

Article

Not peer-reviewed version

Bridge Design Using Finite Element Method (FEM) Software

[Harry Smith](#) *

Posted Date: 28 April 2025

doi: 10.20944/preprints202504.2346.v1

Keywords: Finite Element; Bridge Design



Preprints.org is a free multidisciplinary platform providing preprint service that is dedicated to making early versions of research outputs permanently available and citable. Preprints posted at Preprints.org appear in Web of Science, Crossref, Google Scholar, Scilit, Europe PMC.

Copyright: This open access article is published under a Creative Commons CC BY 4.0 license, which permit the free download, distribution, and reuse, provided that the author and preprint are cited in any reuse.

Article

Bridge Design Using Finite Element Method (FEM) Software

Harris Smith

Independent Researcher; harrismithengr1@gmail.com

Abstract: Finite Element Method (FEM) has become integral to modern bridge engineering, allowing complex structures to be analyzed with high precision. This paper provides an in-depth overview of FEM principles as applied to bridge design and structural analysis. The capabilities of leading FEM software (including MIDAS Civil, SAP2000, ANSYS, LUSAS, and CSI Bridge) are discussed, highlighting features such as advanced modeling tools, comprehensive load analysis (e.g. moving loads, seismic, wind), design code compliance checks, and construction stage simulation. Real-world case studies of major bridges are presented to illustrate how FEM software is employed in practice. The paper also examines modeling techniques (e.g. choice of element types and level of detail), load application methods, and the process of verifying designs against engineering codes. Advantages of using FEM for bridge design (such as accurate representation of complex geometry and load effects) are weighed against its limitations (such as modeling assumptions and computational demands). Finally, recent developments and future directions are explored, including the integration of artificial intelligence (AI) for design optimization and the emergence of digital twin technology for bridge health monitoring. The discussion is presented in a structured format with a professional tone, supported by figures, tables, and references to scholarly and industry sources.

Keywords: finite element; bridge design

Introduction

Bridge design is a complex discipline in civil engineering that demands both creativity and rigorous analysis. Engineers must ensure that bridges can safely carry various loads (from vehicles, pedestrians, wind, earthquakes, etc.) while meeting criteria for strength, serviceability, and longevity. Traditional analysis methods (such as simplified beam models or hand calculations) often fall short when dealing with intricate geometries or load patterns of modern bridges. The Finite Element Method (FEM) addresses these challenges by breaking down a complex structure into smaller, manageable elements, enabling detailed simulation of structural behavior under numerous scenarios. Over the past few decades, FEM-based software tools have evolved into essential everyday instruments for bridge engineers, thanks to advances in computing power and numerical methods. These tools allow engineers to model everything from simple span bridges to record-breaking long-span suspension and cable-stayed bridges with a high degree of confidence in the analytical results. This paper provides a comprehensive review of how FEM is applied in bridge design, covering fundamental principles, software platforms, practical modeling and analysis techniques, case studies, and emerging trends such as AI-driven design and digital twins.

Finite Element Principles in Bridge Engineering The finite element method is a numerical technique for solving structural mechanics problems by discretizing a structure into many small, interconnected components (finite elements). In the context of bridge engineering, the bridge is idealized as an assembly of elements (such as beam segments, shell patches of deck, solid volumes for foundations, or truss elements for cables) connected at nodes. Each element has its own simple equations governing its behavior, and when assembled together they form a global system representing the entire bridge. By solving the system of algebraic equilibrium equations, the FEM yields approximate solutions for unknowns like displacements at nodes and forces in members. In

essence, FEM allows engineers to simulate how a bridge will deform and what stresses will develop under various loads and boundary conditions.

FEM is especially powerful for bridges because of their complex geometries and load interactions. It enables an *accurate representation of complex geometry*, the *inclusion of different materials* (e.g. steel, concrete, composite sections) within one model, and the *capture of local effects* like stress concentrations that simpler methods might miss. For example, a suspension bridge's cables, towers, and deck can all be modeled with appropriate element types to capture their distinct behaviors, yet have them act together as one structural system. FEM also handles the combination of multiple load types simultaneously (dead load, live traffic load, wind, temperature, seismic, etc.), providing a complete picture of the bridge response.

