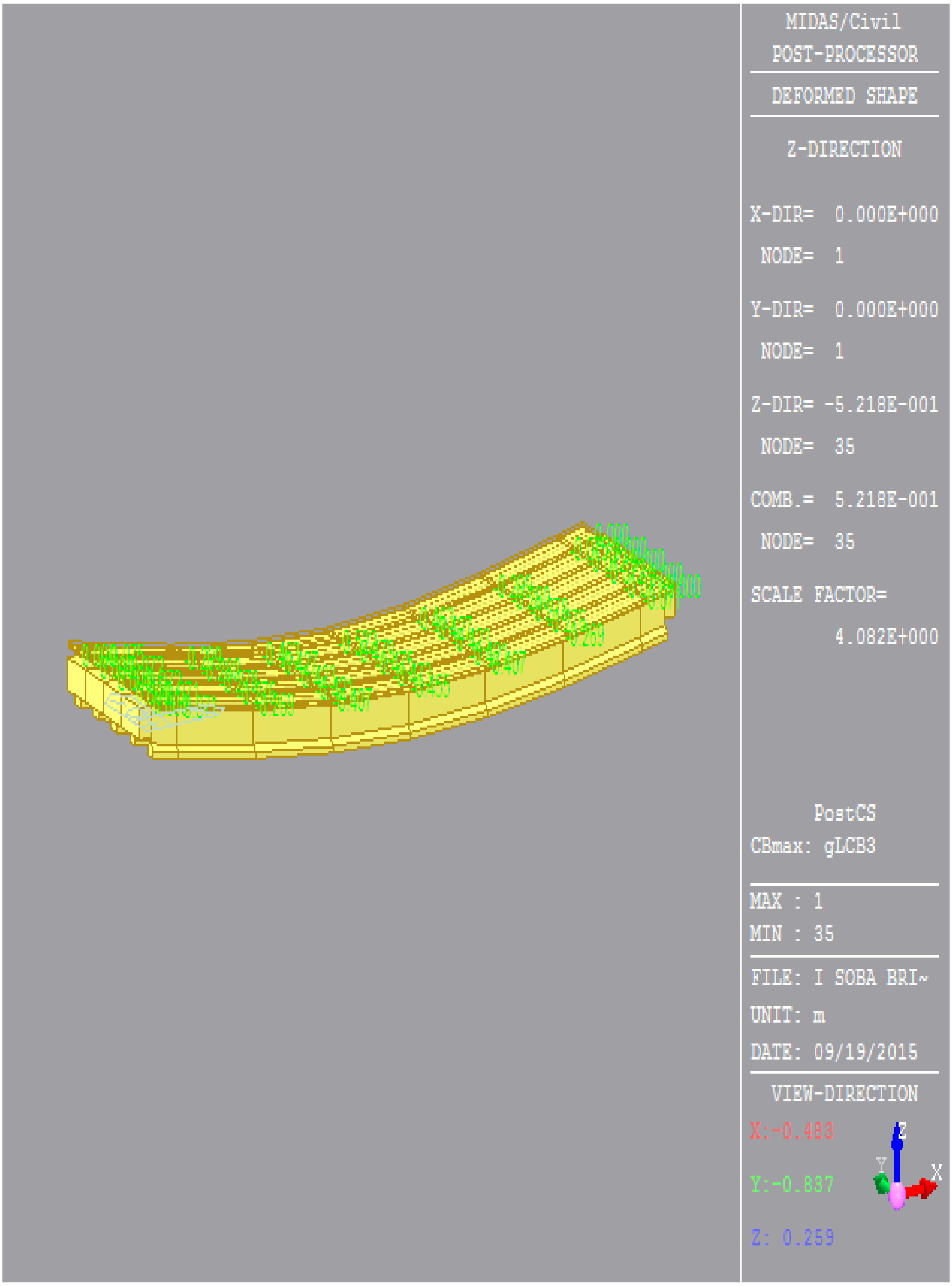


Figure 4-67: Deflection in the girders for LCB3

Table 4-15 Results of analysis

Case		Shear force Z-Z (KN)	Torsion Moment X-X (KN. m)	Moment Y-Y (KN. m)	Moment Z-Z (KN. m)	Deformation (m)
GLCB1	Max	4365.38	0.00	27153.32	0.00	0.757
	Min (-)	0.00	0.00	-18357.9	0.00	0.00
GLCB3	Max	4060.25	0.00	24600.00	0.00	-0.604
	Min (-)	0.00	0.00	-18630.22	0.00	0.00

4. 3 Substructure of Soba Bridge:-

as shown in figure (4-68) and (4-69)

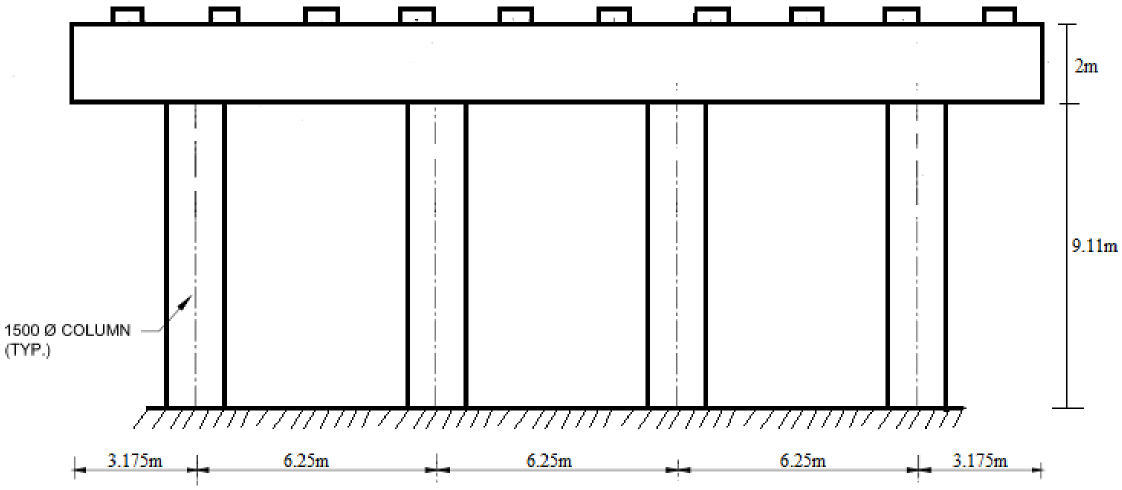


Figure 4-68: Side View of Substructure of Soba Bridge

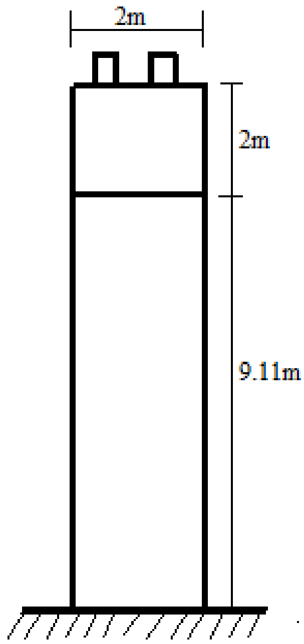


Figure 4-69: Elevation of Substructure of Soba Bridge

4. 3.1 Pier Section

The section of pier is circular section show in figure (4-70)

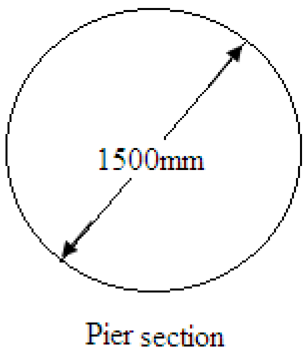


Figure 4-70: section of Pier

4.3.2 Analysis of Pier of Soba Bridge

Steps are:

