APPENDIX D

FEA REQUIREMENTS FOR SYSTEM ANALYSIS

Based on the work performed to characterize the redundancy of the bridges studied, a set of requirements and guidance for the modeling was developed. Given that the characterization of redundancy requires the consideration of every alternate load path and complex interactions between the components of a bridge, 3-D finite element analysis is considered as the most suitable analysis tool. The requirements to adequately perform finite element analysis for redundancy are presented below.

D.1 Analysis Software and Solution Procedure Requirements

Although the methodology that has been developed was implemented using Abaqus, the analyst may utilize any software and solution procedure provided that:

a. The geometry of the finite element model is three-dimensional.
b. Inertial effects are either neglected (static analysis) or negligible (quasi-static analysis). The kinetic energy shall not be larger than 5% of the strain energy of the system.
c. The system is stable for the time increments specified. For an implicit analysis the system will be unconditionally stable. For an explicit analysis the system stability will be conditional to a stable time increment that depends on the size, mass and stiffness of the utilized methods.
d. Non-linear geometry (large deformation theory with finite strains and finite rotations) is used.
e. The pouring sequence of the slab is considered. Particularly, the finite element analysis must consider:
    - The slab does not carry any significant portion of dead loads prior to hardening.
    - When dead loads are applied, the slab does not provide any significant stiffness prior to hardening.
    - The slab must deform in accordance with the deformation of the structural steel during the application of dead load. The slab shall not sag nor slump between girders excessively, although minor deformations reasonably consistent with real in-situ behavior are acceptable.
    - Once dead load is applied, and the slab has deformed appropriately, it must be able to carry live loads and contribute to the stiffness of the system after it has hardened.
f. Gradual failure of a primary member can be modeled without significantly increasing kinetic energy. Stress and displacement amplifications are considered through the use of fracture amplification factors. The analyst shall not model the dynamic behavior of the structure due to sudden failure of a primary steel member; but shall model how failure of a main member alters static load distribution. In order to achieve that, the structure shall be loaded in the undamaged state and then subjected to failure of a primary member.
g. Material non-linearity, particularly inelastic behavior of steel and concrete inelasticity, can be explicitly modeled.

h. The software is capable of modeling kinematic constraints. Particularly the following constraints must be performed:
   - Embedment: For a truss or beam element embedded in a host solid element, the translational degrees of freedom of the nodes of the embedded element are constrained to the interpolated values of the corresponding degrees of freedom of the host element. This constraint is used to model the interaction between rebar and concrete.
   - Tie: The motion of a slave surface or node group is set equal to the motion of a master surface or node group.
   - Coupling: The motion of a slave surface or node group is constrained to the motion of a single master node.

i. The software is capable of modeling the contact interaction between the slab and the steelwork. This includes normal contact (i.e., hard pressure-overclosure relation) and Coulomb's frictional behavior. The contact enforcement method could be direct, through a penalty stiffness, or any other method provided that the Engineer has stablished its adequacy.

j. The following finite elements are implemented in the software:
   - 8-node linear bricks with reduced integration (ideally with hourglass control).
   - 4-node shells with reduced integration and finite membrane strains (ideally with hourglass control).
   - 2-node linear shear-flexible (Timoshenko) beam elements.
   - 2-node truss elements with linear displacement.
   - 2-node three-dimensional spring elements (also known as connectors, links, etc.). Coupled force-displacement and moment-rotation relations, elastic and inelastic behavior shall be available in these elements.

k. Surface tractions, body forces and prescribed displacements can be applied to the geometries of the finite element model.

D.2 Requirements for Modeling the Behavior of Steel Components

All steel components must follow a linear elastic-kinematic hardening plastic material constitutive model. Unless shown otherwise through material testing, the following assumptions can be made regarding steel material properties:

a. The modulus of elasticity can be assumed to be 29,000 ksi and the Poisson's ratio shall be 0.3.
b. Hardening shall be linear, with yield onset at nominal yield strength and reaching nominal ultimate strength at a plastic strain of 0.05.
c. The steel element shall fail or be deleted once a plastic strain of 0.05 is reached to simulate ductile fracture.