It is important to note that FEM solutions are approximate and depend on the assumptions made in modeling. Bridge engineers using FEM must understand the underlying theories and assumptions to ensure the model is valid. For instance, assuming linear material behavior (elasticity) and small displacements might be reasonable for many bridge analyses, but significant nonlinear effects (like concrete cracking, cable sagging, or large deflections in very flexible bridges) require more advanced nonlinear FEM analysis. Despite these considerations, FEM has proven to be a reliable workhorse in bridge design, with its results forming the basis of design decisions and code compliance checks in projects worldwide. Studying or analyzing a bridge with FEM is often referred to as performing a finite element analysis (FEA), and this has become synonymous with cutting-edge structural analysis in civil engineering.

Leading FEM Software for Bridge Design

A variety of FEM software packages are available to structural engineers, but only a handful are particularly tailored or widely used for bridge design and analysis. These programs provide specialized tools to model bridge components and apply bridge-specific loads and code checks. Below, we discuss some of the leading software platforms and their capabilities in the context of bridge engineering.

MIDAS Civil

MIDAS Civil is a dedicated bridge analysis and design software that has become one of the top choices for bridge engineers worldwide. It combines an intuitive modeling interface with powerful analysis features geared specifically toward bridges. MIDAS Civil is often praised for its user-friendly graphical interface and *bridge-oriented modeling wizards*, which allow engineers to rapidly assemble common bridge types (such as slab bridges, girder bridges, or cable-stayed bridges) by inputting key parameters. The software can auto-generate an entire bridge model "in the blink of an eye" using these wizards, which can then be refined as needed.

In terms of analysis capabilities, MIDAS Civil offers a comprehensive suite that covers everything from basic static analysis to advanced simulations. It can perform moving load analysis for traffic loads, construction stage analysis, modal (eigenvalue) analysis, and both linear and nonlinear static analysis. For more complex bridge scenarios, it also supports buckling analysis, cable force tuning (for cable-supported bridges), rail-structure interaction analysis, and soil-structure settlement analysis. This means engineers can study phenomena like the stability of a long-span arch, the optimum tension in cable stays, the effect of train braking on a rail bridge, or differential settlement of supports within one integrated tool.

MIDAS Civil also integrates design code provisions and automated checks. It includes a library of international bridge design codes (such as AASHTO LRFD, Canadian CHBDC, Eurocodes, etc.) and can carry out code-based design checks and load rating within the program. For instance, after an analysis, an engineer can have MIDAS Civil automatically evaluate whether each steel girder meets AASHTO LRFD stress and deflection limits, or determine the load rating factor for an existing bridge for posting purposes. The software's output can produce detailed reports with calculations, code references, and even formulas, giving engineers confidence in the results and saving significant

time in manual verification. MIDAS Civil's emphasis on bridges, from modeling through analysis to design, makes it a "one-stop" solution for bridge projects of all scales.

SAP2000

SAP2000 is a general-purpose structural analysis and design program that has been an industry standard for decades. Introduced over 45 years ago, the name SAP2000 is synonymous with state-of-the-art analytical methods in structural engineering. While SAP2000 is not exclusively for bridges, it is widely used for bridge analysis, especially for complex or unusual structures that may not fit into preset templates of bridge-only software. SAP2000 provides a single user interface for modeling, analysis, design, and reporting of structures, which many engineers appreciate for its consistency and efficiency.

One of SAP2000's strengths is its advanced modeling and meshing capabilities. Engineers can create 3D models of bridges using beam, shell, solid, or link elements and have fine control over mesh generation. SAP2000 allows merging of independently defined meshes between different object types (frame, shell, solid), ensuring compatibility and refinement where needed. This is useful, for example, if a bridge deck is modeled with shell elements while the girder is a beam element – the software can mesh them in a way that they connect correctly. SAP2000 also supports sophisticated material models and section definitions (its Section Designer utility can handle custom cross-sections, including nonprismatic or composite sections, with nonlinear properties).

