DIFFERENT TECHNIQUES FOR THE MODELING OF POST-TENSIONED CONCRETE BOX-GIRDER BRIDGES

Deepak Rayamajhi
Graduate Student
Structural Engineering
Asian Institute of Technology
Bangkok, Thailand
rayamajhi_deepak@yahoo.com

Naveed Anwar
Associate Director
ACECOMS
Asian Institute of Technology
Bangkok, Thailand
nanwar@ait.ac.th

Jimmy Chandra
Graduate Student
Structural Engineering
Asian Institute of Technology
Bangkok, Thailand
chandra.jimmy@yahoo.com

ABSTRACT:

This paper presents a comparison between response of post-tensioned concrete box-girder bridges analyzed by using different modeling techniques. Each type of modeling technique requires a certain set of assumptions to simplify the problem and thus results obtained from these techniques vary according to the assumptions made. In this study, a typical concrete box-girder bridge is analyzed using commercially available finite element software for two analysis cases, static and modal. Static analysis is used to study the responses of the bridges for dead load, moving load and post-tensioning load cases whereas modal analysis is used to study modal dynamic responses of the bridge. Finally, response of the bridge is compared in terms of natural time periods, mode shapes, support reactions, deformations, and internal forces. This study shows that with proper assumptions and modeling techniques, the approximate response of box-girder bridges can be predicted by different models.

KEYWORDS: Post-Tensioned, Box-Girder Bridge, Finite Element Modeling, Bridge Response.

1. Introduction

Nowadays, the use of computer models to perform structural analysis in the field of bridge engineering has become a common practice. Engineers are supposed to use proper model in order to accurately predict the response of the bridge model for design purposes. In recent years, researchers have been developing many modeling techniques that can be used to model a bridge. Each type of modeling techniques requires some set of assumptions to simplify the problem and thus results obtained from these techniques vary according to the assumptions made [2]. Moreover, each type of modeling techniques has its own advantages and disadvantages. Therefore, sometimes engineers use several modeling techniques to model complex bridge structures in order to compare the results and to use them for various purposes.

Several bridge modeling techniques are discussed by Hambly [3] for the modeling of different types of bridges. The modeling techniques being used by engineers to model a bridge range from the simplest one to the highly complex one. The simplest model (spine model) usually comprises of only one single girder to model the bridge deck and hinge or roller supports to model the bearing and abutment. Even though this model is very simple, it is able to give reasonable prediction of the bridge responses under dead load such as maximum displacement and moment at the mid span and support reactions. However, there are many limitations of this model such as transverse analysis of moving load in bridge deck cannot be performed, inaccurate prediction of modal analysis, etc. Therefore, nowadays most of engineers use this model only to do preliminary analysis or sizing of a bridge's components.

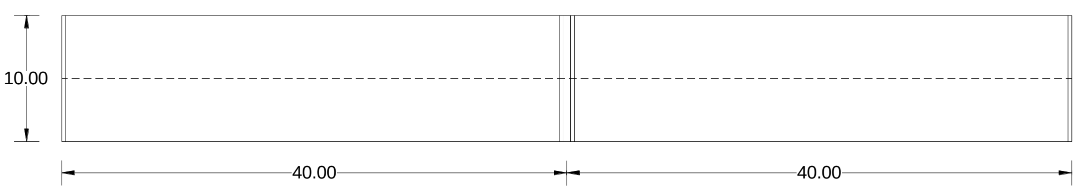
To overcome some limitations in spine model, a frame/grid model was developed. In this model, the bridge deck is modeled using frames and these frames are connected each other using resenting the diaphragms at supports. In this model, transverse analysis of moving load in bridge deck can be performed. Furthermore, this model gives more accurate prediction of modal responses as compared to spine model. Nevertheless, slab behavior cannot be modeled properly in this model, especially the two-way response including twisting.

Frame shell model can be used to improve the accuracy of frame/grid model. In this model, the slab is modeled using shell elements, which means the effects of the slab (both for out-of-plane loads and in-plane stresses) are included explicitly in the model for analyzing purposes. Thus, it can improve the accuracy of analysis results under several load cases.

