APPENDIX F

PROPOSED GUIDE SPECIFICATION APPLICATION EXAMPLES

F.1 Introduction

Based on the work performed to characterize the redundancy of the bridges studied, a set of requirements was developed so an Engineer can establish whether a steel bridge possesses an adequate level of redundancy after the failure of a main tension member. 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 (FEA) is considered as the most suitable analysis tool.

The FEA methodology developed in NCHRP Project 12-87a is applicable to typical steel bridges that contained members designated as fracture critical members (FCMs): simple span and continuous I-girder and tub-girder bridges, through-girder bridges, truss bridges, and tied-arch bridges. The methodology has not been thoroughly benchmarked for non-typical steel bridges, i.e., cable stayed bridges, suspension bridges, etc. However, the overall methodology discussed hereafter may be used to evaluate non-typical bridges at the discretion of the Owner and/or Engineer. For a bridge superstructure which has one or more members that may be considered as fracture critical members (FCMs), the redundancy analysis consists of the following required steps:

1. Selection of an adequate finite element analysis tool as described in section F.1.1.
2. Construction of a three-dimensional finite element model capable of simultaneously capturing material and geometrical non-linearity and the formation of alternative load paths. The construction of these model requires the analysist to:
    - Construct the geometry, meshes and of the different components of the superstructure steelwork as described in section F.1.2.
    - Construct the geometry, meshes and of the concrete slab and concrete barriers as described in section F.1.3.
    - Model the connections between the steelwork components, including connection failure and/or flexibility when required. This requirement is described in section F.1.4.
    - Model the frictional contact interaction between the slab and the steelwork, and model shear stud behavior when required, as described in section F.1.5.
    - Include the effect of the flexibility of the substructure as described in section F.1.6.
3. Identification of steel members that are subjected to net tension across its entire or a portion of its cross-section, and which failure is suspected to result in collapse or loss of serviceability. At the very least, the Engineer must include that least the following, considering one complete member failure, i.e., the entire cross-section of the member is failed,, at a time:
    - In girder bridges (I-girder, tub-girder, wide flange girder, and through-girder bridges), at least, the following member failures shall be investigated:

- In continuous spans of girder bridges, member failure shall be assumed in both an end span and at least one interior span at the most critical location in the positive moment region of each span.
- In simple-span girder bridges, member failure shall be assumed at the most critical location for positive moment within the span.
- In continuous I-girder bridges, in regions with high shear and negative moment, e.g., interior supports, member failure shall be assumed at the most critical location.

- In truss bridges, at least, the following member failures shall be investigated: In simple-span truss bridges, member failure shall be assumed in at least one tension shear diagonal and one tension chord.
  - In continuous spans of truss bridges, member failure shall be assumed in at least one tension chord in the positive and negative moment regions in both an end and an interior span.
  - In continuous spans of truss bridges, member failure shall be assumed in at least one shear diagonal in the positive and negative moment regions in both an end and an interior span.
  - In multi-span truss bridges, where an interior span is to be considered, the member failure scenarios shall be considered for the longest interior span.
  - In all truss bridges, member failure shall be assumed in a single truss hanger.
- In tied-arch bridges, at least, the following member failures shall be investigated:
  - In tied-arch bridges, member failure shall be assumed in the tension tie at a critical location within the span. This location may be at midspan or at a location where the deck and stringers are discontinuous, such as at a deck joint at some location near mid span.
  - In tied-arch bridges, member failure shall be assumed in the tension tie near the intersection with the arch.
- Regarding floor beam systems in girder bridges, in single interior floor beams, member failure shall be assumed at mid span of the floor beam. Floor beams located where stringers are not continuous and/or where deck joint are present shall be investigated.
- Regarding cross-girders, also known as integral pier caps, steel bent caps, etc., the failure of a cross-girder must be assumed at mid span and near the support or column.

4. Analysis of the failure of each steel tension member in the constructed finite element model. In the analysis, the structure must be subjected to load conditions representative of two scenarios: (1) the occurrence of the member failure (Redundancy I), and (2) an extended period of service between the failure event and the structure being repaired or replaced (Redundancy II). The required analysis procedures are described in section F.1.8.
5. Comparison of the results of each individual analysis with a set of minimum performance requirements. This includes satisfaction of strength and serviceability requirements as described in section F.1.9.

These analysis steps are summarized in the flowchart shown in Figure F-1. The requirements are further explained in section F.1.1 through section F.1.9.

Additionally, the following examples of the application of the analysis methodology can be found for further clarification of the analysis methodology:

a. A continuous two span curved twin tub girder bridge is analyzed in section F.2.
b. A simple span through-girder is analyzed in section F.3.
c. A simple span tied-arch bridged is analyzed in section F.4.
d. A continuous three span plate girder bridge with a three girder cross section is analyzed in section F.5.

In these examples the behavior of the bridge after the failure of a primary steel tension member (that may be designated as a fracture critical member) is evaluated. If the faulted structure is able to meet the requirements described in the proposed guide specification in Appendix E, the member which failure is assumed may be re-designated as a system redundant member (SRM); otherwise, such member shall be designated as a fracture critical member (FCM).

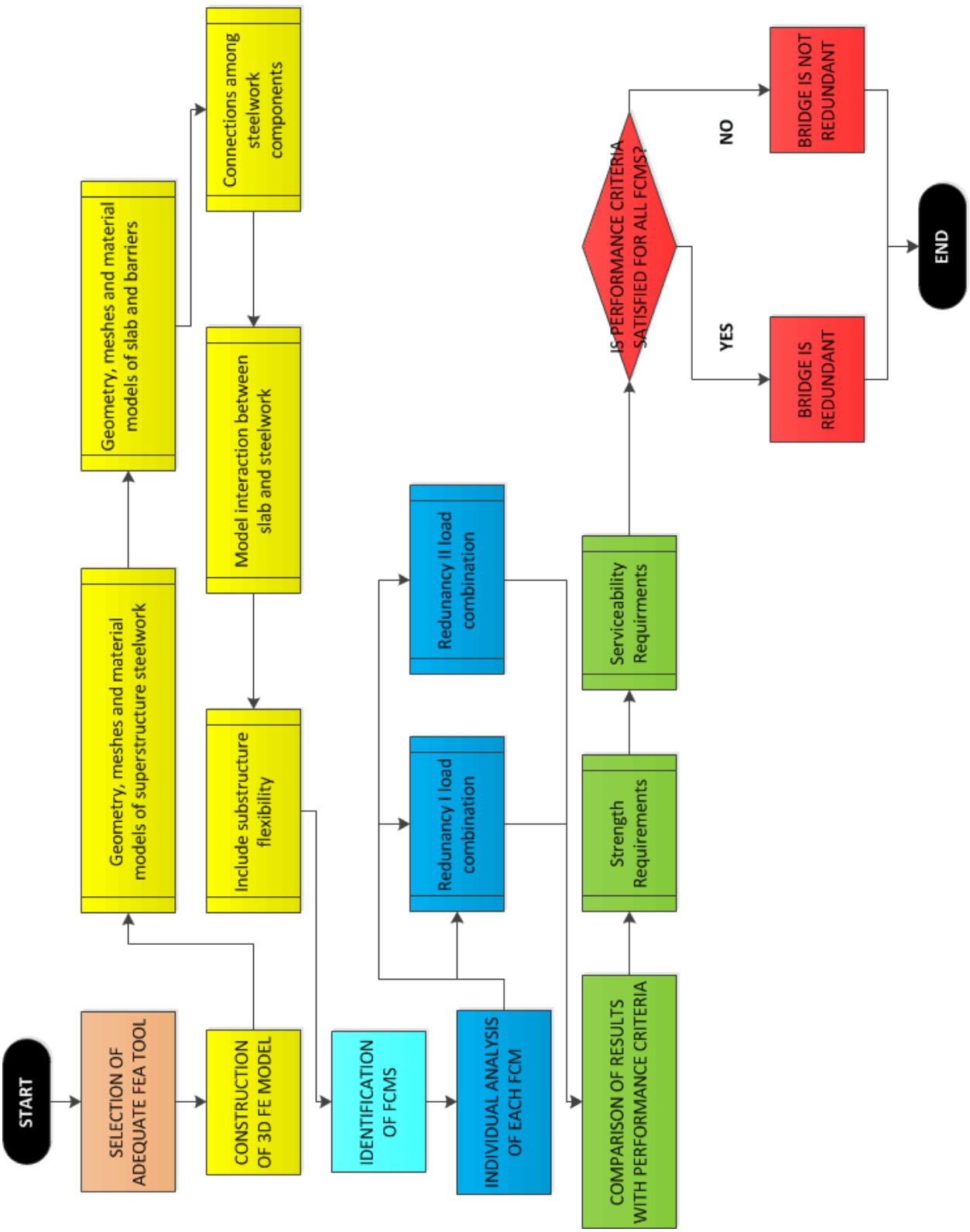


Figure F-1. Flowchart showing overall system analysis procedure.

F.1.1 Analysis Software and Solution Procedure Requirements

Although the methodology that has been developed was implemented using Abaqus, the Engineer 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, or, in other words, large deformation theory with finite strains and finite rotations is considered.
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 Engineer 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 subjected to its own factored dead load in the undamaged state and then subjected to failure of a primary member. Factored live loads and dynamic load allowance, and amplification of load per the dynamic amplification factor are applied after the failure of a primary steel tension member is modeled.
g. Material non-linearity, particularly plasticity 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.
    - Coupling: The motion of a slave surface or node group is constrained to the motion of a master node.
i. The software is capable of modeling the contact interaction between the slab and the steelwork. This includes normal contact and frictional behavior.
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. 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.

F.1.2 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.

Depending on the type of member modeled, the following elements and meshing procedures shall be followed:

a. The following steel members shall be modeled with 4-node shells with reduced integration and finite membrane strains (ideally with hourglass control):
    - 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.

    At least four elements must be used in the along the component width, flange width and/or along the web height must be used. The element maximum aspect ratio shall be kept under 5, and corner angles kept between 60 and 120 degrees.
b. Other members shall be modeled with 2-node linear shear-flexible (Timoshenko) beam elements. At least three elements must be used along the length of the element. Examples of other steel members are:
    - Lateral bracing.
    - Truss floor beams.
    - Other secondary slender elements not typically designed to carry primary loads.

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.

F.1.3 Requirements for Modeling the Behavior of Concrete Slabs

The geometry of the concrete slab shall be modeled per one or a combination of the following approaches:

a. With 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.
    - 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 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.
- Concrete barriers and their reinforcement may be included as part of the slab system.

b. With shell elements that implicitly consider the effect of the steel reinforcement layers:

- 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.

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') in ksi. Initially, the material shall be linear elastic, with modulus of elasticity as follows:

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

where w_c is the density of concrete in kcf, and Poisson's ratio of 0.3.

Concrete inelasticity shall be different for tension and compression. The tension behavior is based on the provisions in the fib Model Code for Concrete Structures (2010). In tension, the behavior is linear elastic until tensile the strength of concrete, f_t, defined as follows:

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

is reached. At that point failure shall occur with a fracture energy, G_t (in ksi-in), defined 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:

$$n = 0.4 f_c' + 1.0$$

$$\varepsilon_c = 0.00124 \sqrt[4]{f_c'}$$

$$\varepsilon_{plastic} = \varepsilon - \frac{f(\varepsilon)}{E_c}$$

F.1.4 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 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. The capacity of the connection shall be computed in accordance with the provisions in the AASHTO LRFD BRIDGE Design Specifications (2014) (AASHTO LRFD BDS), which nominal (unfactored) values are to be input in the finite element model as needed. For bolted connections, it is recommended to consult the provisions in Eurocode 3 (CEN, 2007) and/or Henriques et al. (2014) to calculate the stiffness the connection assembly, and the work of Sarraj (2007) to determine the maximum displacement at failure.

F.1.5 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 Engineer 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. 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. The prescribed force-displacement behavior shall be calculated in accordance with the governing failure mode of the shear stud group. E.g., the shear force-displacement curve shall capture the shear stud yielding or concrete crushing; the axial force-displacement curve shall capture the concrete breakout or split out. Shear stud modeling recommendations are detailed in Appendix A.

F.1.6 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 relations 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 either the superstructure or the substructure are asymmetric, or if the bridge is skewed. 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.

F.1.7 Required Minimum Loads in Analysis

Two load combinations, Redundancy I and Redundancy II, must be evaluated to ensure that a bridge has sufficient capacity after the failure of a main tension component. The appropriate load factors depend upon whether the steel bridge under analysis is constructed in accordance to the Fracture Control Plan (FCP) or not. For bridges constructed to Section 12 of the AWS D1.5 (FCP), the load combinations are as follows:

$$Redundancy\ I\ (FCP): (1 + DA_R)(1.05DC + 1.05DW + 0.85LL)$$

$$Redundancy\ II\ (FCP): 1.05DC + 1.05DW + 1.30(LL + IM)$$

For bridges that do not meet FCP requirements, the load combinations are as follows:

$$Redundancy\ I\ (Non - FCP): (1 + DA_R)(1.15DC + 1.25DW + 1.00LL)$$

$$Redundancy\ II\ (Non - FCP): 1.15DC + 1.25DW + 1.50(LL + IM)$$

In these combinations DC is the dead load of structural components and nonstructural attachments, DW is the dead load of wearing surfaces and utilities, LL is vehicular live load, DA_R is the dynamic amplification during the fracture event, and IM is vehicular dynamic load allowance.

The vehicular live load (LL) applied in the Redundancy I and Redundancy II load combinations is the HL-93 live load model. This load model is composed of the design truck (or tandem) and a lane load of 0.64 klf distributed over a 10 feet width.

The application of the vehicular live load is different for Redundancy I and Redundancy II load combinations:

a. For Redundancy I:
   - Only the striped or normal travel lane(s) shall be considered.
   - The design truck or design tandem and the design lane load of the HL-93 vehicular live load model shall be centered within the striped or normal travel lane(s).

b. For Redundancy II:
   - The number of lanes to be considered shall be taken as specified in Article 3.6.1.1.1 in the AASHTO LRFD BDS.
   - The design lanes shall be positioned transversely to produce the largest demands on the remaining intact components of the bridge.
   - The HL-93 live load model shall be transversely place within the design lanes to produce the largest demands on the remaining components of the structure.
   - The design truck or design tandem shall be positioned transversely such that the center of any wheel load is not closer than 2.0 ft from the edge of the design lane.

The longitudinal positioning of live load for the redundancy evaluation of longitudinal primary members shall be as follows:

- Where the failure section is in a region of positive moment under dead load, the design tandem or centroid of the design truck of the HL-93 vehicular live load model shall be positioned longitudinally coincident with the location of the assumed damage in the faulted member.
- When the failure section is in a region of negative moment under dead load, the HL-93 vehicular live load model shall be applied as described in the third bullet of Article 3.6.1.3.1 in the AASHTO LRFD BDS.

Multiple presence factors shall be applied as specified in Article 3.6.1.1.2 in the AASHTO LRFD BDS in both, Redundancy I and Redundancy II, load combinations.

For the Redundancy I load combination, the applied dead and live loads shall be amplified by a fracture amplification factor, DA_R. The dynamic amplification during the fracture event must be 40% of the combined and factored DC, DW and LL, unless the structure is a continuous twin tub girder with individual span shorter than 225 ft where it must be 20% of the combined and factored DC, DW and LL. Other values may be used when backed by the comprehensive and detailed dynamic analysis.

For the Redundancy II load combination, the vehicular live loads shall be amplified by a vehicular dynamic load allowance, IM. The vehicular dynamic load allowance must be 15% of the factored design truck or design tandem portion of the HL-93 vehicular live load.

The HL-93 live load model, subjected to the appropriate load factors, multiple presence factors, DA_R and/or IM shall be applied to minimize any possible inertial effects. It is recommended to apply them through a smooth amplitude curve such as the following:

$$LR(t) = 6\left(\frac{t}{T}\right)^5 - 15\left(\frac{t}{T}\right)^4 + 10\left(\frac{t}{T}\right)^3$$

where $LR(t)$ is the fraction of load at a load application time t, and T is the duration of the load application. The duration of the load application must be larger than the fundamental period of the structure to minimize oscillatory behavior in the final explicit dynamic analysis.

F.1.8 Required Analysis Procedure

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 load of structural components and nonstructural attachments, DC. 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.
3. Application of factored dead load of wearing surfaces and utilities, DW. The load effect of the pavement may be modeled by specifying a layer of relatively soft solid or shell elements (Engineer should refer to SHRP-A-388 as asphalt stiffness varies very significantly with temperature) and apply the correspondent body force.
4. 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. 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.

5. Application of the dynamic amplification factor (DA_R) to DC and DW. This step is only carried out in the Redundancy I load combination.
6. Application of factored and amplified vehicular live loads, LL. It shall be noted that LL is subjected to loads factors and multiple presence factors, as well as dynamic amplification (DA_R) in the Redundancy I load combination or dynamic load allowance (IM) in the Redundancy II load combination. These shall be applied as surface tractions normal to the slab surface. Details regarding the application and positioning of the live loads are described in section F.1.7.
7. Application of an additional 15% of live loads. This is not to be confused with the 15% related to vehicular dynamic load allowance.

F.1.9 Minimum Required Performance in the Faulted State

In order to establish whether the primary steel tension member which failure is introduced in the analysis is a fracture critical members (FCM) or a system redundant member (SRM) it is necessary to evaluate the faulted structure subjected to the loads described for the Redundancy I and Redundancy II load combinations. If the system meets all of the requirements described in Section F.1.9.1 and Section F.1.9.2, the member which failure is introduced in the analysis may be re-designated as a SRM, otherwise it shall remain as a FCM. It shall be noted that the system must previously be subjected to a screening criteria, as described in Appendix C and in the proposed guide specification in Appendix E.

F.1.9.1 Strength Criteria

All of the strength requirements described hereof are to be checked at the end of step 6 in the analysis procedure described in Section F.1.8 for both load combinations, Redundancy I and Redundancy II, unless otherwise noted. A set of requirements apply to primary members of the superstructure, these are:

- In a component, such as a web or a flange of a primary steel member, the average strain is less than five times the material yield strain.
- In a component, such as a web or a flange of a primary steel member, the average strain is less than 0.01.
- The maximum strain anywhere in a primary steel tension member is less than of 0.05 is reached. Higher strain limits are permitted when supported by experimental testing.
- The combined flexural, torsional and axial force effects computed in primary compression members is below the nominal compressive resistance of the member, unless these limit states are predicted by the FEA.
- Although localized crushing in the slab is allowed, the slab shall not reach 0.003 compression strain in a portion sufficiently large to compromise the overall system load carrying capacity.
- The system fails shall support, i.e., satisfy static equilibrium, an additional 15% of the factored live load at the end of step 8.

Additionally, the substructure must meet the following requirements:

- At any support location, the reaction forces and moments is less than the nominal resistance of a substructure element or the support system.
- The substructure can safely accommodate the displacements and reactions of the superstructure in the faulted state.

F.1.9.2 Serviceability Criteria

The serviceability requirements described hereof are to be checked at the end of step 4 in the analysis procedure described in Section F.1.8 for the Redundancy II load combination only, these are the following:

- The maximum vertical deflection is less than L/50, where L is the span length for primary members oriented longitudinally.
- When considering a scenario in which failure of a floor beam is assumed, the maximum vertical deflection of the floor beam is larger than L/50, where L is the distance between floor beams that are assumed not to have failed.

Additionally, the Owner and/or the Engineer may considered other serviceability related parameters such as uplift at slab joints, changes in the cross-slope of the structure, and other phenomena that negatively impacts the ability of the structure to provide service in a safe manner.

F.2 Continuous Curved Twin Tub Girder Bridge

The redundancy of a continuous two span twin tub girder bridge is analyzed by developing a finite element model in accordance with the methodology described in the proposed guide specification in Appendix E. It is assumed that the structure does not possess any of the detrimental attributes described in the screening criteria and that it is built to Section 12 of the AWS D1.5. In this case, the failing tension member is assumed to be the exterior tub girder. The entire cross-section of the exterior girder is assumed to have failed at a cross section located 50'-8" north of the continuous support (pier) as shown in Figure F-2.

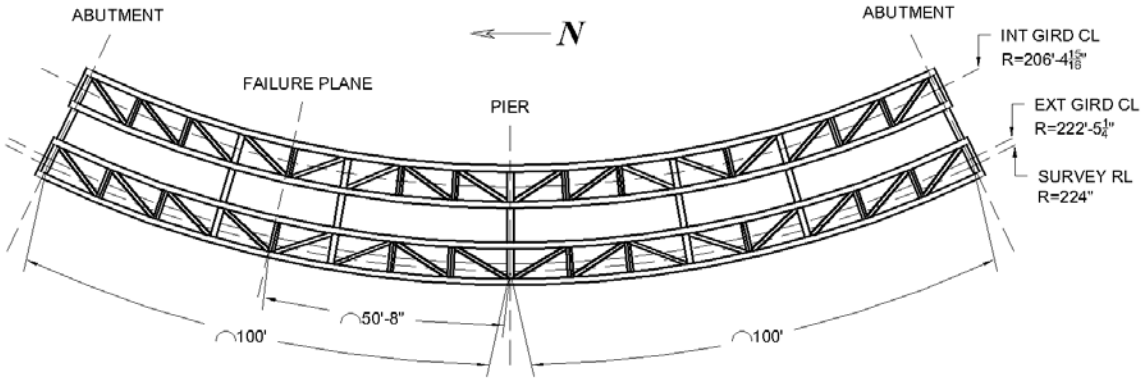


Figure F-2. Steelwork geometry and failure location.

The structure has two spans measuring 100 feet long, and it is uniformly curved with a radius of 224 feet (measured along the surveying reference line). The two trapezoidal box girders have 63 inch wide bottom flanges, 61.875 inch high webs and 16 inch wide top flanges; with variable plate thicknesses. Stability of the girders is provided by seven diaphragms joining both girders and a system of K-frames, struts and braces within each girder. Figure F-3 shows the steelwork framing plan and Figure F-5 provides details of the different diaphragms an internal bracing members.

The reinforced concrete slab is approximately 27 feet wide between interior edges of concrete barriers (approximately 30 feet wide between the outer exterior edges of concrete barriers) and is fully composite with the girders' top flanges through shear studs. The end supports are multi-rotational unidirectional bearings, and the support over the pier is a multi-rotational fixed bearing. All steel plates, used in the girders and diaphragms are made of ASTM A709 HPS 50W. All rolled sections, used in interior stability system are made of ASTM A709 Grade 50. All concrete has a minimum specified compressive strength of 4ksi and all rebar has 60 ksi yield strength. In the analysis of this structure, longitudinal and transverse slopes, as well as camber adjustments will me neglected. Figure F-4 shows the cross section of the structure with the slab, barrier and reinforcement details.

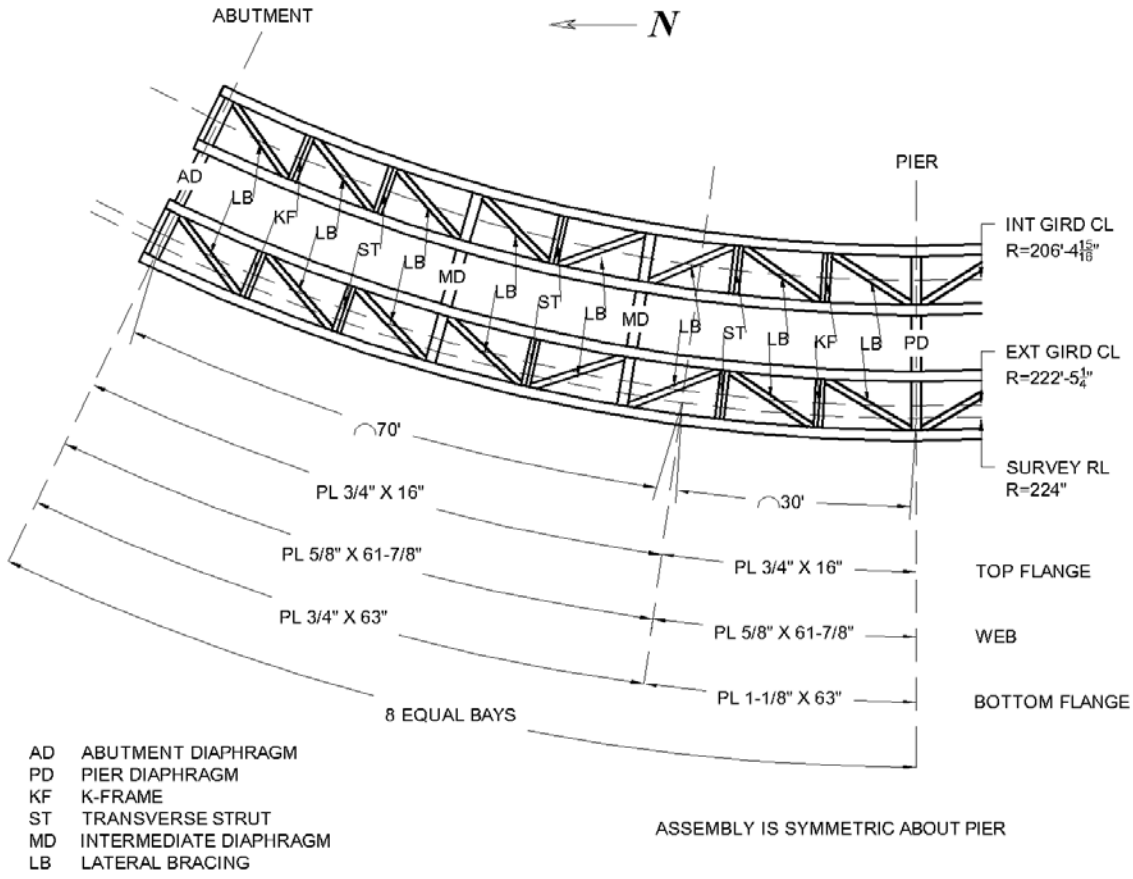


Figure F-3. Girders, diaphragms and internal bracing plan.

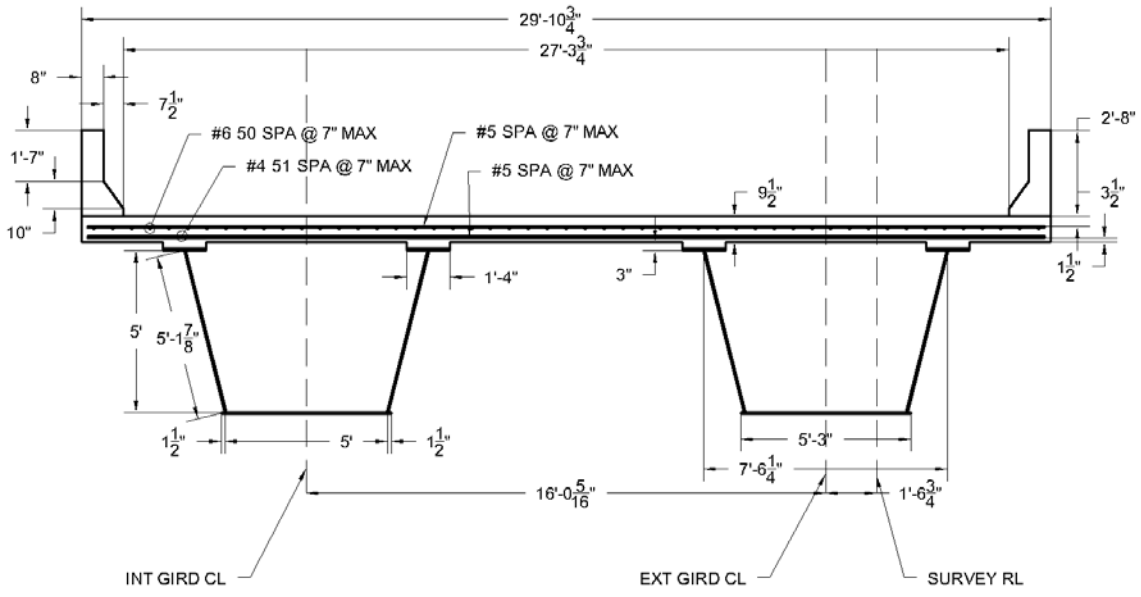


Figure F-4. Typical cross-section and slab reinforcement details.

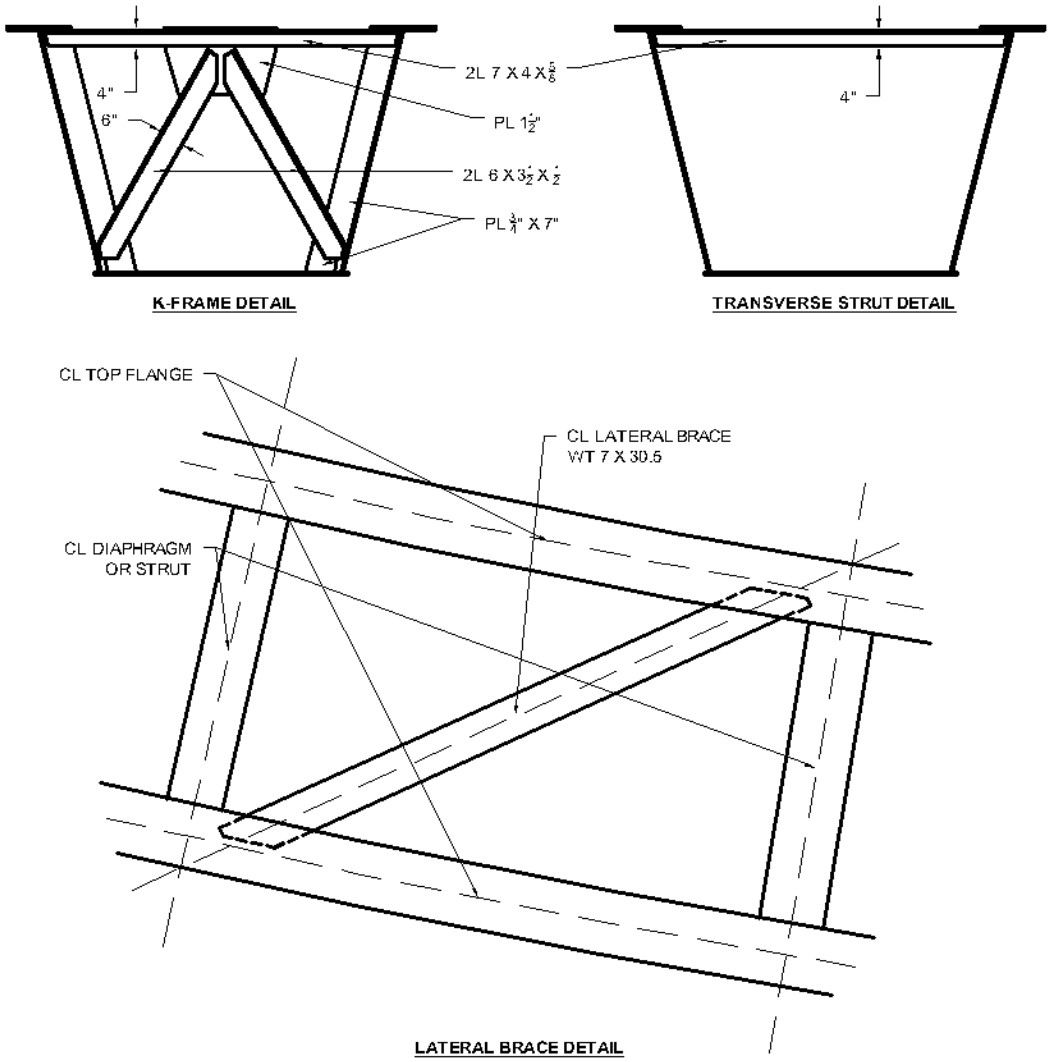


Figure F-5. Internal bracing and diaphragm details.

F.2.1 Analysis Procedure

The analysis is performed to establish if the system demonstrates acceptable performance in the faulted condition. In the example, the term "faulted condition" specifically refers to the case in which a primary steel tension member is assumed to have failed. For this analysis, load factors for both dead and live load are applied as described in the proposed guide specification in Appendix E. In this example, the described analysis procedure is composed of an initial implicit static analysis and a final explicit dynamic analysis, into which the results from the initial implicit static analysis are imported. While it is not mandatory for the Engineer to follow these particular steps, it has been found that this procedure optimizes the computational time required.

F.2.1.1 Initial Implicit Static Analysis

Implicit static analysis was utilized to calculate the state of the structure prior to hardening of the concrete in the slab. An implicit static analysis was used for the initial steps because, although non-linearity is considered in the analysis, the bridge behavior is linear and inertial effects can be neglected as the bridge is in the undamaged condition. As the slab does not carry any load and does not contribute to the stiffness of the system before concrete hardening, two modifications are required in the finite element analysis during this initial implicit static analysis as follows:

- A very low stiffness is specified for the elements composing the slab, i.e., the elements modeling concrete and rebar. A reduced stiffness of 1/1,000 of the respective modulus of elasticity of each material was used. This is done so the load carried by the slab and rebar have negligible contribution to the stiffness of the system. No modifications to the stiffness should be applied to the steelwork.
- Instead of defining contact interaction between the slab and the steelwork, a mesh tie was specified. The nodal displacements of the concrete slab elements are tied to the displacements of the top flanges of girders, floor beams, and stringers which occur due to dead load. As a result, the slab deforms with the steelwork and does not 'sag' between the girders, floor beams, and stringers.

It is worth noting that the remainder of the finite element modeling is identical between the initial implicit static analysis and the final explicit dynamic analysis. The specific steps in the initial implicit static analysis are described as follows:

1. Apply load due to self-weight of the structural steel components as a body force.
2. Apply load due to self-weight of the wet slab components as a body force.
3. The system is then fixed in terms of position, that is, the displacement degrees of freedom are not allowed to change.
4. The elements composing the slab (elements modeling rebar and concrete) are then deactivated.
5. The elements composing the slab are then reactivated. During this reactivation the strain in the elements composing the slab is reset to zero.

Steps 3 through 5 are necessary since even though very low stiffness was specified for the slab, these elements do undergo strain. Setting the strains to zero eliminates "locked in" artificial stresses in later steps.

F.2.1.2 Final Explicit Dynamic Analysis

As contact algorithms, softening material behavior, and non-linear geometry are required to be part of the finite element analysis, implicit solution procedures present unavoidable convergence problems in most FEA solvers. In order to calculate the capacity of the bridge after sudden failure of a tension component, a dynamic explicit analysis needs to be carried out. Therefore, the results obtained from the initial implicit static analysis are imported into the final explicit dynamic analysis. In other words, the state of the system (stresses, strains, displacements and forces) at the beginning of the final explicit dynamic analysis is defined by the state of the system computed at the end of the initial implicit static analysis.

As previously stated, during the initial implicit static analysis, the slab was modeled with largely reduced stiffness to reflect that it is not hardened and a mesh tie constraint was used to assure that the slab deformed with the steelwork. This approach also prevents excessive sag of the soft slab. After the state of the system is imported, the following changes are made to capture the response of the structure after the concrete has hardened:

- The modulus of elasticity of the concrete and rebar elements in the slab is changed to their final actual values. It is noted that no modifications need to be applied for the steelwork.
- The mesh tie constraint between the slab concrete elements and the top flanges of the steelwork is replaced by a frictional contact interaction. Additionally, since the structure under analysis is composite, elements which accurately model the behavior of shear studs are added.

All of the body forces applied during the initial implicit static analysis (i.e., the dead load of the structure) are maintained throughout the final explicit dynamic analysis.

To evaluate the capacity of the structure in the faulted state, the following steps were carried out in the final explicit dynamic analysis:

6. The stiffness of the elements located at the fracture location under consideration were slowly reduced. The stiffness was slowly reduced in order to minimize any dynamic effects. It is noted that the actual fracture and subsequent vibration of the structure is not modeled. This dynamic effect is accounted for using the DA_R factor as discussed before. If dynamic effects are found to be significant even if the stiffness is slowly reduced, the system must be allowed to oscillate until these effects are dampened.
7. Factored loads due to traffic are applied as surface tractions. For the Redundancy I load combination all loads are amplified by DA_R, for the Redundancy II load combination the dynamic load allowance (IM) is applied. These loads were applied very slowly to minimize any dynamic effects, as well. If dynamic effects are significant, the system must be allowed to oscillate until these effects are dampened.
8. An additional 15% of live load is gradually applied.

F.2.2 Material Models

Four material models are needed in the finite element model. Three of those are utilized to model different steel types, and one is utilized to model the response of concrete. For the development of the steel material models, it is necessary to know the yield strength and ultimate strength of each steel type. In this example, since no test values are known to the Engineer, nominal values specified in the respective standards are utilized. These are summarized in Table F-1. A mass density of 0.494 kcf was specified for all steel types.

Table F-1. Material properties for steel material models.

Material	Nominal Yield Strength	Nominal Ultimate Strength	Standard
ASTM A709 HPS 50W	50 ksi	70 ksi	ASTM A709/A709M
ASTM A709 Gr. 50	50 ksi	65 ksi	ASTM A709/A709M
Grade 60 Rebar	60 ksi	90 ksi	ASTM A615/A615M

The stress-strain relation for all steel components will follow an initial linear elastic steel with a Young's modulus of 29,000 ksi and Poisson's ratio of 0.3. Once the nominal yield strength is reached the stress-strain relation is defined by Von Mises (J2) plasticity with kinematic linear hardening, until the nominal ultimate strength is reached at a total strain of 0.05. Once the nominal ultimate strength or a total strain of

0.05 is reached, the material is assumed to fail. Figure F-6 shows the uniaxial material response for the steel employed in this finite element model with the 'X' denoting the stress at the failure strain of 0.05.

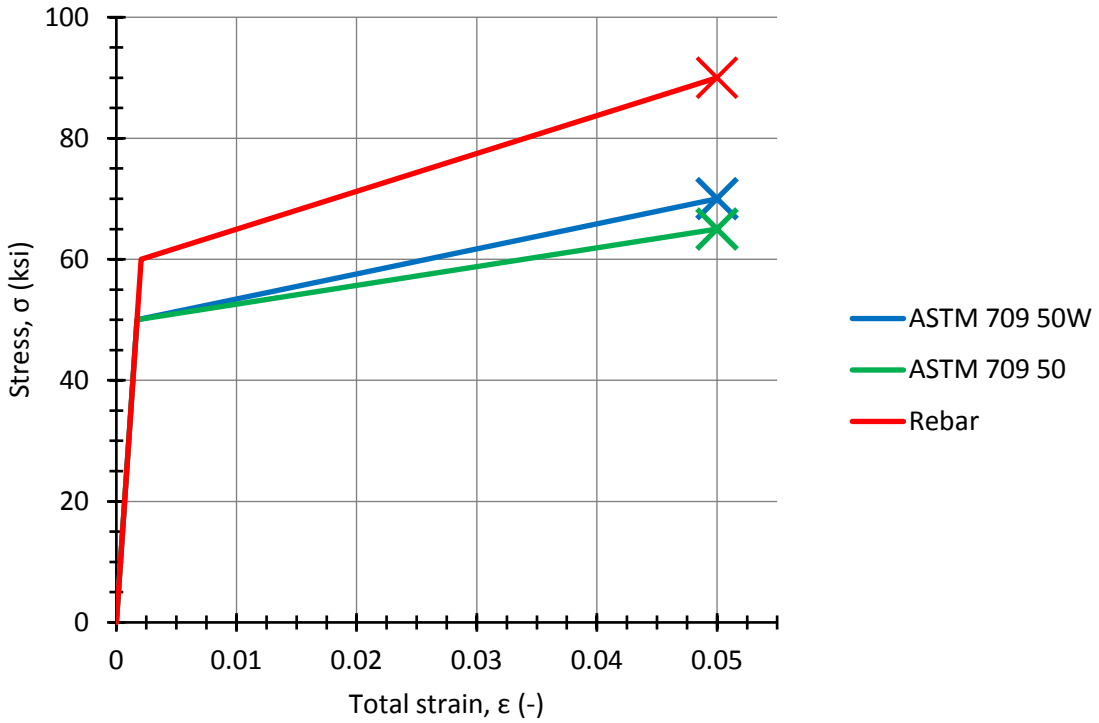


Figure F-6. Stress-strain curves of steel material models.

The material model used in concrete is defined entirely by the specified compressive strength, which in this case is 4 ksi. This quantity is also used to calculate the tensile strength, the total strain at compressive strength, ε_c, and the material parameter, n. Table F-2 summarizes the calculation of these values. A mass density of 0.150 kcf was specified for concrete.

Table F-2. Material properties for concrete material model.