1. Enter the materials : (See figure 4-71)

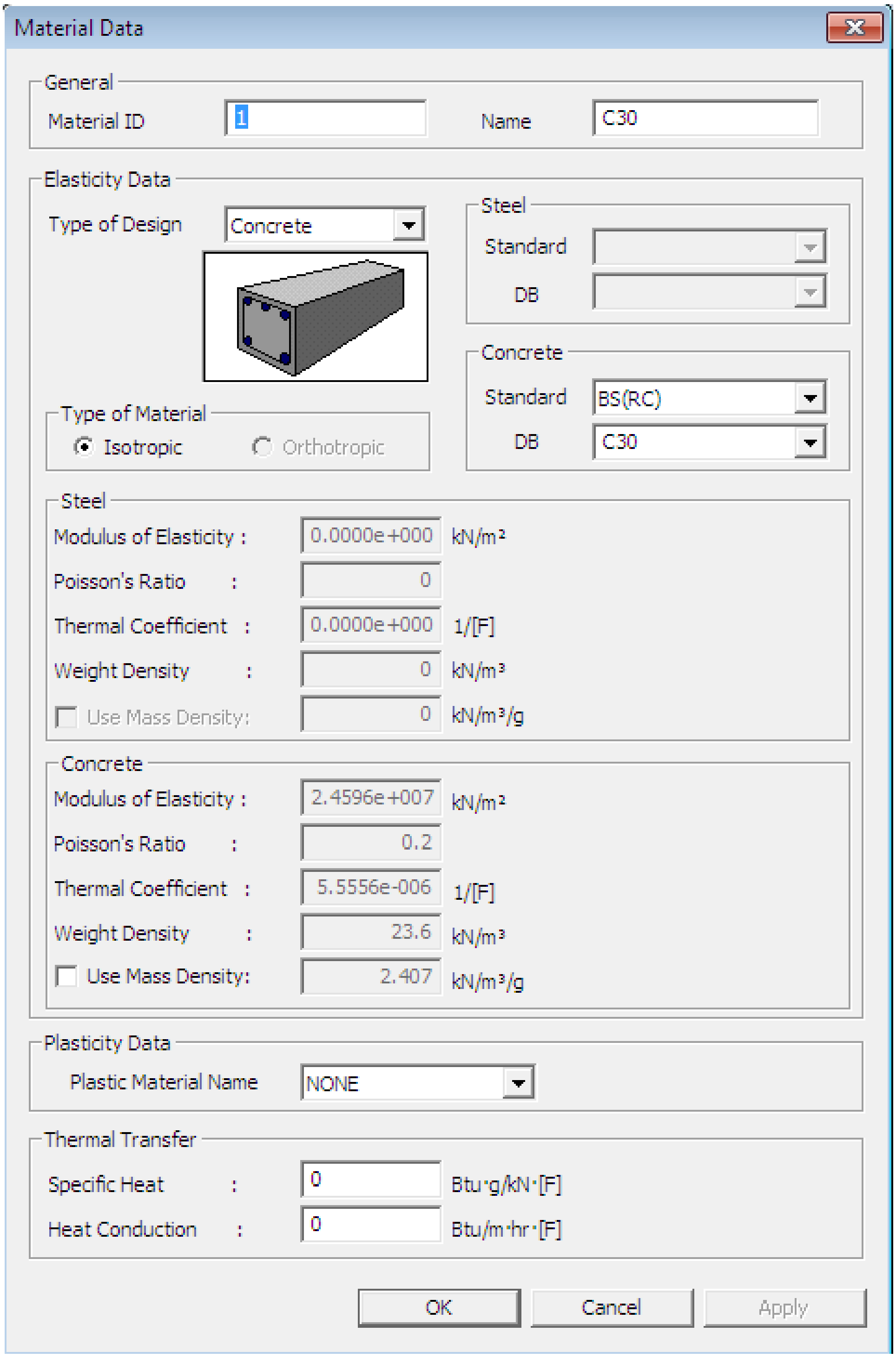


Figure 4-71: Material of Pier

2. Create nodes and elements:

The Piers was modeled as solid element with: (See figure 4-72 , 4-73 , 4-74 and 4-75)