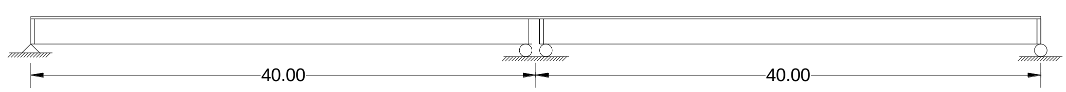
Full shell model is considered as complex modeling technique for bridge. In this model, all elements are modeled using shell elements. Due to its complexity, sometimes it is difficult to extract information from analysis results for design purposes. However, with the help of powerful analysis software and computers that are available nowadays, this problem can be overcome. Therefore, many engineers have started to use this model in order to get accurate prediction of bridge responses under many load cases.

2. Description of Bridge

In this study, a typical concrete box-girder bridge is modeled and analyzed. The bridge has total length of 80 m. The box-girder bridge has two spans of 40 m each and 10 m wide. The clearance of the bridge above the ground level is 7 m. The elevation and cross section of the bridge can be seen in Figure 1 and 2, respectively.



(a)



(b)

Figure 1: (a) Plan view of the bridge; (b) Elevation view of the bridge.

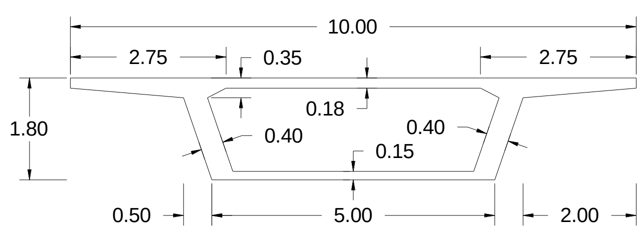


Figure 2: Cross section of the bridge

Here, the concrete compressive strength (fc') for the box-girder is 41 MPa and modulus of elasticity of the concrete is 30442 MPa. In this study, the selected bridge has only two lanes for the vehicles.

The connection between the deck and pier I restricts the displacement of the deck in X, Y, and Z direction and permits the rotation about X, Y, and Z direction (i.e. hinge support). The connections between the deck and pier II and III permit the displacement along X direction and rotation about X, Y, and Z direction but restrict the displacement along Y and Z direction (i.e. roller support).

3. Modeling Techniques

In this study, finite element models of bridge are developed in SAP2000. The bridge is modeled with five different modeling approaches in which the complexity of the model is gradually increased. They are spine model, frame model, grid model, frame shell model and full shell model. Description, details as well as assumptions made in each model are discussed below.

Modeling Approach 1: Spine Model

In spine model, the bridge is modeled using a single girder which represents the whole cross section of the bridge. The section designer function in SAP2000 is used to model the cross section. Since the centroid of the single girder which represents the whole cross section of the bridge is not located at the bottom side of the box-girder, it thus gives improper supports location. Therefore, the location of the supports is modified with the addition of rigid links which connect the single girder and the supports. Furthermore, the supports are constrained to eliminate the instability in torsional response of the deck (refer to figure 3). Later on, the results between models with unmodified and modified supports are compared.