Quantity	Symbol	Equation	Result
Young's modulus	E_c	$E_c = 33{,}000 w_c^{1.5}\sqrt{f_c'} \leq 1{,}802.5\sqrt{f_c'}$	3,600 ksi
Tensile strength	f_t	$f_t = 0.158(f_c')^{\frac{2}{3}}\ for\ f_c' \leq 7.25ksi$ $f_t = 0.307\ln(f_c' + 2.61) - 0.114\ for\ f_c' > 7.25ksi$	0.398 ksi
Fracture energy	G_t	$5.9 \cdot 10^{-4}(f_c' + 1.16)^{0.18}$	$7.93{\cdot}10^{-4}$ kip/in
Total strain at compressive strength	ε_c	$\varepsilon_c = 0.00124\sqrt[4]{f_c'}$	0.00175
Material parameter	n	$n = 0.4f_c' + 1.0$	2.6

The concrete material model is initially linear elastic, defined by a Young's modulus of 3,600 ksi and Poisson's ratio of 0.2, followed by concrete damage plasticity. In tension, once the material reaches its tensile strength, set at 0.398 ksi in this case, a tensile stress-displacement relation characterized by a fracture energy, G_t, of $7.93 \cdot 10^{-4}$ kip-in is followed This fracture energy is applied through a bi-linear tensile stress-displacement relation as shown in Figure F-7, and defined by the following quantities:

$$f_{t1} = \frac{f_t}{5} = 0.0796\ ksi$$

$$\delta_t = \frac{5G_t}{f_t} = 0.00996$$

$$\delta_{t1} = \frac{G_t}{f_t} = 0.00199$$

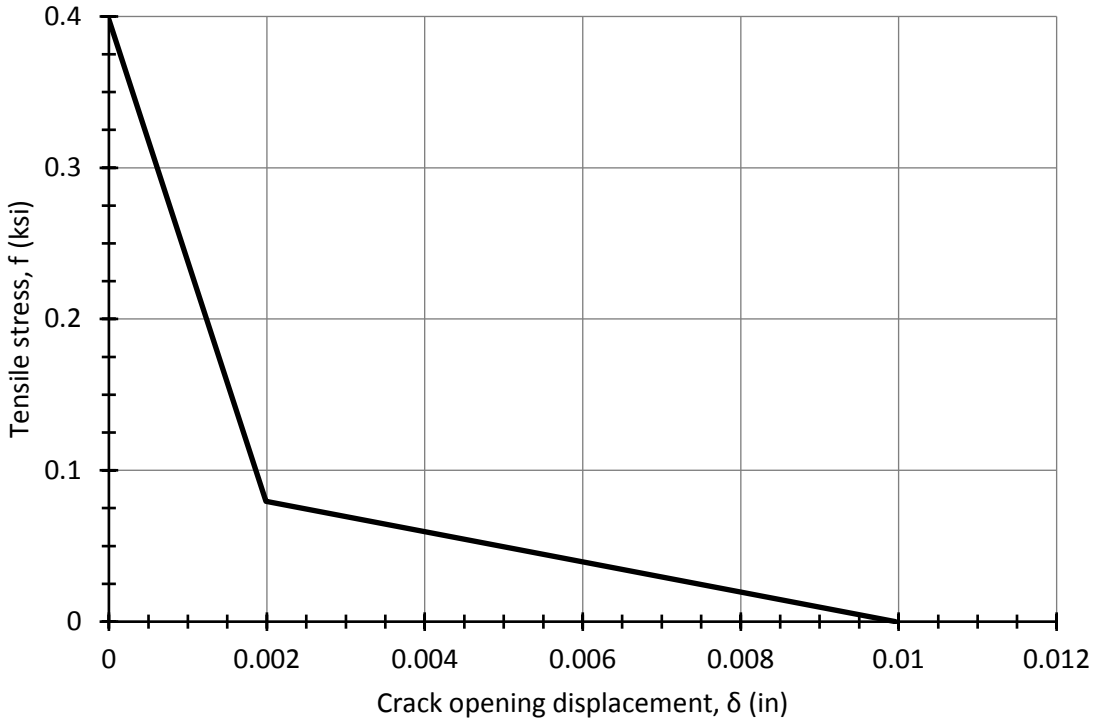


Figure F-7. Tensile stress-crack opening displacement curve for concrete material model.

In compression the material follows the following stress-strain relations:

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

$$\varepsilon_{plastic} = \varepsilon - \frac{f(\varepsilon)}{E_c}$$

Where ε is total (elastic + plastic) strain, $f(\varepsilon)$ is the compressive stress at a given total strain, f_c' is the specified compressive strength, ε_c is the total strain at compressive strength, n is a material parameter, $\varepsilon_{plastic}$ is the plastic strain, and E_c is the concrete Young's modulus. Figure F-8 shows the resulting compressive stress-strain relation.

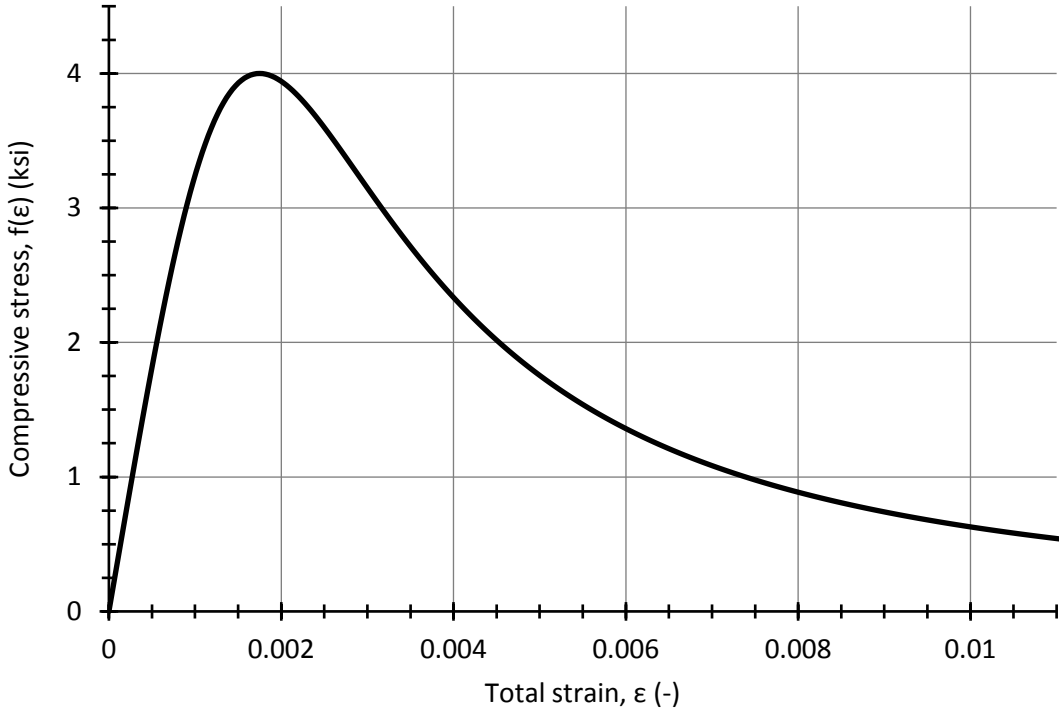


Figure F-8. Compressive stress-strain curve for concrete material model.

F.2.3 Geometries, Meshes and Constraints

The geometry of the structure is based on available design plans and is composed of the following components that must be explicitly modeled:

1. Two horizontally curved trapezoidal tub girders.
2. Seven diaphragms connecting both tub girders:
    a. Two abutment diaphragms.
    b. One pier diaphragm.
    c. Four intermediate diaphragms.
3. A system of internal K-frame within the tub girders (total of 4 frames per girder).
4. A system of transverse struts within the tub girders (total of 6 struts per girder).
5. A system of diagonal bracings within the tub girder (total of 16 braces per girder).
6. A reinforced concrete slab with concrete barriers.

When generating the finite element model, splices, holes, access hatches, etc. are neglected. The structure is assumed to be flat in the vertical plane, in other words, camber and superelevation are ignored. Figure F-9 shows the assembly of all bridge components.

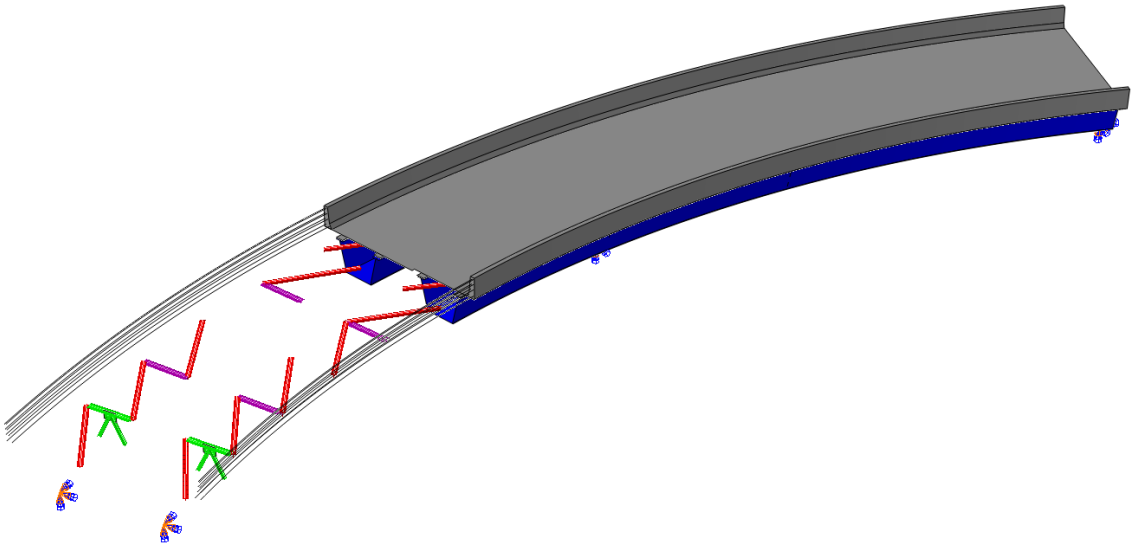


Figure F-9. Twin tub girder bridge geometries. Concrete slab and barriers (grey), tub girders (blue), K-frames (green), transverse struts (magenta), lateral bracing (red), slab reinforcement (black).

Tub girders and diaphragms are modeled with 4-node shell elements with reduced integration. A minimum of four elements are used along flange widths and along web heights. The stiffeners to which the K-frames are attached are modeled with shell elements as well. In this case, the tub girders, diaphragms and stiffeners constitute a single geometry. The maximum aspect ratio was kept below five and corner angles were kept between 60 and 120 degrees. Figure F-10 shows two details of the mesh employed to model the tub girder, diaphragm and stiffener system. The K-frames, transverse struts and diagonal bracings are modeled with 2-node linear shear-flexible (Timoshenko) beam elements. A minimum of three elements are used along the length of the elements. Mesh ties, which are constraints that slave the motion of a surface or node set to the motion of a master surface or node set, are utilized to connect the K-frames to the stiffeners and the struts and diagonals to the webs of the tub girders.

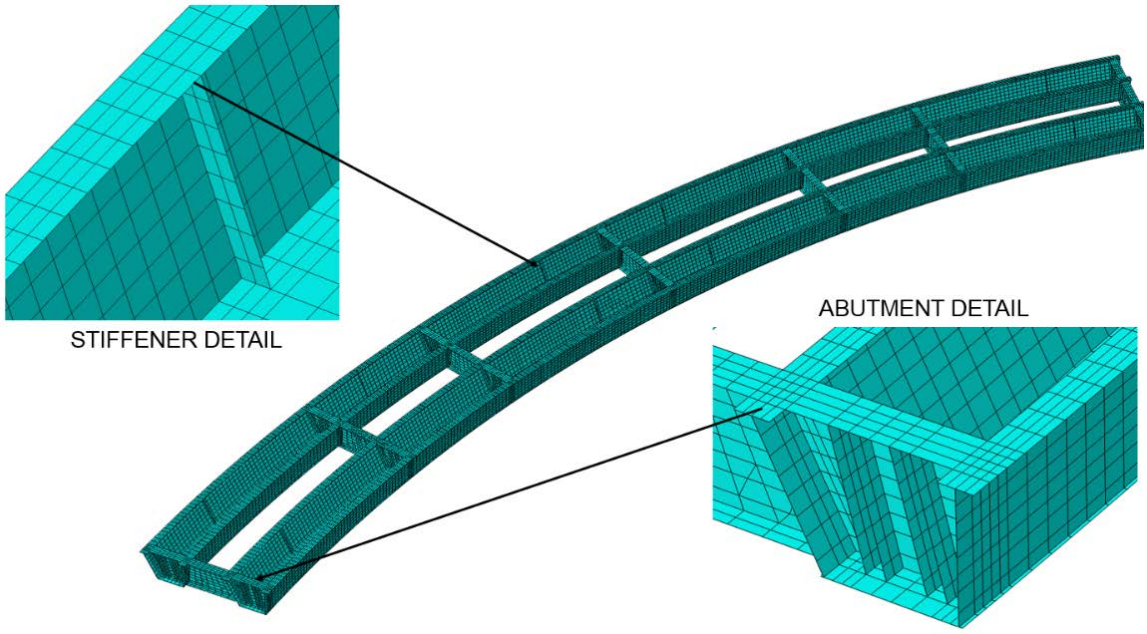


Figure F-10. Mesh details of the tub girders.

The slab is modeled with four-node linear shell elements with reduced integration, finite membrane strains, and a minimum of five Simpson thickness integration points. The transverse and lateral reinforcement in the concrete slab is implicitly included in the section integration of the shell elements, as rebar layers that follow the radial (for transverse reinforcement) and tangential (for longitudinal reinforcement) characteristic of the curvature of the slab.

The barriers are defined using eight-node linear bricks (hexahedral elements) with reduced integration are used to model concrete, and two-node truss (wire) elements with linear displacement to model steel reinforcement. Seven solid concrete elements are used through the height of the parapet with maximum aspect ratio (length of longest edge divided by length of shortest edge) of 5, and corner angles (angle at which two element edges meet) between 40 and 140 degrees. The length of the truss elements used to model the parapet reinforcement were approximately equal to the length of the longest edge of the solid concrete elements. These truss elements are embedded within the solid concrete elements. At the nodes of the embedded truss elements, the translational degrees of freedom are eliminated and the nodal translations were constrained to interpolated values of the nodal translations of the host solid concrete element. The barriers were attached to the slab by mesh ties. Figure F-11 shows a detail of the mesh used for the concrete barrier and slab.

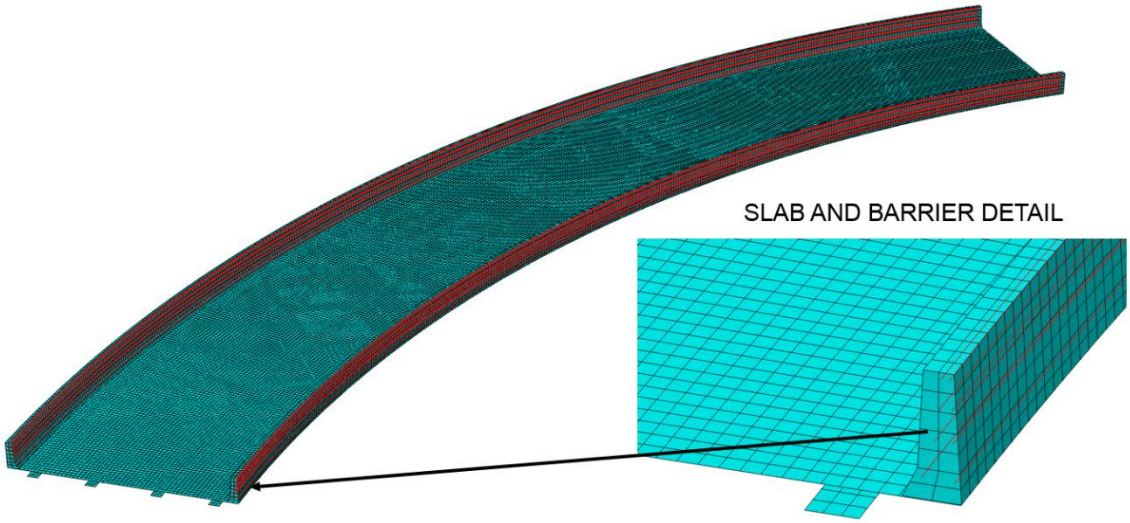


Figure F-11. Mesh details of the reinforced concrete slab and barriers.

F.2.4 Slab-Structural Steel Interaction

As stated, the interaction between the bottom of the concrete slab and the top of the flanges of the tub girders is modeled differently in the two steps described above. In the initial implicit static analysis, when the elements comprising the slab and barriers have 1/1,000th of the modulus of elasticity to model the "wet" condition, a mesh tie is used to slave the motion of the slab to the motion of the surface comprising the top of the steel work. With this procedure, it is ensured that the slab deformation will conform to the deformation of the steelwork while unrealistic sagging of the slab between supporting elements and tipping of the barrier is prevented.

In the final explicit dynamic analysis, when the stiffness of the elements comprising the slab and barriers has been changed to their final real values, the mesh tie previously used is deleted and replaced by a contact interaction and modeling of shear studs. The normal behavior of the contact interaction is modeled through a penalty stiffness. The penalty stiffness is several orders of magnitude larger than the normal stiffness of the underlying contacting elements and allows a very small penetration so a pressure can be calculated. The tangential behavior of the contact interaction is modeled through an algorithm based on Coulomb friction with a limit on the allowable shear. A friction coefficient of 0.55 and an interfacial shear strength of 0.06 ksi are specified.

The simplified stud model, as described in the Appendix A is used to model composite action between the slab and the steelwork. In the simplified stud model, the shear studs were modeled using connector elements which were used to define the axial and interfacial shear interaction between the shear studs and concrete slab. Connector elements are special purpose elements with zero length. These elements model discrete physical connections between deformable or rigid bodies, and are able to model linear or nonlinear force-displacement behavior in their unconstrained relative motion components.

The recommendations of Appendix A were used to define the shear and tensile behavior of shear studs. The shear stud group is composed of three transversely grouped studs which shear strength is 108 kips, following the shear force-displacement relation proposed by Ollgaard et al. (1971) up to maximum shear displacement of 0.2 inches. In tension, the governing failure mode is concrete break-out, resulting in a initial stiffness of 1632 kip/in, and tensile strength of 12.5 kips, and a maximum tensile displacement at failure of 0.049 inches. The tensile behavior follows the characteristic triangular response for transversely grouped shear studs which governing failure mode is concrete break-out or shear stud pullout, as described

in the recommendations of Appendix A. The tension-shear interaction procedure presented in Appendix A is used the combine the effects of shear and tension acting simultaneously on a shear stud group.

F.2.5 Connection Failure Modeling

When a connection is likely to fail before yielding of the member, in addition to the use of mesh ties to attach the components, an additional step may be necessary to capture connection failure. Although it is possible to develop force/moment-displacement/rotation relations which can be applied to a connector element, a simpler approach was developed and utilized herein. In this particular example, it was not necessary to include to model connection failure as the forces developed in the member did not exceed the capacity of the connections. However, fuse elements were included in the finite element model to model the stiffness, capacity, and ductility of the connection shown in Figure F-12. The behavior was modeled by a linear-elastic perfectly-plastic relationship defined by the following:

- An initial elastic stiffness which is defined as a series sum of the contributions due to the axial flexibility of the connection plate, the bearing stiffness of the connection plate, and the shear stiffness of the bolts. The calculation procedure is based on the provision in Eurocode 3 (CEN, 2007) and Henriques et al. (2014).
- The capacity of the connection is calculated per the provision in the AASHTO LRFD BDS. The nominal (unfactored) tensile capacity was specified.
- Once the capacity of the connection is reached, the fuse element behaves perfectly plastic until a maximum failure displacement is reached. This maximum failure displacement is the largest of 2.5 times the ratio of the capacity to the stiffness and 0.18 times the dimeter of the bolt, in accordance with Sarraj (2007).

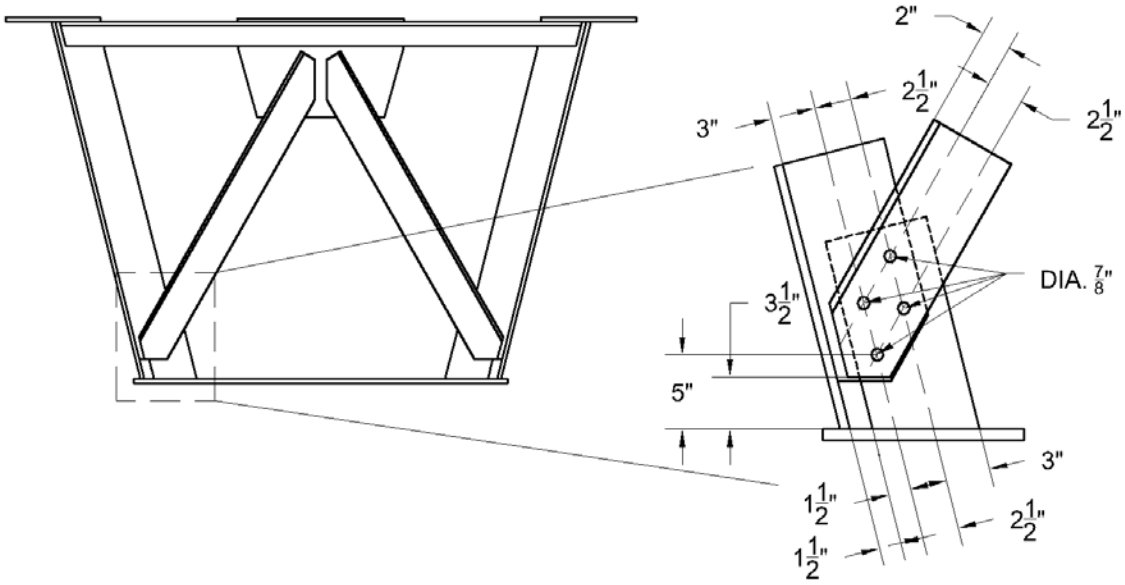


Figure F-12. K-frame connection detail.

F.2.6 Substructure Flexibility Model

In order to account for longitudinal and transverse flexibility of the substructure, connector elements were utilized. These elements allow for the definition of coupled force-deformation relations. The type that was determined to best capture the intended behavior was a Cartesian connector. These elements

provide a connection between two nodes where the change in position is measured in three directions local to the connection. One of the nodes is fixed (or connected to ground) and the other node is the support point in the superstructure. The connector element is rigid in the vertical direction, and has a coupled linear elastic relation in the two horizontal directions (longitudinal and transverse).

In the current case, the structure is assumed to be vertically supported and allowed to translate in the horizontal plane at the abutments; hence the vertical translation at the ends of the structure will be set to zero. At the continuous support, the structure is assumed to be fixed to the pier. As a result, the vertical translation at the pier will be set to zero, while the horizontal stiffness of the pier will be incorporated through connector elements.

In order to obtain the coupled elastic force-displacement relation, a simple finite element analysis of the pier is conducted. The geometry of the pier is drawn according to the design plans, as shown in Figure F-13 and meshed with 8-node linear bricks with reduced integration as shown in Figure F-14. The pier was modeled as linear elastic with modulus of elasticity of 1,800 ksi (in order to account for possible cracking due to combined compression and bending), Poisson's ratio of 0.2 and a density of 0.150 kcf. The base of the pier bears on a rigid mat, prohibiting sliding but allowing uplift as shown in Figure F-14.

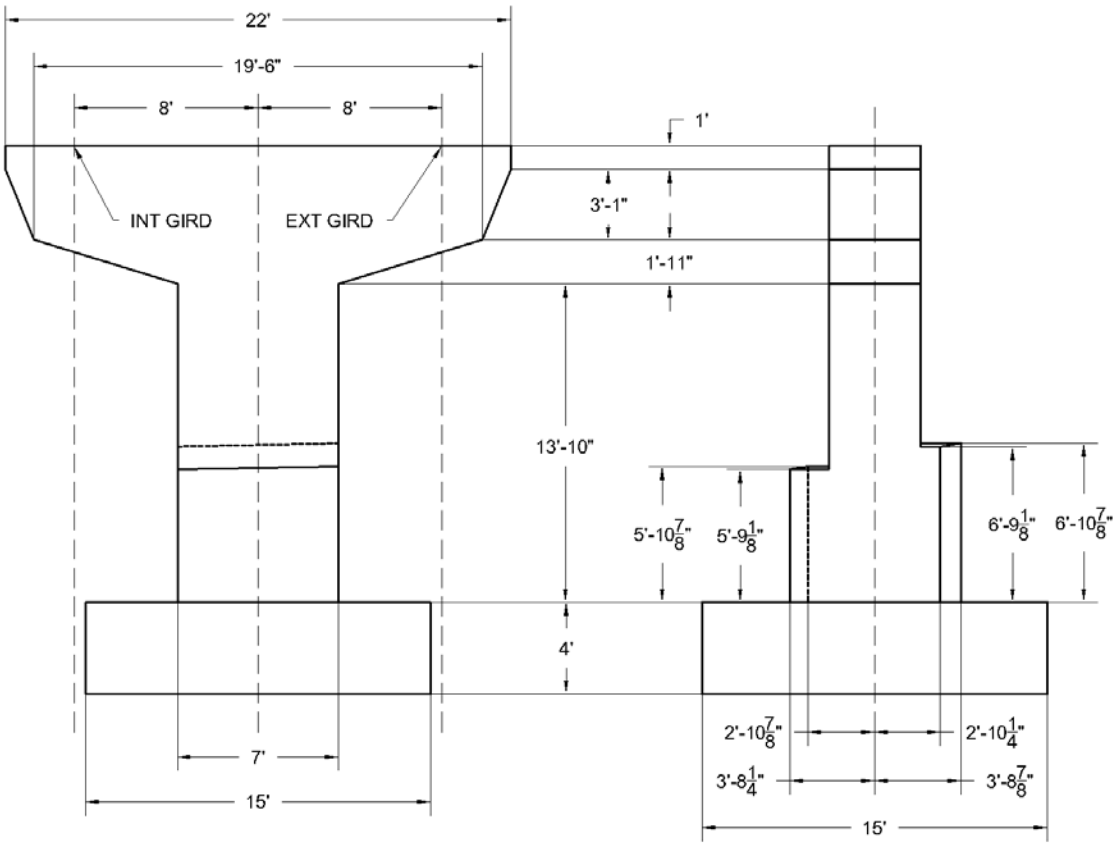


Figure F-13. Geometry of the pier support.

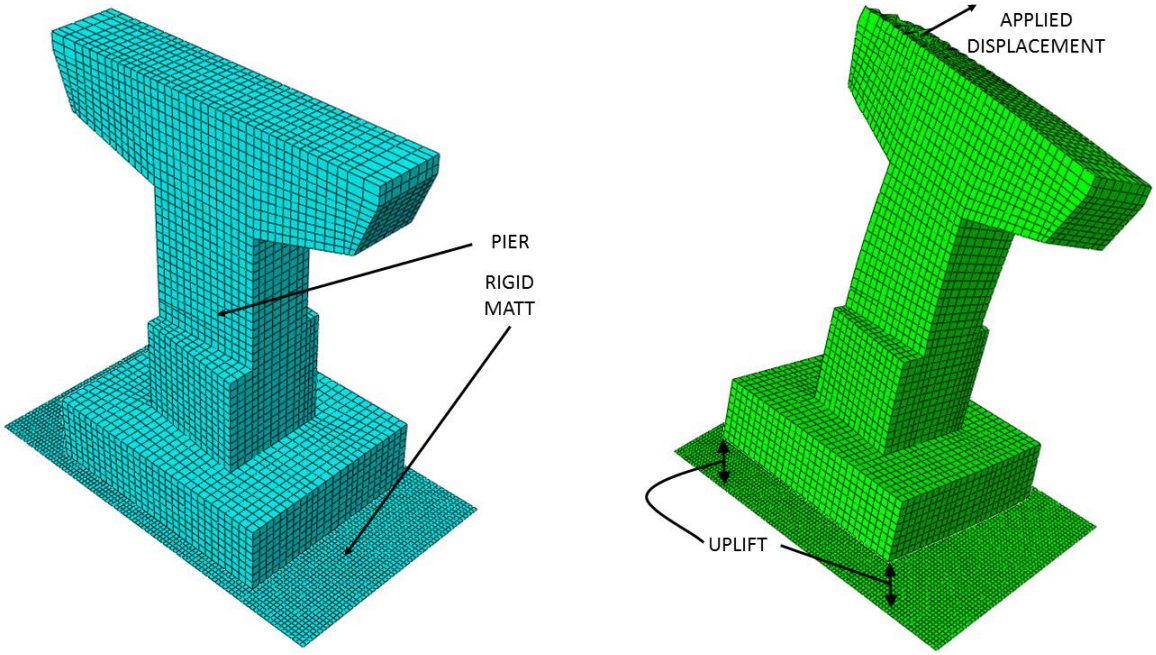


Figure F-14. Detail of mesh geometry used (left) and deformed configuration after application of displacement (right, displacements scale by a factor of 25).

During the first analysis step, dead loads are applied. These are due to (1) the self-weight of the pier, applied as a body force, and (2) bearing of the superstructure on the pier, which were calculated to be 380 kips for the exterior girder and 350 for the interior girder. These are applied as surface tractions over a 28" by 28" patch, the size of the patch is based on the size of the bearings. Once the first step is completed, displacements are applied at the bearing locations so the reaction forces can be calculated. In this case, displacements of 6 inch were applied in the longitudinal and transverse directions (positive and negative signs) so the reaction forces and described in Table F-3 were obtained.

Table F-3. Reaction forces and displacements at support points.

Interior Girder Support				Exterior Girder Support			
U_{TRAN}	U_{LONG}	R_{TRAN}	$R_{LONG.}$	U_{TRAN}	U_{LONG}	R_{TRAN}	R_{LONG}
in	in	kips	kips	in	in	kips	kips
6.00	-0.0620	925	-	6.00	-0.0595	971	-
-6.00	0.108	-1020	-	-6.00	1030	-1060	-
-0.771	6.00	-	787	-0.0738	6.00	-	822
0.141	-6.00	-	-786	0.135	-6.00	-	-823

These are used to build the force deformation relations shown hereof, which are incorporated as the properties of the connector element in the global model to model the flexibility of the pier:

$$\begin{bmatrix} F_{TRANSVERSE} \\ F_{LONGITUDINAL} \end{bmatrix}_{INT\ GIRDER\ SUPPORT} = \begin{bmatrix} 162 & 2.38 \\ 2.38 & 131 \end{bmatrix} \frac{kips}{in} \begin{bmatrix} U_{TRANSVERSE} \\ U_{LONGITUDINAL} \end{bmatrix}_{INT\ GIRDER\ SUPPORT}$$

$$\begin{bmatrix} F_{TRANSVERSE} \\ F_{LONGITUDINAL} \end{bmatrix}_{EXT\ GIRDER\ SUPPORT} = \begin{bmatrix} 170 & 2.38 \\ 2.38 & 137 \end{bmatrix} \frac{kips}{in} \begin{bmatrix} U_{TRANSVERSE} \\ U_{LONGITUDINAL} \end{bmatrix}_{EXT\ GIRDER\ SUPPORT}$$

F.2.7 Loads and Boundary Conditions

Two types of loads were applied in the finite element models: body forces and surface tractions as required by the proposed guide specification in Appendix E. Body forces were applied for component dead loads ("DC" and "DW" per AASHTO designations). These are simply the product of mass, gravity and applicable load factors. Surfaces tractions were applied for traffic live loads ("LL" per AASHTO designation). The traffic live load is based on the HL-93 load model described in the AASHTO LRFD BDS, which is a combination of the truck loads, shown in Figure F-15, and a 0.64 klf load distributed over a width of 10 ft. The current structure does not include any bituminous pavement (i.e., DW is zero).

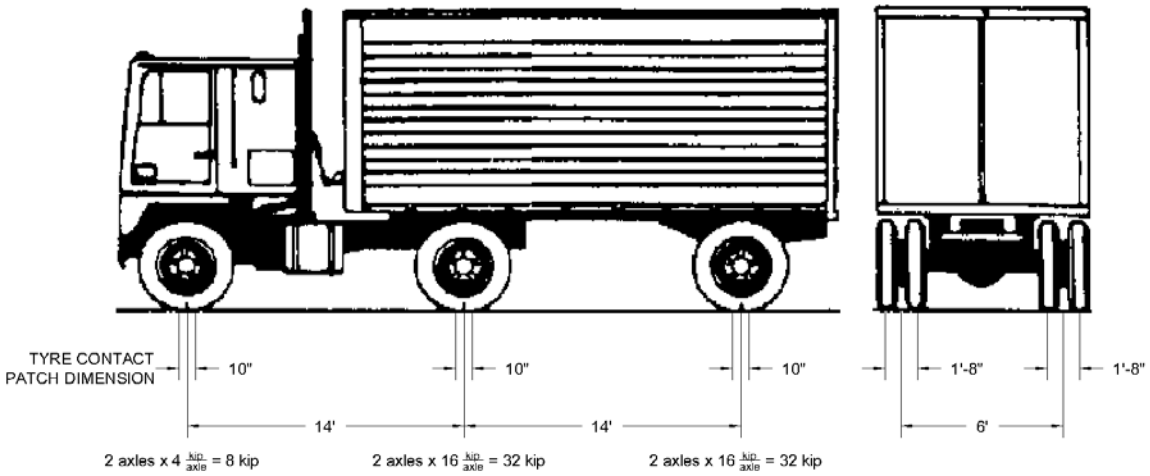


Figure F-15. Truck load components and dimensions of the HL-93 vehicular live load model.

The Redundancy I and Redundancy II loading combinations were used to evaluate the structure in the faulted state. The load factors for these two combinations are as in Table F-4 are based on the provisions in Appendix E for bridges built to Section 12 in the AWS D1.5. The live load (LL) factors are modified by the appropriate multiple presence factors as described in Article 3.6.1.1.2 of the AASHTO LRFD BDS. It must be noted that dynamic amplification factor is equal to 0.2 because the structure is a continuous twin tub girder system, which is applied to DC and LL in the Redundancy I load combination only. Also, the dynamic load allowance is 0.15 of the truck axle loads, and is only applied in the Redundancy II load combination.

Table F-4. Load factors used for Redundancy I and Redundancy II load combinations.

Load Combination	Load Factors				Notes
	DC	LL	DA_R	IM	
Redundancy I	1.05	0.85	0.20	N. A.	$\beta = 1.5$
Redundancy II	1.05	1.30	N. A.	0.15	$\beta = 1.5$

Longitudinally, the loads are positioned in the most critical positions in both the Redundancy I and Redundancy load combinations. For the failure scenario considered in the current case (failure of the exterior girder near mid span on the north span as shown in Figure F-2 and Figure F-16), the most critical position of the truck axle loads which results in the truck facing south with its middle axis positioned at the failure plane, as shown in Figure F-16. The distributed load portion of the HL-93 load is applied along the northernmost span, from the north abutment to the pier.

As described in the proposed guide specification in Appendix E, the transverse positioning of the HL-93 live load model differs between the Redundancy I and Redundancy II load combinations, as illustrated in Figure F-17. Since the vehicular loads in the Redundancy I load combination are meant to represent the applied load at the instant in time in which the assumed member failure occurs, the HL-93 vehicular live load model is transversely positioned centered (both the 10 ft loaded width and the truck axle loads) within the marked (striped) lanes, in this case one lane. Hence, as the bridge is only striped for one lane, there is only one load case for the Redundancy I load combination.

On the other hand, the objective of the Redundancy II load combination is to evaluate the strength of the system after the failure of the primary steel tension member has occurred, so the number of design lanes is established in accordance with Article 3.6.1.1.1 in the AASHTO LRFD BDS, which in this case results in two design lanes with a width of 12 ft. In the Redundancy II load combination, the HL-93 vehicular live load model is transversely positioned (both the 10 ft loaded width and the truck axle loads) to produce extreme force effects within each design lane; however, the truck axle loads are transversely positioned such that the center of any wheel load is not closer than 2 ft from the edge of the design lane. Hence, there are two load cases for the Redundancy II load combination: two design lanes loaded, or one design lane loaded.

Component dead loads were linearly applied in the initial implicit static analysis. Traffic live loads were applied in the final explicit dynamic analysis. Their dynamic effects were minimized by applying them slowly through the use of smooth step, as in the following equation:

$$LR(t) = 6\left(\frac{t}{T}\right)^5 - 15\left(\frac{t}{T}\right)^4 + 10\left(\frac{t}{T}\right)^3$$

where $LR(t)$ is the fraction of load at a load application time t, and T is the duration of the load application. The duration of the load application must be larger than the fundamental period of the structure to minimize oscillatory behavior in the final explicit dynamic analysis.

Regarding prescribed boundary conditions, vertical translation is prescribed to be zero at all support location since uplift would not occur under the loading employed in the current case. Horizontal translations are discussed in Section F.2.6 as they are enforced through connector elements that model the flexibility of the substructure.

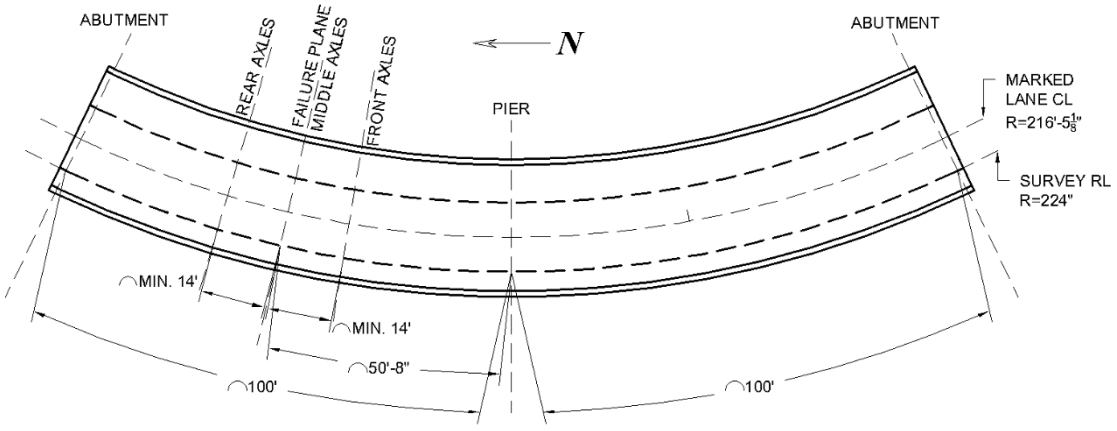
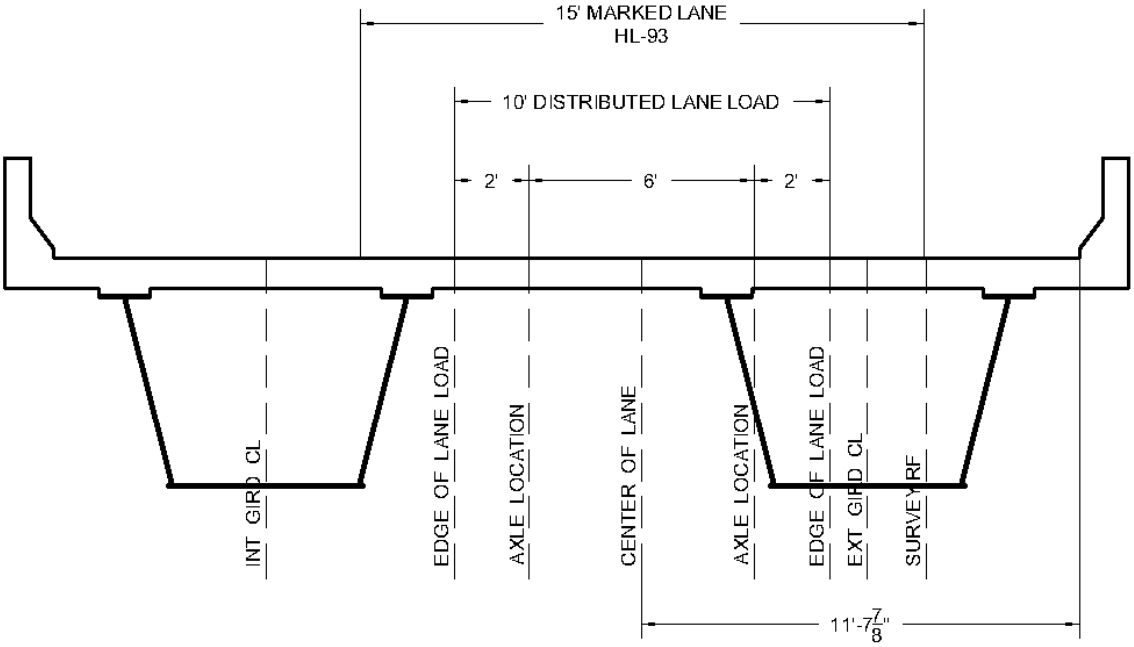
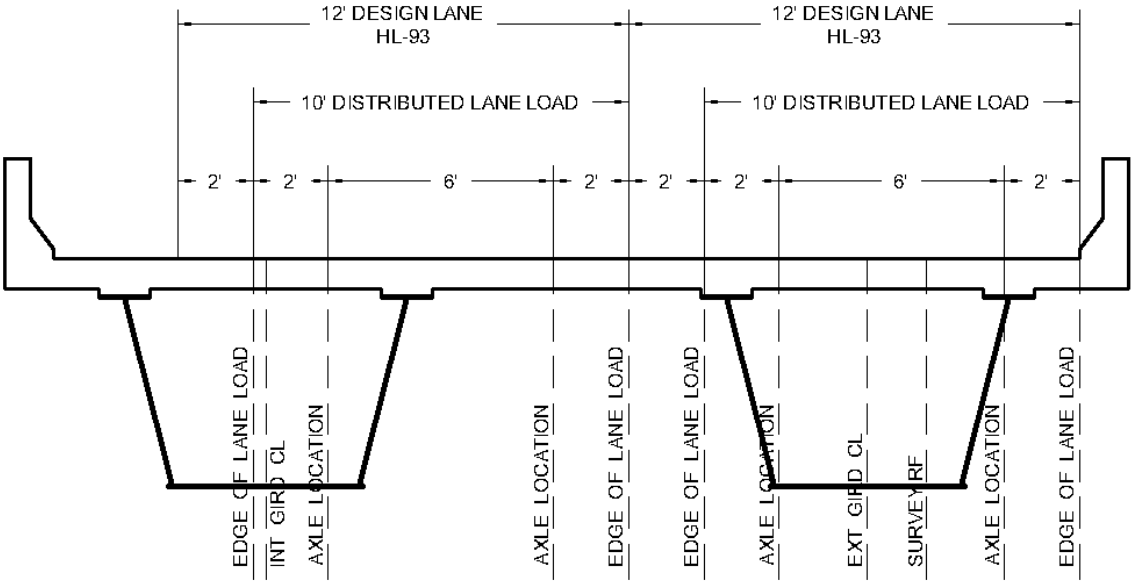


Figure F-16. Longitudinal position of the HL-93 live load model.