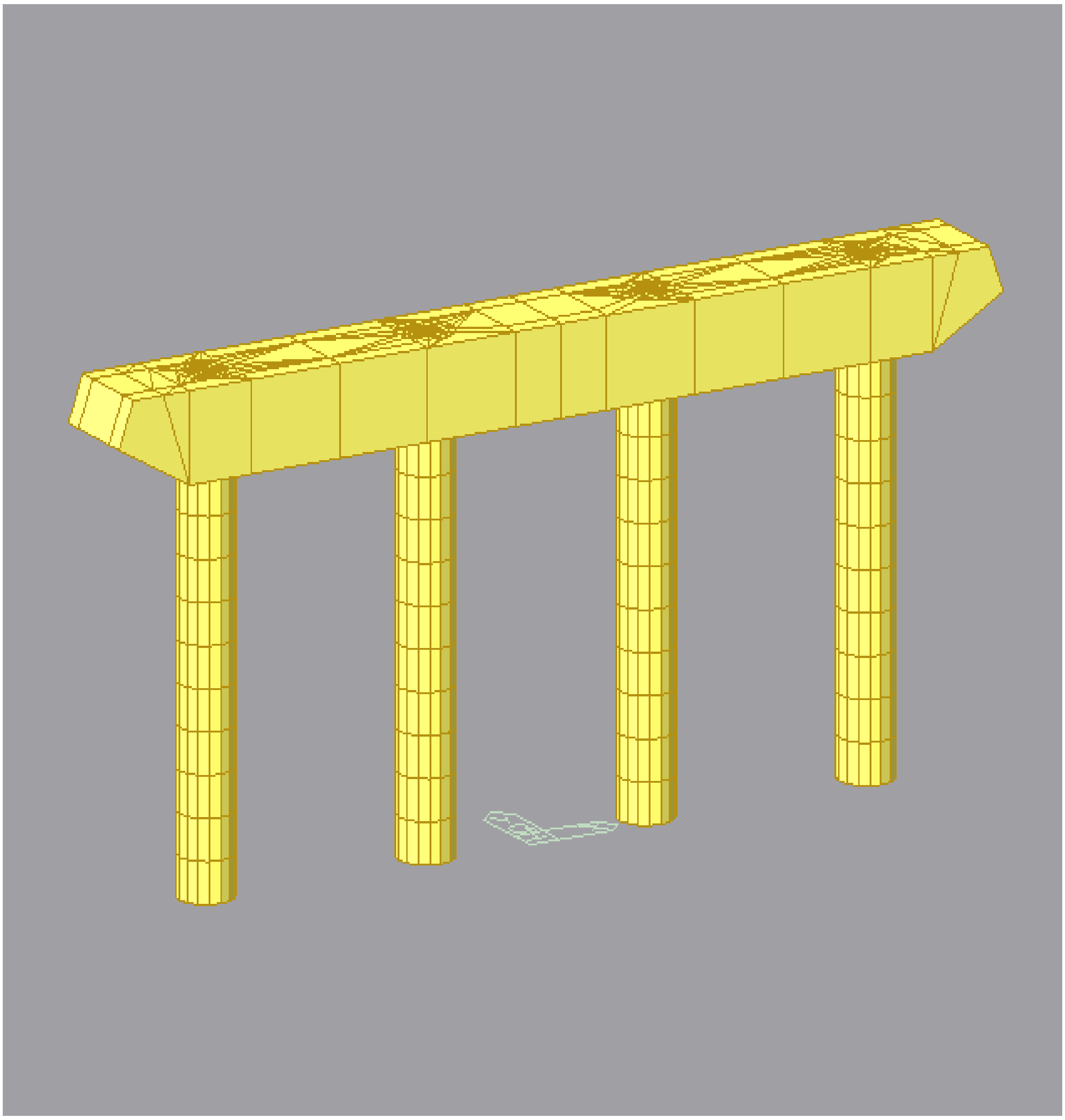


Figure 4-72: 3-D of Modeling of Piers and cross head

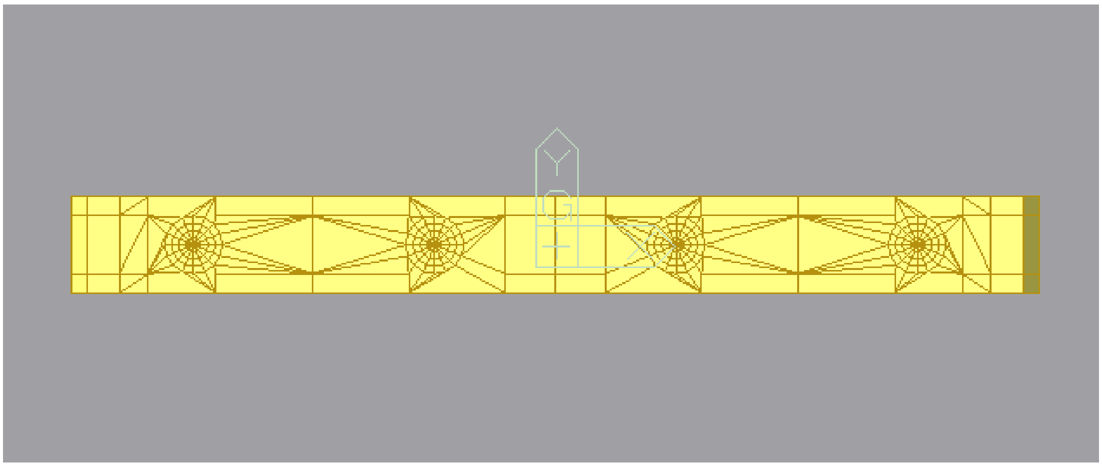


Figure 4-73: Plan of Modeling of Piers and cross head

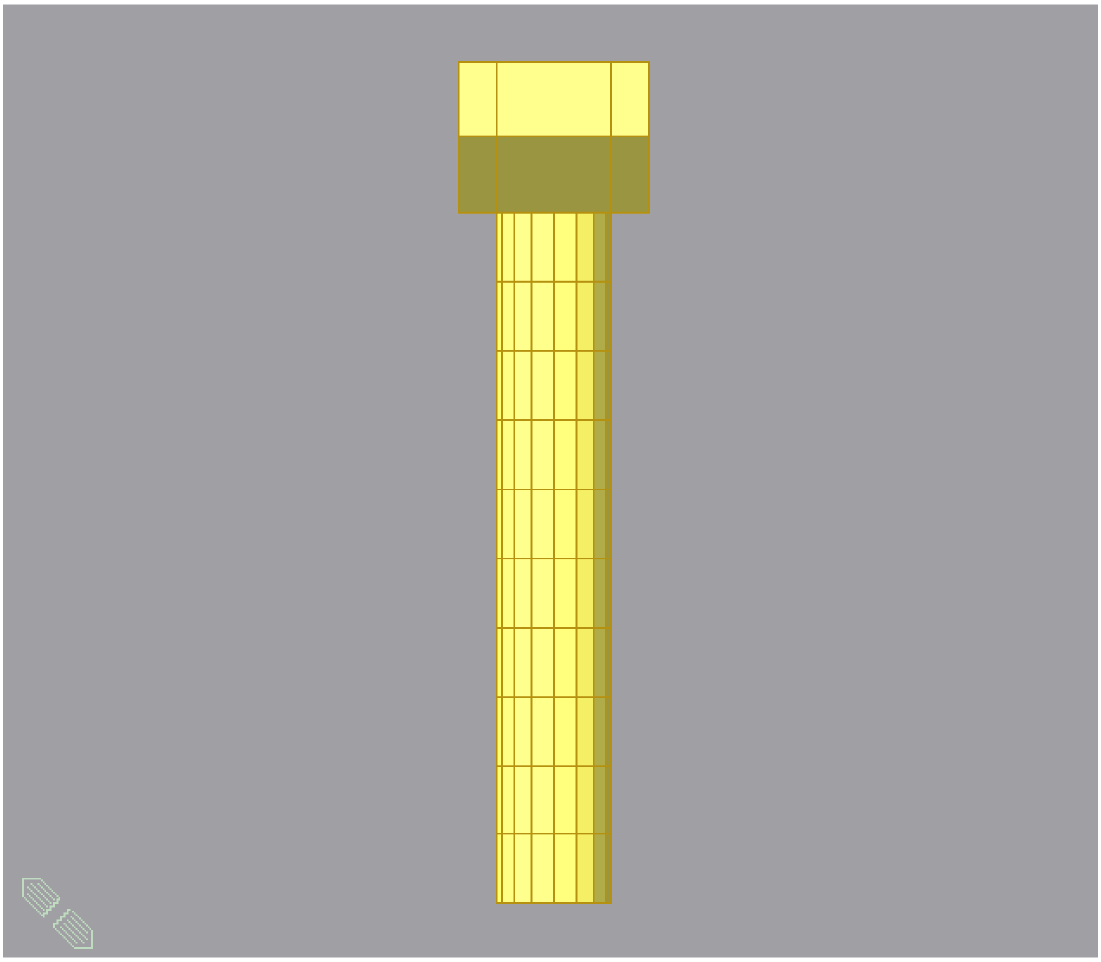


Figure 4-74: Elevation of Modeling of Piers and cross head

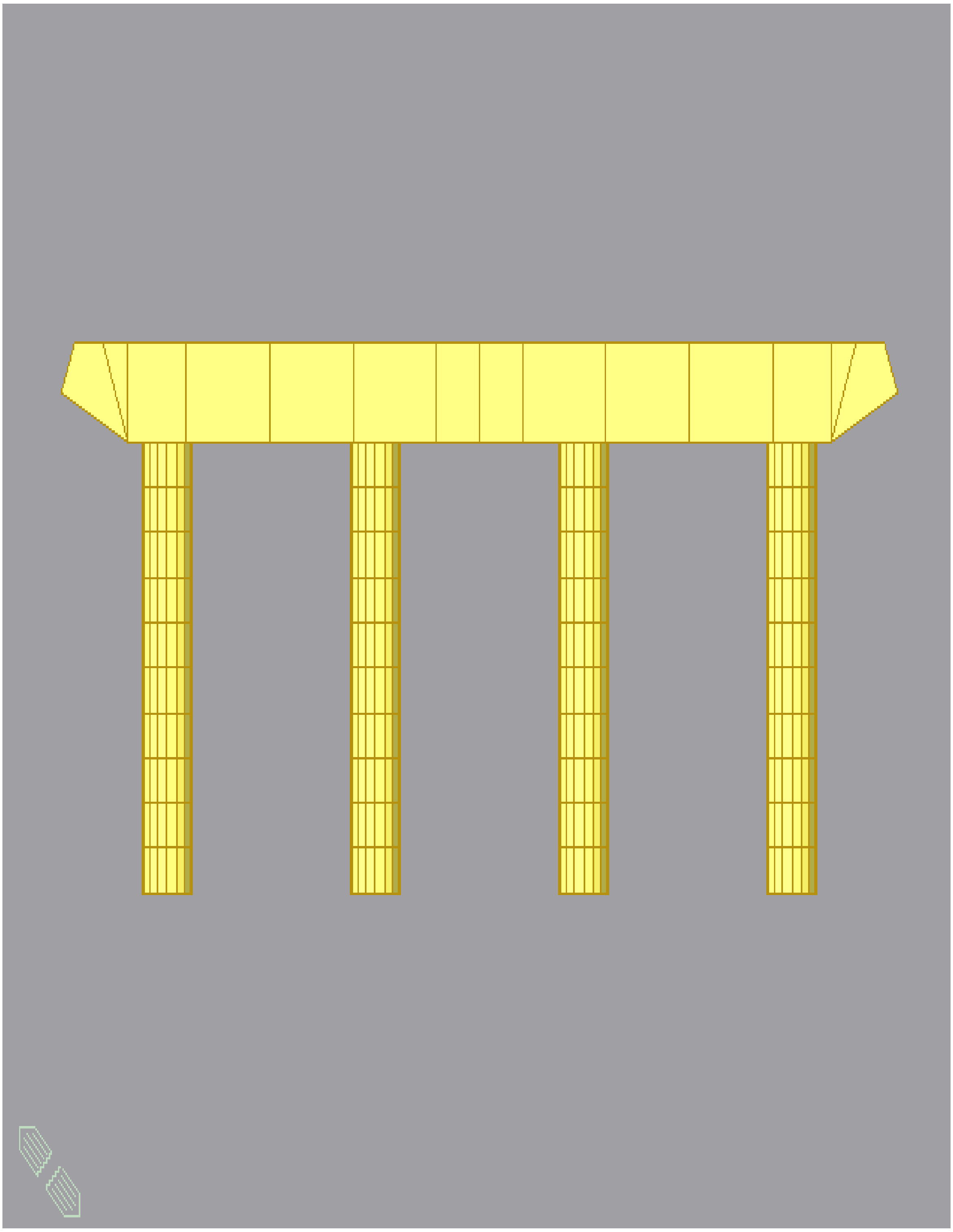


Figure 4-75: Side View of Modeling of Piers and cross head

3. Enter the boundary conditions :(See figure 4-76)

At Bottom of Piers (fixed support) D_x, D_y, D_z, R_x, R_y, R_z

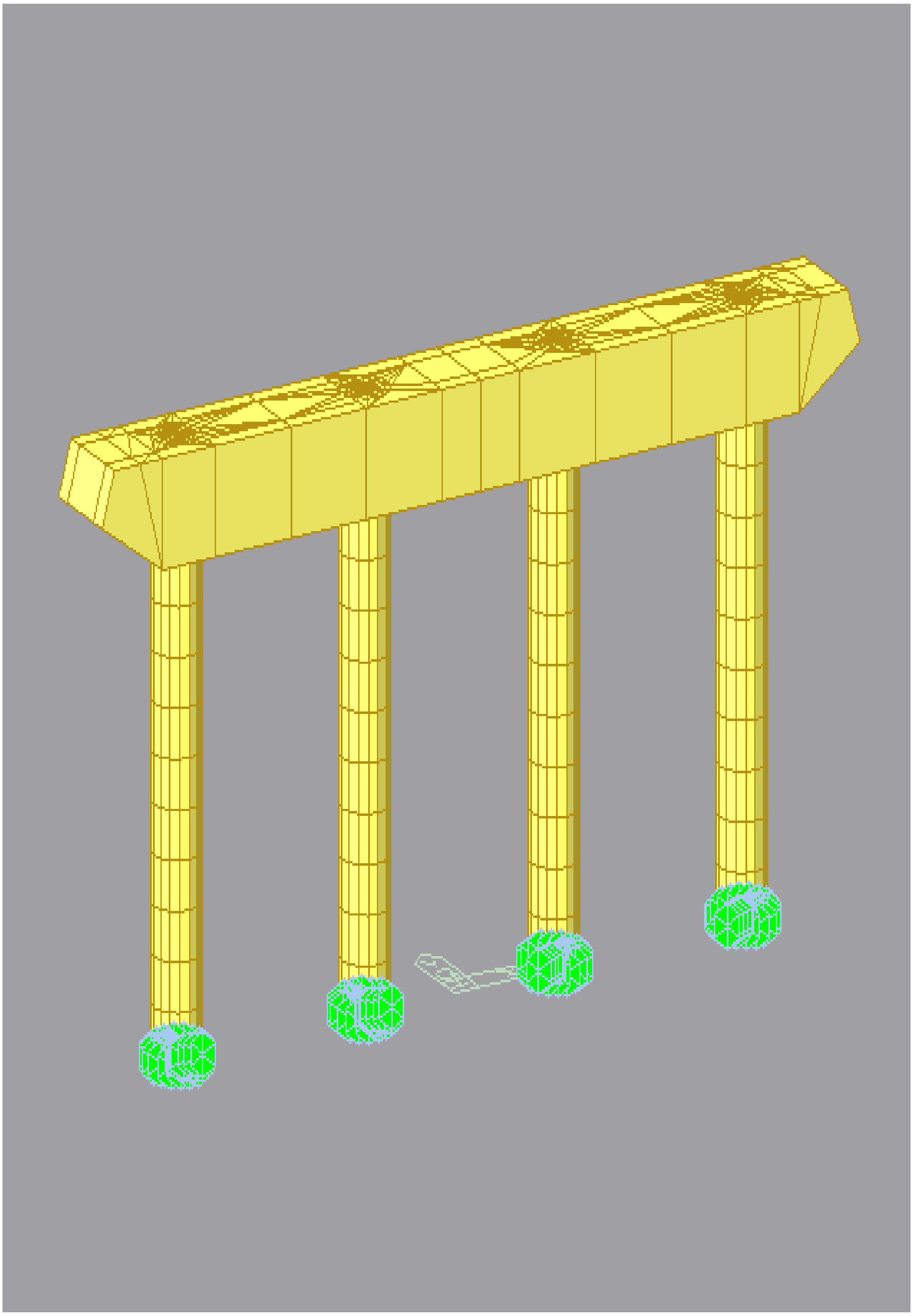


Figure 4-76: Boundary condition

4. Enter the Loads