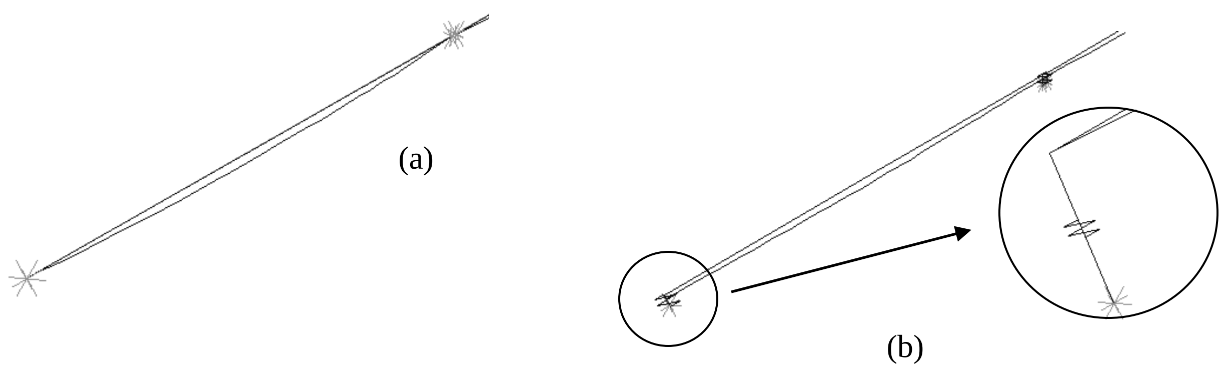


Figure 3: (a) Spine model; (b) Spine model with modified supports

Modeling Approach 2: Frame Model

In frame model, instead of modeling the bridge with a single girder, the bridge is divided into two girders (along the longitudinal direction) where each of them represents a half part of the whole cross section. These two girders are connected transversally each other with diaphragms at the support locations (refer to figure 4). As in the case of spine model, rigid links are used to modify the supports location and the results between models with unmodified and modified supports are compared.

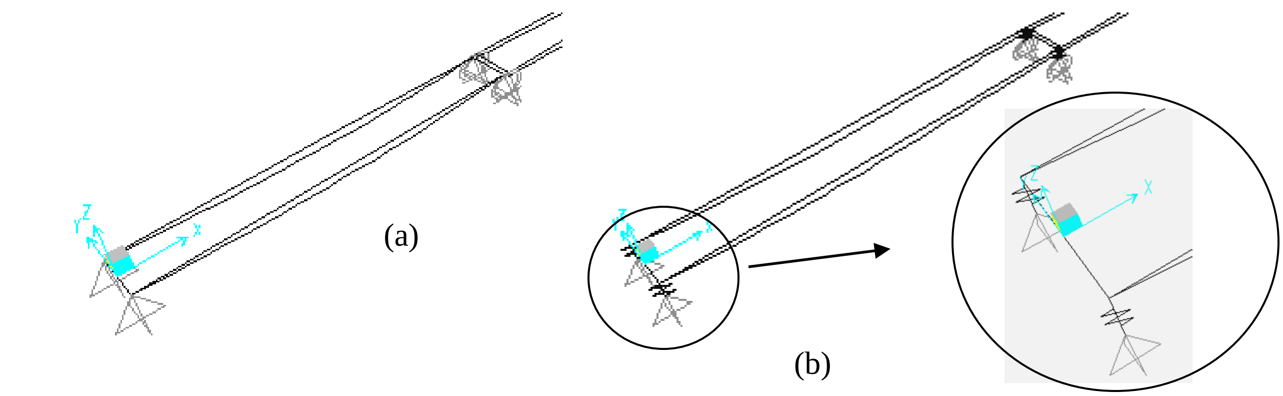


Figure 4: (a) Frame model; (b) Frame model with modified supports

Modeling Approach 3: Grid Model

First the bridge cross section is divided into two sections, top concrete slab (or deck) of 0.18 m depth and remaining 'U' shape section. The remaining section is also divided into two girders along the longitudinal direction where each of them represents a half part of 'U' shape section. The two girders are connected in transverse direction by frame elements with spacing of one meter. These frame elements represents the top concrete slab as defined before. As in the case of spine model, in grid model, rigid links are also used to modify the supports location and the results between models with unmodified and modified supports are compared (refer to figure 5).