TRANSVERSE LOAD POSITION FOR REDUNDANCY I



TRANSVERSE LOAD POSITION FOR REDUNDANCY II

Figure F-17. Transverse position of the HL-93 live load model.

F.2.8 Analysis of Results for Redundancy

Once the analysis is completed the obtained results are evaluated using the requirements described in Article 8 of the proposed guide specifications in Appendix E. It was found that the structure met the strength

and serviceability requirements and is considered redundant against failure of the exterior tub girder. Specific details regarding the performance requirements and the results are summarized in Table F-5.

Table F-5. Summary of the redundancy evaluation.

Performance Requirement		Most Critical Load Combination	Result	Acceptable?
Strength Requirements	Strain Primary Steel Members	-	No plasticity in primary members	**YES**
	Slab Concrete Crushing	-	No concrete crushing in the slab	**YES**
Serviceability Requirements	Vertical Deflection	Only Redundancy II DL considered	0.516 in	**YES**

Notes:
1. The analysis showed that the structure was capable of resisting an additional 15% of the applied factored live load.
2. In order to complete the evaluation, the displacements and reaction forces calculated at support locations should be used as factored demands to check against the nominal capacity of the supports and substructure members.

F.2.8.1 Minimum Strength Requirements

All of the strength requirements were met by the system in the faulted state while subjected to any one of the load cases included in the Redundancy I and Redundancy II load combinations. Since the system met all of the strength requirement it may be re-designated as a system redundant member (SRM) as soon as the minimum serviceability requirements are met; otherwise it shall remain designated a fracture critical member (FCM).

The first set of strength requirements apply to any primary member of the superstructure, which in this case are the tub girders, diaphragms, and concrete slab. These requirements are the following:

- In a component, such as a web or a flange of a primary steel member, the average strain is less than five times the material yield strain.
- In a component, such as a web or a flange of a primary steel member, the average strain is less than 0.01.
- A strain level of 0.05 is not reached anywhere in a primary steel member.
- The combined flexural, torsional and axial force effects computed in primary compression members are below the nominal compressive resistance of the member (these limit states are predicted by the FEA).
- If a compression strain greater than 0.003 is reached in the slab, the portion where that limit is exceeded does not compromise the overall system load carrying capacity.
- The system in the faulted condition is able to support an additional 15% of the factored live load.

No yielding was observed in the primary steel members, further critical buckling loads were not reached in any primary steel member. No plastic strains were calculated in the tub girders or the diaphragms after the failure of the exterior girder for the Redundancy I or Redundancy II load combinations; therefore, the strain requirements on primary steel members are met, as illustrated in Figure F-18. As the FEA accurately predicts potential failure of primary steel compression member subjected to combined flexural, torsional, and axial force effects, and quasi-static equilibrium is reached for both load combinations, the requirements of primary steel compression members are met.

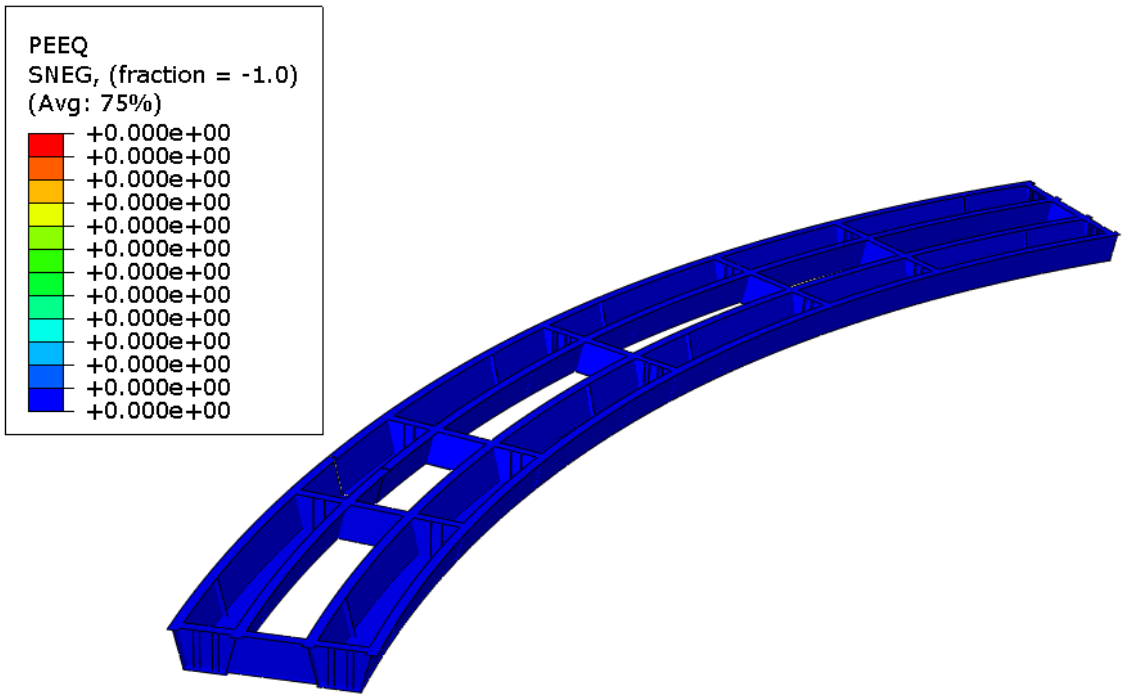


Figure F-18. Absence of plastic equivalent strain in primary steel members.

Regarding the concrete slab, concrete crushing and tension cracking is allowed and expected to take place. However, if the portion of the slab where a total compressive strain of 0.003 has been exceeded is large enough to compromise the overall system load carrying capacity or if significant hinging occurs, the structure should not be considered as sufficiently redundant. In this example, the Redundancy II load combination resulted in the largest compressive strains in the slab, which were located in the haunches over the exterior tub girder at the failure location, as shown in Figure F-19. Although there is some localized compressive damage in the slab it was extremely confined to a small area. Thus, it was not enough to result in a reduction in load carrying capacity.

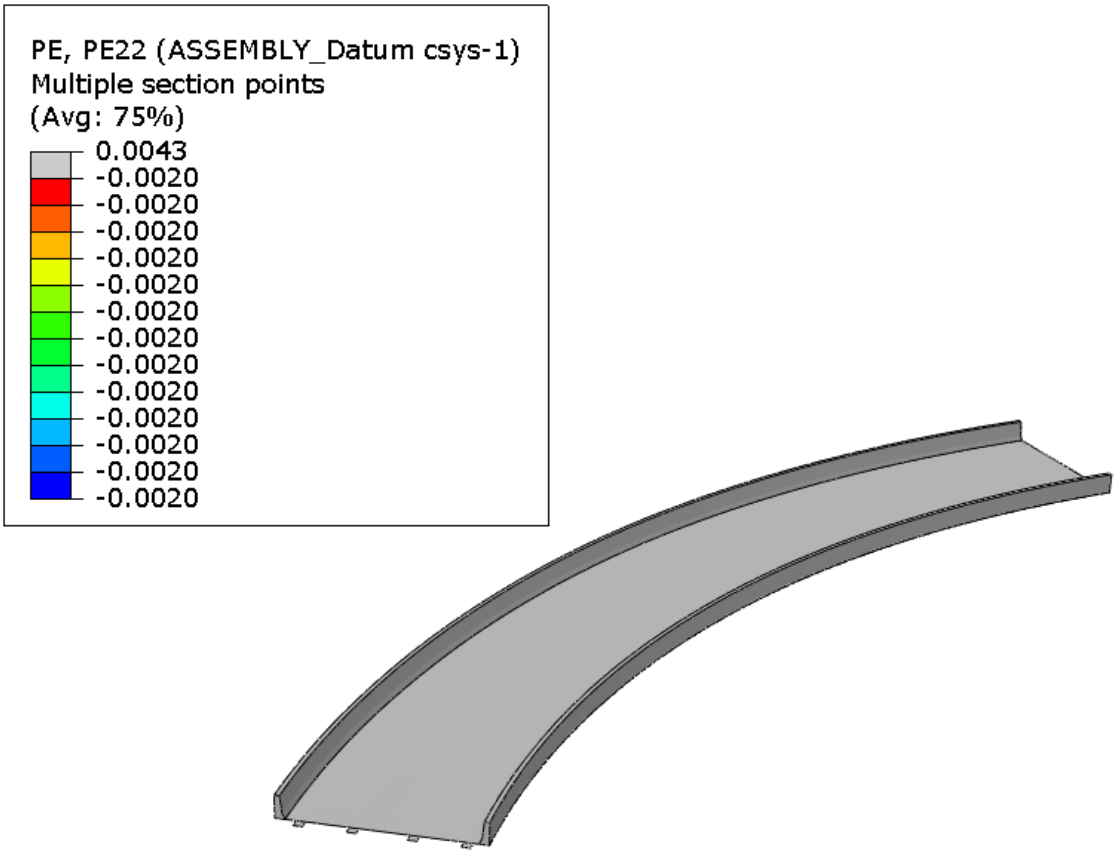


Figure F-19. Absence of concrete crushing in slab.

Although the substructure is not explicitly included in the finite element model, the reaction forces at support locations are calculated in the analysis. These should be taken as the factored demands that the substructure must be able to safely sustain, which are summarized in Table F-6. In this example, the Redundancy I load combination resulted in the largest vertical reaction forces, except for the vertical reaction at exterior girder support in the north abutment which largest value was calculated under the one loaded lane case of the Redundancy II load combination. On the other hand the largest transverse reaction forces take place under the two loaded lanes case of the Redundancy II load combination at the pier. The unfactored nominal capacity of the abutments and the pier need to be checked against these load demands. Similarly the pier and abutments must accommodate the horizontal displacements that are calculated in the analysis at the support locations. In this example, Redundancy I and Redundancy II load combinations resulted in similar small horizontal displacements which are summarized in Table F-7.

825 **Table F-6. Calculated reaction forces for redundancy evaluation.**

Support	Girder	Reaction Force	Result for Redundancy I (1 Lane)	Result for Redundancy II (1 Lane)	Result for Redundancy II (2 Lanes)
South Abutment	Interior	Vertical	107.3 kips	86.3 kips	91.0 kips
	Exterior	Vertical	160.8 kips	105.1 kips	116.9 kips
Pier	Interior	Vertical	596.7 kips	581.5 kips	508.9 kips
		Longitudinal	0.0 kips	0.0 kips	0.0 kips
		Transverse	15.2 kips	15.9 kips	18.7 kips
	Exterior	Vertical	578.2 kips	567.9 kips	551.9 kips
		Longitudinal	0.0 kips	0.0 kips	0.0 kips
		Transverse	-15.2 kips	-15.9 kips	-18.7 kips
North Abutment	Interior	Vertical	123.2 kips	108.9 kips	47.0 kips
	Exterior	Vertical	233.1 kips	284.4 kips	283.7 kips

Notes
1. Longitudinal direction is normal to the radius of the curve. Positive longitudinal direction points south.
2. Transverse direction is parallel to the radius of the curve. Positive transverse direction points away from the origin of the curve.

826

827 **Table F-7. Calculated displacements at support locations for redundancy evaluation.**

Support	Girder	Displacement	Result for Redundancy I (1 Lane)	Result for Redundancy II (1 Lane)	Result for Redundancy II (2 Lanes)
South Abutment	Interior	Longitudinal	0.167 in	0.112 in	0.143 in
		Transverse	0.146 in	0.101 in	0.126 in
	Exterior	Longitudinal	0.180 in	0.141 in	0.173 in
		Transverse	0.147 in	0.102 in	0.125 in
Pier	Interior	Longitudinal	0.000 in	0.000 in	0.000 in
		Transverse	0.093 in	0.098 in	0.115 in
	Exterior	Longitudinal	0.000 in	0.000 in	0.000 in
		Transverse	-0.089 in	-0.094 in	-0.110 in
North Abutment	Interior	Longitudinal	0.290 in	0.251 in	0.319 in
		Transverse	-0.367 in	-0.449 in	-0.474 in
	Exterior	Longitudinal	0.397 in	0.446 in	0.510 in
		Transverse	-0.362 in	-0.447 in	-0.472 in

Notes:
1. Longitudinal direction is normal to the radius of the curve. Positive longitudinal direction points south.
2. Transverse direction is parallel to the radius of the curve. Positive transverse direction points away from the origin of the curve.

828

829 Additionally, the system demonstrated a reserve margin of at least 15% of the applied live load in the
830 Redundancy I and II load combinations. Effectively, this requirement ensures the slope of the load vs
831 displacement curve for the system structure remains positive (i.e., there is still significant remaining reserve
832 capacity).

F.2.8.2 Minimum Serviceability Requirements

The only serviceability requirement in the Appendix E is that the increase of deflection after the failure of a primary steel tension member cannot be greater than L/50. This requirement is to be checked in the Redundancy II load combination under factored dead load only. In the current case, the limit is 24 inches, which was not surpassed since the maximum additional deflection computed in the FEA was 0.516 inches. This is illustrated in Figure F-20.Figure F-20. Deflection after failure of primary steel tension member.

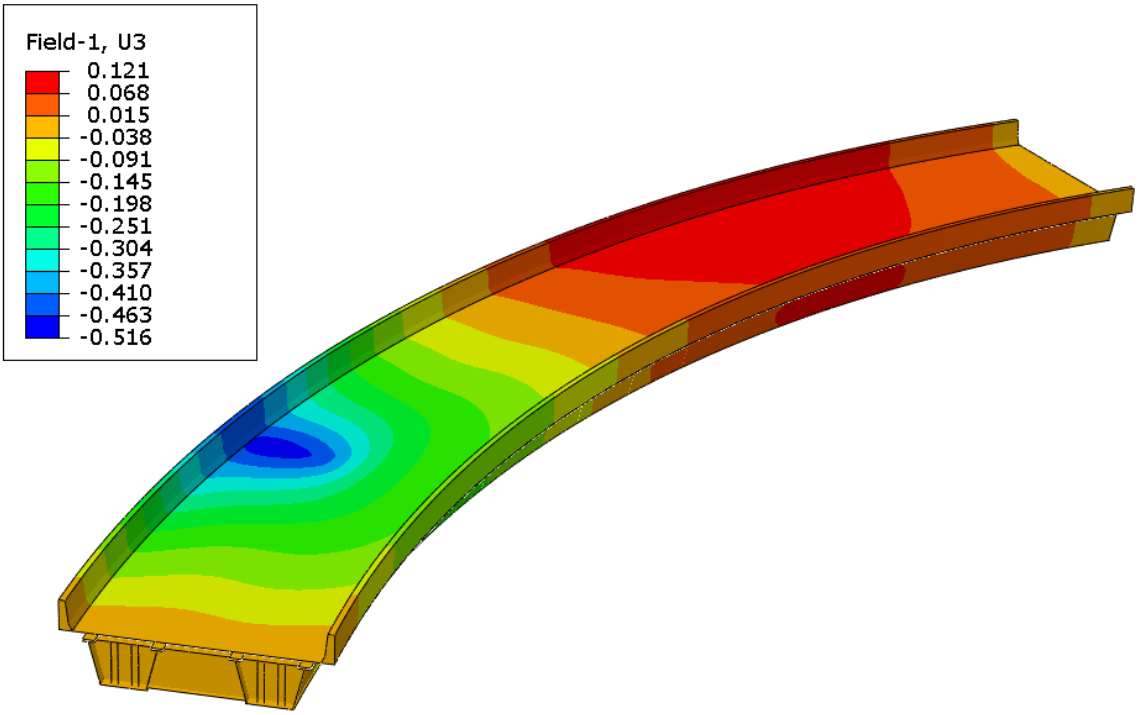


Figure F-20. Deflection after failure of primary steel tension member.

F.2.9 Conclusions

The redundancy of a curved continuous two span twin tub girder bridge after the failure of the exterior tub girder was analyzed in accordance with the methodology described in the proposed guide specification in Appendix E. Based on the comparison between the obtained results and the minimum performance requirements, the structure is not likely to fail nor undergo a significant serviceability loss as result after the failure of the exterior tub girder. Hence the exterior tub girder may be re-designated as a system redundant member (SRM).

F.3 Single Span Through Girder Bridge

The redundancy of a single span straight through girder bridge is analyzed by developing a finite element model in accordance with the methodology described in the proposed guide specification in Appendix E. It is assumed that the structure does not possess any of the detrimental attributes described in the screening criteria and that it is built to Section 12 of the AWS D1.5. In this case, the failing tension member is the east girder. The entire cross-section, including the compression flange, is assumed to fail at mid span as shown in Figure F-21.

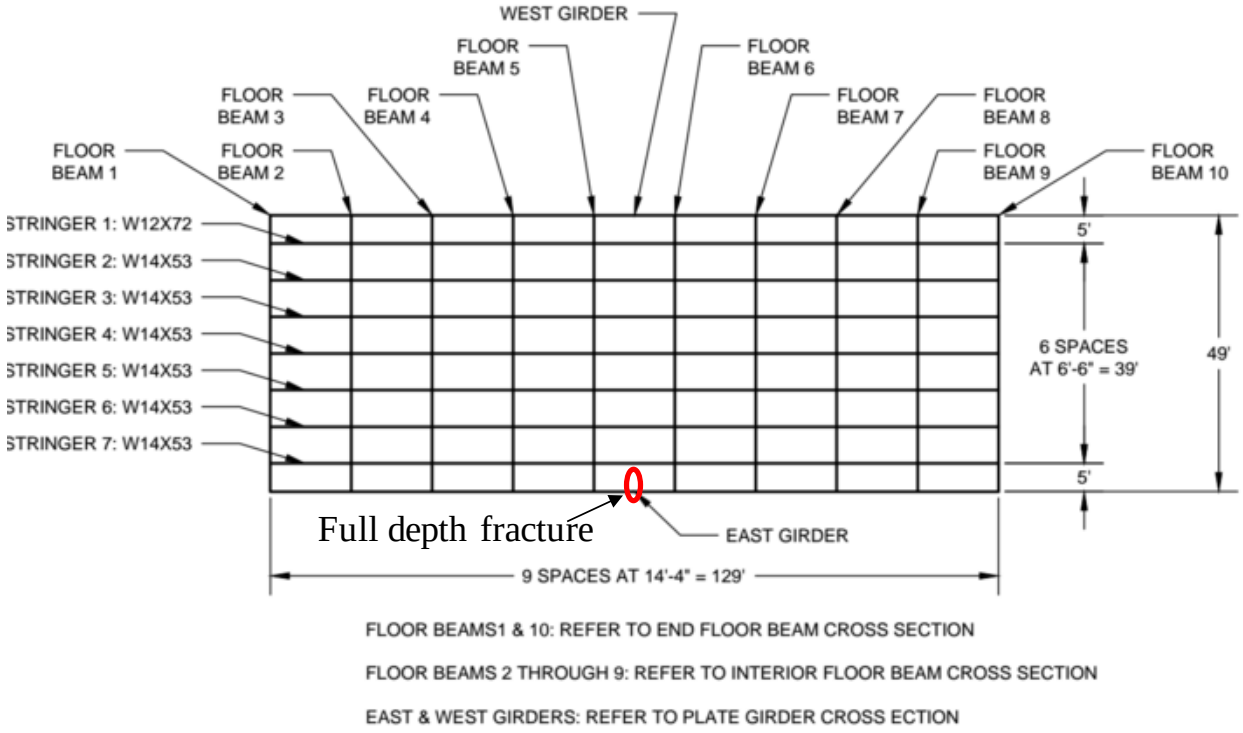


Figure F-21. Steelwork geometry and failure location.

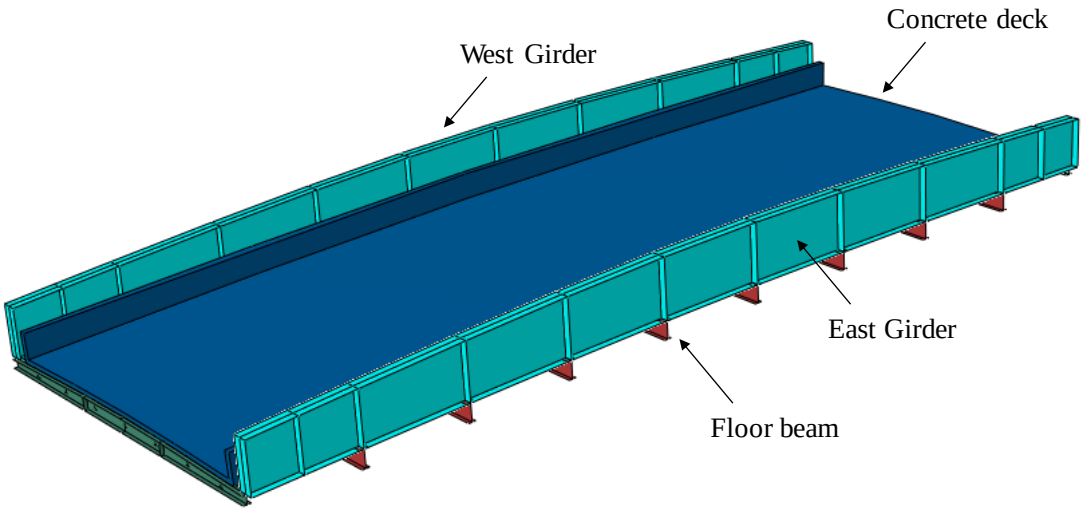


Figure F-22. Overview of the bridge.

The structure is a through girder bridge with girders spaced at 49 feet and a single span measuring 129 feet long. Figure F-21 shows an overall sketch of the structure. As shown in Figure F-22, the bridge consists of the steel girder system and a reinforced concrete slab with railings. The steel girder system is composed of two plate girders (west and east), 10 floor beams, and seven stringers, as shown in Figure F-21. The concrete slab is 44.5 feet wide. The cross section of the bridge is shown in Figure F-23. Figure F-24 shows the connection details between the floor beams and plate girders. Figure F-25 shows the connection details between the stringers and floor beams. As can be seen, this connection is a typical of through girder bridges. The bridge is composite by using shear studs to connect the concrete slab to the steel system. Three shear studs are placed in each row on the top flange of the floor beams, with a typical spacing of 10 in. Two shear studs are placed in each row on the top flange of the stringers, with a typical spacing of 6 in. All steel plates, used in the girders are made of ASTM A709 Grade 50. Slab reinforcement is made of ASTM A706 Grade 60. The concrete compressive strength of the slab is 4 ksi. The cross slope and girder camber were ignored in the development of the geometry used in the model.

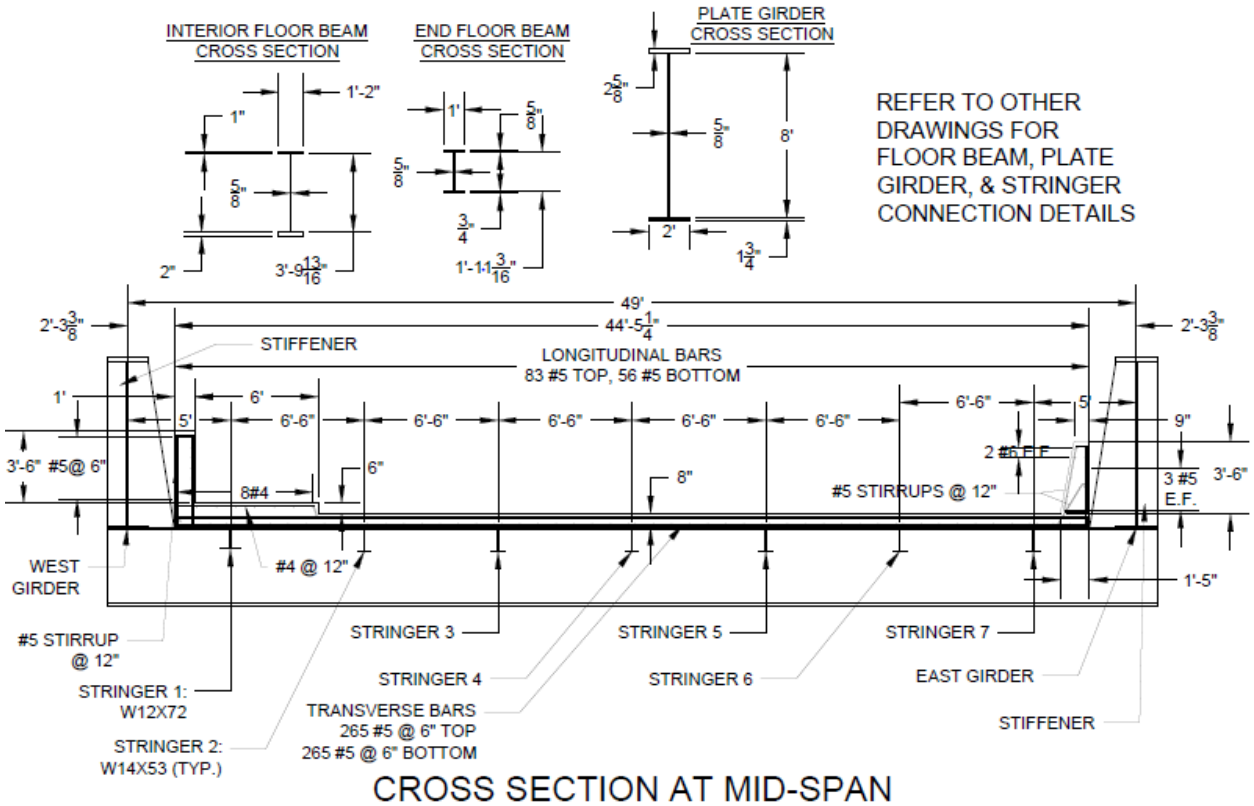


Figure F-23. Typical cross-section and slab reinforcement details.

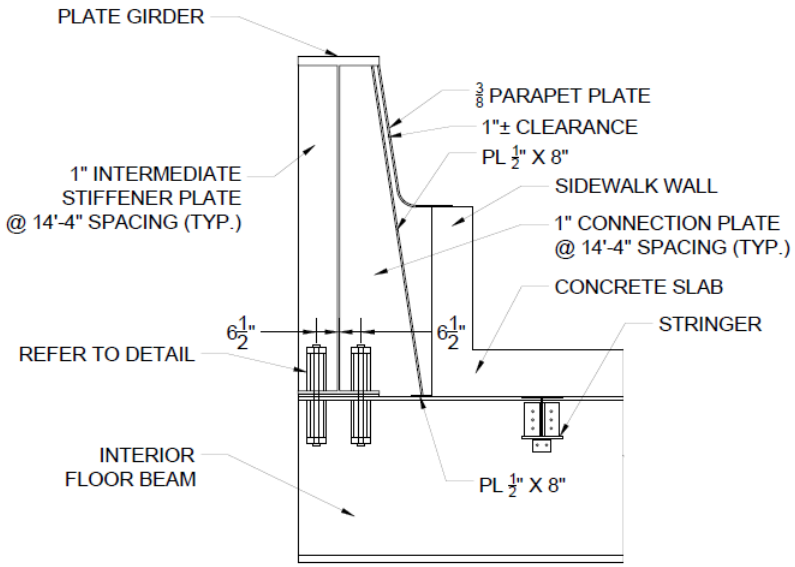


Figure F-24. Connection details between the floor beams and the plate girder.

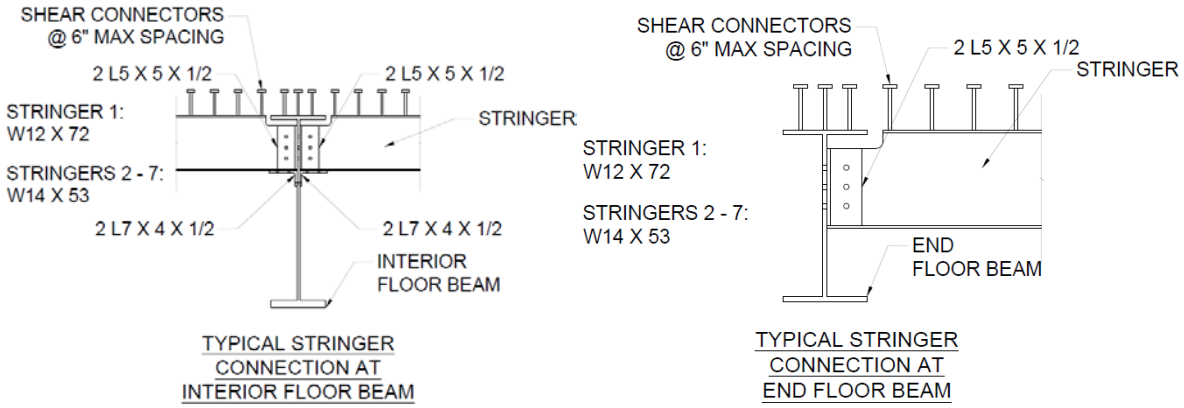


Figure F-25. Connection details between the stringers and the floor beams.

F.3.1 Analysis Procedure

The analysis is performed to establish if the system demonstrates acceptable performance in the faulted condition. In the example, the term "faulted condition" specifically refers to the case in which a primary steel tension member is assumed to have failed. For this analysis, load factors for both dead and live load are applied as described in the proposed guide specification in Appendix E. In this example, the described analysis procedure is composed of an initial implicit static analysis and a final explicit dynamic analysis, into which the results from the initial implicit static analysis are imported. While it is not mandatory for the Engineer to follow these particular steps, it has been found that this procedure optimizes the computational time required.

F.3.1.1 Initial Implicit Static Analysis

Implicit static analysis was utilized to calculate the state of the structure prior to hardening of the concrete in the slab. An implicit static analysis was used for the initial steps because, although non-linearity is considered in the analysis, the bridge behavior is linear and inertial effects can be neglected as the bridge is in the undamaged condition. As the slab does not carry any load and does not contribute to the stiffness of the system before concrete hardening, two modifications are required in the finite element analysis during this initial implicit static analysis as follows:

- A very low stiffness is specified for the elements composing the slab, i.e., the elements modeling concrete and rebar. A reduced stiffness of 1/1,000 of the respective modulus of elasticity of each material was used. This is done so the load carried by the slab and rebar have negligible contribution to the stiffness of the system. No modifications to the stiffness should be applied to the steelwork.
- Instead of defining contact interaction between the slab and the steelwork, a mesh tie was specified. The nodal displacements of the concrete slab elements are tied to the displacements of the top flanges of girders, floor beams, and stringers which occur due to dead load. As a result, the slab deforms with the steelwork and does not 'sag' between the girders, floor beams, and stringers.

It is worth noting that the remainder of the finite element modeling is identical between the initial implicit static analysis and the final explicit dynamic analysis. The specific steps in the initial implicit static analysis are described as follows:

1. Apply load due to self-weight of the structural steel components as a body force.
2. Apply load due to self-weight of the wet slab components as a body force.
3. The system is then fixed in terms of position, that is, the displacement degrees of freedom are not allowed to change.

4. The elements composing the slab (elements modeling rebar and concrete) are then deactivated.
5. The elements composing the slab are then reactivated. During this reactivation the strain in the elements composing the slab is reset to zero.

Steps 3 through 5 are necessary since even though very low stiffness was specified for the slab, these elements do undergo strain. Setting the strains to zero eliminates "locked in" artificial stresses in later steps.

F.3.1.2 Final Explicit Dynamic Analysis

As contact algorithms, softening material behavior, and non-linear geometry are required to be part of the finite element analysis, implicit solution procedures present unavoidable convergence problems in most FEA solvers. In order to calculate the capacity of the bridge after sudden failure of a tension component, a dynamic explicit analysis needs to be carried out. Therefore, the results obtained from the initial implicit static analysis are imported into the final explicit dynamic analysis. In other words, the state of the system (stresses, strains, displacements and forces) at the beginning of the final explicit dynamic analysis is defined by the state of the system computed at the end of the initial implicit static analysis.

As previously stated, during the initial implicit static analysis, the slab was modeled with largely reduced stiffness to reflect that it is not hardened and a mesh tie constraint was used to assure that the slab deformed with the steelwork. This approach also prevents excessive sag of the soft slab. After the state of the system is imported, the following changes are made to capture the response of the structure after the concrete has hardened:

- The modulus of elasticity of the concrete and rebar elements in the slab is changed to their final actual values. It is noted that no modifications need to be applied for the steelwork.
- The mesh tie constraint between the slab concrete elements and the top flanges of the steelwork is replaced by a frictional contact interaction. Additionally, since the structure under analysis is composite, elements which accurately model the behavior of shear studs are added.

All of the body forces applied during the initial implicit static analysis (i.e., the dead load of the structure) are maintained throughout the final explicit dynamic analysis.

To evaluate the capacity of the structure in the faulted state, the following steps were carried out in the final explicit dynamic analysis:

6. The stiffness of the elements located at the fracture location under consideration were slowly reduced. The stiffness was slowly reduced in order to minimize any dynamic effects. It is noted that the actual fracture and subsequent vibration of the structure is not modeled. This dynamic effect is accounted for using the DA_R factor as discussed before. If dynamic effects are found to be significant even if the stiffness is slowly reduced, the system must be allowed to oscillate until these effects are dampened.
7. Factored loads due to traffic are applied as surface tractions. For the Redundancy I load combination all loads are amplified by DA_R, for the Redundancy II load combination the dynamic load allowance (IM) is applied. These loads were applied very slowly to minimize any dynamic effects, as well. If dynamic effects are significant, the system must be allowed to oscillate until these effects are dampened.
8. An additional 15% of live load is gradually applied.

F.3.2 Material Models r

Three material models are needed in the finite element model. Two of these are utilized to model different steel types, and one is utilized to model the response of concrete. For the development of the steel material models, it is necessary to know the yield strength and ultimate strength of each steel type. In this example, since no test values are known to the Engineer, nominal values specified in the respective

standards are utilized. These are summarized in Table F-8. A mass density of 0.494 kcf was specified for all steel types.

Table F-8. Material properties for steel material models.

Material	Nominal Yield Strength	Nominal Ultimate Strength	Standard
ASTM A709 Gr. 50	50 ksi	65 ksi	ASTM A709/A709M
ASTM A 706 Gr. 60	60 ksi	75 ksi	ASTM A706/A706M

The stress-strain relation for all steel components will follow an initial linear elastic steel with a Young's modulus of 29,000 ksi and Poisson's ratio of 0.3. Once the nominal yield strength is reached, the stress-strain relation is defined by Von Mises (J2) plasticity with kinematic linear hardening, until the nominal ultimate strength is reached at a total strain of 0.05. Once the nominal ultimate strength or a total strain of 0.05 is reached, the material is assumed to fail. Figure F-26 shows the uniaxial material response for the steel employed in this finite element model with the 'X' denoting the stress at the failure strain of 0.05.

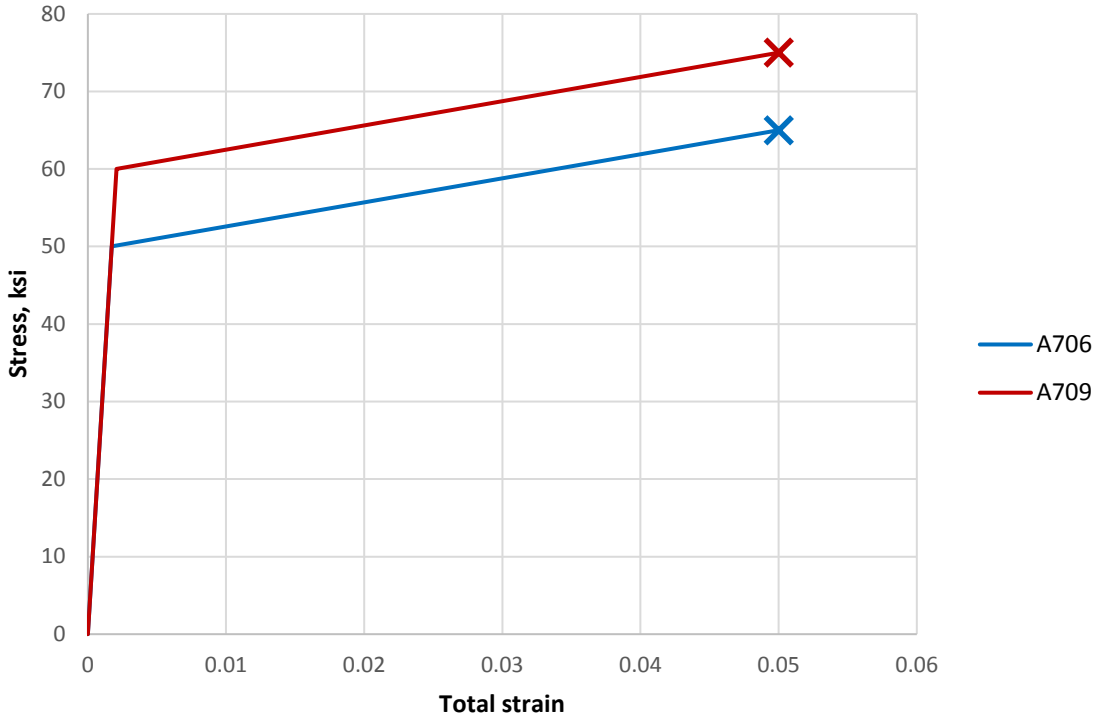


Figure F-26. Stress-strain curves of steel material models.

The material model used in concrete is defined entirely by the specified compressive strength, which in this case is 4 ksi. This quantity is also used to calculate the tensile strength, the total strain at compressive strength, ε_c, and the material parameter n. Table F-9 summarizes the calculation of these values. A mass density of 0.150 kcf was specified for concrete.

977 **Table F-9. Material properties for concrete material model.**

Quantity	Symbol	Equation	Result
Young's modulus	E_c	$E_c = 33{,}000 w_c^{1.5}\sqrt{f_c'} \leq 1{,}802.5\sqrt{f_c'}$	3,600 ksi
Tensile strength	f_t	$f_t = 0.158(f_c')^{\frac{2}{3}}\ for\ f_c' \leq 7.25ksi$ $f_t = 0.307\ln(f_c' + 2.61) - 0.114\ for\ f_c' > 7.25ksi$	0.398 ksi
Fracture energy	G_t	$5.9 \cdot 10^{-4}(f_c' + 1.16)^{0.18}$	$7.93{\cdot}10^{-4}$ kip/in
Total strain at compressive strength	ε_c	$\varepsilon_c = 0.00124\sqrt[4]{f_c'}$	0.00175
Material parameter	n	$n = 0.4f_c' + 1.0$	2.6

978

979 The concrete material model is initially linear elastic, defined by a Young's modulus of 3,600 ksi and
980 Poisson's ratio of 0.2, followed by concrete damage plasticity. In tension, once the material reaches its
981 tensile strength, set at 0.398 ksi in this case, a tensile stress-displacement relation characterized by a fracture
982 energy, G_t, of $7.93{\cdot}10^{-4}$ kip-in is followed This fracture energy is applied through a bi-linear tensile stress-
983 displacement relation as shown in Figure F-27, and defined by the following quantities:

984 $$f_{t1} = \frac{f_t}{5} = 0.0796\ ksi$$

985 $$\delta_t = \frac{5G_t}{f_t} = 0.00996$$

986 $$\delta_{t1} = \frac{G_t}{f_t} = 0.00199$$

987

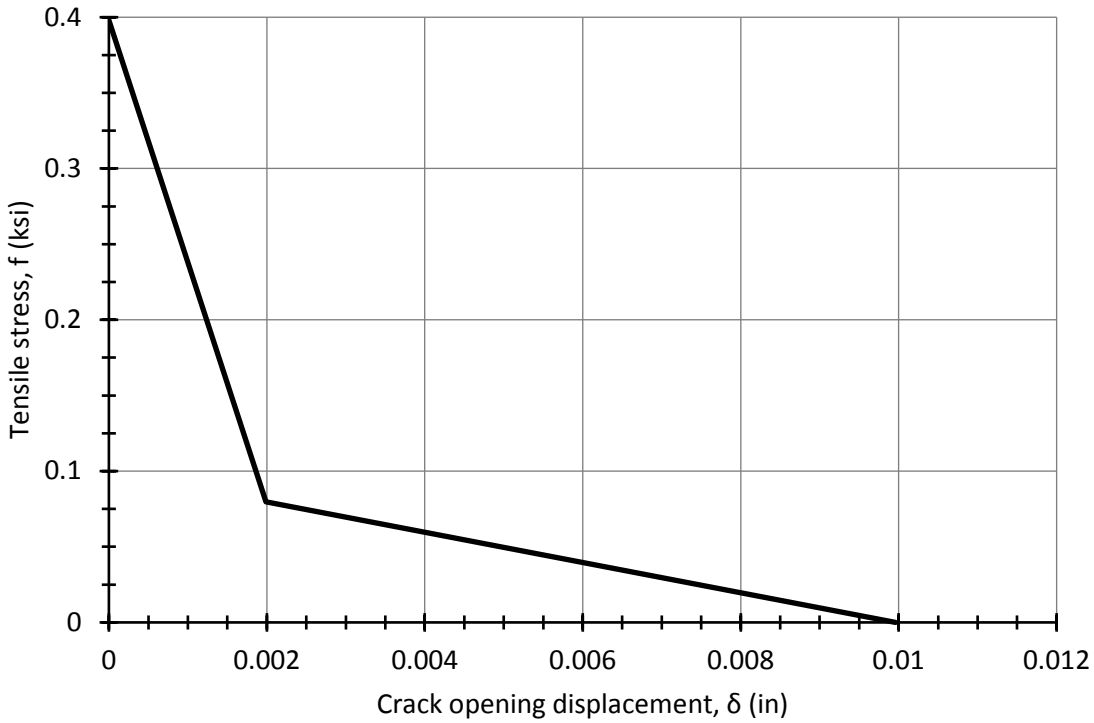


Figure F-27. Tensile stress-crack opening displacement curve for concrete material model.