In terms of analysis, SAP2000 can handle linear and nonlinear static and dynamic analyses. It uses the SAPFire analysis engine, which supports multiple 64-bit solvers and can perform eigen and Ritz analyses for modal extraction. Time-history analyses, geometric nonlinearity (large displacements, $P-\Delta$ effects), and various forms of nonlinear hinges or springs are available for advanced studies. SAP2000 also includes a wide range of design code checks for different materials – it can design steel frames, concrete frames, cold-formed steel, aluminum structures, etc., according to many international codes. While not specialized solely for bridges, SAP2000's broad capabilities mean it has been used for numerous bridge projects, from small pedestrian bridges to large cable-supported bridges, particularly when custom modeling or unique analysis approaches are required.

ANSYS

ANSYS is a high-end finite element analysis software well known in multiple engineering fields (structural, mechanical, aerospace, etc.) for its powerful simulation capabilities. In civil engineering, ANSYS is often employed for detailed analysis of bridges when complex phenomena need to be studied at a granular level. Unlike MIDAS or SAP2000, ANSYS is not a turnkey bridge design package with built-in code checks; rather, it is a general FEA tool that offers tremendous flexibility in modeling and solving but often requires more manual setup for civil applications.

One of the key advantages of ANSYS is its ability to handle advanced material models and multiphysics coupling. Engineers can simulate not just linear elastic behavior, but also nonlinear material behavior (plasticity of steel, cracking and crushing of concrete, etc.), contact interfaces (e.g. friction between bridge deck segments or bearings), and even transient phenomena like impact or explosion effects on bridge components. ANSYS's suite of structural analysis tools enables simulation of shock and vibration, impact and crash scenarios, thermal effects, and other detailed responses. For example, ANSYS has been used to perform 3D simulations of long-span bridge aerodynamics (fluid-structure interaction with wind) or to assess the progressive collapse behavior of bridges under extreme events.

In bridge design practice, ANSYS is often used in tandem with other tools: an engineer might use SAP2000 or MIDAS for the overall structural design and code compliance, and then use ANSYS to dive deeper into specific issues such as the fatigue life of a welded connection or the local stress distribution in an anchorage zone. ANSYS provides a rich set of element types (from 1D beams to 3D solid elements, including special cable elements and shell elements suited for thin-walled sections) which can be mixed to create a detailed bridge model. The software's high level of customization (scripting, APDL, or Workbench for parametric studies) allows automation and optimization as well.

Engineers can run parametric analyses to examine multiple design scenarios quickly, which helps in refining designs for performance and efficiency. For instance, ANSYS was utilized in a study to create a *digital twin* of the Golden Gate Bridge by Ozen Engineering, where a reduced-order model of the bridge was developed to predict its behavior under varying wind loads. This demonstrates ANSYS's capability to integrate with cutting-edge workflows like digital twins.

Overall, ANSYS's strength lies in solving very complex structural problems with high accuracy. Its limitation in a bridge design office context is that it does not inherently include bridge design codes or quick templates – it requires more expertise and time to set up models. However, for critical projects where detailed understanding of behavior is required (such as long-span bridges pushing the limits of design, or forensic analysis of failures), ANSYS is an indispensable tool.

LUSAS

LUSAS Bridge is a specialized finite element analysis package specifically developed for bridge engineering. It has a strong reputation in the industry, particularly for advanced analysis of large or unusual bridge structures. LUSAS is routinely used for all types of architecturally complex bridges – for example, those with **slender or curved shapes**, or where **dynamic loading is important**. This makes it popular for long-span cable-stayed bridges, suspension bridges, arch bridges, and other iconic designs that require meticulous analysis of effects like wind-induced vibrations, seismic response, or pedestrian-induced vibrations.

One of LUSAS's hallmark features is its advanced nonlinear and dynamic analysis capabilities. It offers specialized solvers for geometrical nonlinearity (large displacements), material nonlinearity (like concrete cracking, steel yielding), and transient dynamics. LUSAS has been used in projects for seismic analysis of bridges and for studying construction sequences in detail. For instance, engineers can use LUSAS to simulate the construction of a cable-stayed bridge segment-by-segment, including tensioning of cables and the gradual application of dead load, to ensure the final geometry and stresses match the design intent. It also has tools for analyzing vibration and impact (important for railway bridges or footbridges subjected to rhythmic loading).