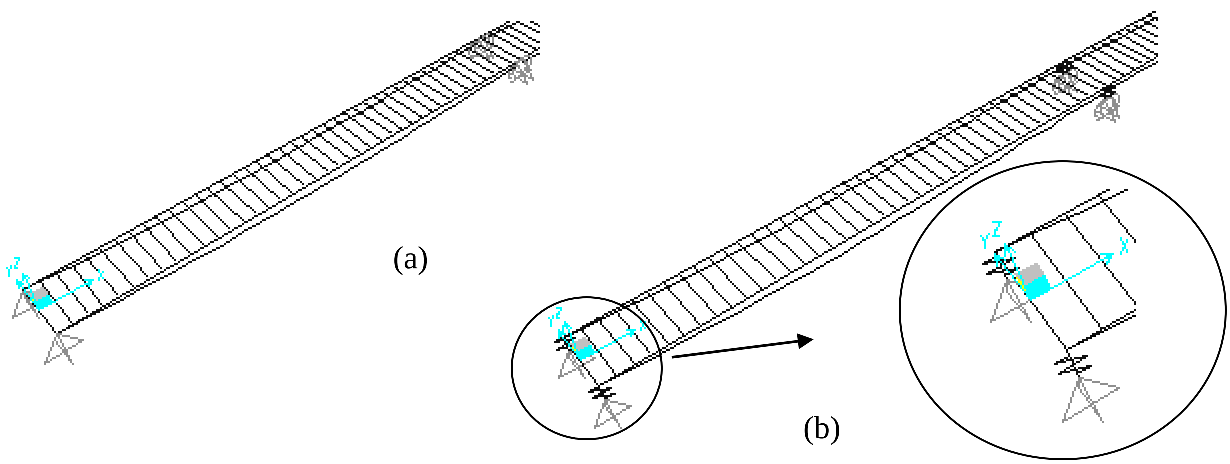


Figure 5: (a) Grid model; (b) Grid model with modified supports

Modeling Approach 4: Frame Shell Model

In this model also the bridge cross section is divided into two sections, top concrete slab (or deck) of 0.18 m depth and remaining 'U' shape section. The remaining section is divided into two girders along the longitudinal direction where each of them represents a half part of 'U' shape section. The concrete slab (or deck) is modeled using shell elements and girders are modeled using frame elements. Generally, in FEA model if the slab and girder are drawn normally without any modifications, it will give improper cross section shape as well as properties. This is because the mid-plane of the shell elements will be located at the same elevation with that of frame elements. In this study, two methods to solve this problem are presented. The first one is to modify the insertion point of the frame elements and the second one is to draw the frame elements at different elevation than that of shell elements and connect them with rigid links. In frame shell model, rigid links are used to modify the supports location as in the case of spine model. However, only the results with modified supports are presented (refer to figure 6).