In compression the material follows the following stress-strain relations:

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

$$\varepsilon_{plastic} = \varepsilon - \frac{f(\varepsilon)}{E_c}$$

Where ε is total (elastic + plastic) strain, $f(\varepsilon)$ is the compressive stress at a given total strain, f_c' is the specified compressive strength, ε_c is the total strain at compressive strength, n is a material parameter, $\varepsilon_{plastic}$ is the plastic strain and E_c is the concrete Young's modulus. Figure F-28 shows the resulting compressive stress-strain relation.

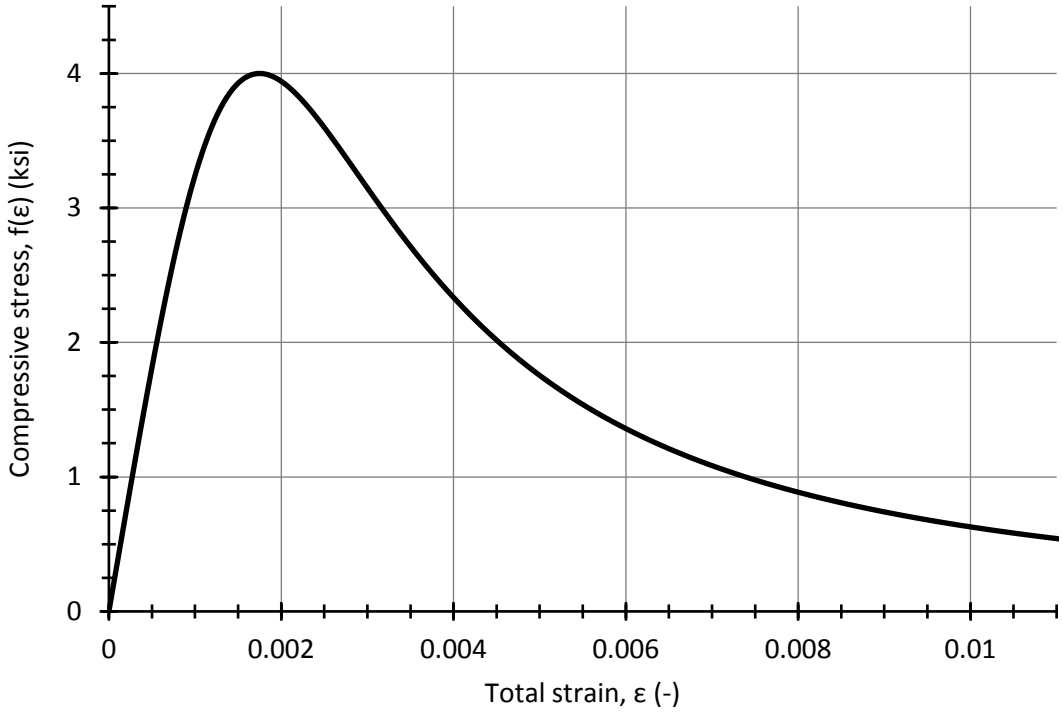


Figure F-28. Compressive stress-strain curve for concrete material model.

F.3.3 Geometries, Meshes and Constraints

The geometry of the structure (see Figure F-29) is based on available design plans and is composed of the following components that must be explicitly modeled:

1. West and east plate girders.
2. 10 floor beams.
3. Seven stringers.
4. A reinforced concrete slab with concrete railings.
5. Shear studs.

When generating the finite element model, splices, holes, access hatches, etc. are neglected. The structure is assumed to be flat in the vertical plane, in other words, camber and superelevation are ignored.

Plate girders, floor beams and stringers are modeled with 4-node shell elements with reduced integration. A minimum of four elements are used along flange widths and along web heights. Stiffening attachments are modeled with shell elements as well. Figure F-30 shows two details of the mesh employed to model the steel system. As shown in this figure, the plate girders, floor beams and stiffeners constitute a single geometry.

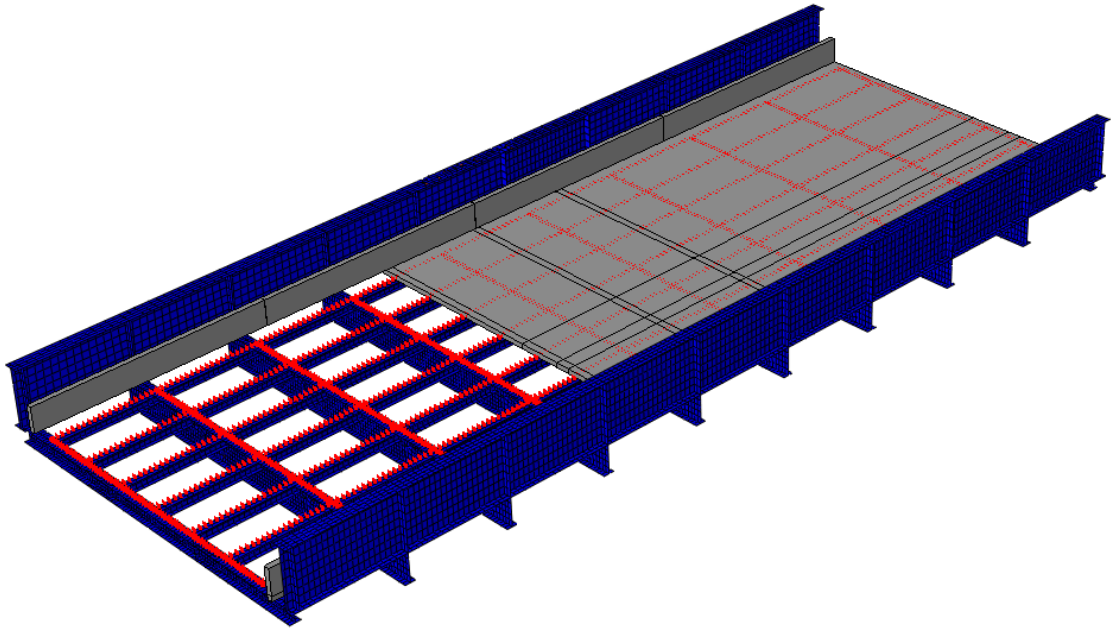


Figure F-29. Through girder bridge geometries. Concrete slab and barriers (grey), steel system (blue), and shear studs (red).

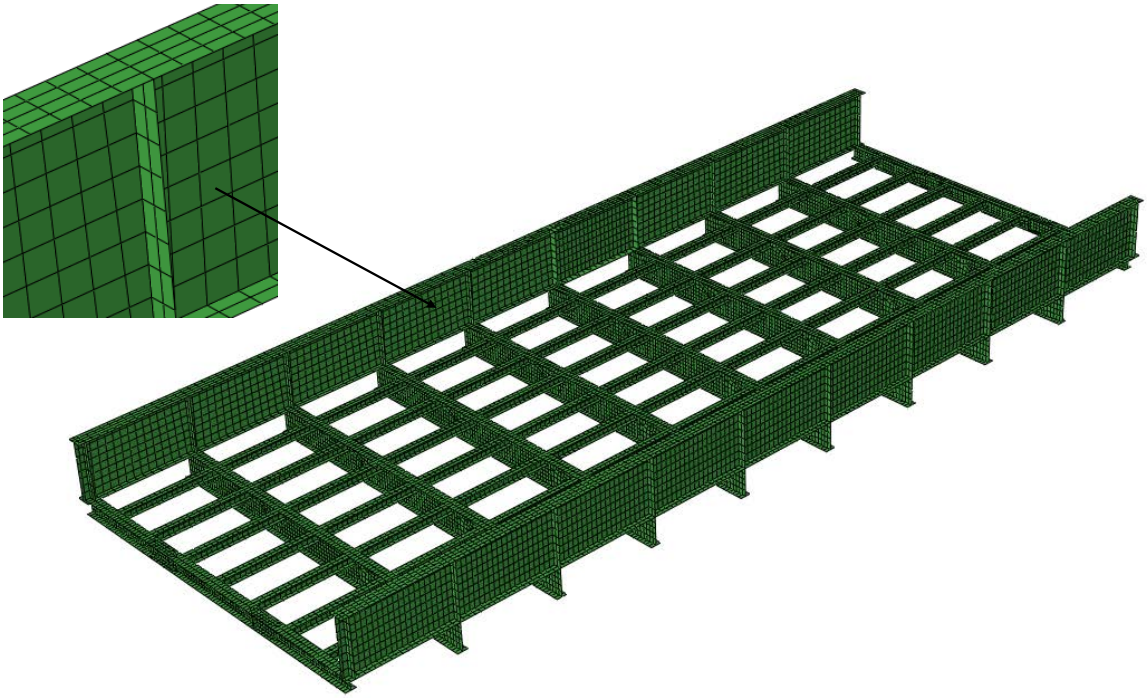


Figure F-30. Mesh details of the steel system.

The reinforced concrete slab was modeled with two types of elements. Specifically, 8-node linear bricks with reduced integration were used to model concrete and 2-node truss elements with linear displacement

were used to model steel reinforcement. Eight solid concrete elements are used through the thickness of the slab with maximum aspect ratio (length of longest edge divided by length of shortest edge) of 5, and corner angles (angle at which two element edges meet) between 40 and 140 degrees. The length of the truss elements used to model the reinforcement were approximately equal to the length of the longest edge of the solid concrete elements. These truss elements were embedded within the solid concrete elements. At the nodes of the embedded truss elements, the translational degrees of freedom are eliminated and the nodal translations were constrained to interpolated values of the nodal translations of the host solid concrete element. The railings were meshed with four elements along its height and two elements across its width, it was attached to the slab by a mesh tie. The reinforcement of the concrete railing was neglected.

F.3.4 Slab-Structural Steel Interaction

As stated, the interaction between the bottom of the concrete slab and the top of the flanges of the steelwork is modeled differently in the two steps described above. In the initial implicit static analysis, when the elements comprising the slab and barriers have $1/1{,}000^{th}$ of the modulus of elasticity to model the "wet" condition, a mesh tie is used to slave the motion of the slab to the motion of the surface comprising the top of the steel work. With this procedure, it is ensured that the slab deformation will conform to the deformation of the steelwork while unrealistic sagging of the slab between supporting elements and tipping of the barrier is prevented.

In the final explicit dynamic analysis, when the stiffness of the elements comprising the slab and barriers has been changed to their final real values, the mesh tie previously used is deleted and replaced by a contact interaction and modeling of shear studs. The normal behavior of the contact interaction is modeled through a penalty stiffness. The penalty stiffness is several orders of magnitude larger than the normal stiffness of the underlying contacting elements and allows a very small penetration so a pressure can be calculated. The tangential behavior of the contact interaction is modeled through an algorithm based on Coulomb friction with a limit on the allowable shear. A friction coefficient of 0.55 and an interfacial shear strength of 0.06 ksi are specified.

The simplified stud model, as described in the Appendix A is used to model composite action between the slab and the steelwork. In the simplified stud model, the shear studs were modeled using connector elements which were used to define the axial and interfacial shear interaction between the shear studs and concrete slab. Connector elements are special purpose elements with zero length. These elements model discrete physical connections between deformable or rigid bodies, and are able to model linear or nonlinear force-displacement behavior in their unconstrained relative motion components.

The recommendations of Appendix A were used to define the shear and tensile behavior of shear studs. The stiffness, strength, and displacement at failure for the different shear stud assemblies included in the model are in Table F-10. In shear, the stud groups follows the shear force-displacement relation proposed by Ollgaard et al. (1971) up to maximum shear displacement of 0.2 inches. In tension, the governing failure mode is concrete break-out, and follows the characteristic triangular response for transversely grouped shear studs which governing failure mode is concrete break-out or shear stud pullout, as described in the recommendations of Appendix A. The tension-shear interaction equation presented in Appendix A is used the combine the effects of shear and tension acting simultaneously on a shear stud group.

Table F-10. Shear stud group properties.

Location	Shear		Tension		
	Strength (kips)	Max. Slip (in)	Initial Stiffness (kip/in)	Strength (kips)	Max. Displ. (in)
End floor beam	108	0.2	3000	11.6	0.04
Interior floor beam	108	0.2	3000	13.9	0.05
Stringer 1	72.2	0.2	2500	7.31	0.03
Stringers 2-7	72.2	0.2	2500	4.64	0.02

F.3.5 Connection Failure Modeling

When a connection is likely to fail before yielding of the member, in addition to the use of mesh ties to attach the components, an additional step may be necessary to capture connection failure. In this particular example, it was not necessary to include to model connection failure as the forces developed in the member did not exceed the capacity of the connections.

F.3.6 Substructure Flexibility Model

Details of the substructure were not available. Therefore, the substructure was not modeled in this case. In the case that such modeling is necessary, please refer to the proposed guide specification in Appendix E.

F.3.7 Loads and Boundary Conditions

Two types of loads were applied in the finite element models: body forces and surface tractions as required by the proposed guide specification in Appendix E. Body forces were applied for component dead loads ("DC" and "DW" per AASHTO designations). These are simply the product of mass, gravity and applicable load factors. Surfaces tractions were applied for traffic live loads ("LL" per AASHTO designation). The traffic live load is based on the HL-93 load model described in the AASHTO LRFD BDS, which is a combination of the truck loads, shown in Figure F-31, and a 0.64 klf load distributed over a width of 10 ft. The current structure does not include any bituminous pavement (i.e.: DW is zero).

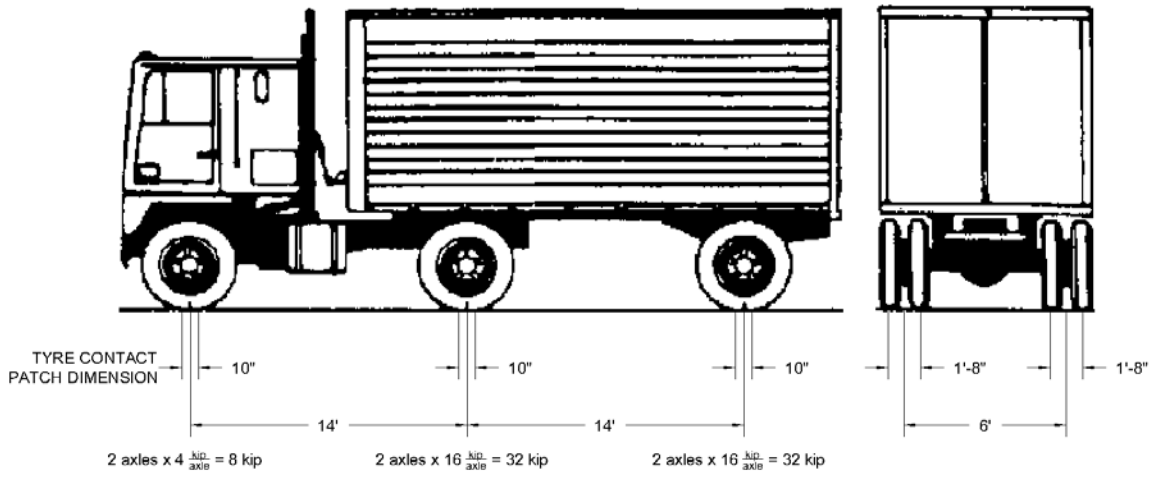


Figure F-31. Truck load components and dimensions of the HL-93 vehicular live load model.

The Redundancy I and Redundancy II loading combinations were used to evaluate the structure in the faulted state. The load factors for these two combinations are as in Table F-11 are based on the provisions in Appendix E for bridges built to Section 12 in the AWS D1.5. The live load (LL) factors are modified by the appropriate multiple presence factors as described in Article 3.6.1.1.2 of the AASHTO LRFD BDS. It must be noted that dynamic amplification factor is equal to 0.4, which is applied to DC and LL in the Redundancy I load combination only. Also, the dynamic load allowance is 0.15 of the truck axle loads, and is only applied in the Redundancy II load combination.

Table F-11. Load factors used for Redundancy I and Redundancy II load combinations.

Load Combination	Load Factors				Notes
	DC	LL	DA_R	IM	
Redundancy I	1.05	0.85	0.40	N. A.	$\beta = 1.5$
Redundancy II	1.05	1.30	N. A.	0.15	$\beta = 1.5$

Longitudinally, the loads are positioned in the most critical positions in both the Redundancy I and Redundancy II load combinations. For the failure scenario considered in the current case (failure of the east girder at mid span as shown in Figure F-21), the most critical position of the truck axle loads which results in the truck facing south with its middle axis positioned at the failure plane, as shown in Figure F-32. The distributed load portion of the HL-93 load is applied along the span of the bridge.

As described in the proposed guide specification in Appendix E, the transverse positioning of the HL-93 live load model differs between the Redundancy I and Redundancy II load combinations, as illustrated in Figure F-32. Since the vehicular loads in the Redundancy I load combination are meant to represent the applied load at the instant in time in which the assumed member failure occurs, the HL-93 vehicular live load model is transversely positioned centered (both the 10 ft loaded width and the truck axle loads) within the marked (striped) lanes, in this case two lanes. Hence, as the bridge is striped for two lanes, there are two load cases for the Redundancy I load combination: two design lanes loaded, or one design lane loaded.

On the other hand, the objective of the Redundancy II load combination is to evaluate the strength of the system after the failure of the primary steel tension member has occurred, so the number of design lanes is established in accordance with Article 3.6.1.1.1 in the AASHTO LRFD BDS, which in this case results in three design lanes with a width of 12 ft. In the Redundancy II load combination, the HL-93 vehicular live load model is transversely positioned (both the 10 ft loaded width and the truck axle loads) to produce extreme force effects within each design lane; however, the truck axle loads are transversely positioned such that the center of any wheel load is not closer than 2 ft from the edge of the design lane. Hence, there are three load cases for the Redundancy II load combination: three design lanes loaded, two design lanes loaded, or one design lane loaded.

Component dead loads were linearly applied in the initial implicit static analysis. Traffic live loads were applied in the final explicit dynamic analysis. Their dynamic effects were minimized by applying them slowly through the use of smooth step, as in the following equation:

$$LR(t) = 6\left(\frac{t}{T}\right)^5 - 15\left(\frac{t}{T}\right)^4 + 10\left(\frac{t}{T}\right)^3$$

where $LR(t)$ is the fraction of load at a load application time t, and T is the duration of the load application. The duration of the load application must be larger than the fundamental period of the structure to minimize oscillatory behavior in the final explicit dynamic analysis.

Regarding prescribed boundary conditions, the structure is simply supported at the ends of the span. Based on the construction plans the bridge was allowed to translate longitudinally and transversely.

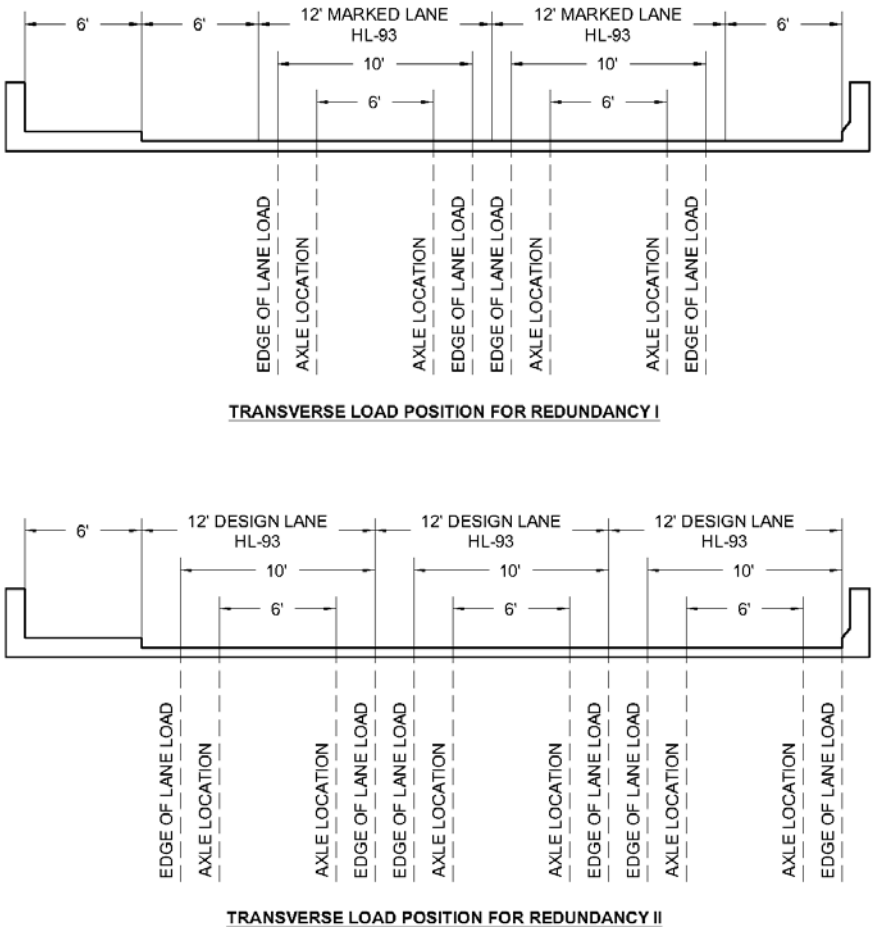


Figure F-32. Transverse position of the HL-93 live load model.

F.3.8 Analysis of Results for Redundancy

Once the analysis is completed the obtained results are evaluated using the requirements described in Article 8 of the proposed guide specification in Appendix E. It was found that the structure did NOT meet the strength and serviceability requirements and is considered non-redundant against failure of the exterior tub girder. Specific details regarding the performance requirements and the results are summarized in Table F-12.

Table F-12. Summary of the redundancy evaluation.

Performance Requirement		Most Critical Load Combination	Result	Acceptable?
Strength Requirements	Strain Primary Steel Members	Redundancy I	Significant yielding (Maximum 31.9%)	**NO**
	Slab Concrete Crushing	Redundancy I	No concrete crushing in the slab	**YES**
Serviceability Requirements	Vertical Deflection	Only Redundancy II DL considered	137.1 in.	**NO**

Notes:
1. In order to complete the evaluation, the displacements and reaction forces calculated at support locations should be used as factored demands to check against the nominal capacity of the supports and substructure members.

F.3.8.1 Minimum Strength Requirements

The strength requirements were not met by the system in the faulted state. The structure in the faulted state did not meet the strength requirements when subjected to factored and amplified dead loads, under either the Redundancy I load combination or the Redundancy II load combination. Therefore, the response of the bridge in the faulted state with traffic loads (LL and IM) was not evaluated. Since the system did not meet all of the strength requirements the east girder must remain a fracture critical member (FCM).

The first set of strength requirements apply to any primary member of the superstructure, which in this case are the tub girders, diaphragms, and concrete slab. These requirements are the following:

- In a component, such as a web or a flange of a primary steel member, the average strain is less than five times the material yield strain.
- In a component, such as a web or a flange of a primary steel member, the average strain is less than 0.01.
- A strain level of 0.05 is not reached anywhere in a primary steel member.
- The combined flexural, torsional and axial force effects computed in primary compression members are below the nominal compressive resistance of the member (these limit states are predicted by the FEA).
- If a compression strain greater than 0.003 is reached in the slab, the portion where that limit is exceeded does not compromise the overall system load carrying capacity.
- The system in the faulted condition is able to support an additional 15% of the factored live load.

In this case, the load combination that resulted in the largest strain in steel primary members was Redundancy I, although the resulting behaviors were similar for both Redundancy I and Redundancy II load combinations. The largest plastic strain was 0.319 and took place on the top flange of the fractured girder at the north support. Also, the average plastic strain of all stringers was greater than 0.01 at the support, and the average plastic strain of floor beams 4-7 was greater than 0.01 at the span between stringer 7 and the fractured girder, as shown in Figure F-33. This is indicated by the elements that are grey in the figure. Thus, the requirements regarding strain in primary steel members are exceeded.

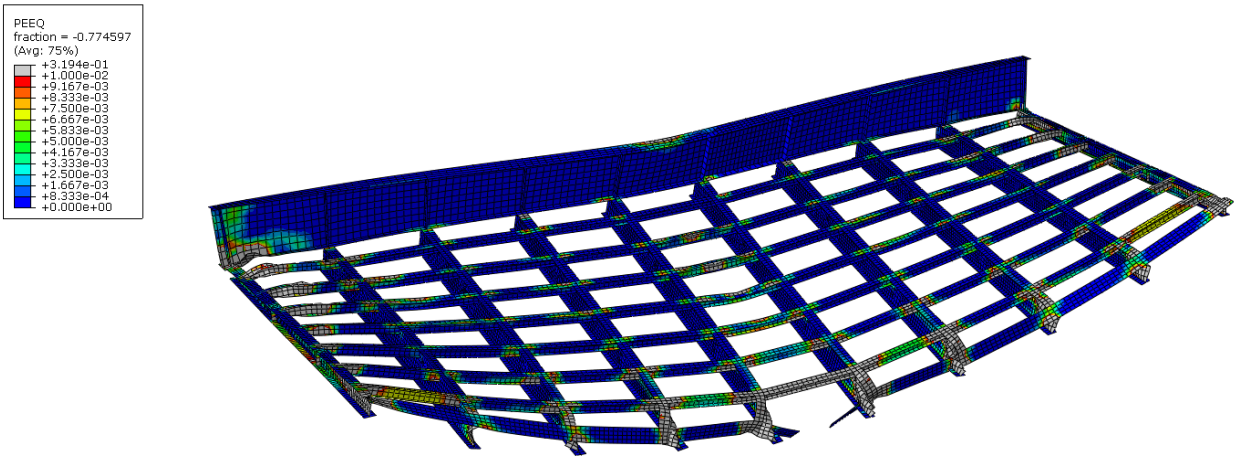


Figure F-33. Equivalent plastic strain in primary steel members.

Regarding the concrete slab, concrete crushing and tension cracking is allowed and expected to take place. However, if the portion of the slab where a total compressive strain of 0.003 has been exceeded is large enough to compromise the overall system load carrying capacity or if significant hinging occurs, the structure should not be considered as sufficiently redundant. In this example, due to the presence of the concrete railings, there was no significant concrete crushing of the slab for both the Redundancy I and Redundancy II load combination. After the fracture occurred, the studs pulled out at two regions: (i) the region where fracture occurred, and (ii) the supports of the west girder (see Figure F-34). Throughout the rest of the slab no obvious stud pullout was observed and, as stated, there was no concrete crushing.

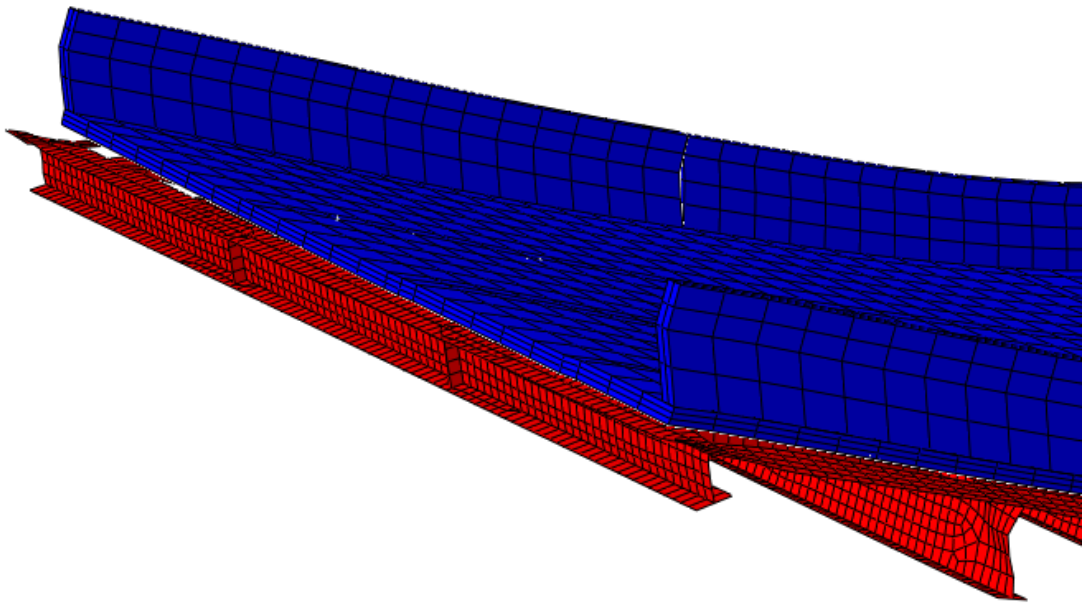


Figure F-34. Pullout of the shear studs at the supports.

Although the substructure is not explicitly included in the finite element model, the reaction forces at support locations are calculated in the analysis. These should be taken as the factored demands that the

substructure must be able to safely sustain, which are summarized in Table F-13. In this example, the Redundancy I load combination resulted in the largest vertical reaction forces. The unfactored nominal capacity of the abutments and the pier need to be checked against these load demands. Similarly the end supports must accommodate the horizontal displacements that are calculated in the analysis at the support locations. In this example, Redundancy I load combination resulted in particularly large transverse displacements which are summarized in Table F-14.

Table F-13. Calculated reaction forces for redundancy evaluation.

Support	Girder	Reaction Force	Result for Redundancy I
South Abutment	West	Vertical	326.3 kips
	East	Vertical	489.5 kips
North Abutment	West	Vertical	327.1 kips
	East	Vertical	489.8 kips

Table F-14. Calculated displacements at support locations for redundancy evaluation.

Support	Girder	Displacement	Result for Redundancy I
South Abutment	West	Longitudinal	0 in.
		Transverse	0 in.
	East	Longitudinal	0 in.
		Transverse	0 in.
North Abutment	West	Longitudinal	1.1 in.
		Transverse	15.6 in.
	East	Longitudinal	4.2 in.
		Transverse	19.2 in.

Notes:
1. Longitudinal direction is along the span direction. Positive longitudinal direction points south.
2. Transverse direction is parallel to span direction. Positive transverse direction points east.

F.3.8.2 Minimum Serviceability Requirements

The only serviceability requirement in the Appendix E is that the increase of deflection after the failure of a primary steel tension member cannot be greater than L/50. This requirement is to be checked in the Redundancy II load combination under factored dead load only. In the current case, the limit is 31.4 inches, which was surpassed since the maximum additional deflection computed in the FEA was 137.1 inches. This is illustrated in Figure F-35.

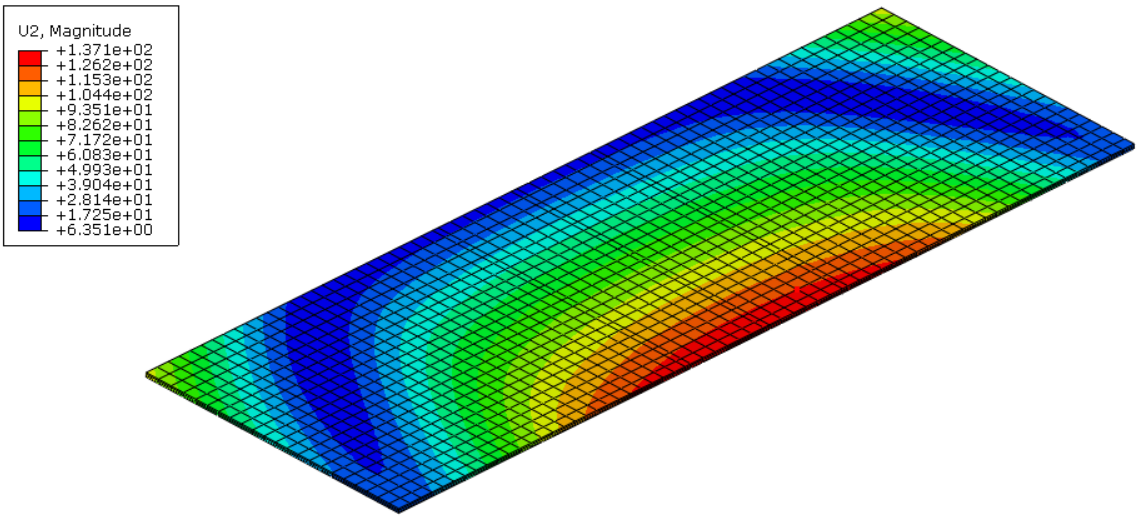


Figure F-35. Deflection after failure of primary steel tension member for the through-girder bridge.

F.3.9 Conclusions

Redundancy of a single span straight through girder bridge after the failure of the east plate girder was analyzed in accordance with the methodology described in the proposed guide specification in Appendix E. Based on the comparison between the obtained results and the minimum performance requirements, the structure is likely to fail and undergo significant serviceability loss after the failure of the east girder. Thus, the east girder must be designated as a fracture critical member (FCM).

F.4 Tied Arch Bridge

The redundancy of a tied arch bridge is analyzed by developing a finite element model in accordance with the methodology described in the proposed guide specification in Appendix E. It is assumed that the structure does not possess any of the detrimental attributes described in the screening criteria and that it is built to Section 12 of the AWS D1.5. In this case, the failing tension member is assumed to be southernmost tie girder. The failure is assumed to take place at a location in which there is a discontinuity in the slab and the stringers are relieved as this presents a worst-case condition since the beneficial participation from the slab is reduced. Figure F-36 shows the location of the failure.

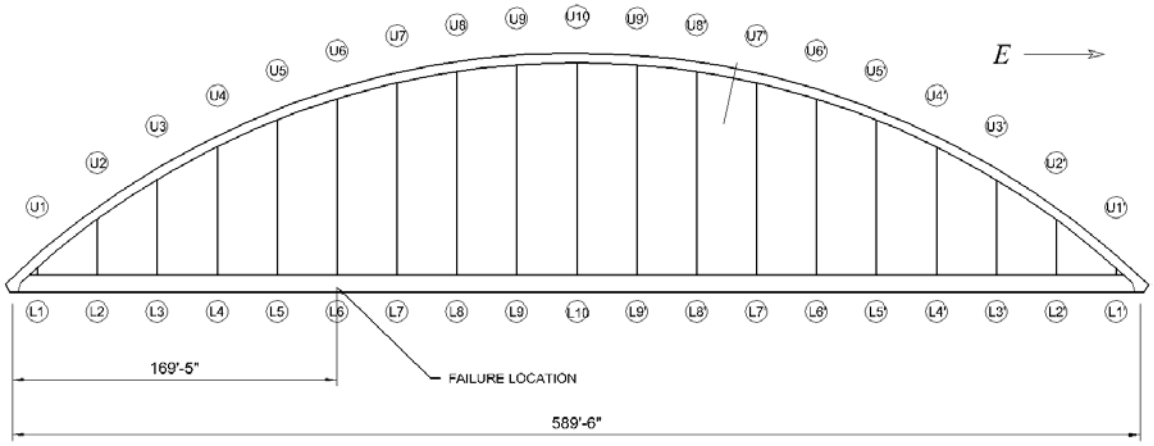
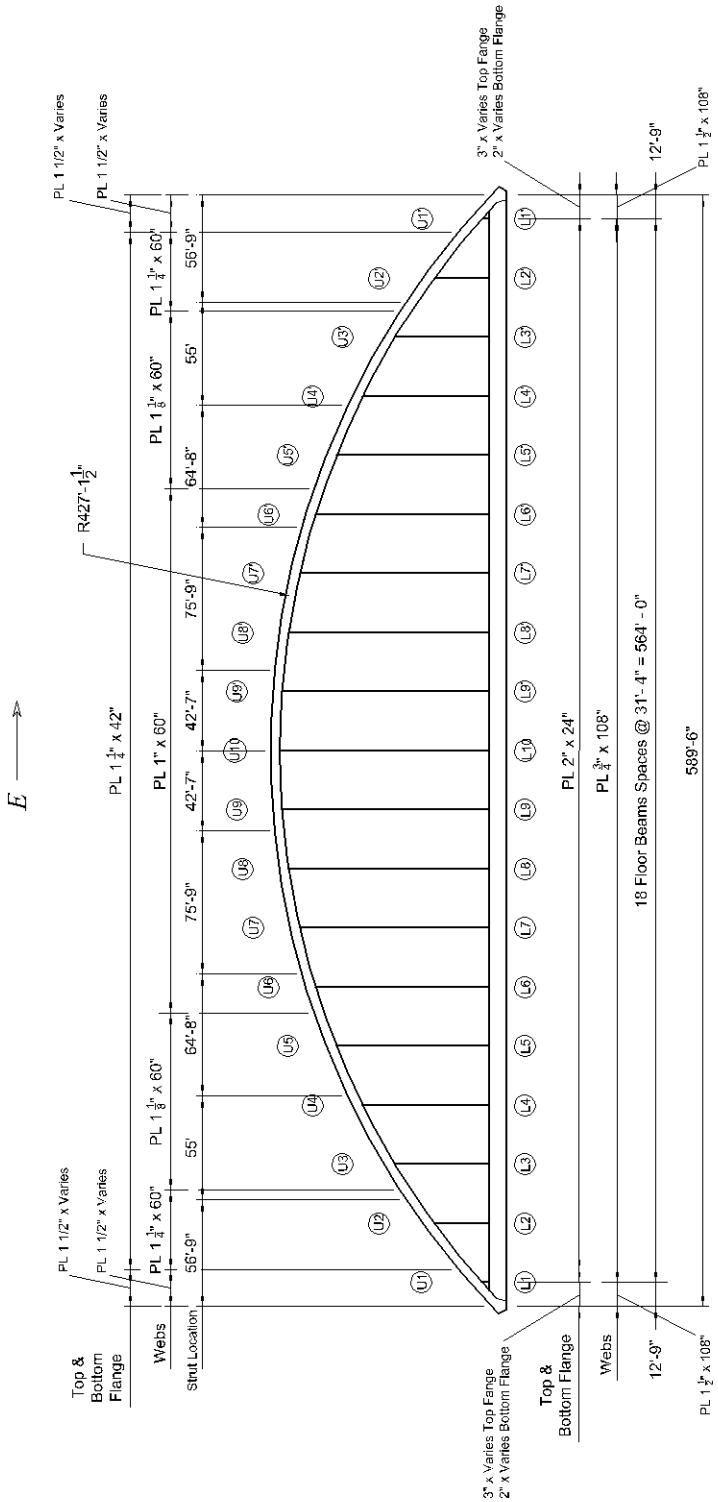


Figure F-36. Steelwork geometry and failure location.

The structure is a single straight span measuring 589.5 feet. Two curved boxes compose each of the arches. The tie is comprised of an I shaped plate girder. The box section of each arch has 42" wide flanges and 60" deep webs. The tie girders have 108" deep webs and 24" wide flanges, as shown in Figure F-37. These dimensions change at the knuckles, which geometry is depicted in Figure F-40. The floor system is comprised of floor beams which provide support to the stringers. The typical cross-section of the lower assembly is shown in Figure F-39. Stability of the lower steel assembly is provided by lateral bracing and diaphragms connecting the stringers. The framing of the lower assembly is shown in Figure F-38. Lateral bracing of the arches is provided by struts as shown in Figure F-38. Location of the hangers is as shown in Figure F-37 and a detail of the hanger anchorages is shown in Figure F-41.