The software provides a library of bridge-specific elements and supports influence line analysis and moving vehicle load generation for highway and railway bridges. LUSAS's modeling environment is flexible – users can create beam models, shell models, or solid models of bridges. In fact, it allows multi-scale modeling: a global model might use beam elements for efficiency, whereas a local submodel (cut out of the global model) might use detailed shell or solid elements to examine a critical region (such as a steel girder connection or a concrete anchorage block) in greater detail. This approach of local refinement is valuable in bridge design to ensure both global performance and local safety.

Many **real-life major bridges have been analyzed or designed using LUSAS**, as evidenced by its extensive case study library (projects spanning from the UK's Second Severn Crossing to long-span bridges in Asia). Engineers turn to LUSAS when they need a *trusted* tool for complex scenarios, and its results have been validated against field measurements and independent checks in numerous instances. In summary, LUSAS Bridge software is known for its innovative analysis options, flexibility in modeling, and reliability, making it a go-to for challenging bridge engineering problems.

CSI Bridge

CSI Bridge (also known as CSiBridge) is a bridge-specific analysis and design program from Computers and Structures, Inc. (the makers of SAP2000). CSI Bridge can be thought of as a version of SAP2000 that has been enhanced and streamlined for bridge applications. It offers a **single user interface to perform modeling, analysis, design, scheduling, load rating, and reporting** specifically tailored to bridge projects. This integration means that bridge engineers can carry a project from conception to final rating without leaving the software environment.

CSI Bridge includes powerful parametric modeling features. It provides **Quick Bridge Templates** that allow users to start with a basic template for common bridge types (such as a multi-span box-girder highway bridge, a precast I-girder bridge, etc). These templates generate a baseline

model which can then be modified by the engineer. The software supports a wide array of parametric bridge components, including different types of girder cross-sections (concrete box, precast I/U girders, steel I/U girders, etc.), all of which can be customized. Changing a parameter (like deck width or girder spacing) will automatically update the entire model geometry, which is very convenient for design iteration. Additionally, CSI Bridge can model bridge substructures in detail – piers (bents), abutments, bearings, and foundations can be represented. Foundation flexibility can be incorporated using spring elements (with options for 6×6 stiffness matrices or nonlinear p-y curves for soil springs) to simulate soil-structure interaction.

On the analysis side, CSI Bridge supports all the analysis types available in SAP2000 and more. It can run static analysis, seismic analysis (modal, response spectrum, time history), moving load analysis, and staged construction analysis, among others. Notably, CSI Bridge has dedicated routines for **vehicle loading**: engineers can define traffic lanes and vehicle classes, and the software will generate the worst-case load placements (in influence-based or brute-force pattern search) for moment, shear, etc., on the bridge. Dynamic vehicle effects can also be analyzed by moving a vehicle across the bridge with a specified speed to simulate impact and bounce, which is useful for vibration checks. CSI Bridge’s dynamic analysis capabilities include computing vibration modes (using eigen or Ritz vectors), linear or nonlinear time-history analysis (for earthquakes or special loads), and even steady-state vibration analysis for things like machine-induced vibrations.

In terms of design, CSI Bridge automates many code checks specifically for bridges. It includes design modules for steel and concrete bridge members per AASHTO and other standards. For example, CSI Bridge can perform steel girder design (calculating required section moduli, checking stresses and deflections, etc.) and concrete pier design (column capacity checks, etc.) within the program. It also has a load rating feature to evaluate the bridge for various rating vehicles according to code. The program outputs design results and rating factors in a user-friendly report.

CSI Bridge essentially brings together the robust analysis engine of SAP2000 with bridge-specific knowledge (templates, vehicles, code checks, and rating). This makes it highly efficient for bridge design offices, as routine highway bridge designs can be done more quickly than building them from scratch in a general program. Furthermore, because it’s built on SAP2000’s platform, it retains the ability to dive into advanced analysis if needed. Many state Departments of Transportation and bridge consulting firms use CSI Bridge for the design and load rating of new and existing bridges.