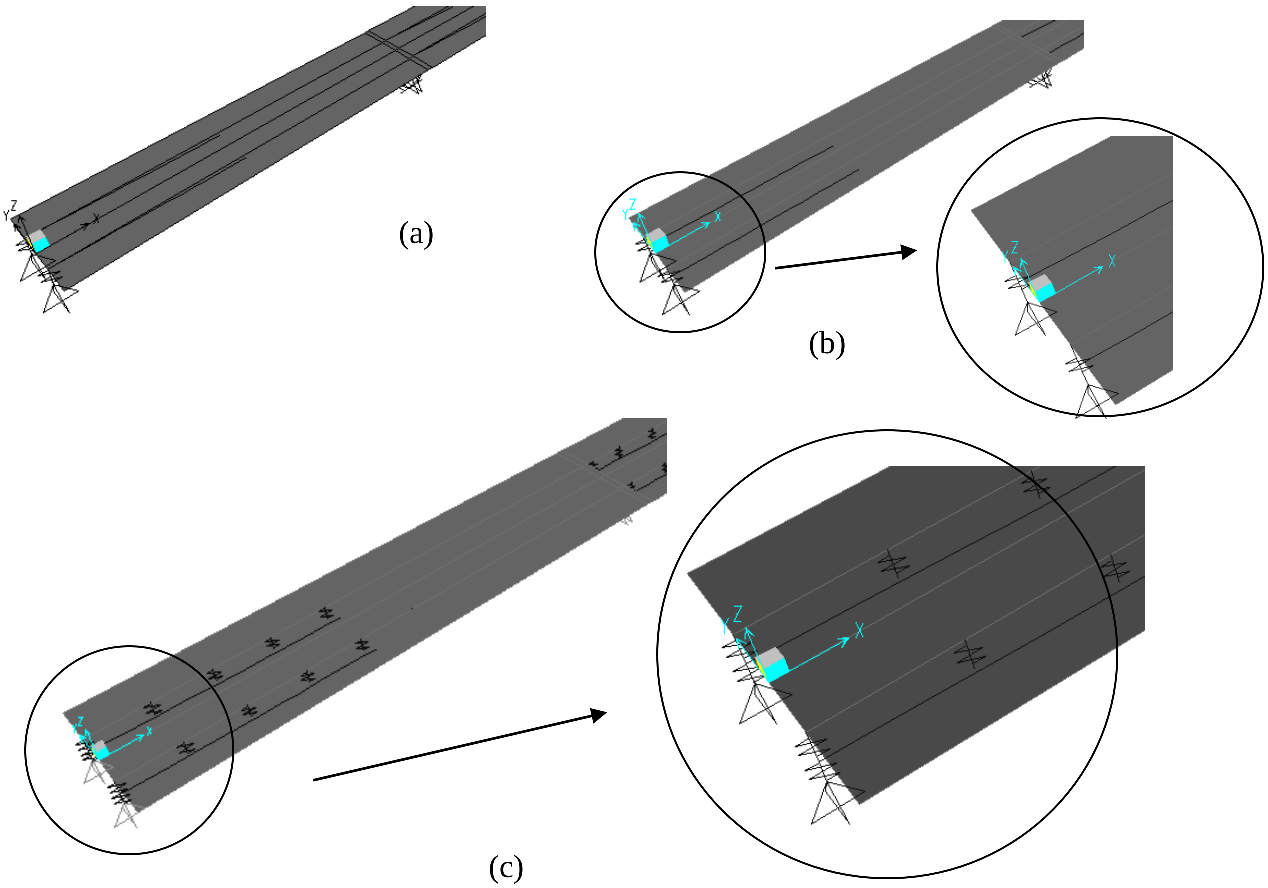


Figure 6: (a) Frame shell model; (b) Frame shell model with insertion point; (c) Frame shell model with beam offset

Modeling Approach 5: Full Shell Model

In full shell model, whole box-girder section is modeled using shell elements. One of the advantages of this model is the supports can be directly put under the Box-girders. However, rigid links are still being used to model the bearings (refer to figure 7).

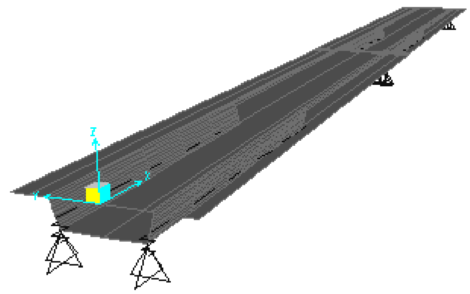


Figure 7: Full shell model

The summary of all modeling techniques is presented in Table 1.

Table1. Summary of modeling techniques

Code	Model Type	Code	Model Type
1a	Spine model	3b	Grid model with modified supports
1b	Spine model with modified supports	4a	Frame shell model
2a	Frame model	4b	Frame shell model with insertion point
2b	Frame model with modified supports	4c	Frame shell model with beam offset
3a	Grid model	5	Full shell model

4. Load Cases and Analysis Cases

In this study, the bridge is analyzed and reviewed for static and modal analysis cases. Static analysis is used to study the responses of the bridge for dead load, moving load and post-tensioning load cases whereas modal analysis is used to study the mode shapes of the bridge. Dead load is considered from the self weight of the bridge. The standard truck HSn-44 in accordance with American Association of State Highway and Transportation Officials (AASHTO) [1] is used for moving load cases. The moving truck loads are applied in both two lanes in opposite direction with particular vehicle speed. For post-tensioning load cases, tendon elements are used as a load pattern on the bridge. The post-tensioning is estimated in such a way that moments produced by the post-tensioning effect should be able to balance most of the moment from dead load.

5. Analysis Results and Discussions

The response results from different modeling techniques are compared in terms of natural time periods or frequencies, mode shapes, support reactions, deformations and internal forces. The comparison of these analysis results is shown on following tables.