1224
1225 ***Figure F-37. Steel assembly of tied arch bridge (side view).***

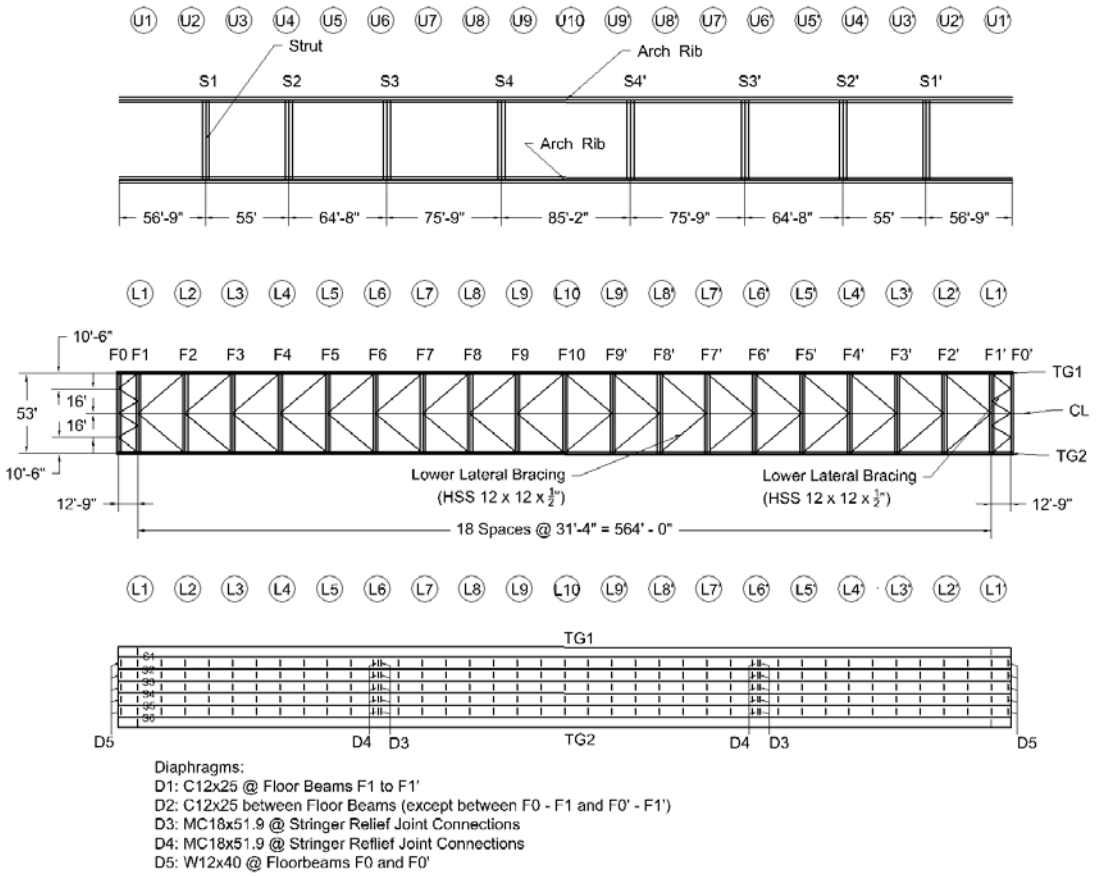


Figure F-38. Steel assembly of tied arch bridge (top views).

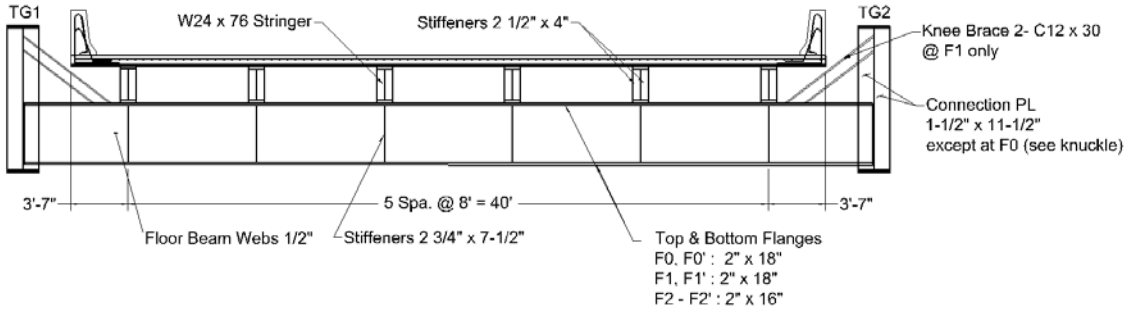
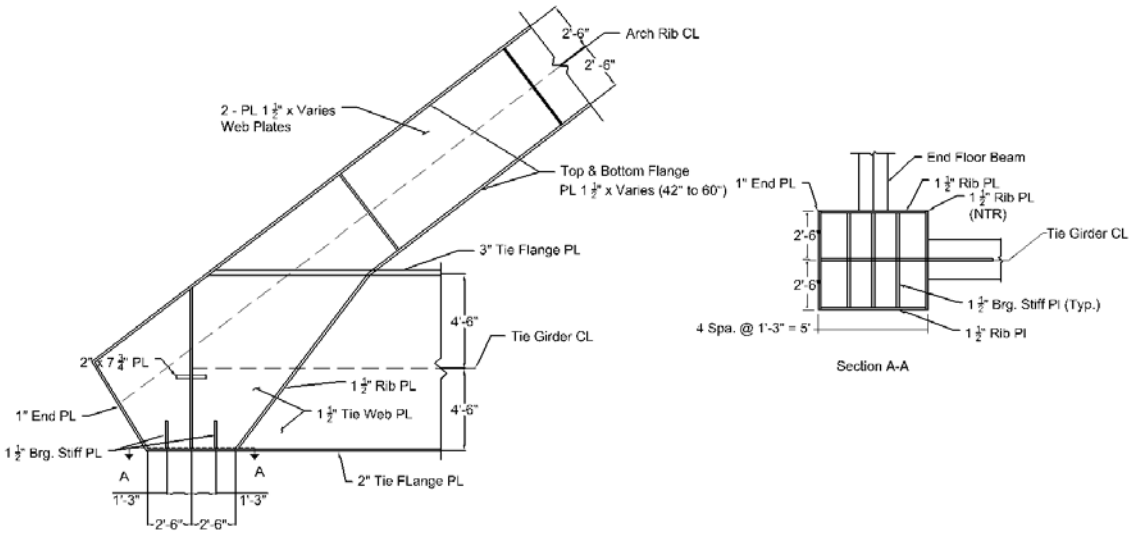
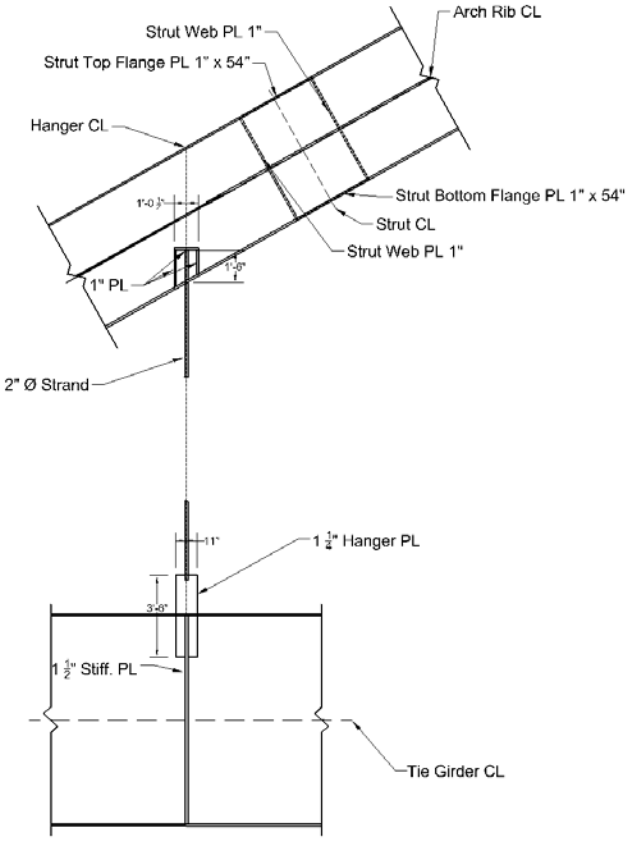


Figure F-39. Cross section of tied arch bridge, lower assembly.



1231
1232 ***Figure F-40. Detail of knuckle of tied arch bridge.***



1233
1234 ***Figure F-41. Detail of hanger anchorage of tied arch bridge.***

The reinforced concrete slab is 44 feet wide between interior edges of concrete barriers (approximately 30 feet wide between the outer exterior edges of concrete barriers) and is fully composite with the stringers' top flanges through shear studs. The end supports at the piers are pin supports, in which translations are fixed but rotations are free. The tie girders are fabricated in ASTM A709 HPS 70W steel, the suspenders' strand is ASTM A586 Grade 1 Class A and the rest of the steel components are fabricated ASTM A709 Grade 50 steel. All concrete has a minimum specified compressive strength of 3.5 ksi and all rebar has 60 ksi yield strength. In the analysis of this structure, the longitudinal and transverse slope of the slab will be neglected (i.e., the slab is assumed flat). Figure F-42 shows the reinforcement schedule of the slab and Figure F-43 shows the reinforcement details.

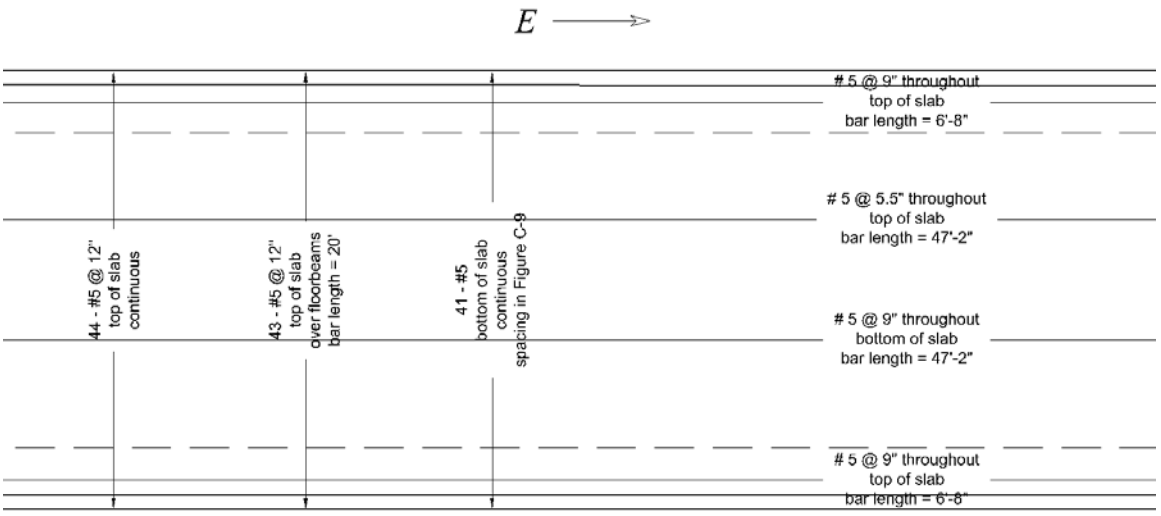
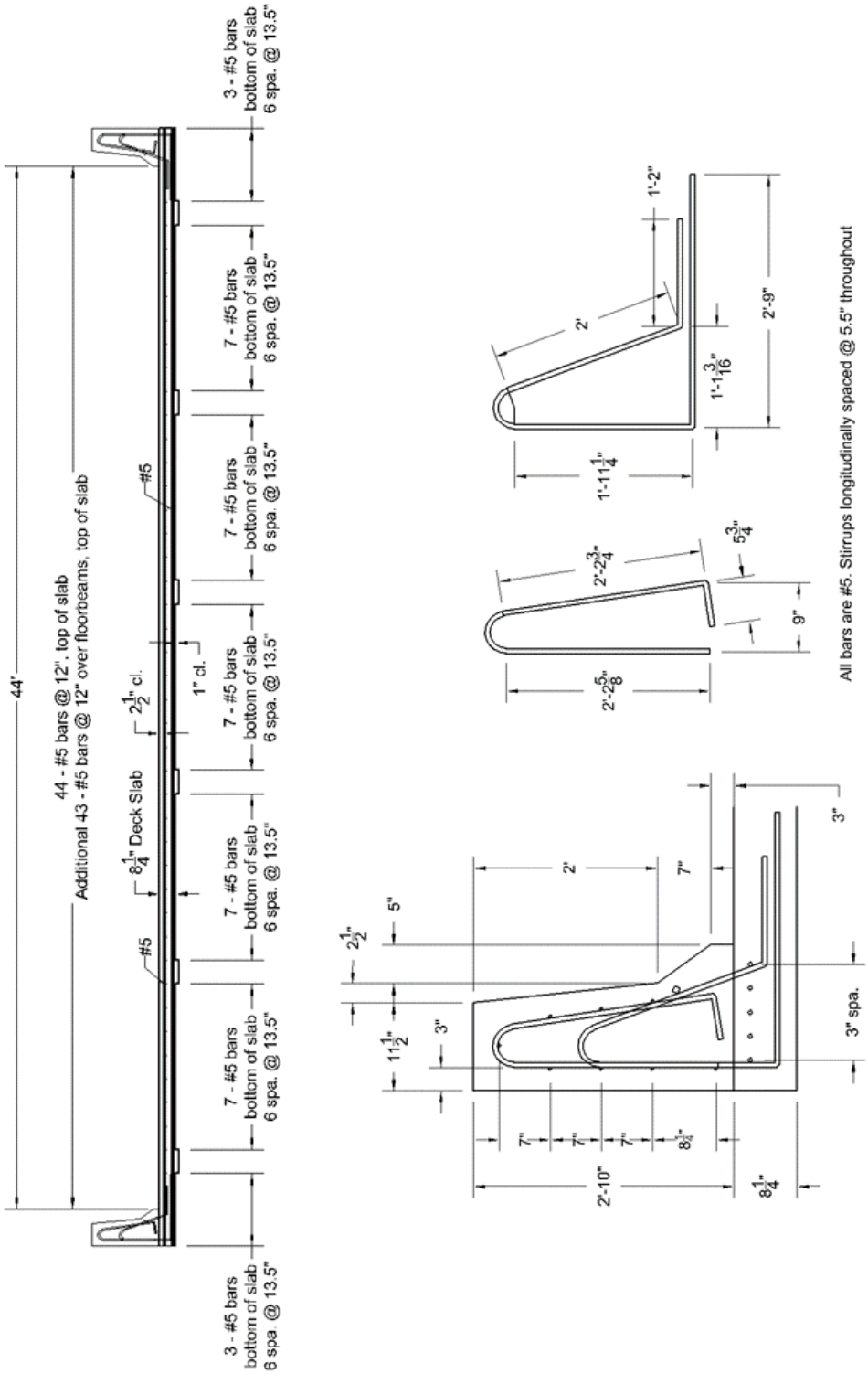


Figure F-42. Schedule of reinforcement of the slab.



1247
1248 ***Figure F-43. Cross section of slab with reinforcement spacing and barrier details.***

F.4.1 Analysis Procedure

The analysis is performed to establish if the system demonstrates acceptable performance in the faulted condition. In the example, the term "faulted condition" specifically refers to the case in which a primary steel tension member is assumed to have failed. For this analysis, load factors for both dead and live load are applied as described in the proposed guide specification in Appendix E. In this example, the described analysis procedure is composed of an initial implicit static analysis and a final explicit dynamic analysis, into which the results from the initial implicit static analysis are imported. While it is not mandatory for the Engineer to follow these particular steps, it has been found that this procedure optimizes the computational time required.

F.4.1.1 Initial Implicit Static Analysis

Implicit static analysis was utilized to calculate the state of the structure prior to hardening of the concrete in the slab. An implicit static analysis was used for the initial steps because, although non-linearity is considered in the analysis, the bridge behavior is linear and inertial effects can be neglected as the bridge is in the undamaged condition. As the slab does not carry any load and does not contribute to the stiffness of the system before concrete hardening, two modifications are required in the finite element analysis during this initial implicit static analysis as follows:

- A very low stiffness is specified for the elements composing the slab, i.e., the elements modeling concrete and rebar. A reduced stiffness of 1/1,000 of the respective modulus of elasticity of each material was used. This is done so the load carried by the slab and rebar have negligible contribution to the stiffness of the system. No modifications to the stiffness should be applied to the steelwork.
- Instead of defining contact interaction between the slab and the steelwork, a mesh tie was specified. The nodal displacements of the concrete slab elements are tied to the displacements of the top flanges of girders, floor beams, and stringers which occur due to dead load. As a result, the slab deforms with the steelwork and does not 'sag' between the girders, floor beams, and stringers.

It is worth noting that the remainder of the finite element modeling is identical between the initial implicit static analysis and the final explicit dynamic analysis. The specific steps in the initial implicit static analysis are described as follows:

1. Apply load due to self-weight of the structural steel components as a body force.
2. Apply load due to self-weight of the wet slab components as a body force.
3. The system is then fixed in terms of position, that is, the displacement degrees of freedom are not allowed to change.
4. The elements composing the slab (elements modeling rebar and concrete) are then deactivated.
5. The elements composing the slab are then reactivated. During this reactivation the strain in the elements composing the slab is reset to zero.

Steps 3 through 5 are necessary since even though very low stiffness was specified for the slab, these elements do undergo strain. Setting the strains to zero eliminates "locked in" artificial stresses in later steps.

F.4.1.2 Final Explicit Dynamic Analysis

As contact algorithms, softening material behavior, and non-linear geometry are required to be part of the finite element analysis, implicit solution procedures present unavoidable convergence problems in most FEA solvers. In order to calculate the capacity of the bridge after sudden failure of a tension component, a dynamic explicit analysis needs to be carried out. Therefore, the results obtained from the initial implicit static analysis are imported into the final explicit dynamic analysis. In other words, the state of the system (stresses, strains, displacements and forces) at the beginning of the final explicit dynamic analysis is defined by the state of the system computed at the end of the initial implicit static analysis.

As previously stated, during the initial implicit static analysis, the slab was modeled with largely reduced stiffness to reflect that it is not hardened and a mesh tie constraint was used to assure that the slab deformed with the steelwork. This approach also prevents excessive sag of the soft slab. After the state of the system is imported, the following changes are made to capture the response of the structure after the concrete has hardened:

- The modulus of elasticity of the concrete and rebar elements in the slab is changed to their final actual values. It is noted that no modifications need to be applied for the steelwork.
- The mesh tie constraint between the slab concrete elements and the top flanges of the steelwork is replaced by a frictional contact interaction. Additionally, since the structure under analysis is composite, elements which accurately model the behavior of shear studs are added.

All of the body forces applied during the initial implicit static analysis (i.e., the dead load of the structure) are maintained throughout the final explicit dynamic analysis.

To evaluate the capacity of the structure in the faulted state, the following steps were carried out in the final explicit dynamic analysis:

6. The stiffness of the elements located at the fracture location under consideration were slowly reduced. The stiffness was slowly reduced in order to minimize any dynamic effects. It is noted that the actual fracture and subsequent vibration of the structure is not modeled. This dynamic effect is accounted for using the DA_R factor as discussed before. If dynamic effects are found to be significant even if the stiffness is slowly reduced, the system must be allowed to oscillate until these effects are dampened.
7. Factored loads due to traffic are applied as surface tractions. For the Redundancy I load combination all loads are amplified by DA_R, for the Redundancy II load combination the dynamic load allowance (IM) is applied. These loads were applied very slowly to minimize any dynamic effects, as well. If dynamic effects are significant, the system must be allowed to oscillate until these effects are dampened.
8. An additional 15% of live load is gradually applied.

F.4.2 Material Models

Five material models are needed in the finite element model. Four of those are utilized to model different steel types, and one is utilized to model the response of concrete. For the development of the steel material models, it is necessary to know the yield strength and ultimate strength of each steel type. In this example, since no test values are known to the Engineer, nominal values specified in the respective standards are utilized. These are summarized in Table F-15. A mass density of 0.494 kcf was specified for all steel types.

Table F-15. Material properties for steel material models.

Material	Nominal Yield Strength	Nominal Ultimate Strength	Standard
ASTM A709 HPS 70W	70 ksi	100 ksi	ASTM A709/A709M
ASTM A709 Gr. 50	50 ksi	65 ksi	ASTM A709/A709M
Grade 60 Rebar	60 ksi	90 ksi	ASTM A615/A615M
ASTM A586 Gr. 1 Cl. A	150 ksi	220 ksi	ASTM A586/A586M

The stress-strain relation for all steel components will follow an initial linear elastic steel with a Young's modulus of 29,000 ksi and Poisson's ratio of 0.3. Once the nominal yield strength is reached the stress-strain relation is defined by Von Mises (J2) plasticity with kinematic linear hardening, until the nominal ultimate strength is reached at a total strain of 0.05. Once the nominal ultimate strength or a total strain of

0.05 is reached, the material is assumed to fail. Figure F-44 shows the uniaxial material response for the steel employed in this finite element model with the 'X' denoting the stress at the failure strain of 0.05.

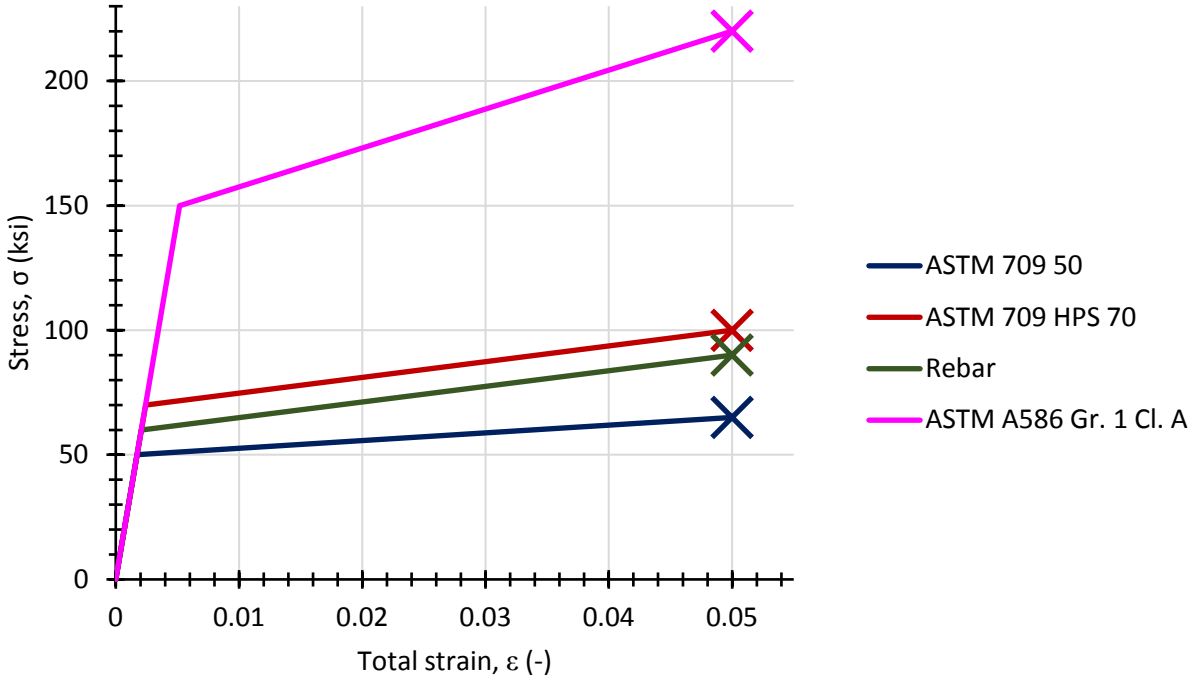


Figure F-44. Stress-strain curves of steel material models.

The material model used in concrete is defined entirely by the specified compressive strength, which in this case is 3.5 ksi. This quantity is also used to calculate the tensile strength, the total strain at compressive strength, ε_c, and the material parameter n. Table F-16 summarizes the calculation of these values. A mass density of 0.150 kcf was specified for concrete.

Table F-16. Material properties for concrete material model.

Quantity	Symbol	Equation	Result
Young's modulus	E_c	$E_c = 33{,}000 w_c^{1.5}\sqrt{f_c'} \leq 1{,}802.5\sqrt{f_c'}$	3,370 ksi
Tensile strength	f_t	$f_t = 0.158(f_c')^{\frac{2}{3}}\ for\ f_c' \leq 7.25ksi$ $f_t = 0.307\ln(f_c' + 2.61) - 0.114\ for\ f_c' > 7.25ksi$	0.364 ksi
Fracture energy	G_t	$5.9 \cdot 10^{-4}(f_c' + 1.16)^{0.18}$	$7.78{\cdot}10^{-4}$ kip/in
Total strain at compressive strength	ε_c	$\varepsilon_c = 0.00124\sqrt[4]{f_c'}$	0.00170
Material parameter	n	$n = 0.4f_c' + 1.0$	2.4

The concrete material model is initially linear elastic, defined by a Young's modulus of 3,600 ksi and Poisson's ratio of 0.2, followed by concrete damage plasticity. In tension, once the material reaches its tensile strength, set at 0.364 ksi in this case, a tensile stress-displacement relation characterized by a fracture energy, G_t, of $7.28 \cdot 10^{-4}$ kip-in is followed This fracture energy is applied through a bi-linear tensile stress-displacement relation as shown in Figure F-45, and defined by the following quantities:

$$f_{t1} = \frac{f_t}{5} = 0.0728\ ksi$$

$$\delta_t = \frac{5G_t}{f_t} = 0.0107$$

$$\delta_{t1} = \frac{G_t}{f_t} = 0.00214$$

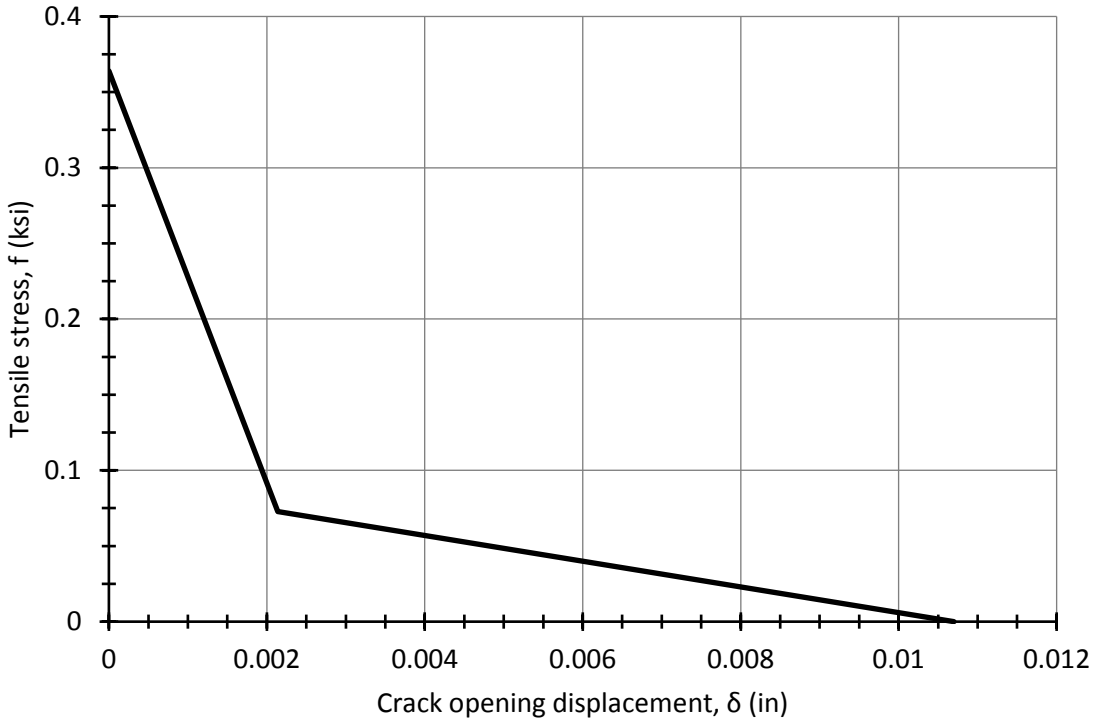


Figure F-45. Tensile stress-crack opening displacement curve for concrete material model.

In compression the material follows the following stress-strain relations:

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

$$\varepsilon_{plastic} = \varepsilon - \frac{f(\varepsilon)}{E_c}$$

where ε is total (elastic + plastic) strain, $f(\varepsilon)$ is the compressive stress at a given total strain, f_c' is the specified compressive strength, ε_c is the total strain at compressive strength, n is a material parameter, $\varepsilon_{plastic}$ is the plastic strain and E_c is the concrete Young's modulus. Figure F-46 shows the resulting compressive stress-strain relation.

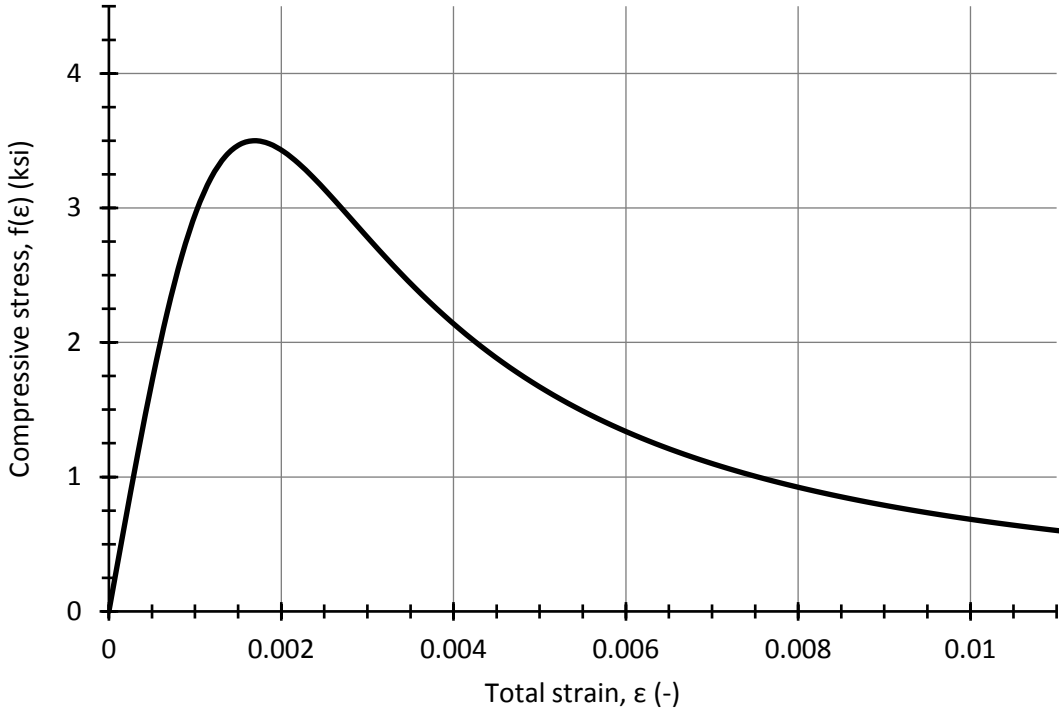


Figure F-46. Compressive stress-strain curve for concrete material model.

F.4.3 Geometries, Meshes and Constraints

The geometry of the structure is based on available design plans and is composed of the following components that must be explicitly modeled:

1. A tied arch system composed of:
    a. Two arch ribs.
    b. Two tie girders.
    c. A set of eight struts.
    d. Hanger anchorage system.
    e. Arch knuckles.
2. Floor beam system composed of 19 floor beams connected to the tie girders.
3. Stringer system composed 6 stringers supported by the floor beam system.
4. A hanger system composed of 76 hangers.
5. A lower assembly bracing system composed of:
    a. W-bracing between floor beam F0 and F1, and F1' and F0', connected to the floor beams and tie girders.
    b. V-bracing among the rest of floor beams, connected to the floor beams and tie girders.
    c. Diaphragms among stingers.
6. A reinforced concrete slab with concrete barriers.

When generating the finite element model, splices, holes, access hatches, etc. are neglected. The structure is assumed to be flat in the vertical plane, in other words, camber and superelevation are ignored. Figure F-47 shows the assembly of all bridge components and Figure F-48 shows a detail of the underside.

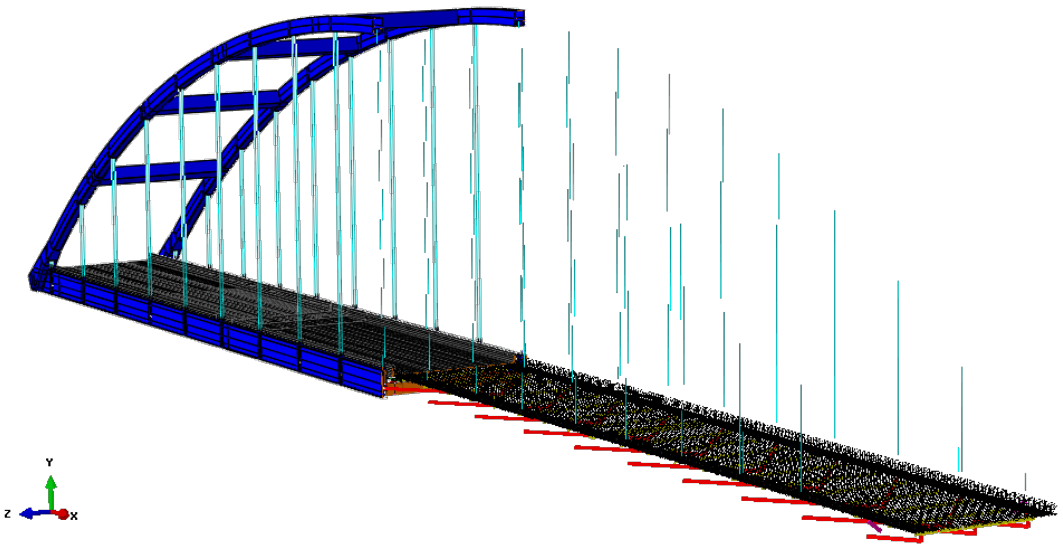


Figure F-47. Tied arch bridge geometries. Concrete slab and barriers (grey), tied arch system (blue), lateral bracing (red), diaphragms (yellow), floor beams (magenta), hangers (cyan), stringers (green), slab reinforcement (black).

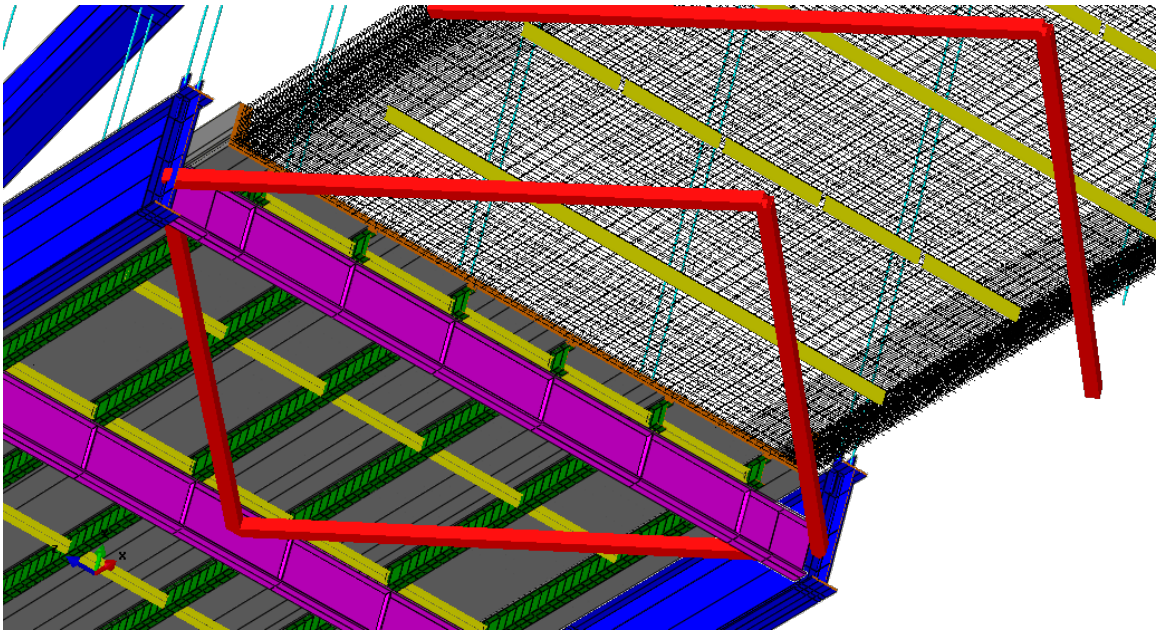


Figure F-48. Detail of the underside of the tied arch bridge geometries. Concrete slab and barriers (grey), tied arch system (blue), lateral bracing (red), diaphragms (yellow), floor beams (magenta), hangers (cyan), stringers (green), slab reinforcement (black).

The tied arch system (arch ribs, tie girders, struts, hanger anchorages and knuckles), floor beams, stringers and diaphragms are modeled with 4-node shell elements with reduced integration. A minimum of four elements are used along flange widths and along web heights. Stiffeners and connection plates are modeled as shell elements as well. The arch ribs, tie girders, struts, hanger anchorages and knuckles

constitute a single geometry. The maximum aspect ratio was kept below five and corner angles were kept between 60 and 120 degrees. Figure F-49 shows two details of the mesh employed to model the tied arch system. All lower assembly lateral bracing (W-bracing, V-bracing and diaphragms) and hangers are modeled with 2-node linear shear-flexible (Timoshenko) beam elements. A minimum of three (3) elements are used along the length of the elements. Mesh ties, which are constraints that slave the motion of a surface or node set to the motion of a master surface or node set, are utilized to connect the all steel components.

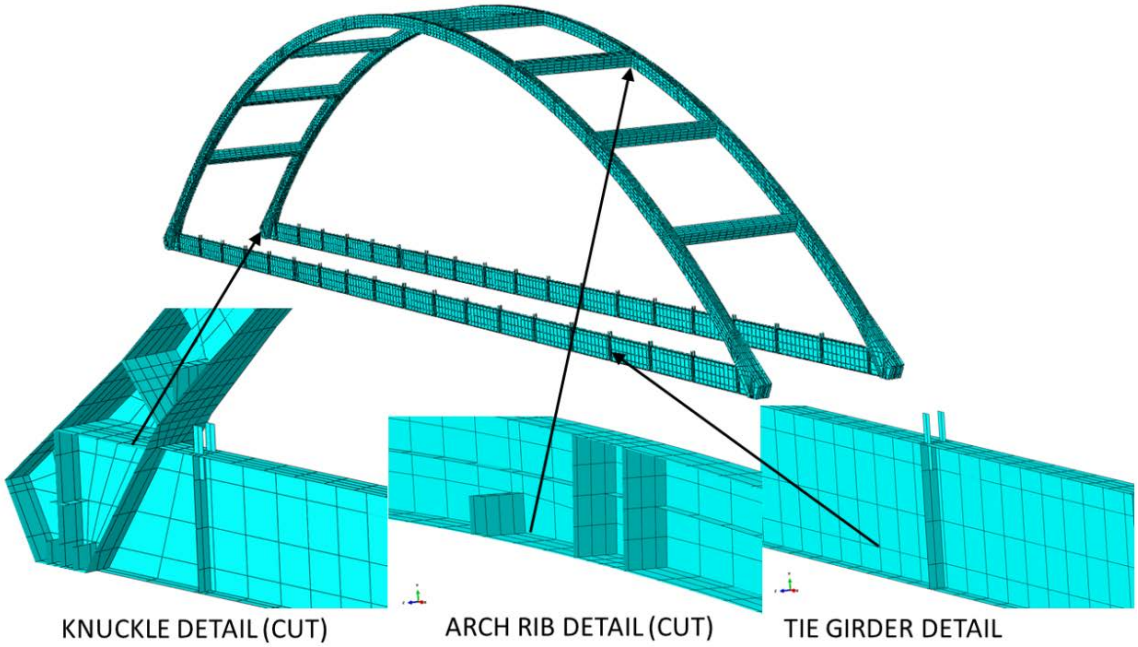


Figure F-49. Mesh details of the tied arch system.

The slab is modeled with two types of elements. Specifically, 8-node linear bricks with reduced integration are used to model concrete and 2-node truss elements with linear displacement to model steel reinforcement. Eight solid concrete elements are used through the thickness of the slab with maximum aspect ratio (length of longest edge divided by length of shortest edge) of 5, and corner angles (angle at which two element edges meet) between 40 and 140 degrees. The length of the truss elements used to model the reinforcement were approximately equal to the length of the longest edge of the solid concrete elements. These truss elements are embedded within the solid concrete elements. At the nodes of the embedded truss elements, the translational degrees of freedom are eliminated and the nodal translations were constrained to interpolated values of the nodal translations of the host solid concrete element. The reinforcement of the concrete barrier was neglected and it was meshed with four elements along its height and two elements across its width, it was attached to the slab by a mesh tie. Figure F-50 shows the concrete slab with the embedded truss elements and a detail of the mesh used for the concrete barrier and slab.