Table 1: A summary comparing the focus and features of leading FEM software for bridge design. Each software has its niche: MIDAS Civil and CSI Bridge provide all-in-one solutions tailored for bridge engineers, SAP2000 offers broad applicability with proven solvers, ANSYS enables deep dives into advanced phenomena, and LUSAS excels in handling the most challenging bridge analysis scenarios.

Table 1. Comparison of Key Features in Leading Bridge FEM Software.

Software	Focus / Strengths	Key Bridge-Specific Capabilities
MIDAS Civil	Bridge-oriented; user-friendly modeling	Bridge type wizards; moving load & staged construction analysis; integrated code checks (AASHTO, Eurocode, etc)
SAP2000	General structural analysis (widely used in bridges)	Versatile modeling (beam/shell/solid); advanced meshing; broad code design checks (steel, concrete, etc.)
ANSYS	High-fidelity FEA; advanced nonlinear analysis	Detailed 3D modeling of complex effects; custom simulations (e.g. wind-structure or blast analysis); integration for digital twins
LUSAS	Specialized for bridges; complex and dynamic analysis	Nonlinear and dynamic analysis for long-span bridges; influence lines and moving loads; detailed modeling of unusual geometry

CSI Bridge	Bridge-specific SAP2000 Parametric bridge templates; lane-based moving variant; integrated loads (static & dynamic); bridge design and load workflow rating in-program
------------	--

Modeling and Analysis Techniques in Bridge Design

The effectiveness of an FEM analysis for a bridge depends heavily on the modeling techniques and analysis procedures employed. Engineers must make thoughtful choices about how to represent the physical bridge in the virtual model – balancing detail with simplicity – and how to apply loads and boundary conditions consistent with reality. This section covers key aspects of modeling (element types and level of detail), load application and analysis methods, checking results against code requirements, and simulating the construction process.

Structural Modeling Approaches

Choosing the right element types and model fidelity is a crucial first step in bridge FEM. Depending on the complexity of the structure and the information needed, engineers may opt for different modeling approaches. For many typical bridges (e.g. girder bridges, trusses), a *beam-element model* is often sufficient for global analysis: the bridge’s girders, deck, and piers are represented as line elements with appropriate properties, capturing bending and axial forces. Such models are computationally efficient and provide results in terms of member forces (moments, shears, axial forces) that align neatly with design calculations. Figure 8.2 in the Bridge Engineering Handbook illustrates a typical beam model, where each element corresponds to a portion of the bridge and force resultants (axial, shear, bending) describe the internal demands. These simplified models make it easy to extract design forces for components and are often used in preliminary design and even final design of standard bridges.

For more complex bridges or local areas of interest, a higher-fidelity model may be necessary. Two-dimensional (plate/shell) or three-dimensional solid elements can capture behavior that beam models cannot. For instance, in a curved steel box-girder bridge, torsional warping and distortional stresses in the box cross-section are significant; a shell element model of the girder and deck can reveal these effects, whereas a simple beam model might not. Similarly, modeling a concrete slab with shell elements can help determine realistic distribution of wheel loads and identify localized bending in the slab between girders. Engineers often use **grillage models** (a network of beam elements representing deck and girders) for distributing loads, or shell models for more continuous slab representation – each approach has its own assumptions.

Hybrid modeling is also common: using beam elements for parts of the structure and shell or solid elements for others. An example would be a cable-stayed bridge where the deck is modeled as a beam for global analysis, but the complex pylon anchorage zone is modeled with solid elements in a sub-model to investigate stress flow around anchor bolts. The global beam model provides forces and displacements at the boundaries of the sub-model, which are then applied to the detailed model for local stress analysis. This sub-modeling technique allows engineers to zoom in on critical details without making the entire model excessively large.