● We will consider only the following two load cases for modeling:

Load Case 1: Self weight (See figure 4-77)

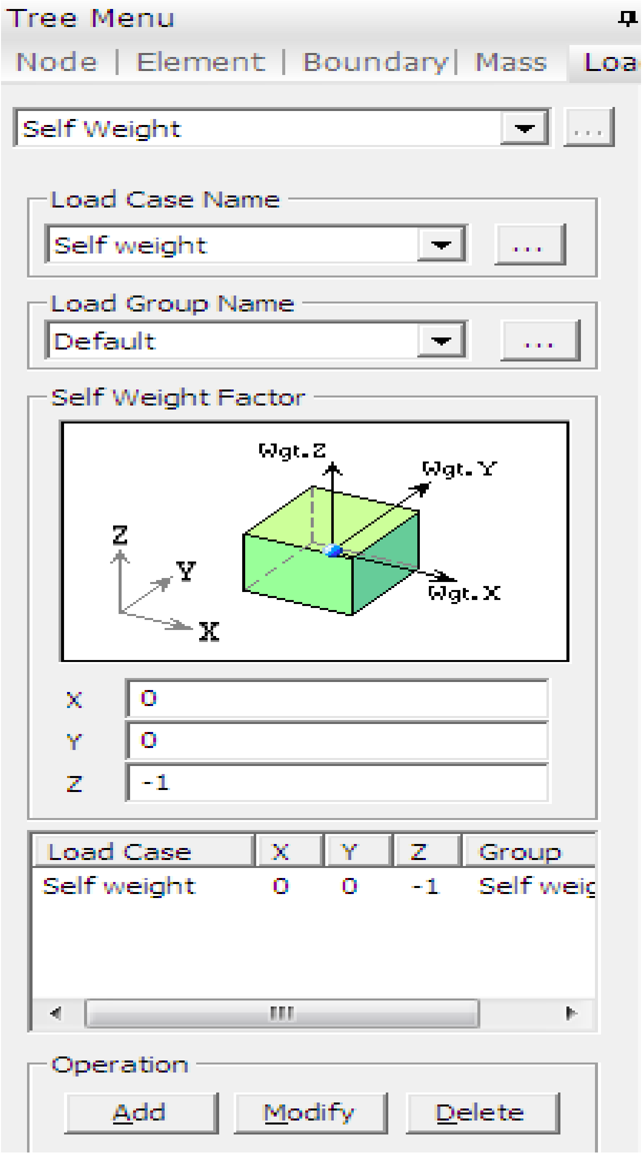


Figure 4-77: Self weight

Load Case 2: Vertical load (See Figure 4-78)

Enter the following reactions from the girders: (See Table 4-16)

Table 4-16 vertical load (reaction of girders)

Reactions at nodes at the left	Reactions at nodes at the right
2283. 5	2269. 4
1792. 9	1778. 2
1798. 9	1782. 9
1817. 5	1799. 8
4352. 2	4414. 8

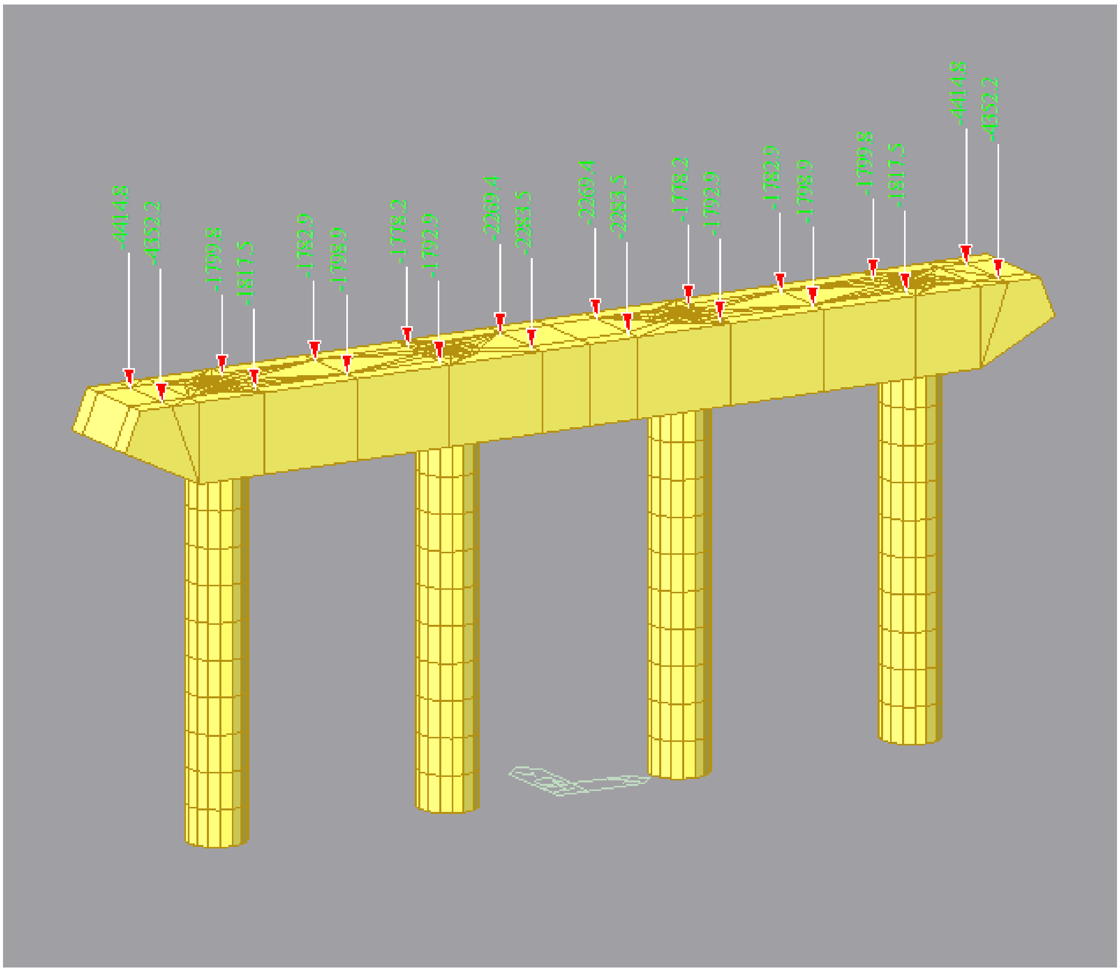


Figure 4-78: Vertical loads

5. Enter the combinations:(See figure 4-79)

CBLI = 1.25(Self weight + Vertical Load)

