

## Appendix B

### GENERAL GUIDELINES FOR REFINED ANALYSIS OF DECK SLABS

Traditionally, deck slabs have been analyzed using approximate methods. The approximate methods are based on calculating moments per unit width of the deck and design the reinforcement to resist these moments. This approach has been used successfully for many decades. However, the approximate methods were generally based on laboratory testing and/or refined analysis of typical decks supported on parallel girders and no skews. In case of deck slabs with unusual geometry, such as sharply skewed decks, the results of the approximate methods may not be accurate. For example, negative moments may develop at the acute corner of a sharply skewed deck. These moments are not accounted for in the approximate methods as they rely on assuming that the deck is behaving as a continuous beam.

In cases of unusual deck geometry, bridge designers may find it beneficial to employ refined methods of analysis. Typically the use of the refined methods of analysis is meant for the design of both of the girders and the deck slab. The design method of analysis most used is the finite element analysis. However, for deck slabs, other methods such as the yield line method and the finite differences method may be used. Following is a general description of the use of the finite elements in analyzing deck slabs.

#### **Finite element modeling of decks**

##### **Type of elements**

The finite element method is based on dividing a component into a group of small components or “finite elements”. Depending on the type of the element, the number of displacements (translations and rotations) varies at each end or corner of the element varies. The displacements are typically referred to as ‘degrees of freedom’. The basic output of the analysis is the displacements at each node. These displacements are then converted into forces at the nodes. The force output corresponding to a rotational degrees of freedom is in the form of a moment while forces correspond to translational degrees of freedom. Following are the types of elements typically used to model a plate structure and the advantage and disadvantages of each type.

**Plate elements:** Plate elements are developed assuming that the thickness of the plate component is small relative to the other two dimensions. The plate is modeled by its middle surface. Each element typically has four corners or nodes. Most computer programs have the ability of handling three-node or triangular plate elements, which are typically treated as a special case of the four-node basic element. Following the general plate theory, plate elements are assumed have three allowed displacements at each node; translation perpendicular to the plate and rotations about two perpendicular axes in the plane of the plate. The typical output includes the moments (usually given as moment per unit width of the face of the elements) and the shear in the plate. This form of output is convenient because the moments may be directly used to design the deck.

The main disadvantage of plate elements is that they do not account for the forces in the plane of the plate. This results in ignoring the stiffness of the plate elements in this plane. This precludes them from being used as part of a three-dimensional model to analyze both the deck and the girders.

The deck supports are modeled as rigid supports along the lines of the supporting components, i.e. girders, diaphragms and/or floor beams. Where it is desirable to consider the effect of the flexibility of the supporting components on the deck moments, the model may include these components that are typically modeled as beams. As the plate elements, theoretically, have no in-plane stiffness, the effect of the composite action on the stiffness of the beams should be considered when determining the stiffness of the beam elements.

**Shell elements:** Shell elements are also developed assuming that the thickness of the component is small relative to the other two dimensions and are also modeled by their middle surface. They differ from plate elements in that they are considered to have six degrees of freedom at each node, three translations and three rotations. Typically the rotation about the axis perpendicular to the surface at a node is eliminated leaving only five degrees of freedom per node. Shell elements may be used to model two dimensional (plate) components or three-dimensional (shell) components. Commercially available computer programs typically allow three-node and four-node elements. The typical output includes the moments (usually given as moment per unit width of the face of the elements) and the shear and axial loads in the element. This form of output is convenient because the moments may be directly used to design the deck.

Due to the inclusion of the translations in the plane of the elements, shell elements may be used as part of a three-dimensional model to analyze both the deck and the girders. When the supporting components are modeled using beam elements, only the stiffness of the noncomposite beams is introduced when defining the stiffness of the beams. The effect of the composite action between the deck and the supporting components is automatically included due to the presence of the in-plane stiffness of the shell elements representing the deck.

**Solid elements:** Solid elements may be used to model both thin and thick components. The thickness of the component may be divided into several layers or, for thin components such as decks, may be modeled using one layer. The solid elements are developed assuming three translations at each node and the rotations are not considered in the development. The typical output includes the forces in the direction of the three degrees of freedom at the nodes. Most computer programs have the ability to determine the surface stresses of the solid elements. This form of output is not convenient because these forces or stresses need to be converted to moments that may be used to design the deck. Notice that, theoretically, there should be no force perpendicular to the free surface of an element. However, due to rounding off errors, a small force is typically calculated.

Similar to shell elements, due to the inclusion of all translations in the development of the elements, solid elements may be used as part of a three-dimensional model to analyze both the deck and the girders. When the supporting components are modeled using beam elements, only the stiffness of the noncomposite beams is introduced when defining the stiffness of the beams.

**Element size and aspect ratio:** The accuracy of the results of a finite element model increases as the element size decreases. The required size of elements is smaller at areas where high loads exist such as location of applied concentrated loads and reactions. For a deck slab, the dividing the width between the girders to five or more girders typically yields accurate results. The aspect ratio of the element (length-

to-width ratio for plate and shell elements and longest-to-shortest side length ratio for solid elements) and the corner angles should be kept within the values recommended by the developer of the computer program. Typically an aspect ratio less than 3 and corner angles between 60 and 120 degrees are considered acceptable. In case the developer recommendations are not followed, the inaccurate results are usually limited to the nonconformant elements and the surrounding areas. When many of the elements do not conform to the developer recommendation, it is recommended that a finer model be developed and the results of the two models compared. If the difference is within the acceptable limits for design, the coarser model may be used. If the difference is not acceptable, a third, finer model should be developed and the results are then compared to the previous model. This process should be repeated until the difference between the results of the last two models is within the acceptable limits.

For deck slabs with constant thickness, the results are not very sensitive to element size and aspect ratio.

**Load application:** Local stress concentrations take place at the locations of concentrated loads applied to a finite element model. For a bridge deck, wheel loads should preferably be applied as uniform load distributed over the tire contact area specified in Article S3.6.1.2.5. To simplify live load application to the deck model, the size of the elements should be selected to eliminate the partial loading of some finite elements, i.e. the tire contact area preferably match the area of one or a group of elements.