Shell element formulation best addresses typical plate behavior characteristic of structural steel members, so, whenever possible structural steel members shall be modeled with 4-node linear shell elements with finite membrane strains and a minimum of five Simpson thickness integration points. A minimum of four elements shall be used along the flange width or the web height. Unless prohibited by the geometry of the member, the maximum element aspect ratio shall be less than 5, and corner angles shall be kept between 60 and 120 degrees. Particularly, the following members do require modeling with shell elements:

- All steel members in contact with the slab.
- Tub girders in tub girder systems.
- Plate girders and stringers in plate girder systems.
- Truss members for primary truss members (i.e., not cross bracing between plate girders).
- Fabricated plate floor beams.
- Connections of primary steel members, e.g., gusset plate connections in truss bridges.

Secondary steel members that are not in contact with the concrete slab may be modeled with 2-node linear shear-flexible beam elements. A minimum of three elements shall be used along the length of the member. The following members may be considered secondary for the application of these provisions:

- Lateral braces, sway braces, and cross-frames or diaphragms;
- Chords and diagonals in truss-bridge floor beams;
- Construction braces.

Vertical stiffeners in plate and tub girder may be either explicitly modeled with shell elements or through a coupling constraint. If shell elements are used, one element through the width of the stiffening element is sufficient, but maximum aspect ratio shall be kept under 5, and corner angles kept between 60 and 120 degrees. If coupling constraints are used, they shall be applied to prevent cross-sectional distortion at the location of the stiffener.

D.3 Requirements for Modeling the Behavior of Concrete Slabs

The concrete slab shall be composed of two elements: 8-node linear bricks with reduced integration (ideally with hourglass control) to model concrete and 2-node truss elements with linear displacement to model reinforcing steel (rebar). To model the interaction between concrete and rebar, the elements modeling rebar must lie within the elements modeling concrete, and the translational degrees of freedom of the nodes of the elements modeling rebar must be constrained to the interpolated values of the corresponding degrees of freedom of the element modeling concrete.

The material behavior of rebar shall follow a linear elastic-kinematic hardening plastic material constitutive model. Unless shown otherwise through material testing, the following assumptions can be made regarding steel material properties:

a. The modulus of elasticity of the rebar shall be assumed to be 29,000 ksi and the Poisson's ratio shall be 0.3.
b. Hardening shall be linear, with yield onset at nominal yield strength and reaching nominal ultimate strength at a plastic strain of 0.05. Once a plastic strain of 0.05 is reached the steel element shall fail or be deleted.

The material behavior of concrete is based on its nominal compressive strength (f_c'). Initially, the material shall be linear elastic, Poisson's ratio of 0.3, and modulus of elasticity, E_c, as follows:

$$E_c = 33{,}000(w_c)^{1.5}(f_c')^{0.5} \leq 1802.5(f_c')^{0.5}$$

Concrete inelasticity shall be different for tension and compression. In tension, the elements modeling concrete cannot reach tensile stress in excess of the tensile failure stress, f_t, defined as follows:

$$f_t = \begin{cases} 0.158(f_c')^{2/3} & for\ f_c' \leq 7.25\ ksi \\ 0.307\ln(f_c' + 2.61) - 0.114 & for\ f_c' > 7.25\ ksi \end{cases}$$

Once at that point the element shall fail or initiate softening with a maximum fracture energy, G_t, as follows:

$$G_t = 5.9 \cdot 10^{-4}(f_c' + 1.16)^{0.18}$$

In compression, the elements modeling concrete cannot reach compressive stress in excess of f_c', and should follow the stress-strain Popovics' stress-strain relation, which is:

$$f(\varepsilon) = f_c'\left(\frac{\varepsilon}{\varepsilon_c}\right)\left[\frac{n}{n - 1 + \left(\frac{\varepsilon}{\varepsilon_c}\right)^n}\right]$$

where f_c' is the compressive strength of concrete, ε_c is the total strain at compressive strength, and n is a parameter calculated from experimental data. It should be noted that ε is the total strain ($\varepsilon = \varepsilon_{elastic} + \varepsilon_{plastic}$). The total strain at compressive strength, ε_c, the experimental parameter, n, and the plastic strain, $\varepsilon_{plastic}$, may be calculated as:

$$n = 0.4f_c' + 1.0$$
$$\varepsilon_c = 0.00124\sqrt[4]{f_c'}$$
$$\varepsilon_{plastic} = \varepsilon - \frac{f(\varepsilon)}{E_c}$$