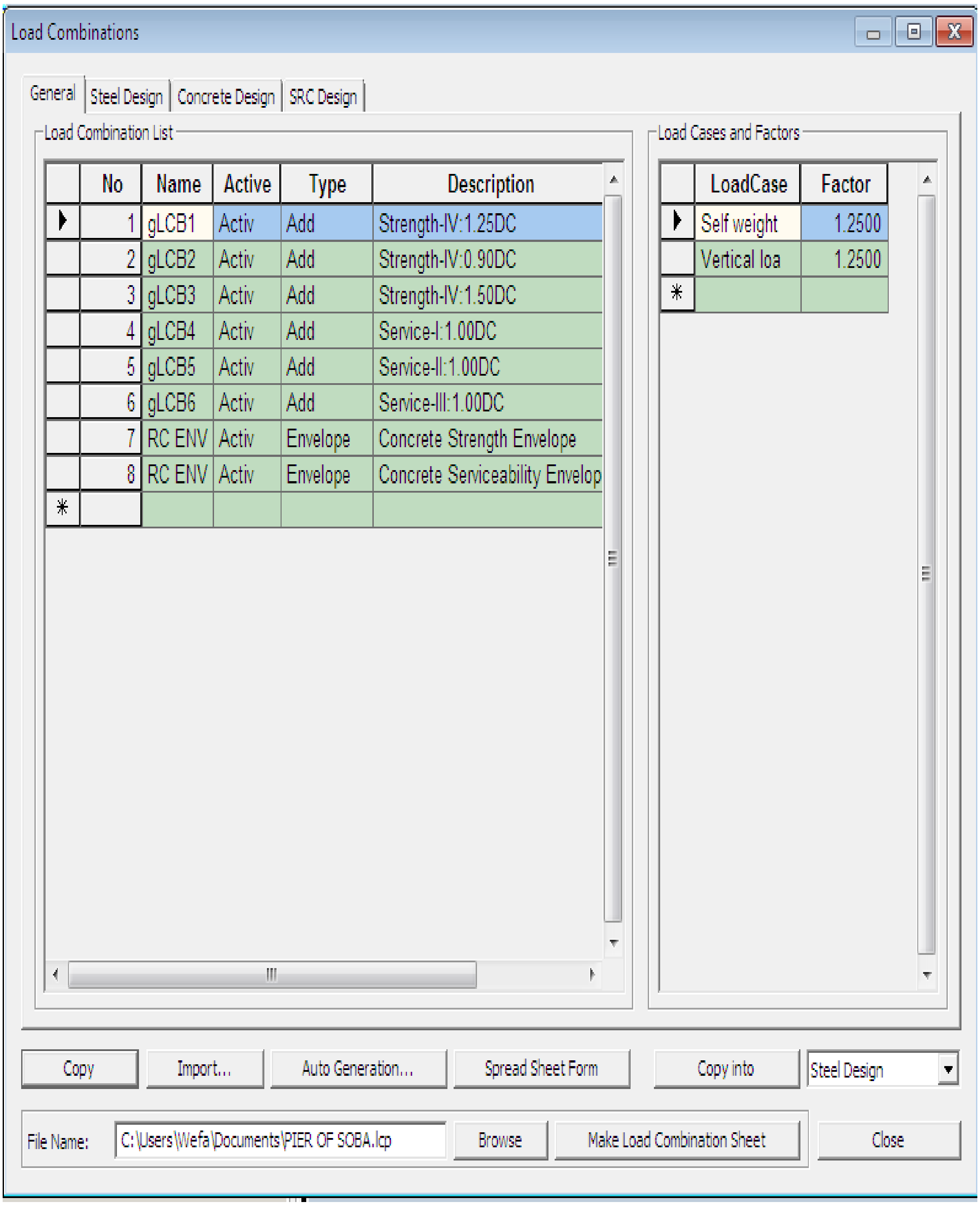


Figure 4-79: Load Combination

❖ ***Deformed shape:-***As shown in figure(4-80)

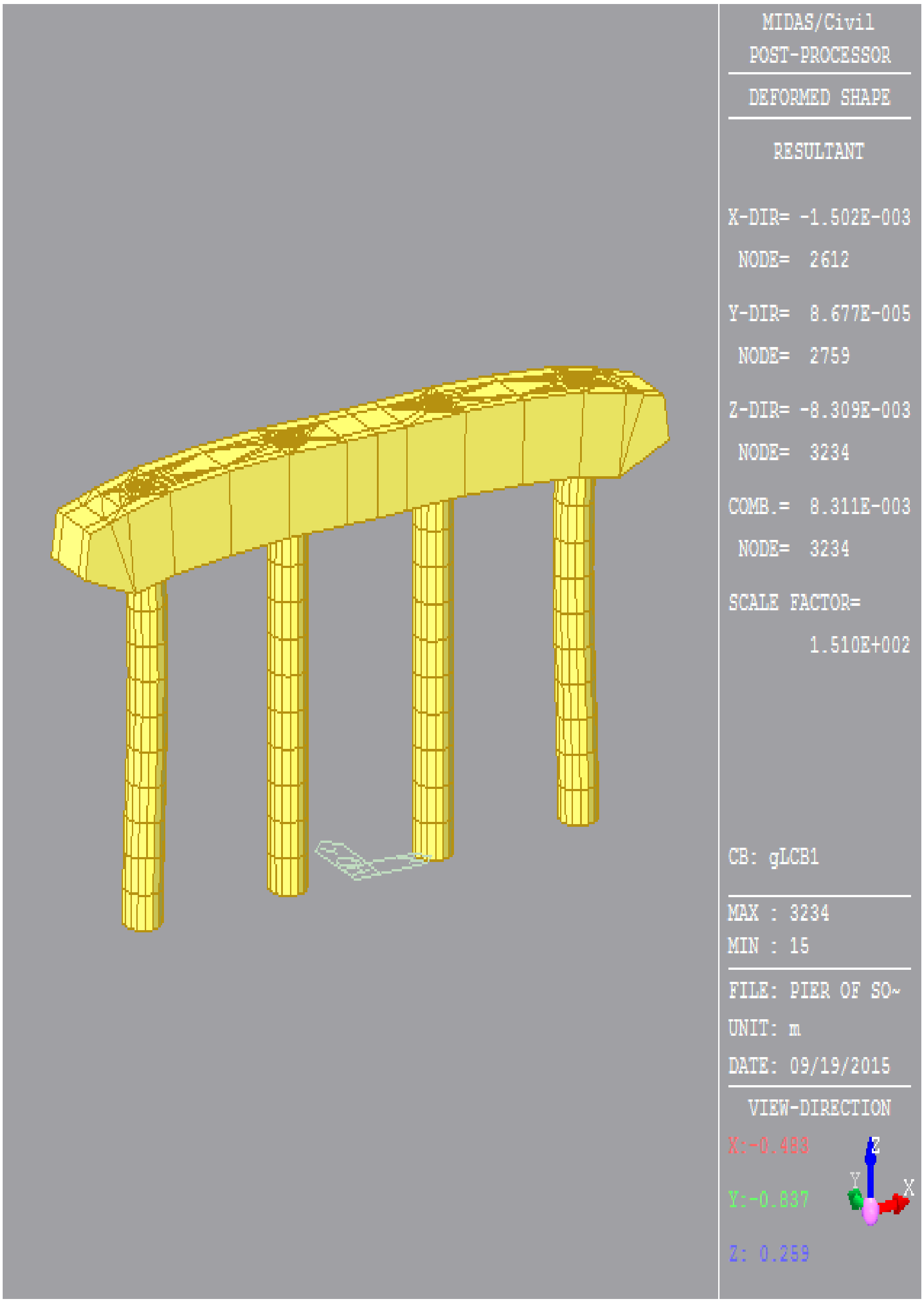


Figure 4-80: Deformation shape

6. Result of analysis of pier:

The maximum Stress from: - (As shown in figure 4-80)

LCB1(+) = 7704 KN/m^2

LCB1(-) = - 6500 KN/m^2

The maximum Shear in Z direction from: -

LCB1(+) = 103.06 KN

LCB1(-) = - 93.67KN

- ***Stresses in Pier***: (See figure 4-81)