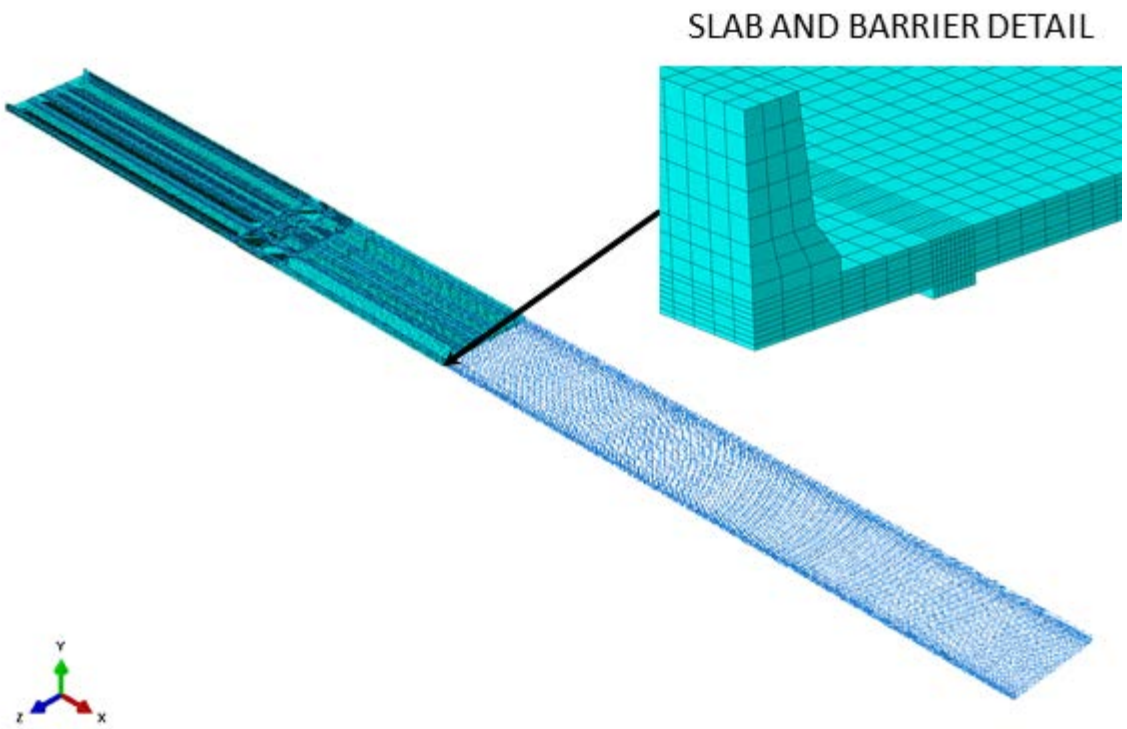


Figure F-50. Mesh details of the reinforced concrete slab and barriers.

F.4.4 Slab-Structural Steel Interaction

As stated, the interaction between the bottom of the concrete slab and the top of the flanges of the steelwork is modeled differently in the two steps described above. In the initial implicit static analysis, when the elements comprising the slab and barriers have 1/1,000th of the modulus of elasticity to model the "wet" condition, a mesh tie is used to slave the motion of the slab to the motion of the surface comprising the top of the steel work. With this procedure, it is ensured that the slab deformation will conform to the deformation of the steelwork while unrealistic sagging of the slab between supporting elements and tipping of the barrier is prevented.

In the final explicit dynamic analysis, when the stiffness of the elements comprising the slab and barriers has been changed to their final real values, the mesh tie previously used is deleted and replaced by a contact interaction and modeling of shear studs. The normal behavior of the contact interaction is modeled through a penalty stiffness. The penalty stiffness is several orders of magnitude larger than the normal stiffness of the underlying contacting elements and allows a very small penetration so a pressure can be calculated. The tangential behavior of the contact interaction is modeled through an algorithm based on Coulomb friction with a limit on the allowable shear. A friction coefficient of 0.55 and an interfacial shear strength of 0.06 ksi are specified.

The simplified stud model, as described in the Appendix A is used to model composite action between the slab and the steelwork. In the simplified stud model, the shear studs were modeled using connector elements which were used to define the axial and interfacial shear interaction between the shear studs and concrete slab. Connector elements are special purpose elements with zero length. These elements model discrete physical connections between deformable or rigid bodies, and are able to model linear or nonlinear force-displacement behavior in their unconstrained relative motion components.

The recommendations of Appendix A were used to define the shear and tensile behavior of shear studs. The shear stud group is composed of three transversely grouped studs which shear strength is 53.0 kips,

following the shear force-displacement relation proposed by Ollgaard et al. (1971) up to maximum shear displacement of 0.2 inches. In tension, the governing failure mode is concrete break-out, resulting in a initial stiffness of 2500 kip/in, and tensile strength of 9.23 kips, and a maximum tensile displacement at failure of 0.04 inches. The tensile behavior follows the characteristic triangular response for transversely grouped shear studs which governing failure mode is concrete break-out or shear stud pullout, as described in the recommendations of Appendix A. The tension-shear interaction equation presented in Appendix A is used the combine the effects of shear and tension acting simultaneously on a shear stud group.

F.4.5 Stringer Relief Joint Modeling

The stringers of the tied arch bridge have relief joints next to floor beams F6 and F6' (see Figure F-38), at those locations the slab is discontinued as well. Figure F-51 shows the sketch of the relief joint. At the discontinuity, the slab may provide inferior load-path redundancy to other location in which the slab is continuous, and the relief joint may not provide sufficient capacity to make up for the loss of load-path redundancy in the slab. Hence, in this example, it was chosen as the critical location.

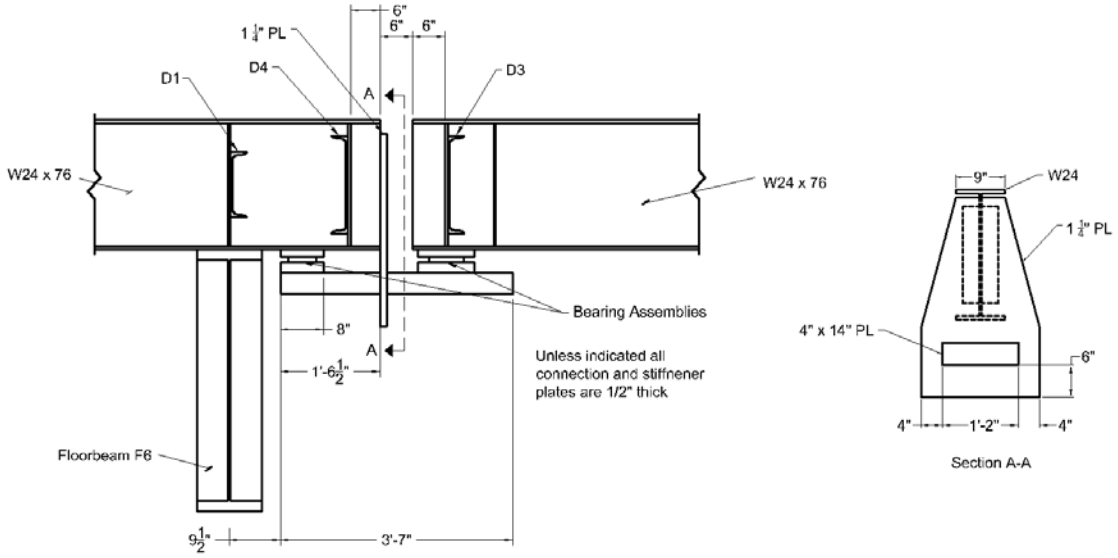


Figure F-51. Sketch of the stringer relief joint.

The relief joint has complex behavior for which is not easy to obtain force/moment-displacement/rotation relations. In this example, all relief joints were explicitly modeled in the finite element analysis. The geometries of the joints were explicit included in the finite element model, except the bearing assemblies. The bearing assemblies were modeled with a kinematic constraint, which equals the translations between two points. Figure F-52 shows a detail of the relief joint as modeled in the finite element analysis, along with the location of the kinematic constrain. The discontinuity in the slab was included in the finite element model as well, as shown in Figure F-53

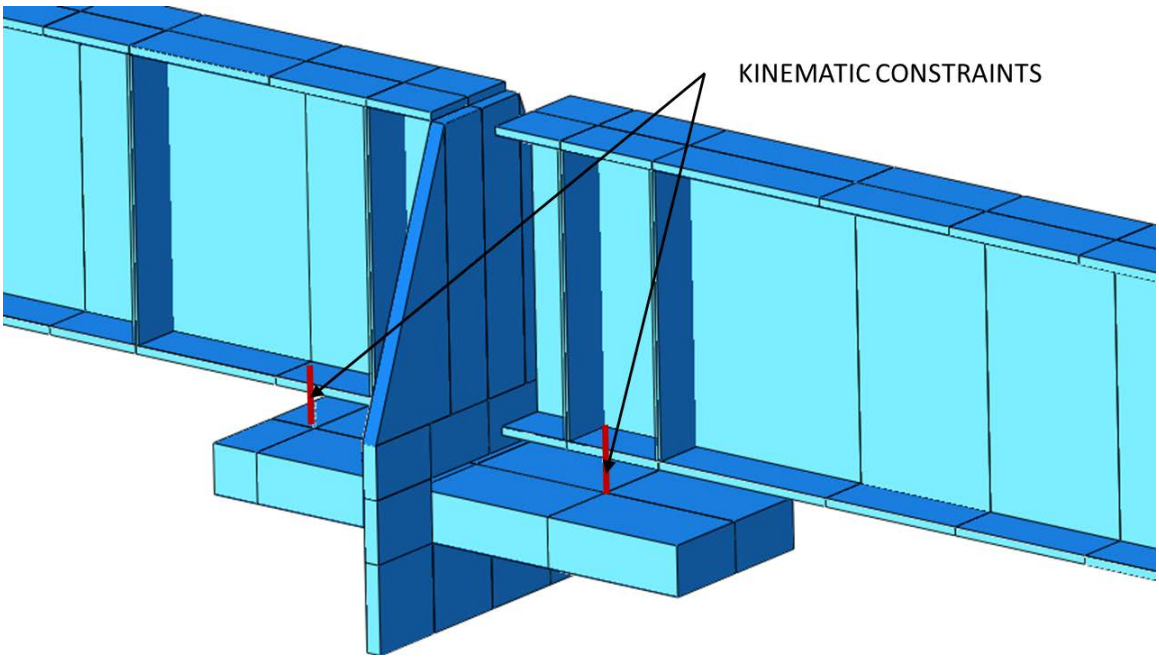


Figure F-52. Detail of stringer relief joint geometry in finite element models.

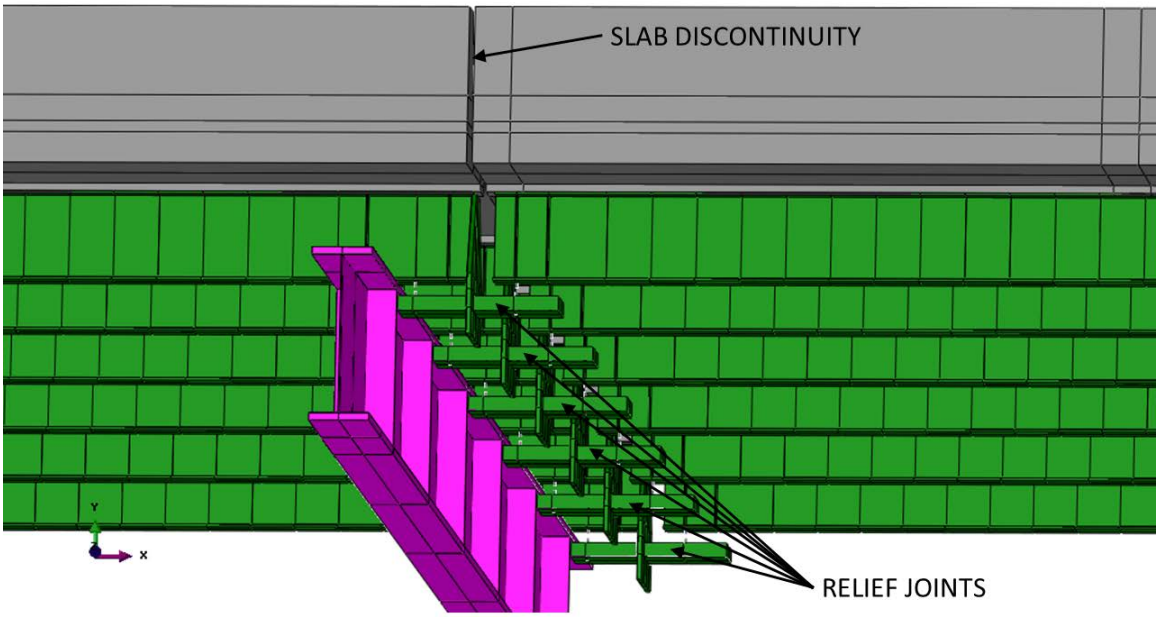


Figure F-53. Detail of discontinuity in slab and stringer relief joints. Slabs (grey), stringers (green) and floor beam (magenta) are shown.

F.4.6 Substructure Flexibility Model

In order to account for longitudinal and transverse flexibility of the substructure, connector elements were utilized. These elements allow for the definition of coupled force-deformation relations. The type that was determined to best capture the intended behavior was a Cartesian connector. These elements provide a connection between two nodes where the change in position is measured in three directions local

to the connection. One of the nodes is fixed (or connected to ground) and the other node is the support point in the superstructure. The connector element is rigid in the vertical direction, and has a coupled linear elastic relation in the two horizontal directions (longitudinal and transverse).

In the current case, the structure is assumed to be pinned at the end spans; as a result, the vertical translation at the bearing location in the knuckles will be set to zero, while the horizontal stiffness of the pier will be incorporated through connector elements. In order to obtain the coupled elastic force-displacement relation, a simple finite element analysis of the pier is conducted. The geometry of the pier is drawn according to the design plans, as shown in Figure F-54 and meshed with 8-node linear bricks with reduced integration as shown in Figure F-55. The pier was modeled as linear elastic with modulus of elasticity of 1,800 ksi (in order to account for possible cracking due to combined compression and bending), Poisson's ratio of 0.2 and a density of 0.150 kcf. The base of the pier bears on a rigid mat, prohibiting sliding but allowing uplift as shown in Figure F-14.

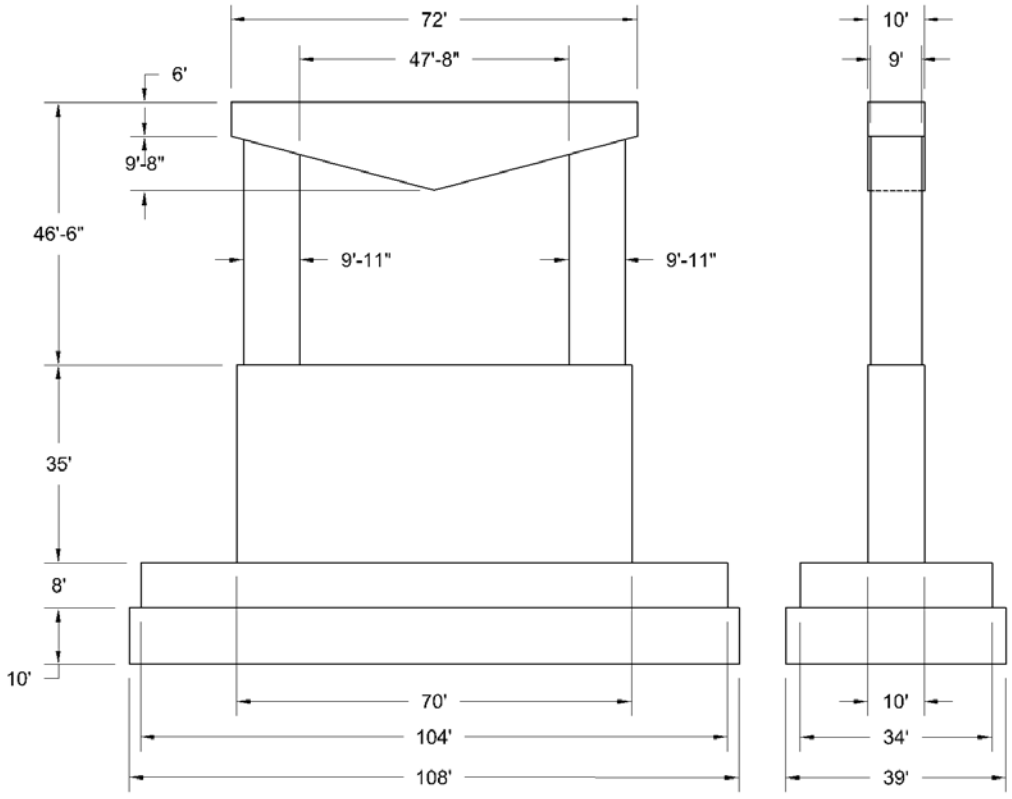


Figure F-54. Geometry of the pier support.

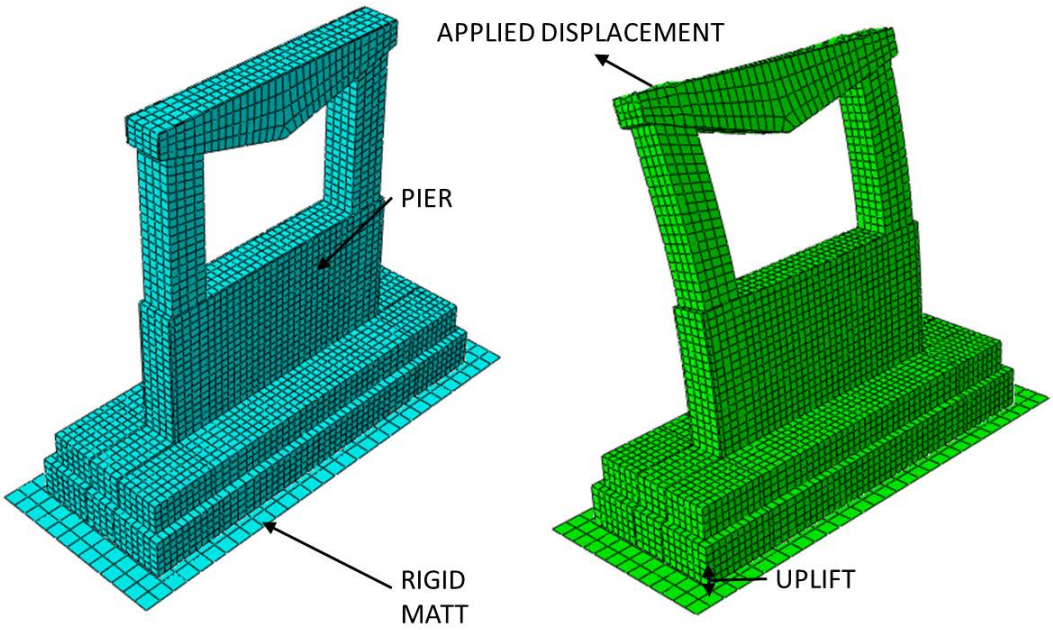


Figure F-55. Detail of mesh geometry used (left) and deformed configuration after application of displacement (right, displacements scale by a factor of 50).

During the first analysis step, dead loads are applied. These are due to (1) the self-weight of the pier, applied as a body force, and (2) bearing of the superstructure on the pier, which were calculated to be 1750 kips at each bearing location. These are applied as surface tractions over a 60" by 60" patch, the size of the patch is based on the size of the bearings. Once the first step is completed, displacements are applied at the bearing locations so the reaction forces can be calculated. In this case, displacements of 10 inch were applied in the longitudinal and transverse directions (positive and negative signs) so the reaction forces and described in Table F-3 were obtained.

Table F-17. Reaction forces and displacements at support points.

Pier Support			
$U_{TRANSVERSE}$	$U_{LONGIT.}$	$R_{TRANSVERSE}$	$R_{LONGIT.}$
in	in	kips	kips
10.0	-7.25E-10	222000	-
-10.0	7.03E-10	-222000	-
-6.89E-10	10.0	-	38400
7.02E-10	-10.0	-	-38400

These are used to build the force deformation relations shown hereof, which are incorporated as the properties of the connector element in the global model to model the flexibility of the pier:

$$\begin{bmatrix} F_{TRANSVERSE} \\ F_{LONGITUDINAL} \end{bmatrix}_{SUPPORT} = \begin{bmatrix} 22{,}200 & 0 \\ 0 & 3{,}840 \end{bmatrix} \frac{kips}{in} \begin{bmatrix} U_{TRANSVERSE} \\ U_{LONGITUDINAL} \end{bmatrix}_{SUPPORT}$$

F.4.7 Loads and Boundary Conditions

Two types of loads were applied in the finite element models: body forces and surface tractions as required by the proposed guide specification in Appendix E. Body forces were applied for component dead loads ("DC" and "DW" per AASHTO designations). These are simply the product of mass, gravity and applicable load factors. Surfaces tractions were applied for traffic live loads ("LL" per AASHTO designation). The traffic live load is based on the HL-93 load model described in the AASHTO LRFD BDS, which is a combination of the truck loads, shown in Figure F-56, and a 0.64 klf load distributed over a width of 10 ft. The current structure does not include any bituminous pavement (i.e., DW is zero).

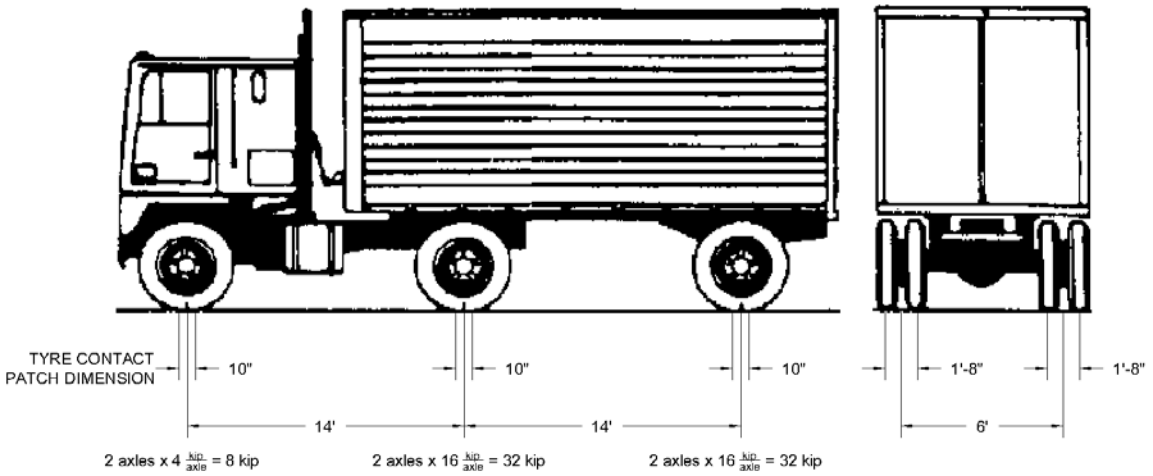


Figure F-56. Truck load components and dimensions of the HL-93 vehicular live load model.

The Redundancy I and Redundancy II loading combinations were used to evaluate the structure in the faulted state. The load factors for these two combinations are as in Table F-18, based on the provisions in Appendix E for bridges built to Section 12 in the AWS D1.5. The live load (LL) factors are modified by the appropriate multiple presence factors as described in Article 3.6.1.1.2 of the AASHTO LRFD BDS. It must be noted that dynamic amplification factor is equal to 0.4, which is applied to DC and LL in the Redundancy I load combination only. Also, the dynamic load allowance is 0.15 of the truck axle loads, and is only applied in the Redundancy II load combination.

Table F-18. Load factors used for Redundancy I and Redundancy II load combinations.

Load Combination	Load Factors				Notes
	DC	LL	DA_R	IM	
Redundancy I	1.05	0.85	0.40	N. A.	$\beta = 1.5$
Redundancy II	1.05	1.30	N. A.	0.15	$\beta = 1.5$

Longitudinally, the loads are positioned in the most critical positions in both the Redundancy I and Redundancy load combinations. For the failure scenario considered in the current case (failure of the southernmost tie girder at the stringer relief section, shown in Figure F-36 and Figure F-57, the most critical position of the truck axle loads which results in the truck facing west with its middle axis positioned at the failure plane, as shown in Figure F-57. The distributed load portion of the HL-93 load is applied along the entire span.

As described in the proposed guide specification in Appendix E, the transverse positioning of the HL-93 live load model differs between the Redundancy I and Redundancy II load combinations, as illustrated in Figure F-58. Since the vehicular loads in the Redundancy I load combination are meant to represent the applied load at the instant in time in which the assumed member failure occurs, the HL-93 vehicular live load model is transversely positioned centered (both the 10 ft loaded width and the truck axle loads) within the marked (striped) lanes, in this case two lanes. Hence, as the bridge is striped for two lanes, there are two load cases for the Redundancy I load combination: two marked lanes loaded, or one marked lane loaded.

On the other hand, the objective of the Redundancy II load combination is to evaluate the strength of the system after the failure of the primary steel tension member has occurred, so the number of design lanes is established in accordance with Article 3.6.1.1.1 in the AASHTO LRFD BDS, which in this case results in three design lanes with a width of 12 ft. In the Redundancy II load combination, the HL-93 vehicular live load model is transversely positioned (both the 10 ft loaded width and the truck axle loads) to produce extreme force effects within each design lane; however, the truck axle loads are transversely positioned such that the center of any wheel load is not closer than 2 ft from the edge of the design lane. Hence, there are three load cases for the Redundancy II load combination: three design lanes loaded, two design lanes loaded, or one design lane loaded.

Component dead loads were linearly applied in the initial implicit static analysis. Traffic live loads were applied in the final explicit dynamic analysis. Their dynamic effects were minimized by applying them slowly through the use of smooth step, as in the following equation:

$$LR(t) = 6\left(\frac{t}{T}\right)^5 - 15\left(\frac{t}{T}\right)^4 + 10\left(\frac{t}{T}\right)^3$$

where $LR(t)$ is the fraction of load at a load application time t, and T is the duration of the load application. The duration of the load application must be larger than the fundamental period of the structure to minimize oscillatory behavior in the final explicit dynamic analysis.

Regarding prescribed boundary conditions, vertical translation is prescribed to be zero at all support location since uplift would not occur under the loading employed in the current case. Horizontal translations are discussed in Section F.4.6 as they are enforced through connector elements that model the flexibility of the substructure.

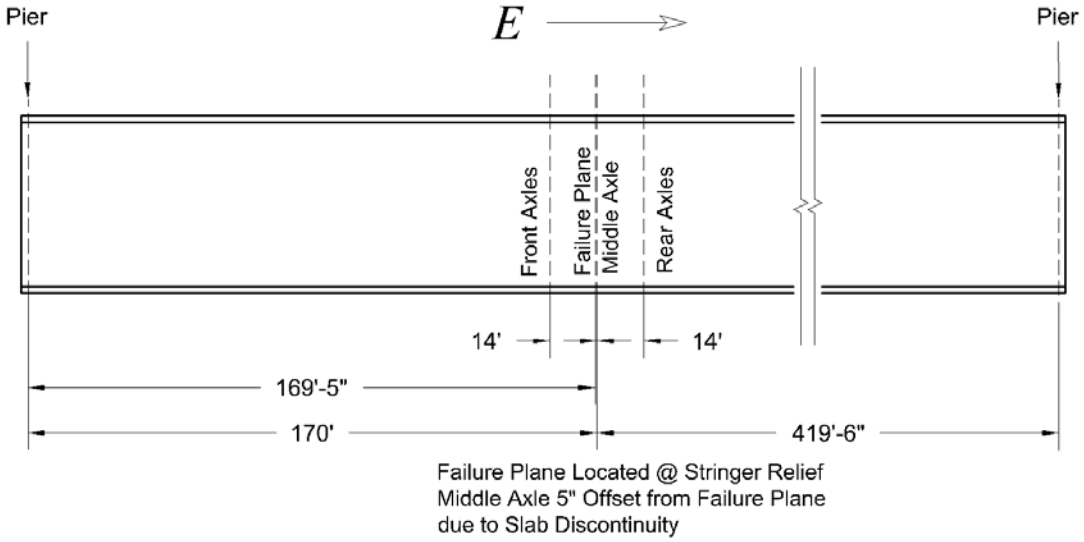
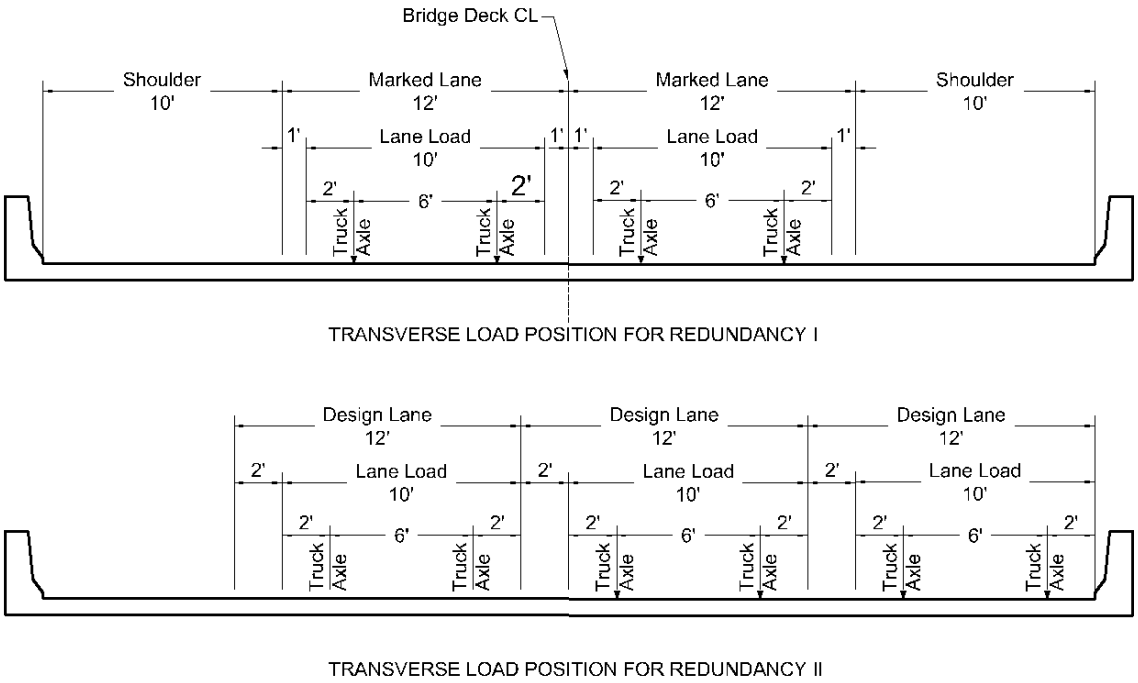


Figure F-57. Longitudinal position of truck axles.



1563
1564 ***Figure F-58. Transverse position of traffic loads.***

1565 ## F.4.8 Analysis of Results for Redundancy

1566 Once the analysis is completed the obtained results are evaluated using the requirements described in
1567 Article 8 of the proposed guide specification in Appendix E. It was found that the structure met the strength
1568 and serviceability requirements and is considered redundant against failure of the southernmost tie girder
1569 at the stringer relief joint. Specific details regarding the performance requirements and the results are
1570 summarized in Table F-19.

1571 **Table F-19. Summary of the redundancy evaluation after the failure of south tie girder.**

Performance Requirement		Most Critical Load Combination	Result	Acceptable?
Strength Requirements	Strain Primary Steel Members	Redundancy I 1 Lane	0.0055	**YES**
	Slab Concrete Crushing	-	No concrete crushing in the slab	**YES**
Serviceability Requirements	Vertical Deflection	Only Redundancy II DL considered	9.99 inches	**YES**

Notes:
1. The analysis showed that the structure was capable of resisting an additional 15% of the applied factored live load.
2. In order to complete the evaluation, the displacements and reaction forces calculated at support locations should be used as factored demands to check against the nominal capacity of the supports and substructure members.

F.4.8.1 Minimum Strength Requirements

All of the strength requirements were met by the system in the faulted state while subjected to any one of the load cases included in the Redundancy I and Redundancy II load combinations. Since the system met all of the strength requirement it may be re-designated as a system redundant member (SRM) as soon as the minimum serviceability requirements are met; otherwise it shall remain designated a fracture critical member (FCM).

The first set of strength requirements apply to any primary member of the superstructure, which in this case are the tub girders, diaphragms, and concrete slab. These requirements are the following:

- In a component, such as a web or a flange of a primary steel member, the average strain is less than five times the material yield strain.
- In a component, such as a web or a flange of a primary steel member, the average strain is less than 0.01.
- A strain level of 0.05 is not reached anywhere in a primary steel member.
- The combined flexural, torsional and axial force effects computed in primary compression members are below the nominal compressive resistance of the member (these limit states are predicted by the FEA).
- If a compression strain greater than 0.003 is reached in the slab, the portion where that limit is exceeded does not compromise the overall system load carrying capacity.
- The system in the faulted condition is able to support an additional 15% of the factored live load.

Very small and localized yielding was observed in the primary steel members, further critical buckling loads were not reached in any primary steel member. The plastic strains calculated in the tie girders, tied arch, and floor beams were below 0.01 after the failure of the southernmost tie girder for the Redundancy I or Redundancy II load combinations; therefore, the strain requirements on primary steel members are met. This is illustrated in Figure F-59, in which the equivalent plastic strain is shown for the most critical load case: the Redundancy I load combination with one marked lane loaded. As the FEA accurately predicts potential failure of primary steel compression member subjected to combined flexural, torsional, and axial force effects, and quasi-static equilibrium is reached for both load combinations, the requirements of primary steel compression members are met.

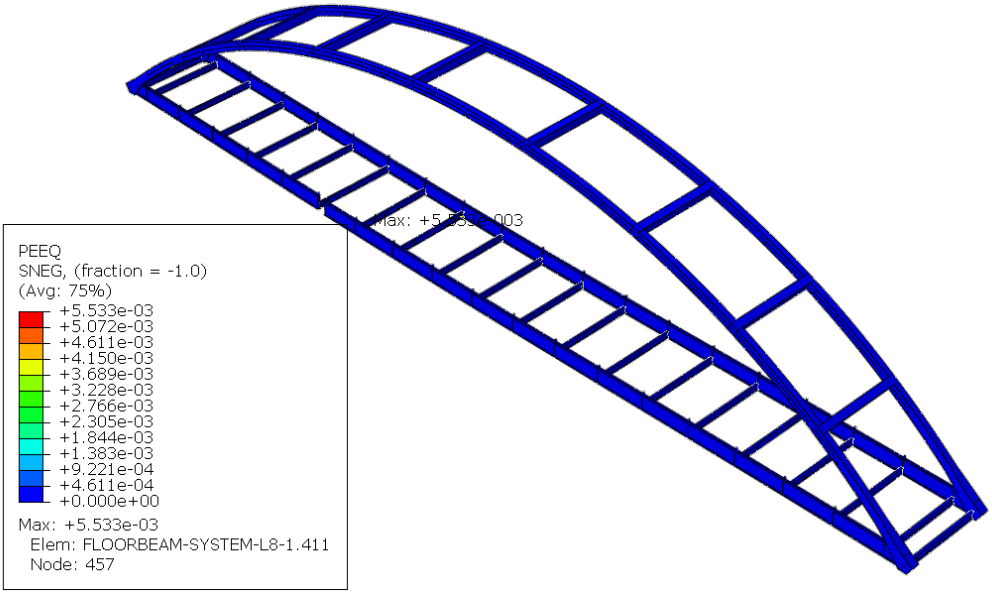


Figure F-59. Location of maximum plastic equivalent strain in primary steel members.

Regarding the concrete slab, concrete crushing and tension cracking is allowed and expected to take place. However, if the portion of the slab where a total compressive strain of 0.003 has been exceeded is large enough to compromise the overall system load carrying capacity or if significant hinging occurs, the structure should not be considered as sufficiently redundant. In this example, the Redundancy II load combination, with three design lanes loaded, resulted in the largest compressive strains in the slab, which were located in the haunches over the southernmost stringers near the failure location. However these strains do not reach 0.003 as shown in Figure F-60; thus, it was not enough to result in a reduction in load carrying capacity.

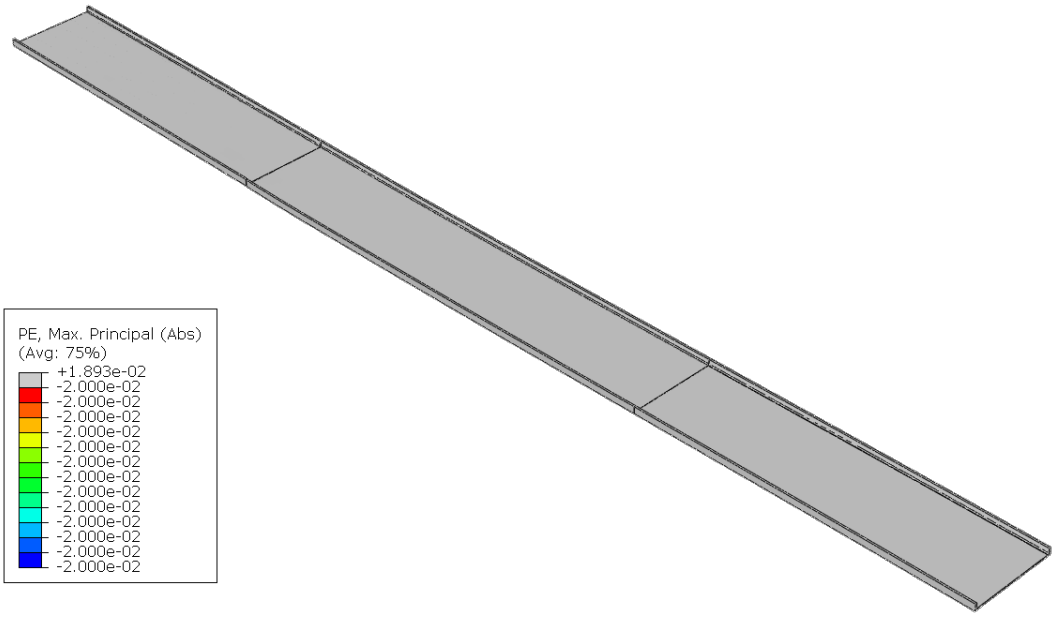


Figure F-60. Absence of concrete crushing in slab.

Although the substructure is not explicitly included in the finite element model, the reaction forces at support locations are calculated in the analysis. These should be taken as the factored demands that the substructure must be able to safely sustain, which are summarized in Table F-20. In this example, the Redundancy I load combination resulted in the largest vertical reaction forces. Similarly the largest longitudinal and transverse reaction forces take place under the Redundancy I load combination. The unfactored nominal capacity of the abutments and the pier need to be checked against these load demands. Similarly the pier and abutments must accommodate the horizontal displacements that are calculated in the analysis at the support locations. In this example, Redundancy I and Redundancy II load combinations resulted in similar small horizontal displacements which are summarized in Table F-21.

1625 **Table F-20. Calculated reaction forces for redundancy evaluations.**

Support	Knuckle	Reaction Force	Redundancy I		Redundancy II		
			1 Lane	2 Lanes	1 Lane	2 Lanes	3 Lanes
East Pier	South	Vertical	2749 kips	2849 kips	2129 kips	2242 kips	2278 kips
		Longitudinal	-1594 kips	-1813 kips	-1045 kips	-1261 kips	-1444 kips
		Transverse	65.3 kips	74.8 kips	43.6 kips	56.5 kips	65.2 kips
	North	Vertical	2670 kips	2816 kips	1886 kips	2015 kips	2155 kips
		Longitudinal	-882 kips	-1211 kips	-421 kips	-621 kips	-888 kips
		Transverse	-11.5 kips	-22.6 kips	-2.71 kips	-11.9 kips	-20.0 kips
West Pier	South	Vertical	2754 kips	2831 kips	2096 kips	2253 kips	2297 kips
		Longitudinal	2184 kips	2386 kips	1496 kips	1753 kips	1936 kips
		Transverse	40.4 kips	41.1 kips	29.3 kips	28.5 kips	27.9 kips
	North	Vertical	2789 kips	2293 kips	1967 kips	2104 kips	2241 kips
		Longitudinal	292 kips	638 kips	-29.6 kips	130 kips	390 kips
		Transverse	-93.0 kips	-93.1 kips	69.5 kips	72.3 kips	72.7 kips

Notes:
1. Positive longitudinal direction points east.
2. Positive transverse direction points south.

1626

1627 **Table F-21. Calculated displacements at support locations for redundancy evaluation.**

Support	Knuckle	Displacement	Redundancy I		Redundancy II		
			1 Lane	2 Lanes	1 Lane	2 Lanes	3 Lanes
East Pier	South	Longitudinal	0.415 in	0.472 in	0.272 in	0.328 in	0.375 in
		Transverse	-2.94E-3 in	-3.37E-3 in	-1.96E-3 in	-2.54E-3 in	-2.94E-3 in
	North	Longitudinal	0.230 in	0.316 in	0.110 in	0.162 in	0.231 in
		Transverse	5.18E-4 in	1.02E-3 in	1.21E-4 in	5.37E-4 in	9.02E-4 in
West Pier	South	Longitudinal	-0.569 in	-0.621 in	-0.390 in	-0.456 in	-0.504 in
		Transverse	-1.82E-3 in	-1.85E-3 in	-1.32E-3 in	-1.28E-3 in	-1.26E-3 in
	North	Longitudinal	-0.0760 in	-0.166 in	7.71E-3 in	-0.0338 in	-0.102 in
		Transverse	4.19E-3 in	4.20E-3 in	3.13E-3 in	3.26E-3 in	3.27E-3 in

Notes:
1. Positive longitudinal direction points east.
2. Positive transverse direction points south.