The reinforced concrete slab may be modeled by (1) using solid elements to model the concrete, and embedded truss (wire) elements to model the reinforcement, or by (2) using shell elements in which the effect of the layers of reinforcements is implicitly included. It is recommended that Engineer approximately includes the geometry of the haunch, and the concrete barriers and their reinforcement may be included as part of the slab system, regardless of the approach utilized to model the concrete slab.

When using the first approach (i.e., truss (wire) elements embedded in solid elements), the elements modeling the concrete slab shall be 8-node linear brick elements with reduced integration. The material model of the solid elements shall model the behavior of concrete, as previously defined. A minimum of eight elements shall be used through the thickness of the slab in the regions close to the fracture, which is generally within a distance of one half the width of the deck on each side of the failure location. Fewer elements may be used through the thickness in other regions, but no fewer than four shall be used. The maximum element aspect ratio shall be less than 5. Unless prohibited by the geometry of the slab, corner angles shall be kept between 40 and 140 degrees. At the locations in contact with steelwork, e.g., bottom slab haunches, the mesh density should be higher than the mesh density of the steelwork to ensure proper enforcement of the contact interaction. The reinforcing steel within the slab shall be modeled by using truss (wire) elements embedded within the solid elements. The material model of the wire elements shall model the behavior of the steel rebar. The elements shall be 2-node linear truss elements. The length of the wire elements shall be approximately equal to the largest dimension of the concrete element.

When using the second approach, (i.e., shell elements in which the effect of the layers of reinforcements is implicitly included), the elements modeling the reinforced concrete slab shall be 4-node linear shells with reduced integration, finite membrane strains, and a minimum of 5 Simpson thickness integration points. The effect of the reinforcement shall be included as a material property or in the integration of the shell section. The Engineer shall test the performance of the shell element when the effects of the reinforcement are included in the element formulation, and verify that the nominal shear resistance of the slab is not exceeded. In general, the mesh density shall be similar to the one utilized for the steel elements. At the locations in contact with steelwork, e.g., bottom slab haunches, the mesh density should be higher than the mesh density of the steelwork to ensure proper enforcement of the contact interaction. Haunches may be modeled with additional superimposed layers of shell elements.

D.4 Requirements for Modeling Attached Steel Components

Connections between individual steel components shall transfer forces in accordance with the behavior of the connection. When calculated capacities of the connected members are lower than the calculated capacity of the connection, it shall be sufficient to model the attachment through appropriate kinematic constraints. In the cases in which the connection capacity is lower than member capacity, the connection capacity shall be considered either by reducing member capacity or by explicitly modeling connection failure. When connection plates exist and they increase the flexibility of the connection, their effect shall be considered. Eccentricity that may exist due to the configuration of the connection shall also be considered, such as when only one leg of an angle is connected to another component.

D.5 Requirements for Modeling Interactions between Slab and Steelwork

Normal and tangential behavior of the contact interaction between slab and structural steel shall be considered in the analysis. Any contact enforcement method and contact algorithm may be used by the analyst provided that:

a. Elements in contact shall be allowed to separate and/or slip.
b. The normal behavior follows a hard pressure-overclosure relation. This means that elements in contact are not allowed to penetrate each other (although negligible penalty penetrations are acceptable), and that the contact pressure is only limited by the bearing capacity of the elements in contact. If the elements are not in contact, the contact pressure shall be zero.
c. The tangential behavior shall be Coulomb's friction (although negligible penalty penetrations are acceptable). The coefficient of friction shall be 0.55 with a maximum interfacial shear stress of 0.06 ksi.

When shear studs exist between the slab and the steelwork, both the axial and shear behavior for the shear studs shall be modeled. A set of modeling recommendations can be found in Appendix A.