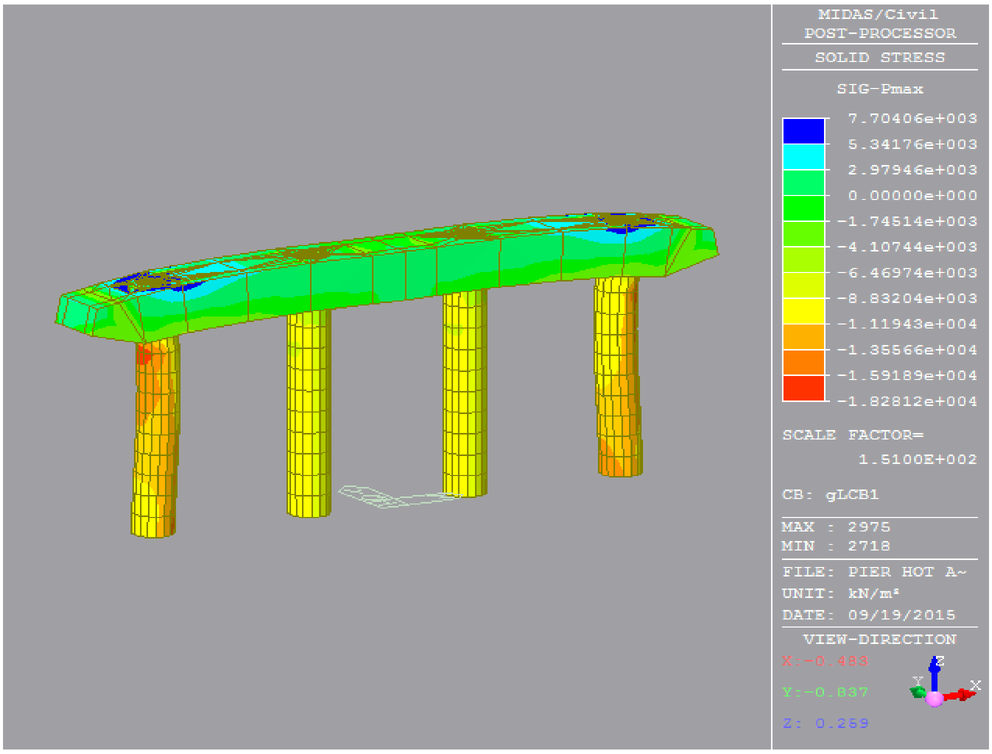


Figure 4-81: stresses in pier

4.4 Discussion of analysis and design results

The analysis of the girder where carried out by using MIDAS 2006 program by which both steel and concrete can be result maximum shear at the end of girder equal to [4365.38 KN] at Z direction and maximum moment at the mid span equal to [27153KN.m] about Y direction. No tensional result because restraining in Y direction, maximum deflection at mid span equal to [0.757 m].

The difference between analyzed manually and analyzed using MIDAS 2006 equal to 11% as shown in table (4-17).

Table 4-17 Results of analysis

Result using manual analysis		Result using program analysis	
Shear Force (KN)	Bending Moment (KN.m)	Shear Force (KN	Bending Moment (KN.m)
3312.2	32730.2	4365.38	27153.32

Chapter 5
Design of single span of Soba Bridge

This chapter introduces the calculations part for the design of Soba Bridge using mathematical formulas as shown on the following pages.

5.1 Design of Slab of Soba Bridge

(See tables of design below)

Ref.	**Calculations**	**Output**

❖ Information of design:-

f'_c= 30MPa f_y= 420MPa S=2.1m

d_{bar} = 16mm A_{bar} = 210mm^2 cover =25mm

❖ Minimum thickness of slab:-

$$h_{min} = \frac{S+10}{30} = \left(\frac{2.1*\frac{10}{3}+10}{30}\right) * 0.3 = 0.17m = 170mm$$

$$h = 180mm > h_{min} = 170mm$$

OK

❖ Effective depth

$$d = h - \frac{d_{bar}}{2} - cover = 180 - \frac{16}{2} - 25 = 147mm$$

❖ Maximum moment Applied on Deck slab:-

M_u = **45.48KN.m**

$$M_n = \frac{M_u}{Ø}$$

Ø = 0.9 for resistance of Moment

$$M_n = \frac{45.48}{0.9} = 50.53KN.m$$

❖ Main Reinforcement:-

$M_n = 0.85*f'_c*b*a\,(d – a/2)$

$50.53*10^6 = 0.85×30×1000\,a\,(147 – a/2)$

$- 0.5a^2 + 147a - 1981.7 = 0$ ⇒ a= **14.16mm**

$$A_s = \frac{0.85*f'_c*b*a}{f_y} = \frac{0.85×30×1000×14.16}{420} = 860mm$$

A_s = **860mm/m**

Ref.	Calculations	Output
	No of bars = $\frac{A_s}{A_b} = \frac{860}{201}$ = **5bars**	
	Spacing = $\frac{L}{\text{No of bars}} = \frac{1000}{5} = 200mm$	
	Use Ø 16mm@ 200mmc/c	
	$a_{new} = \frac{A_s f_y}{0.85\, f'_c\, b} = \frac{5*201*420}{0.85\times30\times1000} = 16.56$ mm	
	$M_r = ØM_n = 0.9\times5\times201\times420\times(147-\frac{16.56}{2})*10^{-6}$	
	M_r = 52.7KN.m	
	❖ **Maximum reinforcement:**	
	For sure that the steel reinforcement not exceed the maximum amount must apply the following condition:-	
	$\frac{c}{d} \leq 0.42$	
	$\beta = 0.85 - 0.05\left(\frac{f_c^{\backslash}-28}{7}\right) \geq 0.6$	
	$\beta = 0.85 - 0.05\left(\frac{30-28}{7}\right) = 0.8$	
	$C = \frac{a_{new}}{\beta} = \frac{16.56}{0.8}$ = **20.7mm**	
	$\frac{C}{d} = \frac{20.7}{147} = 0.141 < 0.42 \Rightarrow$ OK	**OK**
	❖ **Minimum reinforcement:**	
	The amount of steel reinforcement be satisfy if the moment of strength more or at least equal to the leaser of following two moments:	

Ref.	Calculations	Output
	• 1 .2M_{cr}Ø, • 1.33M_u	
	$M_{cr} = S_c f$	
	$S_c = \frac{bh^2}{6} = \frac{1000\times 180^2}{6} = \underline{\underline{5.4\times 10^6 mm^2}}$	
	$f_r = 0.63\sqrt{f'_c} = 0.63\sqrt{30} = \underline{\underline{3.45 N/mm^2}}$	
	$M_{cr} = 5.4\times 10^6 \times 3.45\times 10^{-6} = \underline{\underline{18.63 KNm}}$	
	$1.2M_{cr} = 1.2 * 18.63 = 22.36 KNm$	
	$1.33M_u = 1.33 * 45.48 = 60.5 KN.m$	**OK**
(A 9.7.3.2)	❖ **Distributed Reinforcement:**	
	Distributed steel:	
	This distributed do the distribute wheel in the longitudinal direction (direction of motion) and but in the bottom of slab. And as percentage from main reinforcement for positive moment of the slab.	
	Percentage = $\frac{3840}{\sqrt{S_e}} \le 67\%$	
	Effective length of the slab (S_e):	
	For type of slab supported by steel or concrete girders S_e equal:	
	S_e = 2500-180= 2320mm	

(A5.10.8.2)

Percentage = $\frac{3840}{\sqrt{2320}}$= 80 > 67% ⇒OK

$A^{\backslash}_s$ = 0. 67×posA_s = 0. 67×1005 = **673.35 mm²/m**

Use d_{bar} =12mm A_{bar} = 113mm²

NO of bars = $\frac{A_s}{A_b}$ = $\frac{673.35}{113}$ = **6 bars**

Spacing =$\frac{L}{\text{No of bars}}$ = $\frac{1000}{6}$ = 150mm

Use ∅12mm@150mmc/c

❖ **Reinforcement for Temperature and shrinkage:**

Minimum amount of reinforcement for Temperature and shrinkage for section of girder determine from following:-

$A_s \geq 0.75 \frac{A_g}{f_y}$

$A_s \geq 0.75 \frac{1000\times180}{420}$ = 430mm²

Node: amount of reinforcement steel divide into two parts:

A_s per surface = $\frac{430}{2}$= 215mm²

Use ∅12mm A_p = 113mm²

NO of bars = $\frac{A_s}{A_b}$ = $\frac{215}{113}$ = **2mm**

Spacing = $\frac{1000}{2}$= 500mm

Use ∅12mm@500mmc/c

5.2 The Design of Girders

►**Design Brief**

The girders are to be checked in accordance with the following documents:

● American Association of State Highway and Transportation Of facials [**AASHTO**]

●Load Resistance Factor Design [**LRFD**]

The design loads to be considered and all load combination are defined in the design analysis stage

►**Materials**

Piers Class 25/30
Precast, post tensioned beams Class 35/45
In situ deck Class 25/30
Reinforcement
Grade 420, Type 2, high yield deformed bars ToBS4449-1997with
a minimum yield strength, FY = 420 N/mm2.

►Prestressing

15.2mm 7 wire strand to BS5896, fpu = 1860N/mm2 or similaragreed alt ernative to an internationally recognized standard.