Table2. Comparison of natural periods and mode shapes

Model	Mode 1		Mode 2		Mode 3	
Code	Period (s)	Shape	Period (s)	Shape	Period (s)	Shape
1a	0.442	Longitudinal	0.427	Longitudinal	0.120	Transverse
1b	0.447	Longitudinal	0.430	Longitudinal	0.120	Transverse
2a	0.442	Longitudinal	0.438	Longitudinal	0.427	Longitudinal
2b	0.447	Longitudinal	0.439	Longitudinal	0.430	Longitudinal
3a	0.437	Longitudinal	0.421	Longitudinal	0.375	Torsional
3b	0.441	Longitudinal	0.424	Longitudinal	0.379	Torsional
4a	0.663	Longitudinal	0.621	Longitudinal	0.539	Torsional
4b	0.443	Longitudinal	0.438	Longitudinal	0.399	Torsional
4c	0.449	Longitudinal	0.432	Longitudinal	0.347	Torsional
5	0.476	Longitudinal	0.466	Longitudinal	0.366	Torsional

Table3. Comparison of maximum displacement at middle span under different load cases

Model Code	Dead (mm)	Live (mm)	Post-tensioned (mm)
1a	-61	-7	54
1b	-61	-7	54
2a	-6	-14	54
2b	-61	-14	54
3a	-61	-12	54
3b	-61	-11	54
4a	-136	-25	62
4b	-61	-11	48
4c	-61	-10	52
5	-66	-9	59

Table4. Comparison of maximum moment at middle span under different load cases

Model Code	Dead (kNm)	Live (kNm)	Post-tensioned (kNm)
1a	18436	2682	-15424
1b	18436	2680	-15424
2a	18436	5204	-15352
2b	18436	5154	-15352
3a	18448	4440	-15350
3b	18448	4164	-15352
4a	15282	2828	-7090
4b	17585	2813	-24502
4c	17941	2858	-27850
5	19558	2821	-25872

Table5. Comparison of support reactions under different load cases

Model Code	Exterior Support Reactions			Interior Support Reactions		
	Dead (kN)	Live (kN)	Post-tensioned (kN)	Dead (kN)	Live (kN)	Post-tensioned (kN)
1a	1909	292	52	2024	304	-52
1b	1909	292	54	2024	304	-54
2a	1942	544	54	2058	2093	-54
2b	1942	546	54	2058	2087	-54
3a	1908	544	54	2024	1513	-54
3b	1909	563	54	1910	1399	-54
4a	1682	599	63	1892	2210	-63
4b	1762	605	27	1920	1263	-27
4c	1816	562	29	1854	985	-29
5	2026	611	32	2004	559	-32

From the modal dynamic analysis result, it can be seen that all models give first and second mode as longitudinal mode. However, for third mode, spine model show transverse mode, frame model show longitudinal mode, and the other models show torsional mode. The natural periods for first and second mode obtained from all model in average (excluding frame shell model natural periods for calculating average) is around 0.45s and 0.44s. Other than frame shell model, all the model predicts first and second natural period lower than full shell model. Furthermore, it should be noted that in all three modes, frame shell model without any modifications always gives higher natural period which means the structure is more flexible. This happens due to incorrect girder location which in this model, the girder centroid is located at the same elevation as the slab centroid. Thus, it reduces the moment of inertia of the whole section of the bridge and reduces the bridge stiffness.

For maximum displacements and moments at middle span, almost all models give approximately same values. The major difference can be found in frame shell model without any modifications. As explained before, incorrect modeling of the girder location causes reduction in the bridge stiffness. Therefore, in this model, the displacement values are higher as compared to other models in all load cases (dead, live, and post-tensioned).

6. References

[1] AMERICAN ASSOCIATION OF STATE HIGHWAY AND TRANSPORTATION OFFICIALS (AASHTO), *LRFD Bridge Design Specifications*, 2nd Ed. with Interims, Washington, D.C., U.S.A., 2003.

[2] CHEN, W.F., DUAN, L., *Bridge Engineering Handbook*, CRC Press LLC, Boca Raton, Florida, U.S.A., 2000.

[3] HAMBLY, E. C., *Bridge Deck Behaviour*, Routledge Mot E. F. & N. Spon, 2nd Edition, 1990