1628

Additionally, the system demonstrated a reserve margin of at least 15% of the applied live load in the Redundancy I and II load combinations. Effectively, this requirement ensures the slope of the load vs displacement curve for the system structure remains positive (i.e., there is still significant remaining reserve capacity).

F.4.8.2 Minimum Serviceability Requirements

The only serviceability requirement in the Appendix E is that the increase of deflection after the failure of a primary steel tension member cannot be greater than L/50. This requirement is to be checked in the Redundancy II load combination under factored dead load only. In the current case, the limit is 135 inches, which was not surpassed since the maximum additional deflection computed in the FEA was 9.99 inches. This is illustrated in Figure F 20.

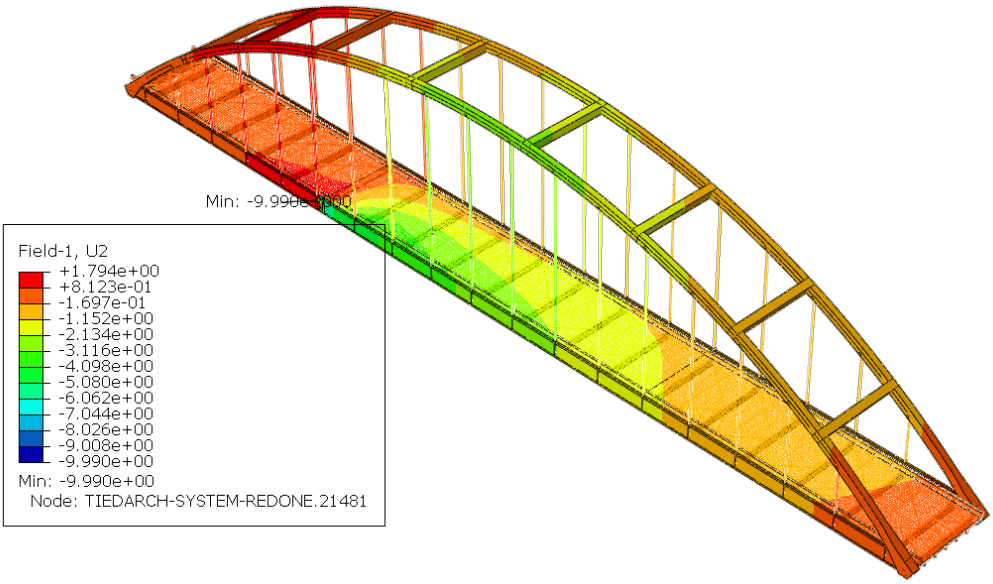


Figure F-61. Deflection after failure of primary steel tension member.

F.4.9 Conclusions

The redundancy of a tied arch bridge after the failure of the southernmost tie girder at a stringer relief joint was analyzed in accordance with the methodology described in the proposed guide specification in Appendix E. Based on the comparison between the obtained results and the minimum performance requirements, the structure is not likely to fail nor undergo a significant serviceability loss as result after the failure of the tie girder at the stringer relief joint. Hence the tie girders may be re-designated as a system redundant member (SRM), provided that it also passes an additional evaluation in which failure of the tie girder is introduced near the knuckle.

F.5 Continuous Three Span Three-Girder Bridge

The redundancy of a continuous three span three-girder bridge is analyzed by developing a finite element model in accordance with the methodology described in Appendix E. It is assumed that the structure does not possess any of the detrimental attributes described in the screening criteria. The bridge is NOT built to Section 12 of the AWS D1.5. In this case, the failing tension member is assumed to be the exterior westernmost girder. The entire cross-section is assumed to have failed at a cross section located south of the northernmost pier (Pier 1) as shown in Figure F-62.

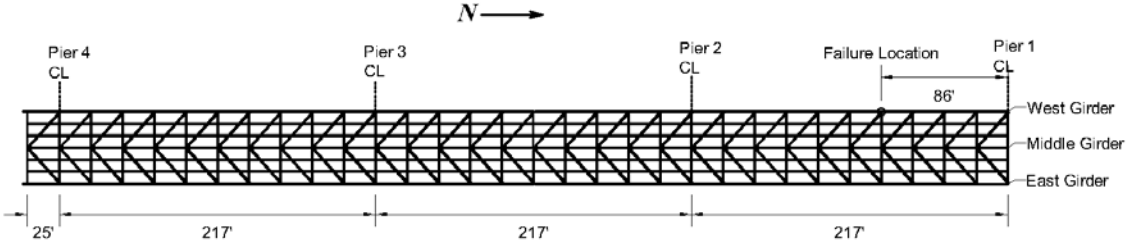


Figure F-62. Steelwork geometry and failure location.

The structure has three spans measuring 217 feet long each. The structure is straight with no skew. Three welded plate girders conform the primary members. Attached to the girders there is a system of plate floor beams which provided support to continuous stringers. Lateral stability of the girders is provided by K-bracing connecting. Figure F-63 shows the steelwork framing plan. The girder schedules are described in Figure F-64 for the exteriors (east and west) plate girders, and Figure F-65 for the middle girder. A sketch of the typical connections among girders, floorbeams and bracing is shown in Figure F-66

The reinforced concrete slab is 52 feet wide between interior edges of concrete barriers (approximately 56 feet wide between the outer exterior edges of concrete barriers) and is non-composite. The supports are multi-rotational fixed bearings at all support points. All steel components are fabricated of ASTM A36, with the exception of the flanges of the plate girders which are fabricated of ASTM A588. All concrete has a minimum specified compressive strength of 4 ksi and all rebar has 60 ksi yield strength. In the analysis of this structure, longitudinal and transverse slopes will be neglected. Figure F-67 shows the cross section of the structure with the slab, barrier and reinforcement details.

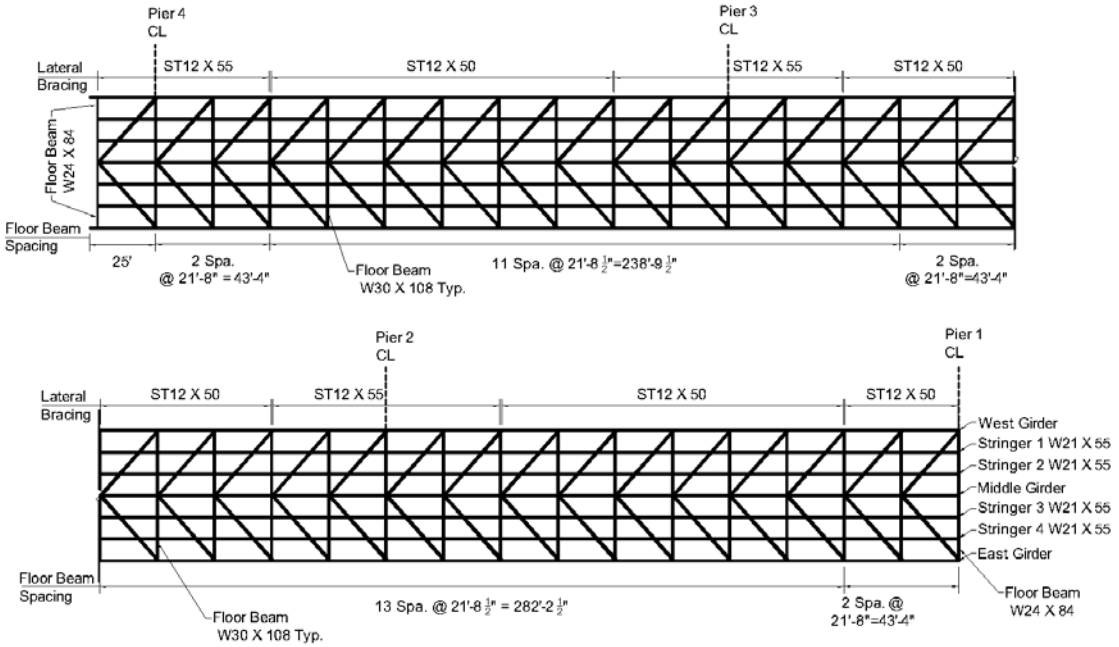


Figure F-63. Assembly of steel components of the three-girder bridge.

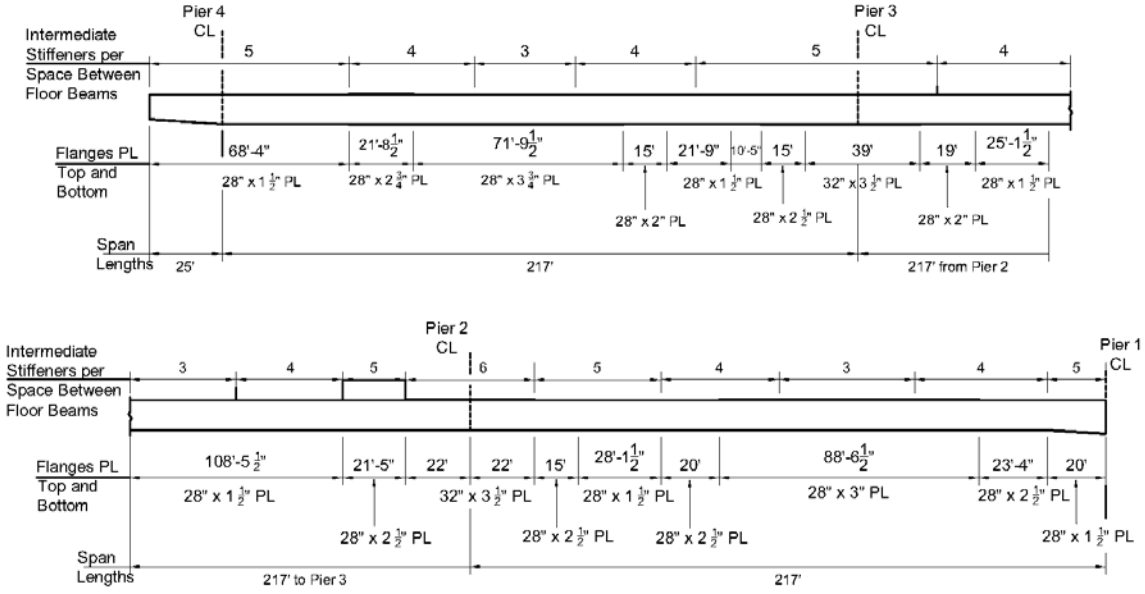


Figure F-64. Schedule of east and west plate girders.

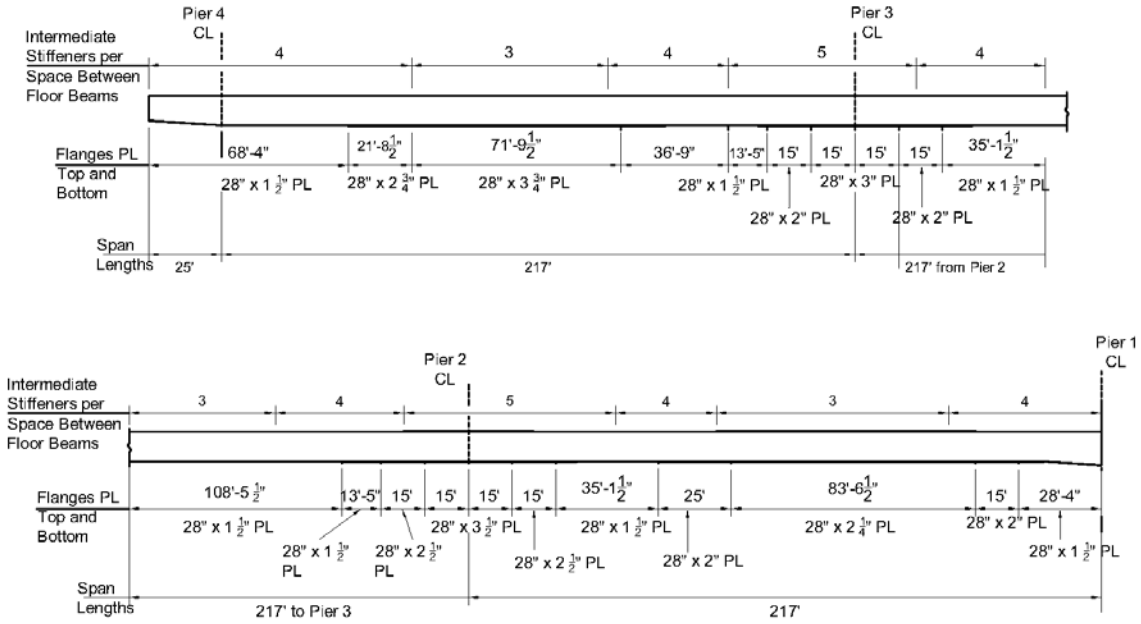


Figure F-65. Schedule of middle plate girder.

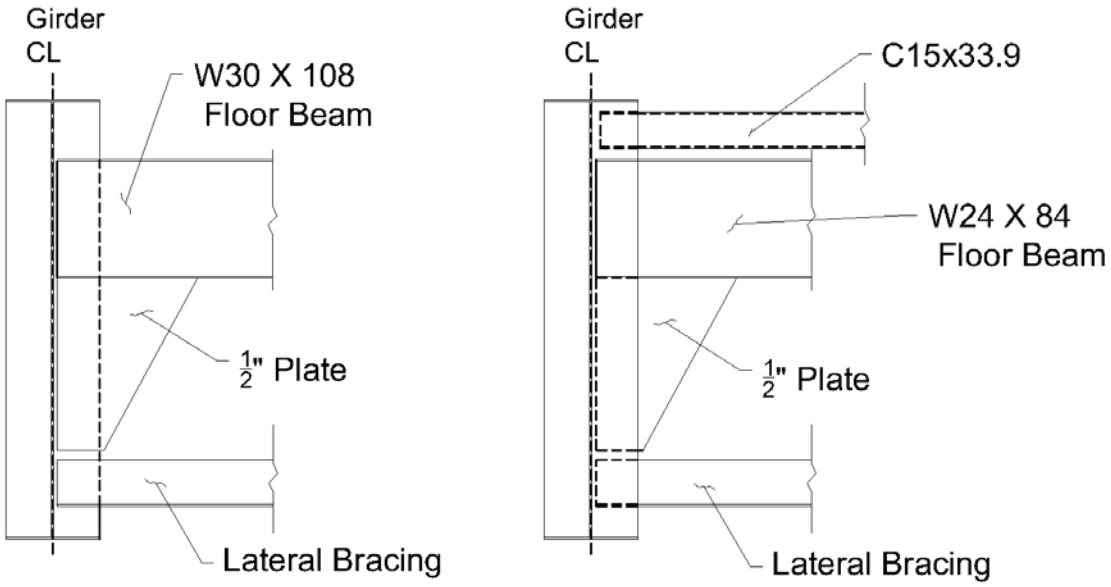


Figure F-66. Typical connection detail.

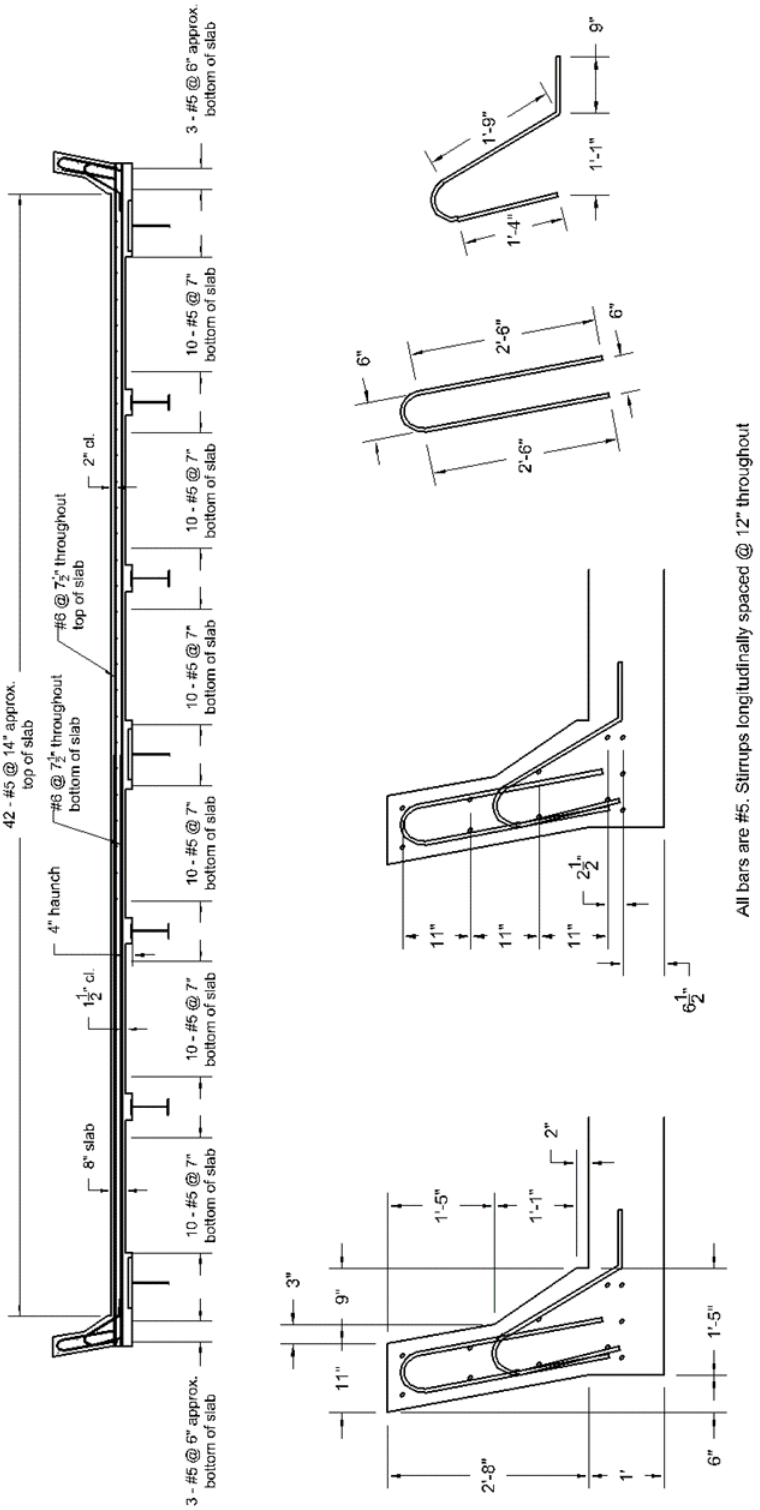


Figure F-67. Slab cross-section and reinforcement details.

F.5.1 Analysis Procedure

The analysis is performed to establish if the system demonstrates acceptable performance in the faulted condition. In the example, the term "faulted condition" specifically refers to the case in which a primary steel tension member is assumed to have failed. For this analysis, load factors for both dead and live load are applied as described in the proposed guide specification in Appendix E. In this example, the described analysis procedure is composed of an initial implicit static analysis and a final explicit dynamic analysis, into which the results from the initial implicit static analysis are imported. While it is not mandatory for the Engineer to follow these particular steps, it has been found that this procedure optimizes the computational time required.

F.5.1.1 Initial Implicit Static Analysis

Implicit static analysis was utilized to calculate the state of the structure prior to hardening of the concrete in the slab. An implicit static analysis was used for the initial steps because, although non-linearity is considered in the analysis, the bridge behavior is linear and inertial effects can be neglected as the bridge is in the undamaged condition. As the slab does not carry any load and does not contribute to the stiffness of the system before concrete hardening, two modifications are required in the finite element analysis during this initial implicit static analysis as follows:

- A very low stiffness is specified for the elements composing the slab, i.e, the elements modeling concrete and rebar. A reduced stiffness of 1/1,000 of the respective modulus of elasticity of each material was used. This is done so the load carried by the slab and rebar have negligible contribution to the stiffness of the system. No modifications to the stiffness should be applied to the steelwork.
- Instead of defining contact interaction between the slab and the steelwork, a mesh tie was specified. The nodal displacements of the concrete slab elements are tied to the displacements of the top flanges of girders, floor beams, and stringers which occur due to dead load. As a result, the slab deforms with the steelwork and does not 'sag' between the girders, floor beams, and stringers.

It is worth noting that the remainder of the finite element modeling is identical between the initial implicit static analysis and the final explicit dynamic analysis. The specific steps in the initial implicit static analysis are described as follows:

1. Apply load due to self-weight of the structural steel components as a body force.
2. Apply load due to self-weight of the wet slab components as a body force.
3. The system is then fixed in terms of position, that is, the displacement degrees of freedom are not allowed to change.
4. The elements composing the slab (elements modeling rebar and concrete) are then deactivated.
5. The elements composing the slab are then reactivated. During this reactivation the strain in the elements composing the slab is reset to zero.

Steps 3 through 5 are necessary since even though very low stiffness was specified for the slab, these elements do undergo strain. Setting the strains to zero eliminates "locked in" artificial stresses in later steps.

F.5.1.2 Final Explicit Dynamic Analysis

As contact algorithms, softening material behavior, and non-linear geometry are required to be part of the finite element analysis, implicit solution procedures present unavoidable convergence problems in most FEA solvers. In order to calculate the capacity of the bridge after sudden failure of a tension component, a dynamic explicit analysis needs to be carried out. Therefore, the results obtained from the initial implicit static analysis are imported into the final explicit dynamic analysis. In other words, the state of the system (stresses, strains, displacements and forces) at the beginning of the final explicit dynamic analysis is defined by the state of the system computed at the end of the initial implicit static analysis.

As previously stated, during the initial implicit static analysis, the slab was modeled with largely reduced stiffness to reflect that it is not hardened and a mesh tie constraint was used to assure that the slab deformed with the steelwork. This approach also prevents excessive sag of the soft slab. After the state of the system is imported, the following changes are made to capture the response of the structure after the concrete has hardened:

- The modulus of elasticity of the concrete and rebar elements in the slab is changed to their final actual values. It is noted that no modifications need to be applied for the steelwork.
- The mesh tie constraint between the slab concrete elements and the top flanges of the steelwork is replaced by a frictional contact interaction. In this case, since the structure is non-composite, the only interaction between the slab and the steel members' top flanges is frictional contact, i.e., no elements that model the behavior of shear studs are added.

All of the body forces applied during the initial implicit static analysis (i.e., the dead load of the structure) are maintained throughout the final explicit dynamic analysis.

To evaluate the capacity of the structure in the faulted state, the following steps were carried out in the final explicit dynamic analysis:

6. The stiffness of the elements located at the fracture location under consideration were slowly reduced. The stiffness was slowly reduced in order to minimize any dynamic effects. It is noted that the actual fracture and subsequent vibration of the structure is not modeled. This dynamic effect is accounted for using the DA_R factor as discussed before. If dynamic effects are found to be significant even if the stiffness is slowly reduced, the system must be allowed to oscillate until these effects are dampened.
7. Factored loads due to traffic are applied as surface tractions. For the Redundancy I load combination all loads are amplified by DA_R, for the Redundancy II load combination the dynamic load allowance (IM) is applied. These loads were applied very slowly to minimize any dynamic effects, as well. If dynamic effects are significant, the system must be allowed to oscillate until these effects are dampened.
8. An additional 15% of live load is gradually applied.

F.5.2 Material Models

Four material models are needed in the finite element model. Three of those are utilized to model different steel types, and one is utilized to model the response of concrete. For the development of the steel material models, it is necessary to know the yield strength and ultimate strength of each steel type. In this example, since no test values are known to the Engineer, nominal values specified in the respective standards are utilized. These are summarized in Table F-22. A mass density of 0.494 kcf was specified for all steel types.

Table F-22. Material properties for steel material models.

Material	Nominal Yield Strength	Nominal Ultimate Strength	Standard
ASTM A36	36 ksi	58 ksi	ASTM A36/A36M
ASTM A588	50 ksi	70 ksi	ASTM A588/A588M
Grade 60 Rebar	60 ksi	90 ksi	ASTM A615/A615M

The stress-strain relation for all steel components will follow an initial linear elastic steel with a Young's modulus of 29,000 ksi and Poisson's ratio of 0.3. Once the nominal yield strength is reached the stress-strain relation is defined by Von Mises (J2) plasticity with kinematic linear hardening, until the nominal ultimate strength is reached at a total strain of 0.05. Once the nominal ultimate strength or a total strain of

0.05 is reached, the material is assumed to fail. Figure F-68 shows the uniaxial material response for the steel employed in this finite element model with the 'X' denoting the stress at the failure strain of 0.05.

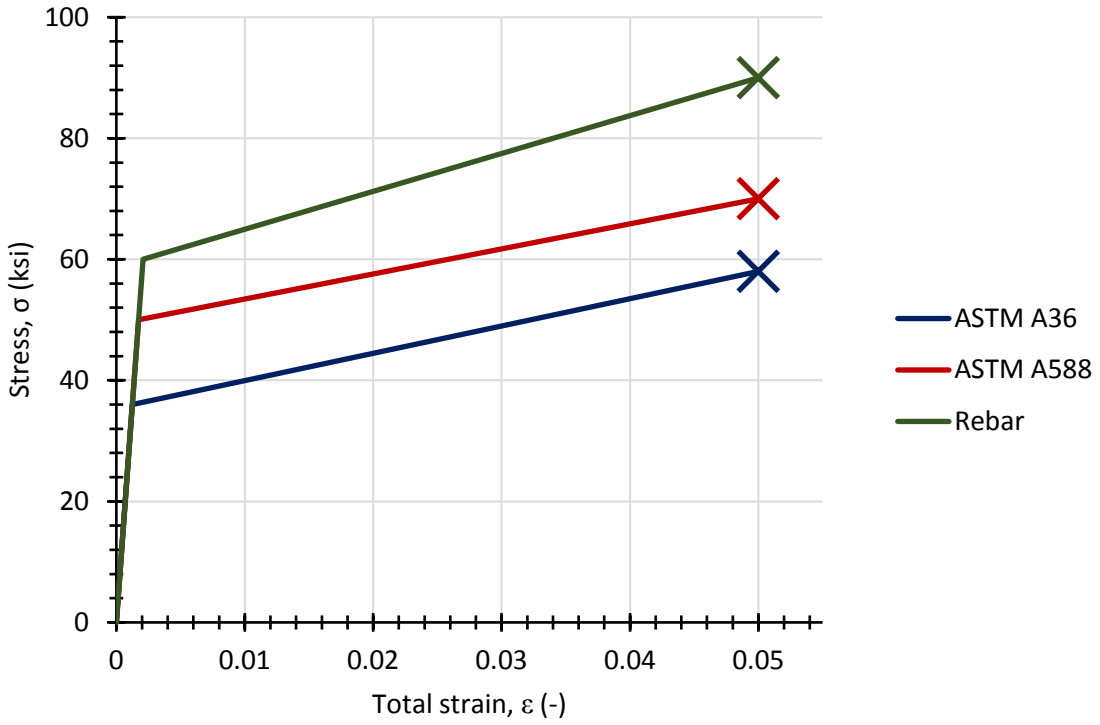


Figure F-68. Stress-strain curves of steel material models.

The material model used in concrete is defined entirely by the specified compressive strength which in this case is 4 ksi. This quantity is also used to calculate the tensile strength, the total strain at compressive strength, ε_c, and the material parameter n. Table F-23 summarizes the calculation of these values. A mass density of 0.150 kcf was specified for concrete.

Table F-23. Material properties for concrete material model.

Quantity	Symbol	Equation	Result
Young's modulus	E_c	$E_c = 33{,}000 w_c^{1.5} \sqrt{f_c'} \leq 1{,}802.5\sqrt{f_c'}$	3,600 ksi
Tensile strength	f_t	$f_t = 0.158(f_c')^{\frac{2}{3}} for\ f_c' \leq 7.25ksi$ $f_t = 0.307 \ln(f_c' + 2.61) - 0.114\ for\ f_c' > 7.25ksi$	0.398 ksi
Fracture energy	G_t	$5.9 \cdot 10^{-4}(f_c' + 1.16)^{0.18}$	$7.93 \cdot 10^{-4}$ kip/in
Total strain at compressive strength	ε_c	$\varepsilon_c = 0.00124\sqrt[4]{f_c'}$	0.00175
Material parameter	n	$n = 0.4f_c' + 1.0$	2.6

The concrete material model is initially linear elastic, defined by a Young's modulus of 3,600 ksi and Poisson's ratio of 0.2, followed by concrete damage plasticity. In tension, once the material reaches its tensile strength, set at 0.398 ksi in this case, a tensile stress-displacement relation characterized by a fracture energy, G_t, of $7.93 \cdot 10^{-4}$ kip-in is followed This fracture energy is applied through a bi-linear tensile stress-displacement relation as shown in Figure F-69, and defined by the following quantities:

$$f_{t1} = \frac{f_t}{5} = 0.0796\ ksi$$

$$\delta_t = \frac{5G_t}{f_t} = 0.00996$$

$$\delta_{t1} = \frac{G_t}{f_t} = 0.00199$$

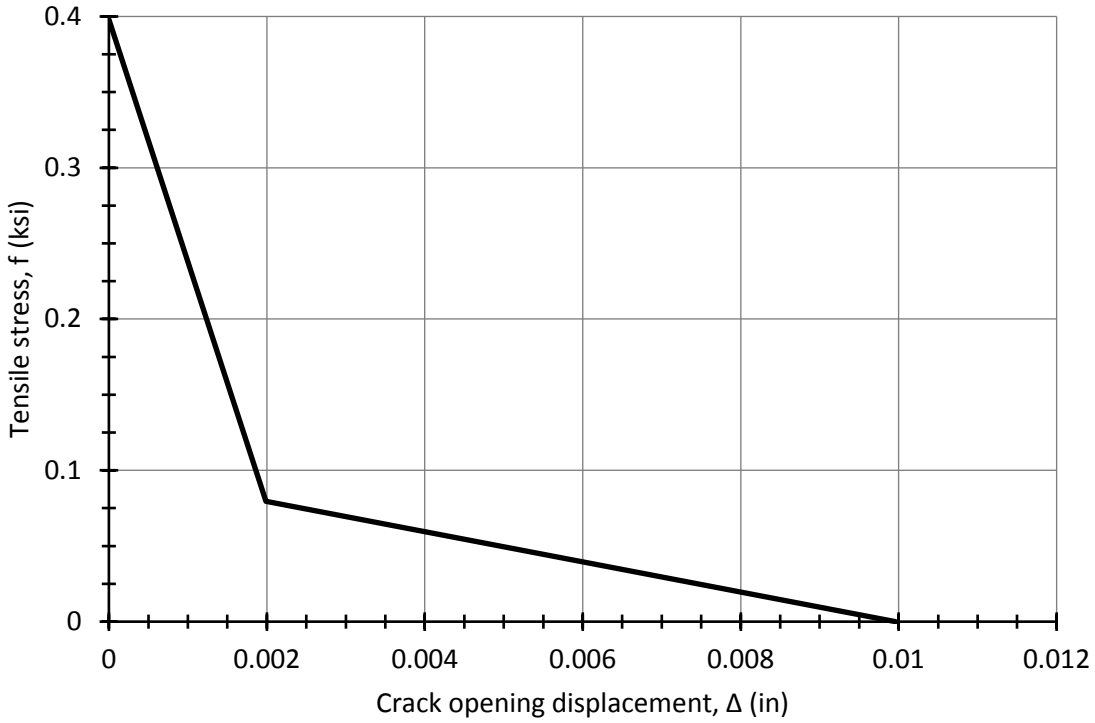


Figure F-69. Tensile stress-crack opening displacement curve for concrete material model.

In compression the material follows the following stress-strain relations:

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

$$\varepsilon_{plastic} = \varepsilon - \frac{f(\varepsilon)}{E_c}$$

Where ε is total (elastic + plastic) strain, $f(\varepsilon)$ is the compressive stress at a given total strain, f_c' is the specified compressive strength, ε_c is the total strain at compressive strength, n is a material parameter, $\varepsilon_{plastic}$ is the plastic strain and E_c is the concrete Young's modulus. Figure F-70 shows the resulting compressive stress-strain relation.

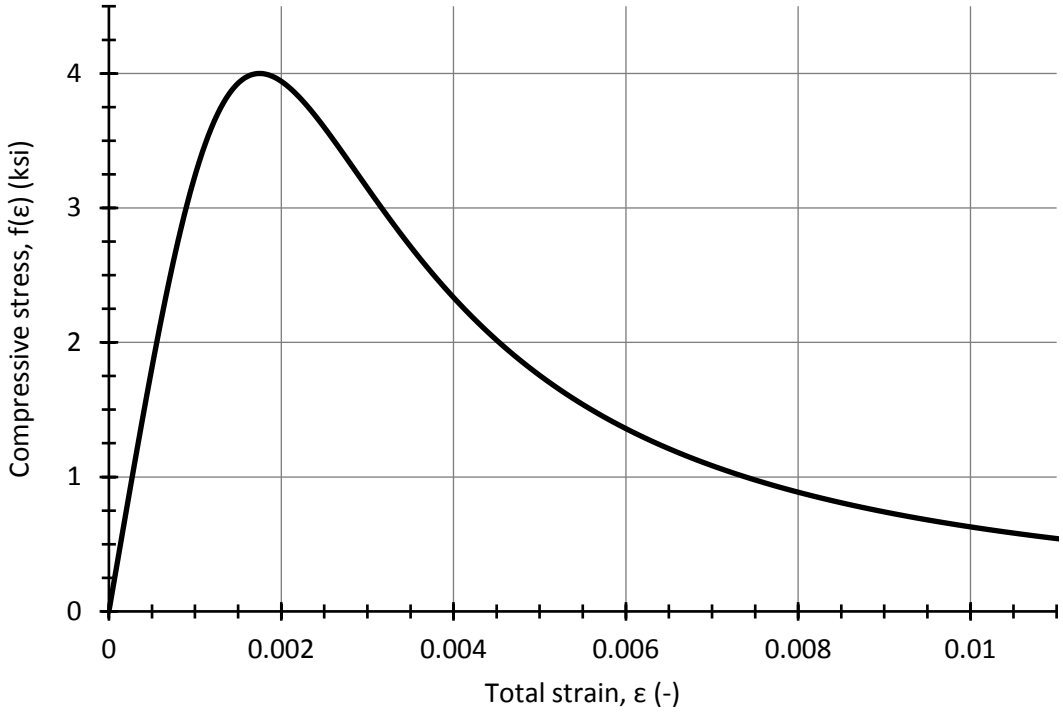


Figure F-70. Compressive stress-strain curve for concrete material model.

F.5.3 Geometries, Meshes and Constraints

The geometry of the structure is based on available design plans and is composed of the following components that must be explicitly modeled:

1. Three plate girders.
2. A system of 64 floor beam.
3. Four continuous stringers supported on the floor beams.
4. A system of lateral braces.
5. Stiffener plates and connection plates.
6. A reinforced concrete slab with concrete barriers.

When generating the finite element model, splices, holes, access hatches, etc. are neglected. The structure is assumed to be flat in the vertical plane, in other words, camber and superelevation are ignored. Figure F-71 shows the assembly of all bridge components looking up from the below the bridge.

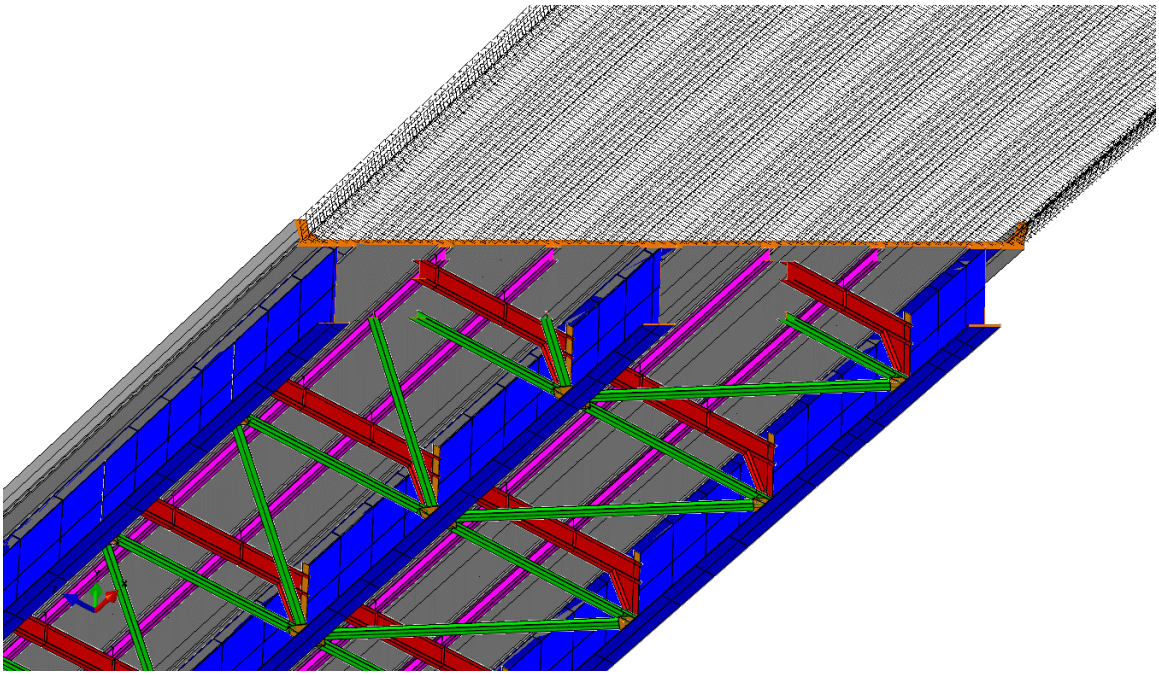
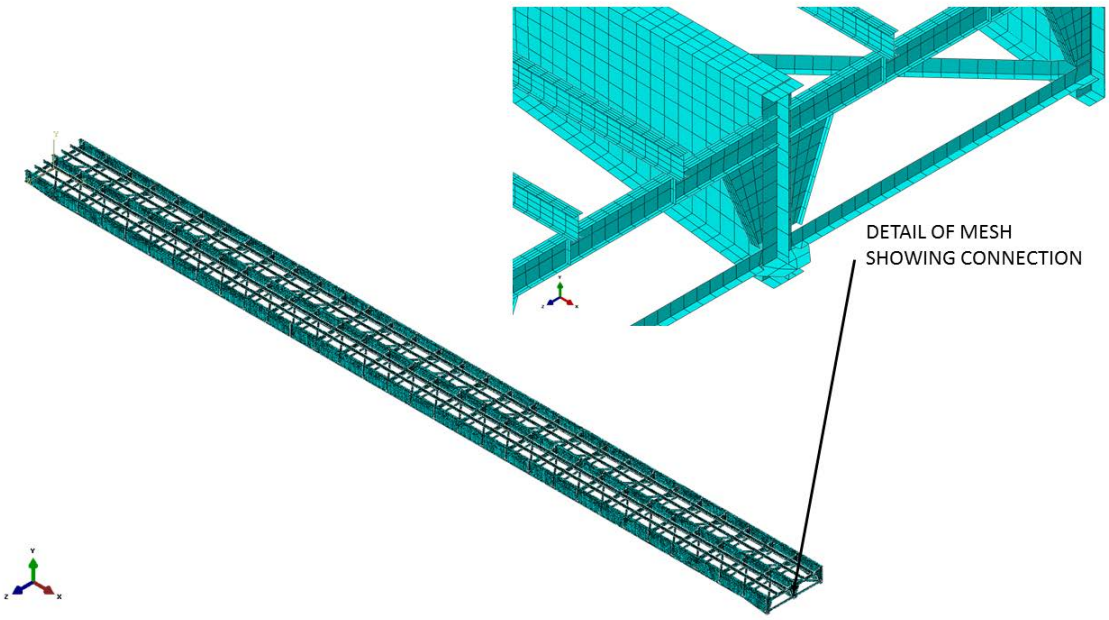


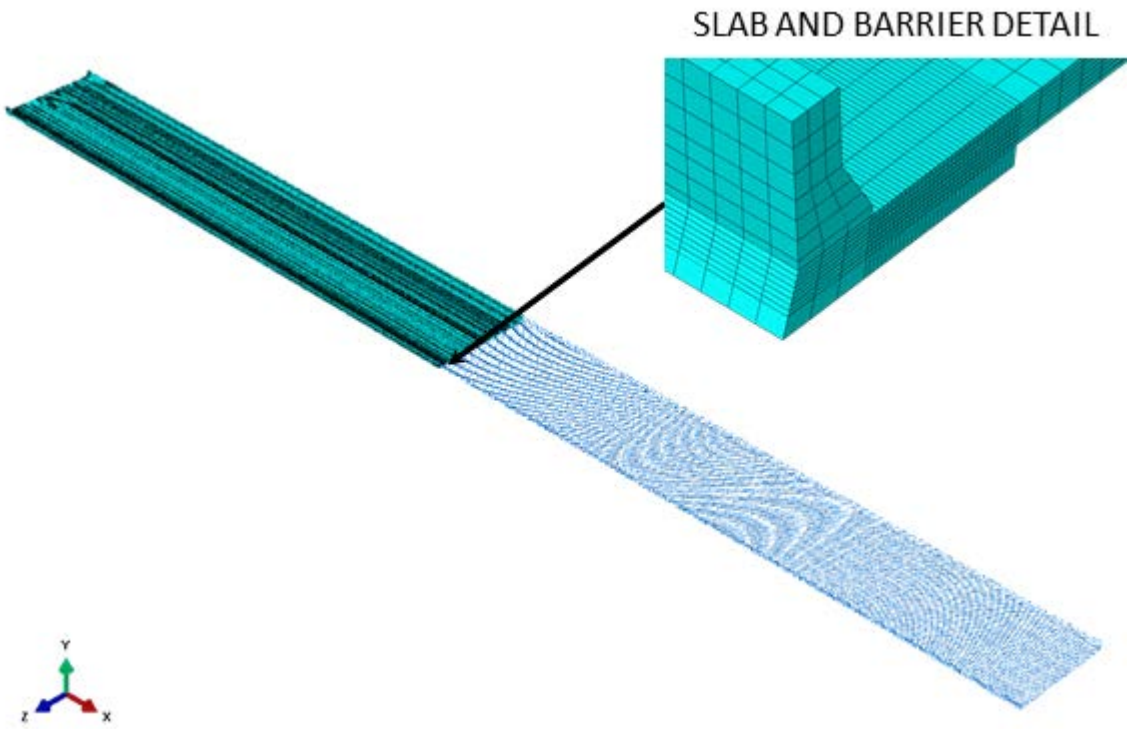
Figure F-71. Three-girder bridge geometries, detailed bottom view. Concrete slab and barriers (grey), plate girdes (blue), K-bracing (green), stringers (magenta), floor beams (red), slab reinforcement (black) and stiffeners and connection details (orange).