Ref.	Calculations	Output
	Design data:	
	Properties of Prestressing Strand and Bar:	
	●Section area of the strand:	
	$A_{\phi} = \frac{\pi \times 15.2^2}{4}$	**182mm²**
	●Elastic modulus of prestress tendon (Mpa):	
	Ep =195000 Mpa	
	●Yield strength of prestressing steel (Mpa):	
	f_{pe}=0. 9f_{pu}=0. 9× 1860	**1674N/mm²**
	● **Check**:	
	Calculate minimum prestress force $f_f$can do tensile stress$f_t$in the bottom of girder, middle span and in	

Ref.	Calculations	Output
	the end of the service (after happen the losses) and under effect of the service loads III.	
Table 5. 9. 4. 2. 2. 1	f_t=0. 25$\sqrt{f'_c}$=0. 25$\sqrt{45}$= **1. 68N/mm²**	**1. 68N/mm²**
	$f_{bg}= \frac{f_t}{A_g}+\frac{f_f \times e_g}{S_{bg}}-\frac{M_{dg}+M_{ds}}{S_{bg}}-\frac{M_{da}+M_{LL}}{S_{bc}}$	
	Eccentricity of strands in the middle span:	
	h_g=0. 1× 2500	**250mm**
	$e_g= Y_{bg}- h_g$=1229. 88- 250	**979. 88mm**
	Note:	
	Moment of wearing surface, barrier and live loads affect the composite section.	
	Moments from other loads affect the section of girder only.	
	$-1.68= \frac{f_t}{982300}+\frac{f_f \times 979.88}{691.612\times 10^6}-\frac{(4812.67+2352.9)10^6}{691.612\times 10^6}-\frac{(9607+9970.234)10^6}{856.44\times 10^6}$	
	$f_t \geq$ **13000KN**	
	•Total area of strands (A_{ps})**:**	
	$A_{ps} \geq \frac{f_f}{0.6 f_{pu}} \geq \frac{13000\times 10^3}{0.6\times 1860} \geq$ 11830mm²	
	NO. $=\frac{A_{ps}}{A_{\phi}}=\frac{11830}{182}$= **65strands**	

Ref.	Calculations	Output
	Check form the area of reinforced steel under the load strength I:	
	$\phi M_n = \phi(0.9h) \times (0.95\, f_{pu} \times A_{ps} + f_y \times A_s) \geq M_u$	
A 5. 5. 5. 2. 1	$\phi = 0.9 + 0.1 \times (PPR)$, $\phi = 1.0$	
	$PPR = \frac{A_{ps} \times f_{py}}{A_{ps} \times f_{py} + A_s + f_y}$	
	$A_s = 0$, PPR = 1. 0	
	$\phi M_n = 0.9(0.9 \times 2680) \times (0.95 \times 1860 \times 11830)$	
	= 50419. 51KNm	
	$A_{ps} \geq \frac{\text{Mu}}{\phi \times 0.95 f_{pu} \times 0.9\text{h}} \geq \frac{50419.51 \times 10^6}{1.0 \times 0.95 \times 1860 \times 0.9 \times 2680}$	**11830mm^2**

Table 5-1 Tendons

N	Y mm	N Y
4	40	160
4	90	360
4	140	560
4	190	760
4	240	960
4	290	1160
2	340	680
2	440	880
2	490	980
2	540	1080
2	590	1180
2	640	1280
2	690	1380
1	740	740
2	779	1558
2	829	1658
2	879	1758
2	929	1858
2	979	1958
2	1029	2058
1	1079	1079
2	1117	2234
2	1167	2334
2	1217	2434
2	1267	2534
2	1317	2634
2	1367	2734
1	1417	1417

horizontal and vertical distance between cable =50mm

Ym$=\frac{40408}{65}$= **621. 7mm**

Ee**em**=Ybg-Ym=1229. 88-621. 7 =**608. 22mm**

At the two end of span

Yend $=\frac{39500}{65}$ = **607. 692mm**

eend = 1229. 88-607. 692 = **622. 188mm**

❖ **prestress losses**

Δfpt =ΔƑ PES +ΔƑ PLT

❖ **Initial losses{elastic strain }**

ΔƑ PES $=\frac{Ep}{Eci} f$cgp

- Ec =0. 043K1×ɣ c $^{1.5}\sqrt{F'C}$
- K1= 1. 0
- ɣ c= 23. 4305 Kg/m^3

Ec = 0. 043×1. 0×$(\frac{23.4305}{10^8})^{1.5}$ ×$\sqrt{45}$ =**13.27×10^{-11}N/mm^2**

Eci =4855$\sqrt{F'ci}$, F´ci =0. 75F´c= 0. 75×45=**33. 75N/mm^2**

Eci = 4855$\sqrt{33.75}$ = **28. 205×10^3 N/mm^2**

❖ **Initial stress in reinforced strand**

Ƒ pi = 0. 675fpu = 0. 675×1860 = **1255. 5N/mm^2**

❖ **Initial prestress forces**

Fi =Fpi ×Aps = 1255. 5×65×182 = **14852. 565 KN**

Ref.	Calculations	Output
	❖ **Equation to calculate elastic strain losses**	
	$\Delta Fpes = \frac{APS \times Fpi[Ig+em2 \times Ag]-em \times Mg \times Ag}{APs[Ig+em2 \times Ag]+\frac{Ag \times Ig \times Eci}{Es}}$	
	$\frac{11830 \times 1255.5[8.506 \times 1011+608.222 \times 982300]-608.22 \times 4812.67 \times 982300}{11830[8.506 \times 1011+608.222 \times 982300]+\frac{982300 \times 8.056 \times 1011 \times 28.205 \times 103}{195000}}$	
	ΔFpes= **14. 745K/mm2**	
	❖ **Timing losses**	
	Calculate timing of losses there result from creep And shrinkage in concrete from equation below	
	$\Delta fplT = 10 \times \frac{Fpi \times Aps}{Ag} \times \gamma h \times \gamma st + 83 \times \gamma h \times \gamma st + \Delta fpR$	
	γ h = 17-0. 01H H = 70% γ st= $\frac{35}{[7+fci]}$	
[A 5. 9. 5. 3]	ΔfpR = 17N/mm^2 Δfpi = 0. 75fpu	
	γ h = 1. 70-0. 01(70) = 1. 0	
	γ st = $\frac{35}{7+33.75}$ = 0. 86	
	fpi = 0. 75×1860 = 1395N/mm^2	
	$\Delta fplT = 10 \times \frac{1395 \times 65 \times 182}{982300} \times 1 \times 0.86 + 83 \times 1 \times 0.86 + 17$	
	ΔfplT= **232. 86 N/ mm^2**	
	Δfpt = Δfpes +Δfplt = 14. 745+232. 86 =**247. 607 N/mm^2**	

Stresses in beam at transverse stage

In this stage the resultant stress for prestress force

And weight of beam effective only

Effective stress in transverse stage

Fpi = 0. 75Fpu - ΔFpEs = 0. 75×1860-14. 745

= 1380. 255N/mm^2

Fp = Fpi ×Aps=1380. 255×65×182 = **16328. 42KN**

em= 608. 22mm

at the mid of girder

tension stress at top flange

Fti = $\frac{Fi}{Ag}$-$\frac{Fi\times em}{stg}+\frac{Mdg}{stg}<0.25\sqrt{Fc}$

Fti = $\frac{13573.66\times10^3}{982300}$-$\frac{13573.66\times10^3\times608.22}{669.701\times10^6}+\frac{7169\times10^6}{669.701\times0^6}$

=**8. 98N/mm^2**$<0.25\sqrt{45}$ **=1. 68K/mm^2**

[compressive stress]…………. Ok

Compressive stress at bottom flange

Fbi = $\frac{Fi}{Ag}+\frac{fi\times em}{sbg}$-$\frac{Mdg}{sbg}<0.45f'c$

=$\frac{16328.42\times10^3}{982300}$+$\frac{16328.4\times10^3\times608.22}{691.612\times10^6}-\frac{4812.67\times10^6}{691.612\times0^6}$

Ref.	**Calculations**	**Output**

Fbi = 24N/mm^2 □ [0. 45F´c=0. 45×45 =20. 25N/mm^2]

………………. . not ok

At the edge of girder

t top flange

Fti = $\frac{Fi}{Ag}$-$\frac{Fi \times eend}{stg} < 0.25\sqrt{Fc}$

Fti = $\frac{16328.42 \times 10^3}{982300}$-$\frac{16328.4 \times 10^3 \times 622.81}{669.701 \times 10^6}$

=**1. 45N/mm^2< $0.25\sqrt{45}$ =1. 68K/mm^2**

Compressive stress at bottom flange

Fbi = $\frac{Fi}{Ag} + \frac{fi \times eend}{sbg}$-$< 0.45f´c$

=$\frac{16328.42 \times 10^3}{982300}$+$\frac{16328.4 \times 10^3 \times 622.188}{691.612 \times 10^6}$

Fbi = 31. 31N/mm^2 □ [0. 45F´c=0. 45×45 =20. 25N/mm^2]