Boundary conditions and supports must be modeled to reflect reality. Bridges often have supports that allow some movements (e.g. expansion bearings) and restrain others. In FEM models, supports can be idealized as pinned, fixed, or springs. For example, an abutment on elastomeric bearings might be modeled as a support with a spring in the transverse direction (to simulate shear deformation of the bearings) and a free or sliding condition in the longitudinal direction (to allow thermal expansion). Foundation flexibility can be included by using spring constants or more advanced soil-structure models (like Winkler springs or continuum elements for soil). CSI Bridge, for instance, allows foundation springs defined as 6×6 stiffness matrices or nonlinear p-y curves for piles, enabling a realistic representation of how the ground supports the bridge.

Geometry considerations: Accurate geometry input is essential. This includes span lengths, curvature (horizontal curvature or vertical curvature of the bridge profile), and cross-section

dimensions. Curved and skewed bridges require careful meshing because geometry irregularity can cause asymmetric load distribution. FEM software often provide tools to generate curved geometry and mesh it appropriately. It is also important to model initial camber or pre-deflected shapes if the bridge will be built with camber (common in long spans to offset deflections). Some programs allow the user to input the fabricated cambered shape and then apply gravity load, ensuring that the computed deflected shape matches the intended final profile.

Finally, **mesh refinement** is a technical consideration: using a finer mesh (smaller elements) generally increases accuracy but at a cost of runtime. Engineers must strike a balance between accuracy and efficiency. Key locations (like mid-span of beams, points of load application, connections) often need finer mesh to capture peak responses, whereas other regions can be coarser. Modern software can automate some of this, but engineering judgment is needed to ensure that stress gradients are captured and that results have converged (i.e. further refinement would not significantly change the results). A rule of thumb is to refine the mesh until changes in peak values are minimal and the shape of stress distributions is smooth.

In summary, modeling techniques in bridge FEM range from simple (beam models) to elaborate (3D solid models). The choice depends on the bridge type and analysis goals. Often a staged approach is used: start simple to get global behavior, then refine as needed for local details. Good modeling practices and understanding of structural behavior are essential to avoid garbage-in-garbage-out – a highly refined model is only useful if it is built on correct assumptions and inputs. As one handbook emphasizes, users must recognize the *assumptions and limitations* of their model and element types. With prudent modeling, FEM can provide extremely powerful insights into bridge behavior.

Load Application and Analysis Methods

Bridges are subject to a variety of loads, and FEM software offers multiple ways to apply and analyze these loads. Key load types include: **dead loads** (self-weight of the structure and fixed attachments), **live loads** (vehicles, pedestrians, trains), **environmental loads** (wind, thermal expansion, earthquake, snow), and **special loads** (impact from vessels or vehicles, erection loads, etc.). How these loads are applied in the model must follow the intent of design codes and reality.

Dead loads are usually straightforward – the software can calculate self-weight automatically based on material density and geometry. Additional dead loads like asphalt wearing surface or sidewalks can be applied as distributed loads on the deck elements. It's important to ensure dead loads that will be present only after certain construction stages (e.g. asphalt applied at the end) are not applied too early in staged analysis (more on that in the construction stage section).

Live load (Traffic) analysis is one of the most critical and distinctive aspects of bridge design. Unlike building loads, vehicle positions can vary, and we must find the worst-case placement for the bridge effects. FEM software has specialized live load generators: for example, CSI Bridge and MIDAS Civil allow the definition of traffic lanes across the deck, along with vehicle types (truck patterns, lane loads, etc. as specified by standards like AASHTO or Eurocode). The program then automatically moves these loads across the span to compute influence lines or influence surfaces and determine the maximum effects. **Moving load analysis** in CSiBridge, for instance, considers every permutation of multiple vehicles in multiple lanes to find the extreme force responses. The result is akin to what an engineer would do with influence lines manually, but automated and exact for the FEM model. Dynamic impact factors (IM) specified by codes can be applied to live loads to account for vehicle bounce – some software do this automatically when classifying a load as “traffic with impact.”

For **dynamic analysis of moving loads**, advanced simulation can be done. CSiBridge allows vehicles with a given speed to traverse the span in time steps, which can capture dynamic amplification beyond the code's static IM factors. This is useful for long-span bridges or high-speed rail bridges where resonance or dynamic amplification is a concern.