D.6 Requirements for Modeling Substructure Flexibility

At the locations in which the superstructure transverse and/or longitudinal displacements are constrained by the substructure, the flexibility of the structure must be considered as well as the strength of the support or bearing. It is not necessary to model the substructure in detail, but, at least, a linear elastic relation between horizontal reaction forces and horizontal displacements shall be applied at the support point. It shall be noted that transverse and longitudinal force-displacement relations are coupled if the superstructure is skewed with respect to the substructure. When calculating the stiffness of the substructure, the loads due to self-weight of the superstructure shall be considered to account for the effects on stability and load stiffening.

The flexibility of the substructure in the vertical direction may be neglected (the substructure may be assumed to be rigid in the vertical direction). Uplift of the superstructure should be allowed if the connection between superstructure and substructure does not provide resistance against uplift.

D.7 Required Analysis Procedure

Two load combinations below must be evaluated to ensure that a bridge has sufficient capacity after the failure of a main tension component. At a minimum the following load combinations shall be utilized in for the redundancy evaluation of bridges with members designated as FCMs or SRMs:

- For bridges that satisfy the fabrication requirements of Section 12 of the AWS D1.5:

$$Redundancy\ I = (1 + DA_R)(1.05\ DC + 1.05\ DW + 0.85\ LL)$$
$$Redundancy\ II = 1.05\ DC + 1.05\ DW + 1.30\ (LL + IM)$$

- For bridges the rest of bridges:

$$Redundancy\ I = (1 + DA_R)(1.15\ DC + 1.25\ DW + 1.00\ LL)$$
$$Redundancy\ II = 1.15\ DC + 1.25\ DW + 1.50\ (LL + IM)$$

The live load, LL, in the previous load combinations is based on the HL-93 vehicular live load model applied as described in. The dynamic amplification factor, DA_R, shall be take as 0.40, unless a different value is obtained by detailed non-linear dynamic FEA, or if the bridge is a continuous twin tub bridge with individual spans less than 225 ft, in which case DA_R may be taken as 0.20. The dynamic load allowance,

IM, is 0.15 of the truck component of the HL-93 vehicular live load. Details regarding the application and positioning of loads are described in Article 3.2 of the proposed guide specification in Appendix E.

The analysis procedure shall be static or quasi-static. If a quasi-static analysis procedure is utilized, experience has shown that the kinetic energy of the system shall not be greater than 5% of the strain energy of the system. The required analysis procedure for Redundancy I and Redundancy II load combinations shall be follow the following four steps:

1. Application of factored dead loads. This step must follow these requirements:
    - Dead loads shall be applied as body forces.
    - The slab shall not contribute to the stiffness of the system and shall not carry any significant portion of the dead load.
    - The slab deformation shall conform to the deformation of the steelwork.
2. The stiffness of the slab elements shall be changed to their final values assuming the concrete is fully cured. This step must follow this requirements:
    - The slab must retain the deformed shape computed in step 1 and not carry any significant portion of dead load.
    - The steelwork must retain the stresses and deformations computed in step 1.
    - Any modifications regarding the contact interaction between the slab and the structural steel members are applied in this step. E.g., if initially the slab was constrained to the structural steel members, that constraint shall now be replace by a frictional contact interaction.
3. The appropriate section of the main member is fractured and the system is allowed to redistribute the factored dead load. The amount of material that is removed or softened shall match that portion which would actually fail as closely as possible. In other words, if a crack is simulated, the width of the member that shall be deleted or softened shall correspond to a very narrow width, say ½ inch. Removal of a large portion of the member in this case is not acceptable. Removal can be accomplished by gradually softening the behavior of the elements that form the failing section. It is noted that at this point the slab does contribute to the stiffness of the system and is able to carry load.
4. Application of factored live loads as surface tractions. For the Redundancy I load combination, either the applied dead and live loads or the total stresses and forces obtained at the end of the analysis shall be amplified by a dynamic amplification factor (DA_R). For the Redundancy II load combination, either the applied live loads or the increase in stresses and forces due to the application of live load shall be amplified by a vehicular dynamic allowance (IM).
5. Application of an additional 15% of the applied factored live load.