………………. . not ok

Stress at girder after all losses

Fpf = 0. 75Fpu - ΔFpT = 0. 75×1860-247. 607

= 1147. 393N/mm^2

FF = FpF ×Aps=1147. 393×65×182 = **13573. 66KN**

	Sudan University Of Science and Technology Collage of Engineering –School of Civil Engineering *Engineering Dept*	
Component	**Designed by:**	**Page No**
	Checked by:	
Ref.	**Calculations**	**Output**

At mid span of composite girder

- Compressive stress at top flange

Ftf = $\frac{Ff}{Ag} - \frac{ff \times em}{stg} + \frac{Mdg+Mds}{stg} + \frac{Mda+Mll}{sic} < 0.45f'c$

$= \frac{13573.66 \times 10^3}{982300} - \frac{13573.66 \times 10^3 \times 608.22}{669.701 \times 10^6} + \frac{[4812.67+2352.39] \times 10^3}{669.701 \times 0\text{^}6}$

$\frac{19577 \times 10^6}{1506.3 \times 0^6}$

=<u>24.8N/mm^2□ 20N/mm2</u>

not ok

- tension stress at top flange

Ftifbf = $\frac{Ff}{Ag} + \frac{Ff \times eend}{sbg} - \frac{Mdg+Mds}{btg} - \frac{Mda+Mdll}{sic} < 0.25\sqrt{Fc}$

Fbf = $\frac{13573.66 \times 10^3}{982300} - \frac{13573.66 \times 10^3 \times 608.22}{691.612 \times 10^6} - \frac{7165 \times 10^6}{691.612 \times 10^6}$

$- \frac{19577 \times 10^6}{1506.3 \times 10^6}$

=**<u>-21.5N/mm^2$< 0.25\sqrt{45}$ =1. 68K/mm^2</u>**

[compressive stress]…………. ok

Compressive stress at top flange under weight of slab,

Asphalt and barrier effect

Ftc= $\frac{Mda+Mll}{sic} < 0.45f'c = \frac{19577 \times 10^6}{1506.3 \times 10^6}$ =**<u>13N/mm^2</u>**

ok

Ref.	Calculations	Output
	• Check for cable area not exceed minimum limit.	
	Reinforced steel it enough when result moment strength	
	Greater than or equal [1. 33Mu or 1. 2Mcr] which lesser]	
	Mcr= Ss [fr + fcpe] - Mdnc[$\frac{sc}{snc}$ – 1] ≥ Sc fr	
[A5. 7. 3. 3. 2. 1]		
	Compressive stress at bottom flange under	
	prestress force	
	Effect [after losses]	
	fcpe = $\frac{Ff}{Ag}$+$\frac{Ff\times em}{sbg}$	
	fcpe = $\frac{13573.66\times10^3}{982300}$+$\frac{13573.66\times10^3\times608.22}{691.612\times10^6}$ =**25. 755N/mm^2**	
[A5. 4. 2. 6]	fr= 0. 63$\sqrt{Fc}$=0. 63$\sqrt{45}$ = 4. 226N/mm^2	
	Mdnc = Mdg+Mds [service load]	
	=4812. 67 + 2352.39 = **7165 KNm**	
	Sc=Sbc=856. 44×10^{6}mm^3	
	Snc =Sbg=691. 61210^{6}mm^3	
	Mcr= 856. 44×10^6 [4. 226 + 25. 755] – 7165×10^6	
	[$\frac{856.44\times10}{691.612\times106}$ – 1] = **4411 KN. m**	

	Sudan University Of Science and Technology Collage of Engineering –School of Civil Engineering *Engineering Dept*	
Component	**Designed by:**	**Page No**
	Checked by:	
Ref.	**Calculations**	**Output**

Scfr =856. 44×10^6×4. 226 = **3619. 32KNm**

Mcr= 4411 KN. M

1. 2Mcr=5923KN. m

1. 33Mcr= 5866KN. m

ΦMn= 50419. 51 □ 1. 2 Mcr ……………ok

❖ Shear

Taketotal nominal shear strength value below which lesser

- Vn=Vc+Vs+Vp
- Vn=0. 25fc bv dv +vp
- Vp=0

Nominal strength of concrete for shear

Vc=0. 083β$\sqrt{fc}$ bv dv

At mid span

de=h-Ym=2680-621. 7= **2058. 3mm**

0. 9de=0. 9×2058. 3=1852. 47mm

dv≥max

0. 72×2680=1929. 6mm

$a=\beta c=0.73\times237.92=173.682mm$

$dv=de-\frac{a}{2}=2058.3-\frac{173.682}{2}=1971.459mm$ □ $1929.6mm$

at end of span

de=h-Yend=2680-607. 692= **2072. 308mm**

$$dv\geq \max\begin{cases}0.9de=0.9\times2072.308=1865.077mm\\ 0.72\times2680=1929.6mm\end{cases}$$

$dv=de-\frac{a}{2}=2072.308-\frac{173.682}{2}=1985.467mm$ □ $1929.6mm$

bv=200mm

take total shear strength value below which lesser

- Vn=Vc+Vs+Vp
- Vn=0. 25fc bv dv +vp
- Vp=0

Vn=0. 25×45×200×1985. 467+0 = **4467. 3KN**

Nominal strength of concrete for shear

$Vc=0.083\beta\sqrt{fc}\ bv\ dv$

Ref.	Calculations	Output

Vc=0. 0830. 73$\sqrt{45}$ ×200× 1985. 467 = **161. 399KN**

Vs=$\frac{Vu}{\varphi} - Vc = 4467.3 - 161.399 = \mathbf{4305.901KN}$

S =$\frac{Av \times fy \times dv \times \cot\theta}{vs} \leq Smax$

Vu=$\frac{vu}{\varphi dvbv}$ $\frac{4020.57\times10^3}{0.9\times1985.467\times200}$=11.25 N/mm2□0. 125 fc □5.63

Smax =0. 4dv =**794. 19mm□ 300mm**

Ds=12mm

Av=2××$\frac{12^2}{4}$226mm2

s≤ $\frac{Av \times fy \times dv \times \cot\theta}{VS}$= $\frac{226\times420\times1985.467\times\cot 45}{V4305.901\times10^3}$

S=43mm

Use Φ12mm@40mm c/c

5.3 Discussion of design results

For The Slab:

Main Reinforcement:-

Use Φ16mm@200 mm c/c

Distributed Reinforcement:

Use Φ12mm@150 mm c/c

Reinforcement for Temperature and shrinkage:

Use Φ12mm@500 mm c/c

For The Girder:

Minimum prestress force f_f can do tensile stress f_f in the button of girder , middle span and in the end of the service (after happen the losses) and under effect of the service load III are fine.The result obtain total area of prestress equal to 3510 mm^2. No. of strand are 65 reinforced of concrete φ 12mm @ 40mmc/c.

Chapter 6
Conclusion and Recommendation

6.1 Conclusions

1. Soba Bridge was analyzed and designed using MIDAS/CIVIL computer program.
2. The bridge was analyzed and designed using AASHTO LRFD.
3. The results obtained for the maximum deflection was 757mm which is acceptable.
4. The results obtained for the maximum Moment was 27153. 32 KN. M which was in the range.
5. The results obtained for the maximum shear was 4365. 38KNwhich is acceptable.
6. The results obtained for the required reinforcement of girder according to the AASHTO.

6.2 Recommendations

6.2.1 General Recommendations

Throughout the research we were faced by many problems like lack of reference specialized in bridges and availability of suitable software, it is recommended that:

1-Referencesrecently published should be made available in sufficient copies.

2-Intensive courses particularly showing how to analyze and design bridge in the up to date style should be gun by the department.

3.Specialized courses in computer applications and use of package like (LUSA S, SAP2000, ANSIS)…. .etc.; should be made available.

6.2.2 Recommendation for Future Studies

1-The design it is recommended that or the Pier and cross head should be carried out to complete this project.

2-This project should be extended to cover the analysis and design of Foundation (piles) for bridge.

3-The use of programs which have the ability to include different sections without specifying the size should be considered.

4-The original versions of the software package should be obtained for use in graduation projects.

References

1 -Wai-fah Chain and Liam Duan , Bridge Engineering Handbook, Florida, CRC press LLC, 2000.

2-Colin O’Conner andPeter A Shaw , Bridge loads , United states of America , Canada , Spon Press , 2000.

3- JimZhau ,Tonios Demetrio’s , Bridge Engineering , USA , McGRAW Hill , 2012.

4-W.Hussin ,A.Bakr and asjad ,Analysis and Design of soba Bridge as a Cable Stayed Bridge,Final year project, Department of Civil Eng SUST,2014.

5-Toma, S.; Duan, L. and Chen, W.F. “Bridge Structures” Structural Engineering Hand book Ed. Chen Wai-FahBoca Raton: CRC Press LLC,1999

6-Guyon, Y., Limit State Design of Prestressed Concrete, Vol. 2,Applied Science Publishers, 1974.

7-American Association of State Highway and Transportation Officials 444 North Capitol Street, NW Suite 249 Washington, DC 20001.