All steel components are modeled with 4-node shell elements with reduced integration. A minimum of four elements are used along flange widths and along web heights, except at bracing elements, were one element was used. The maximum aspect ratio was kept below five and corner angles were kept between 60 and 120 degrees. Figure F-72 shows a detail of the mesh employed to model the plate girders, diaphragm, floor beam, stringers, and stiffeners. Mesh ties, which are constraints that slave the motion of a surface or node set to the motion of a master surface or node set, are utilized to connect the various steel components.

The slab is modeled with two types of elements. Specifically, 8-node linear bricks with reduced integration are used to model concrete and 2-node truss elements with linear displacement to model steel reinforcement. Eight solid concrete elements are used through the thickness of the slab with maximum aspect ratio (length of longest edge divided by length of shortest edge) of 5, and corner angles (angle at which two element edges meet) between 40 and 140 degrees. The length of the truss elements used to model the reinforcement were approximately equal to the length of the longest edge of the solid concrete elements. These truss elements are embedded within the solid concrete elements. At the nodes of the embedded truss elements, the translational degrees of freedom are eliminated and the nodal translations were constrained to interpolated values of the nodal translations of the host solid concrete element. Figure F-73 shows the concrete slab with the embedded truss elements and a detail of the mesh used for the concrete barrier and slab.



1834
1835 ***Figure F-72. Mesh details of the steel component of the three-girder bridge.***



1836
1837 ***Figure F-73. Mesh details of the reinforced concrete slab and barriers.***

F.5.4 Slab-Structural Steel Interaction

As stated, the interaction between the bottom of the concrete slab and the top of the flanges of the steelwork is modeled differently in the two steps described above. In the initial implicit static analysis, when the elements comprising the slab and barriers have 1/1,000th of the modulus of elasticity to model the "wet" condition, a mesh tie is used to slave the motion of the slab to the motion of the surface comprising the top of the steel work. With this procedure, it is ensured that the slab deformation will conform to the deformation of the steelwork while unrealistic sagging of the slab between supporting elements and tipping of the barrier is prevented.

In the final explicit dynamic analysis, when the stiffness of the elements comprising the slab and barriers has been changed to their final real values, the mesh tie previously used is deleted and replaced by a contact interaction and modeling of shear studs. The normal behavior of the contact interaction is modeled through a penalty stiffness. The penalty stiffness is several orders of magnitude larger than the normal stiffness of the underlying contacting elements and allows a very small penetration so a pressure can be calculated. The tangential behavior of the contact interaction is modeled through an algorithm based on Coulomb friction with a limit on the allowable shear. A friction coefficient of 0.55 and an interfacial shear strength of 0.06 ksi are specified.

F.5.5 Connection Modeling

When a connection is likely to fail before yielding of the member, in addition to the use of mesh ties to attach the components, an additional step may be necessary to capture connection failure. In this particular example, it was opted to model every component of the connections between floor beams, girders and bracing. Typical connections are shown in Figure F-74.

The main motive was to capture the flexibility of the connection, the eccentricity of bracing elements at the connection, and possible failure of the braces at the connection. Mesh tie constraints were utilized to attach the components; in a mesh tie constraint the displacement of a slave region is constrain to the displacement of a master region. In this case, the entire stiffener and connection plate is the master region, while the slave regions are the portions of the connected elements that are bolted to the connection as shown in Figure F-75.

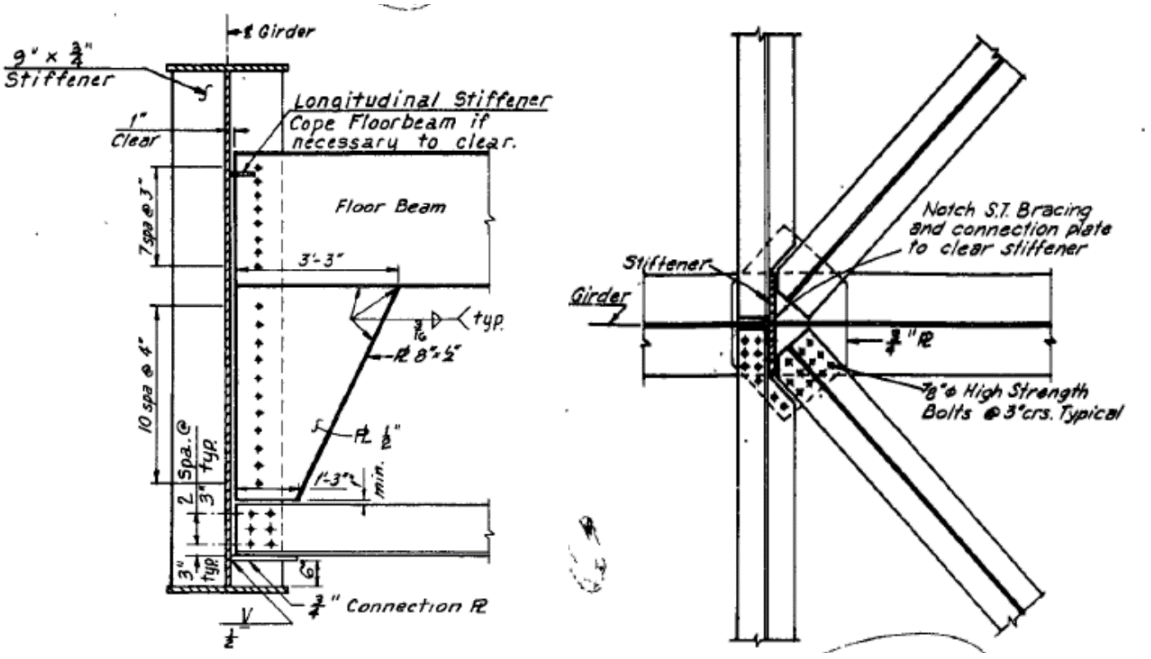


Figure F-74. Connection details. Longitudinal stiffener was not included in finite element model.

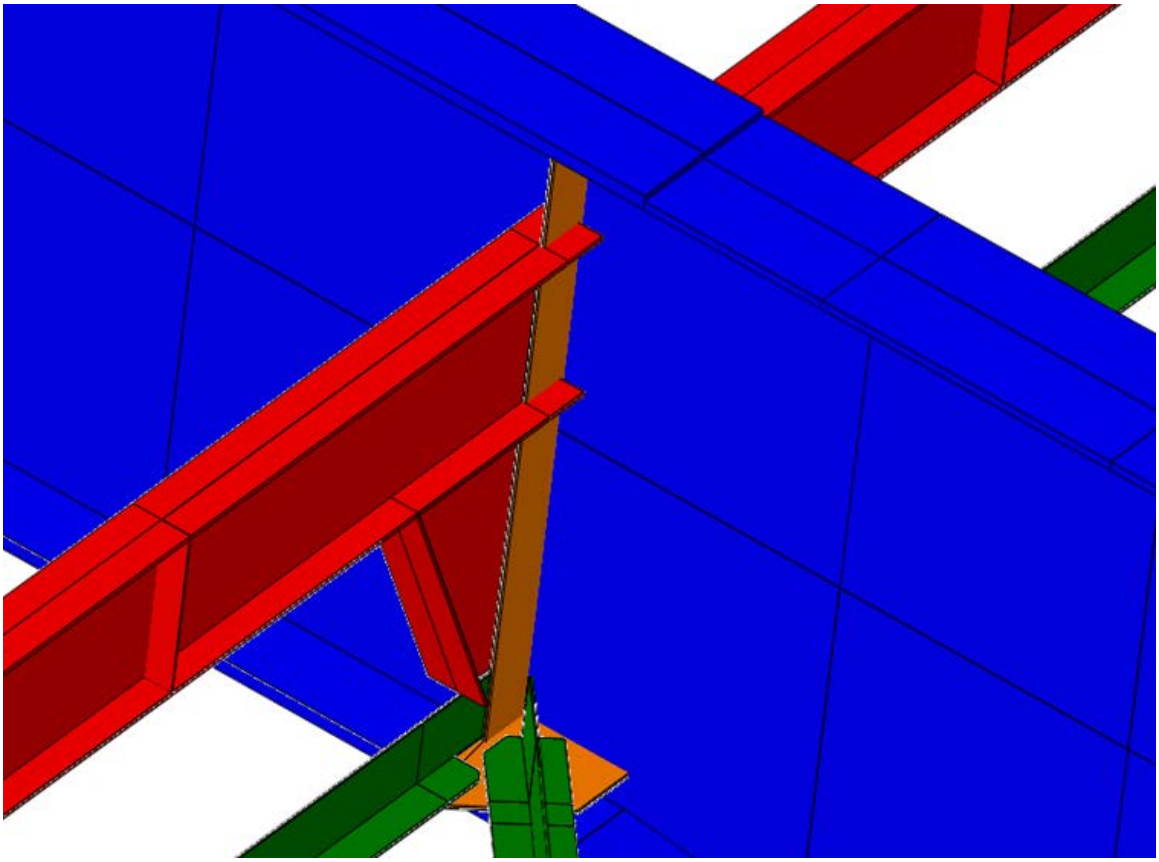


Figure F-75. Detail of connection as modeled in finite element model. Stiffener and connection plate shown in orange. Bracing (green), girder (blue) and floor beam (red) shown as well.

F.5.6 Loads and Boundary Conditions

Two types of loads were applied in the finite element models: body forces and surface tractions as required by the proposed guide specification in Appendix E. Body forces were applied for component dead loads ("DC" and "DW" per AASHTO designations). These are simply the product of mass, gravity and applicable load factors. Surfaces tractions were applied for traffic live loads ("LL" per AASHTO designation). The traffic live load is based on the HL-93 load model described in the AASHTO LRFD BDS, which is a combination of the truck loads, shown in Figure F-76, and a 0.64 klf load distributed over a width of 10 ft. The current structure does not include any bituminous pavement (i.e., DW is zero).

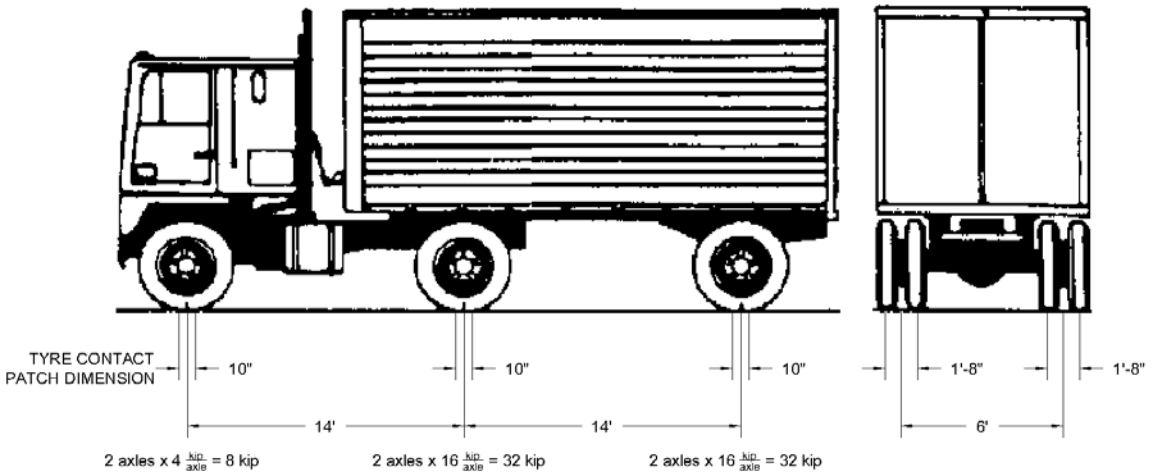


Figure F-76. Truck load components and dimensions of the HL-93 vehicular live load model.

The Redundancy I and Redundancy II loading combinations were used to evaluate the structure in the faulted state. The load factors for these two combinations are as in Table F-24, based on the provisions in Appendix E for bridges NOT built to Section 12 in the AWS D1.5. It must be noted that dynamic amplification factor is equal to 0.4, which is applied to DC and LL in the Redundancy I load combination only. Also, the dynamic load allowance is 0.15 of the truck axle loads, and is only applied in the Redundancy II load combination. :

Table F-24. Load factors used for Redundancy I and Redundancy II load combinations.

Load Combination	Load Factors				Notes
	DC	LL	DA_R	IM	
Redundancy I	1.15	1.00	0.40	N. A.	$\beta = 2.5$
Redundancy II	1.15	1.50	N. A.	0.15	$\beta = 2.5$

Longitudinally, the loads are positioned in the most critical positions in both the Redundancy I and Redundancy load combinations. For the failure scenario considered in the current case (failure of the exterior westernmost girder at 86 feet south or Pier 1, shown in Figure F-62 and Figure F-77), the most critical position of the truck axle loads which results in the truck facing north with its middle axis positioned at the failure plane, as shown in Figure F-57. The distributed load portion of the HL-93 load is applied along the entire span.

As described in the proposed guide specification in Appendix E, the transverse positioning of the HL-93 live load model differs between the Redundancy I and Redundancy II load combinations, as illustrated in Figure F-58. Since the vehicular loads in the Redundancy I load combination are meant to represent the applied load at the instant in time in which the assumed member failure occurs, the HL-93 vehicular live load model is transversely positioned centered (both the 10 ft loaded width and the truck axle loads) within the marked (striped) lanes, in this case three lanes. Hence, as the bridge is striped for three lanes, there are three load cases for the Redundancy I load combination: three marked lanes loaded, two marked lanes loaded, or one marked lane loaded.

On the other hand, the objective of the Redundancy II load combination is to evaluate the strength of the system after the failure of the primary steel tension member has occurred, so the number of design lanes is established in accordance with Article 3.6.1.1.1 in the AASHTO LRFD BDS, which in this case results in four design lanes with a width of 12 ft. In the Redundancy II load combination, the HL-93 vehicular live

load model is transversely positioned (both the 10 ft loaded width and the truck axle loads) to produce extreme force effects within each design lane; however, the truck axle loads are transversely positioned such that the center of any wheel load is not closer than 2 ft from the edge of the design lane. Hence, there are four load cases for the Redundancy II load combination: four design lanes loaded, three design lanes loaded, two design lanes loaded, or one design lane loaded.

Component dead loads were linearly applied in the initial implicit static analysis. Traffic live loads were applied in the final explicit dynamic analysis. Their dynamic effects were minimized by applying them slowly through the use of smooth step, as in the following equation:

$$LR(t) = 6\left(\frac{t}{T}\right)^5 - 15\left(\frac{t}{T}\right)^4 + 10\left(\frac{t}{T}\right)^3$$

where $LR(t)$ is the fraction of load at a load application time t, and T is the duration of the load application. The duration of the load application must be larger than the fundamental period of the structure to minimize oscillatory behavior in the final explicit dynamic analysis.

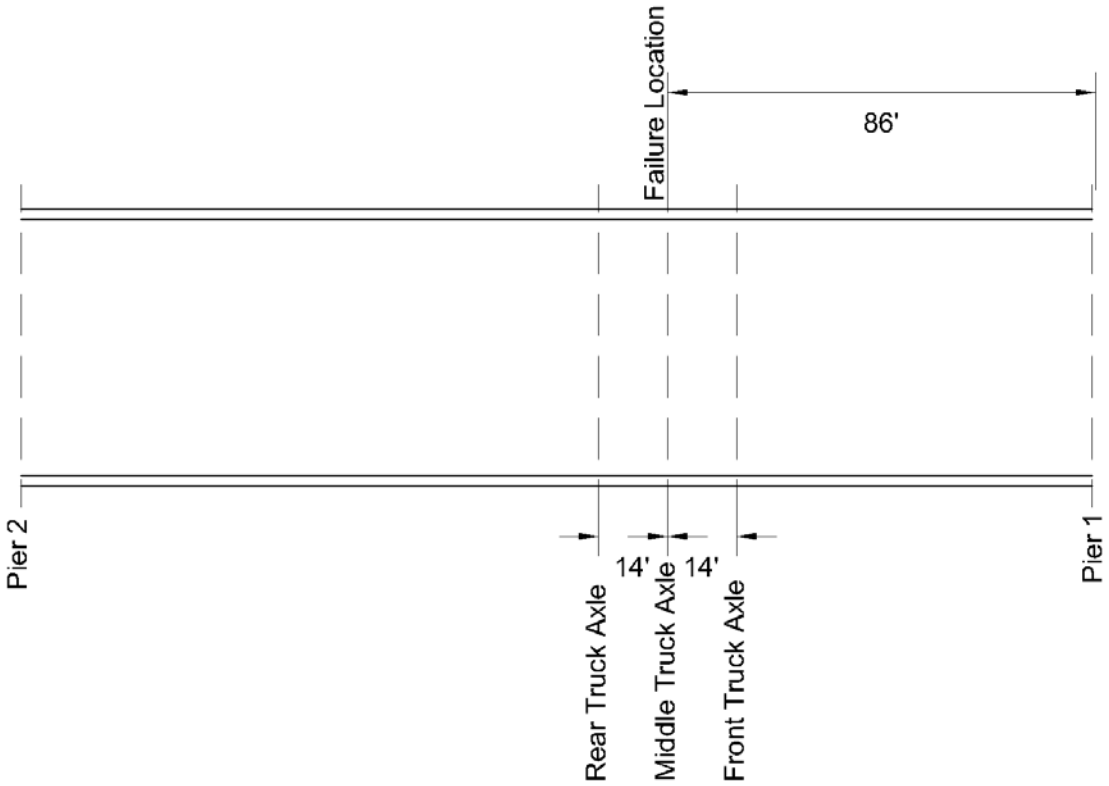


Figure F-77. Longitudinal position of truck axles.

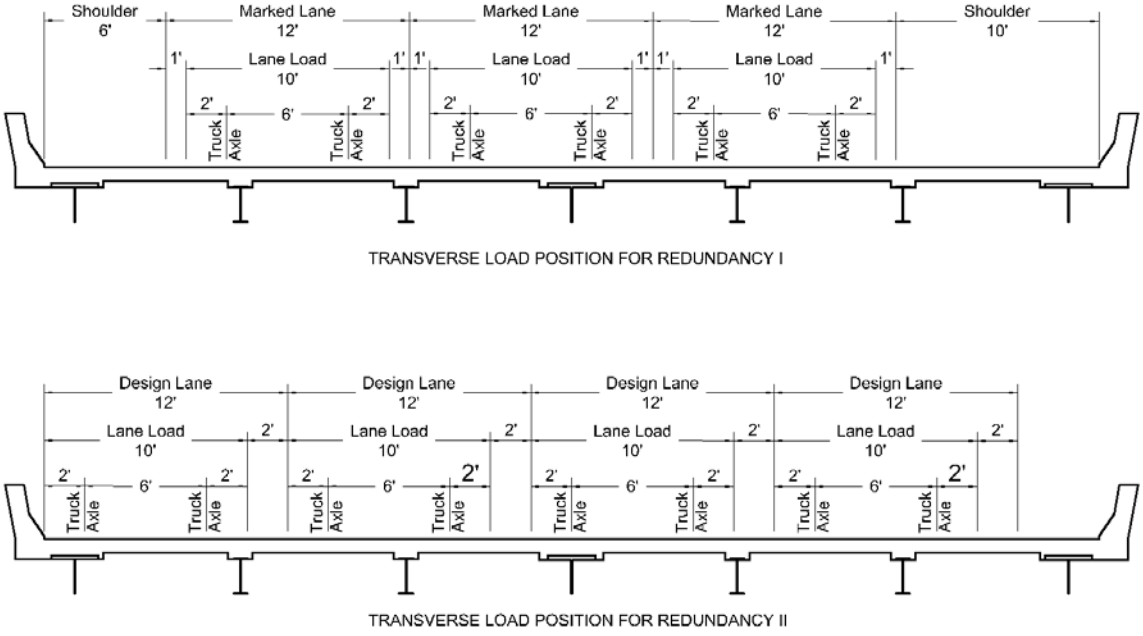


Figure F-78. Transverse position of traffic loads. Fracture is girder is leftmost girder.

F.5.7 Analysis of Results for Redundancy

Once the analysis is completed the obtained results are evaluated using the requirements described in Article 8 of the proposed guide specification in Appendix E. It was found that the structure met the strength and serviceability requirements and is considered redundant against failure of the exterior westernmost girder. Specific details regarding the performance requirements and the results are summarized in Table F-25.

1933 **Table F-25. Summary of the redundancy evaluation.**

Performance Requirement		Most Critical Load Combination	Result	Acceptable?
Strength Requirements	Strain Primary Steel Members	Redundancy I 3 Lanes	0.0022	**YES**
	Slab Concrete Crushing	-	No concrete crushing in the slab	**YES**
Serviceability Requirements	Vertical Deflection	Only Redundancy II DL considered	10.3 inches	**YES**

Notes:
1. The analysis showed that the structure was capable of resisting an additional 15% of the applied factored live load.
2. In order to complete the evaluation, the displacements and reaction forces calculated at support locations should be used as factored demands to check against the nominal capacity of the supports and substructure members.

1934

1935 *F.5.7.1 Minimum Strength Requirements*

1936 All of the strength requirements were met by the system in the faulted state while subjected to any one
1937 of the load cases included in the Redundancy I and Redundancy II load combinations. Since the system
1938 met all of the strength requirement it may be re-designated as a system redundant member (SRM) as soon
1939 as the minimum serviceability requirements are met; otherwise it shall remain designated a fracture critical
1940 member (FCM).
1941 The first set of strength requirements apply to any primary member of the superstructure, which in this
1942 case are the girders, and concrete slab. These requirements are the following:
1943 • In a component, such as a web or a flange of a primary steel member, the average strain is less than
1944 five times the material yield strain.
1945 • In a component, such as a web or a flange of a primary steel member, the average strain is less than
1946 0.01.
1947 • A strain level of 0.05 is not reached anywhere in a primary steel member.
1948 • The combined flexural, torsional and axial force effects computed in primary compression
1949 members are below the nominal compressive resistance of the member (these limit states are
1950 predicted by the FEA).
1951 • If a compression strain greater than 0.003 is reached in the slab, the portion where that limit is
1952 exceeded does not compromise the overall system load carrying capacity.
1953 • The system in the faulted condition is able to support an additional 15% of the factored live load.
1954 Very small and localized yielding was observed in the primary steel members, further critical buckling
1955 loads were not reached in any primary steel member. The plastic strains calculated in the girders were
1956 below 0.01 after the failure of the southernmost tie girder for the Redundancy I or Redundancy II load
1957 combinations; therefore, the strain requirements on primary steel members are met. This is illustrated in
1958 Figure F-79, in which the equivalent plastic strain is shown for the most critical load case: the Redundancy
1959 I load combination with three marked lanes loaded. As the FEA accurately predicts potential failure of
1960 primary steel compression member subjected to combined flexural, torsional, and axial force effects, and
1961 quasi-static equilibrium is reached for both load combinations, the requirements for primary steel
1962 compression members are met.
1963

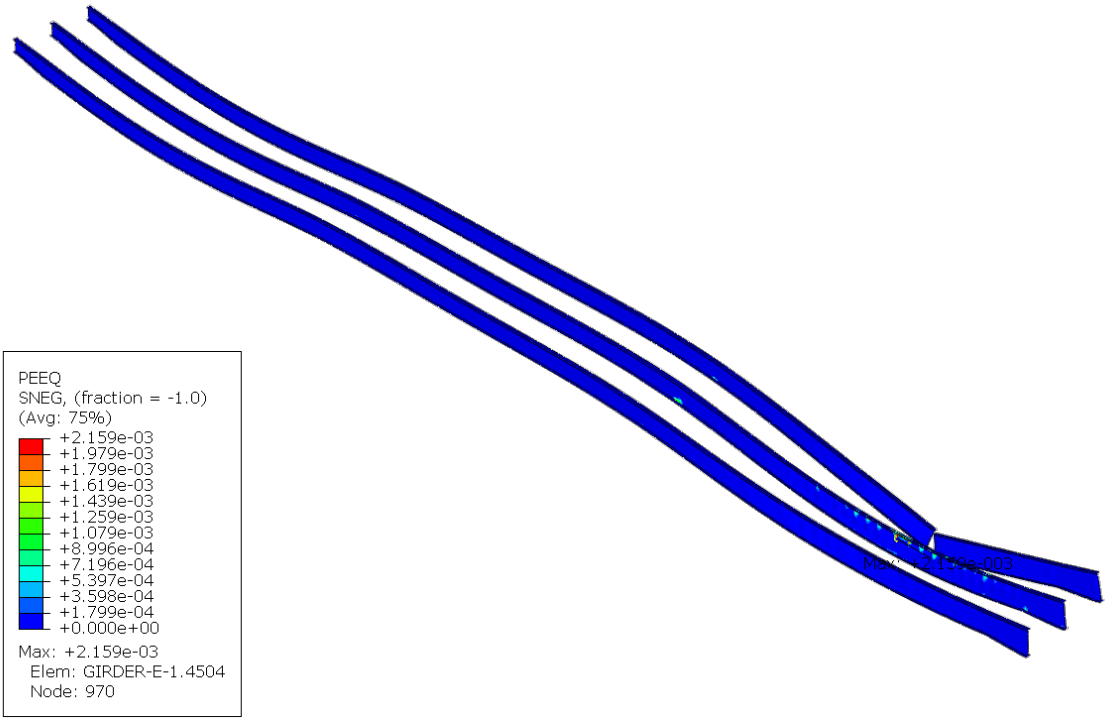


Figure F-79. Location of maximum plastic equivalent strain in primary steel members.

Regarding the concrete slab, concrete crushing and tension cracking is allowed and expected to take place. However, if the portion of the slab where a total compressive strain of 0.003 has been exceeded is large enough to compromise the overall system load carrying capacity or if significant hinging occurs, the structure should not be considered as sufficiently redundant. In this example, the Redundancy II load combination, with four design lanes loaded, resulted in the largest compressive strains in the slab, which were located in the haunches over the middle girder near the failure location. However these strains do not reach 0.003 as shown in Figure F-80; thus, it was not enough to result in a reduction in load carrying capacity.

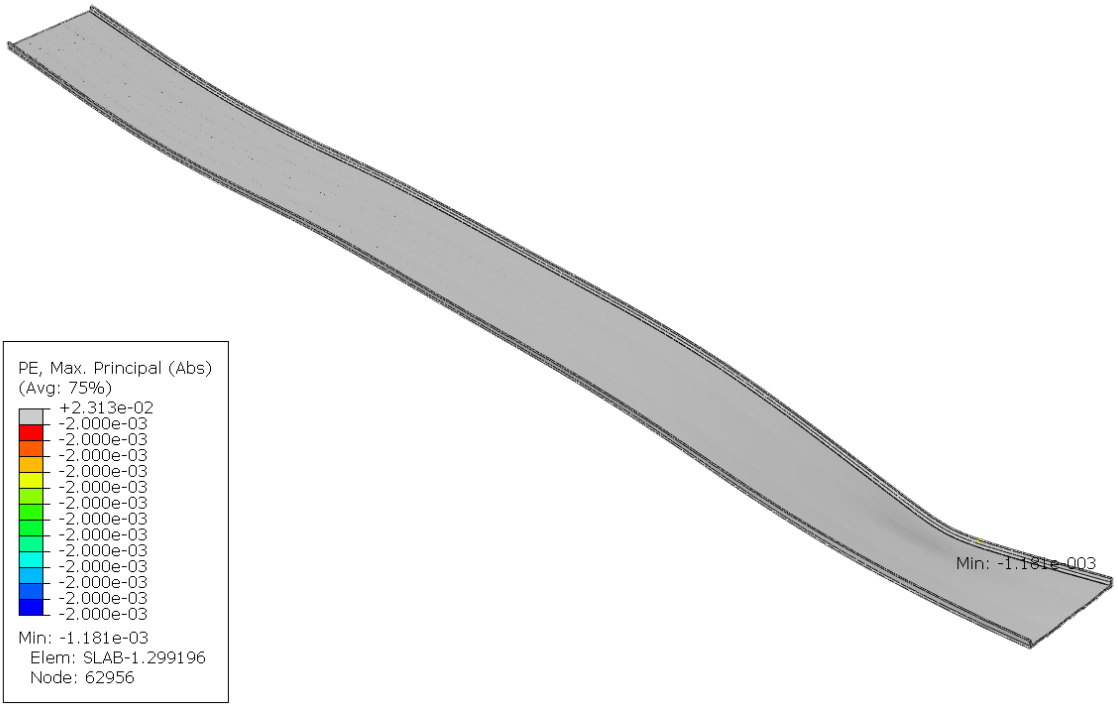


Figure F-80. Absence of concrete crushing in slab.

Although the substructure is not explicitly included in the finite element model, the reaction forces at support locations are calculated in the analysis. These should be taken as the factored demands that the substructure must be able to safely sustain, which are summarized in Table F 20. In this case, the largest vertical reaction forces were calucalted for the Redundancy I load combination. The calculated longitudinal and transverse reaction forces are small for both loading combinations. The unfactored nominal capacity of the members of the susbtructure needs to be checked against these load demands. Similarly these elements of the substructure must accommodate the horizontal displacements that are calculated in the analysis at the support locations. In this example, Redundancy I and Redundancy II load combinations resulted in similar small horizontal displacements which are summarized in Table F 21.

1988 **Table F-26. Calculated reaction forces for redundancy evaluation. All results in kips.**

Support	Girder	Reaction Force	Redundancy I			Redundancy II			
			1 Lane	2 Lanes	3 Lanes	1 Lane	2 Lanes	3 Lanes	4 Lanes
Pier 1	West	Vertical	298	321	315	235	275	276	254
		Longit.	7.90	-10.4	-51.6	12.6	-3.46	-25.8	-55.6
		Transv.	-2.48	3.54	19.9	-7.78	-5.77	3.34	18.7
	Middle	Vertical	902	957	989	726	788	830	811
		Longit.	12.8	2.05	-50.2	16.4	9.62	-7.60	-58.6
		Transv.	-1.96	-0.818	1.46	-5.22	-4.58	-3.22	-0.508
	East	Vertical	315	343	397	178	194	237	297
		Longit.	7.69	7.94	-15.5	7.95	9.54	8.95	-17.2
		Transv.	-0.267	-6.18	-17.4	-0.0554	-3.43	-10.4	-19.3
Pier 2	West	Vertical	1410	1441	1424	1115	1181	1173	1102
		Longit.	-8.90	-0.241	30.3	-11.6	-3.52	6.70	33.8
		Transv.	-1.65	1.43	2.82	1.99	4.74	6.30	4.10
	Middle	Vertical	1805	1893	1941	1350	1443	1511	1492
		Longit.	-6.17	-2.33	28.4	-7.70	-3.90	3.68	33.5
		Transv.	2.17	-1.85	-6.80	7.34	4.82	1.61	-4.23
	East	Vertical	1108	1147	1211	738	761	815	892
		Longit.	-4.22	-2.30	17.8	-4.75	-3.31	0.114	20.1
		Transv.	3.18	2.25	-5.91	5.65	7.33	6.30	-1.44
Pier 3	West	Vertical	1068	1060	1059	738	725	722	735
		Longit.	-0.512	0.888	-1.55	-0.621	-0.313	0.892	-1.35
		Transv.	2.88	3.07	3.25	3.74	3.32	1.97	1.60
	Middle	Vertical	1446	1433	1427	1022	1007	999.2	1004
		Longit.	-0.285	0.872	-0.127	-0.456	-0.167	1.10	0.266
		Transv.	2.84	3.56	6.22	3.53	3.95	3.97	6.40
	East	Vertical	1150	1137	1128	827	814	803	795
		Longit.	-0.117	0.665	0.959	-0.312	-0.0765	1.06	1.48
		Transv.	1.87	2.25	2.16	2.55	2.59	2.11	2.65
Pier 4	West	Vertical	559	561	563	400	403	404	404
		Longit.	-3.57	1.36	17.2	-4.84	-1.70	4.39	17.69
		Transv.	-2.18	-1.91	0.486	-4.19	-4.35	-3.24	-1.13
	Middle	Vertical	734	737	738	526	530	530	530
		Longit.	-3.15	0.864	14.0	-4.28	-1.68	3.47	14.7
		Transv.	-2.25	-2.64	-1.86	-3.99	-4.48	-3.81	-2.35
	East	Vertical	561	563	564	402	405	406	407
		Longit.	-2.14	0.443	9.58	-2.92	-1.25	2.14	10.4
		Transv.	-2.02	-2.55	-3.28	-3.50	-3.79	-3.27	-3.10

Notes:
1. Positive longitudinal direction points north.
2. Positive transverse direction points east.

1989
1990

Table F-27. Calculated displacements at support locations for redundancy evaluation. All results in inches.

Support	Girder	Reaction Force	Redundancy I			Redundancy II			
			1 Lane	2 Lanes	3 Lanes	1 Lane	2 Lanes	3 Lanes	4 Lanes
Pier 1	West	Longit.	-0.0451	0.0594	0.295	-0.0719	0.0198	0.147	0.318
		Transv.	3.13E-3	-4.47E-3	-0.0252	9.84E-3	7.30E-3	-4.22E-3	-0.0236
	Middle	Longit.	-0.0844	-0.0135	0.331	-0.108	-0.0632	0.0500	0.385
		Transv.	2.61E-3	1.09E-3	-1.94E-3	6.94E-3	6.08E-3	4.29E-3	6.75E-4
	East	Longit.	-0.0719	-0.0742	0.145	-0.074	-0.0891	-0.0836	0.161
		Transv.	3.71E-4	8.56E-3	0.0241	7.67E-5	4.74E-3	0.0145	0.0267
Pier 2	West	Longit.	0.0748	2.02E-3	-0.254	0.0972	0.0295	-0.0563	-0.284
		Transv.	3.25E-3	-2.81E-3	-5.55E-3	-3.92E-3	-9.32E-3	-0.0124	-8.07E-3
	Middle	Longit.	0.0610	0.0231	-0.281	0.0762	0.0386	-0.0364	-0.332
		Transv.	-4.48E-3	3.81E-3	0.0140	-0.0151	-9.93E-3	-3.33E-3	8.71E-3
	East	Longit.	0.0620	0.0337	-0.262	0.0699	0.0486	-1.67E-3	-0.295
		Transv.	-7.27E-3	-5.14E-3	0.0135	-0.0129	-0.0168	-0.0144	3.29E-3
Pier 3	West	Longit.	3.69E-3	-6.39E-3	0.0112	4.46E-3	2.25E-3	-6.41E-3	9.74E-3
		Transv.	-4.54E-3	-4.84E-3	-5.12E-3	-5.89E-3	-5.23E-3	-3.10E-3	-2.52E-3
	Middle	Longit.	2.42E-3	-7.39E-3	1.08E-3	3.87E-3	1.41E-3	-9.29E-3	-2.25E-3
		Transv.	-4.71E-3	-5.91E-3	-0.0103	-5.86E-3	-6.57E-3	-6.60E-3	-0.0106
	East	Longit.	1.49E-3	-8.49E-3	-0.0122	3.98E-3	9.76E-4	-0.0135	-0.0188
		Transv.	-3.53E-3	-4.25E-3	-4.07E-3	-4.80E-3	-4.89E-3	-3.97E-3	-5.00E-3
Pier 4	West	Longit.	0.0222	-8.42E-3	-0.107	0.0301	0.0106	-0.0273	-0.110
		Transv.	2.89E-3	2.53E-3	-6.44E-4	5.54E-3	5.76E-3	4.29E-3	1.49E-3
	Middle	Longit.	0.0232	-6.36E-3	-0.103	0.0315	0.0124	-0.0255	-0.108
		Transv.	3.15E-3	3.70E-3	2.61E-3	5.60E-3	6.29E-3	5.35E-3	3.31E-3
	East	Longit.	0.024	-4.99E-3	-0.108	0.0328	0.0141	-0.0241	-0.117
		Transv.	3.23E-3	4.07E-3	5.23E-3	5.59E-3	6.05E-3	5.22E-3	4.95E-3

Notes:
1. Positive longitudinal direction points north.
2. Positive transverse direction points east.

Additionally, the system demonstrated a reserve margin of at least 15% of the applied live load in the Redundancy I and II load combinations. Effectively, this requirement ensures the slope of the load vs displacement curve for the system structure remains positive (i.e., there is still significant remaining reserve capacity).

F.5.7.2 Minimum Serviceability Requirements

The only serviceability requirement in the Appendix E is that the increase of deflection after the failure of a primary steel tension member cannot be greater than L/50. This requirement is to be checked in the Redundancy II load combination under factored dead load only. In the current case, the limit is 52 inches, which was not surpassed since the maximum additional deflection computed in the FEA was 10.3 inches. This is illustrated in Figure F-81.

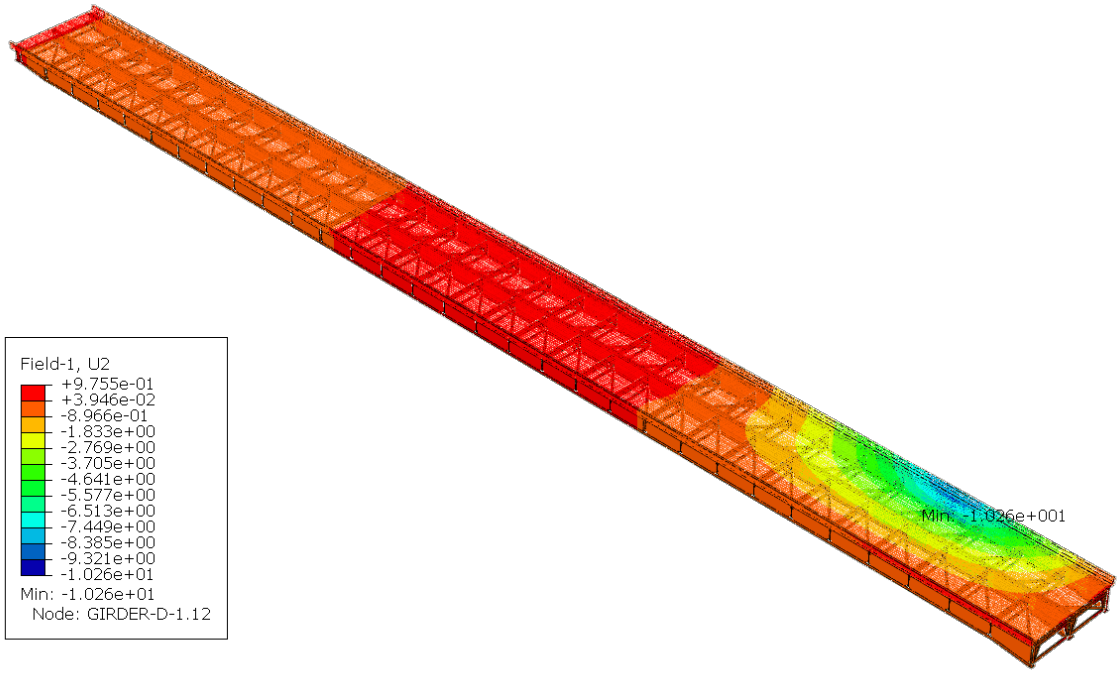


Figure F-81. Deflection after failure of primary steel tension member.

F.5.8 Conclusions

The redundancy of a straight continuous three-span three-girder bridge after the failure of the exterior girder was analyzed in accordance with the methodology described in the proposed guide specification in Appendix E. Based on the comparison between the calculated results and the minimum performance requirements, the structure is not likely to fail nor undergo a significant serviceability loss as result after the failure of the exterior tub girder. Hence, the westernmost girder may be re-designated as a system redundant member (SRM).