Wind loads on bridges (especially long-span) are complex. At a basic level, wind can be applied as a static pressure on the deck and towers. FEM models can include these as lateral distributed loads. For tall bridges, wind load variation along height and across the span is considered (using code-

specified wind profiles). More advanced analysis might involve *aerodynamic* effects: calculating the bridge's natural frequencies and modes, then performing a **response spectrum or time-history analysis for wind** (or even coupling with a CFD simulation in programs like ANSYS). A notable use of FEM in wind engineering is to check for **flutter** and **buffeting** in suspension bridges. For example, the Akashi Kaikyo Bridge (with nearly 2 km span) was studied by building an FEM model and applying wind loading to evaluate its critical flutter wind speed. The FEM helped predict at what wind speed the bridge might become unstable, information essential for designing aerodynamic stabilizing measures.

Seismic loads are typically applied via response spectrum analysis or time-history analysis in bridge FEM. Codes provide design response spectra, and the FEM software can compute the bridge's likely maximum seismic response by combining mode responses (this is linear elastic seismic analysis. In high seismic zones or for important bridges, nonlinear time-history analyses might be done, where an earthquake acceleration record is input and the structure's time-domain response is computed. This allows modeling of inelastic behavior (yielding in ductile members, nonlinear bearings, etc.). Many cable-supported bridges have seismic isolation devices or dampers, which are modeled as nonlinear link elements. Software like LUSAS, SAP2000/CSiBridge provide libraries of link element types (friction pendulum bearings, viscous dampers, gap elements, etc.) to simulate these devices. For example, a base-isolated bridge could be modeled with specialized elements at the base that slip or yield after a certain force, capturing the intended seismic energy dissipation.

Thermal loads are also important: uniform temperature changes cause expansion/contraction (which induce forces if expansion is restrained), and temperature differentials (sun heating the top of the deck more than the bottom) cause curvature. FEM models handle thermal effects by specifying temperature change values to elements. A change in temperature applied to steel girders, for instance, will generate axial forces if the ends are fixed. Temperature gradients are applied as a linear variation of temperature through the depth of members. Bridge design codes specify certain thermal gradients for design, and these can be input to check resulting stresses and deflections.

Analysis types: After applying loads, the type of analysis solver to use must be decided. For most design situations, a linear static analysis (superposition of effects) is sufficient. However, when load magnitudes become large relative to stiffness (e.g. very flexible cable-supported bridges under heavy loads) or when considering stability, a geometric nonlinear analysis (sometimes called P-Delta or large deflection analysis) might be used. Nonlinear analysis is also needed if there are elements that only take one type of force (like a cable that goes slack in compression, or a soil spring that only works in compression but not tension). Cables in bridges are usually given a "tension-only" property in FEM, so the analysis will iteratively adjust forces so that cables don't go into compression – this is a nonlinear effect. Similarly, if an analysis includes cracking of concrete (with reduced stiffness after cracking), it becomes nonlinear. Most bridge FEM software can perform nonlinear static analysis; some, like MIDAS, even have specialized *cable force tuning* features to adjust initial cable tensions in the model for desired geometry.

Modal analysis is run to find the natural frequencies and mode shapes of the bridge. This is important for dynamic performance (checking if any modes fall in concerning ranges for wind or pedestrian excitation, for example). Software can compute a certain number of modes using eigenvalue solvers. The results are used for seismic spectrum analysis or just to assess dynamic characteristics.

In summary, load application in bridge FEM involves reproducing code-specified loads (vehicles, wind, quakes, temperature, etc.) as closely as possible on the model. Tools like moving load generators and response spectrum analyzers greatly assist in this. The engineer must combine load cases into load combinations per code (often software will generate these combinations automatically). With modern tools, one can perform a **comprehensive analysis** considering static loads, moving loads, and dynamic effects all within one model. For example, CSiBridge handles numerous analysis types from static, staged, moving loads to multiple forms of dynamic analysis in

one integrated package. This allows a holistic view of the bridge's structural response and ensures critical conditions are not overlooked.

Design Code Compliance Checks

After obtaining analysis results (member forces, stresses, deflections), the next step in bridge design is to check these against the requirements of design codes (such as AASHTO LRFD in the US, Eurocode in Europe, etc.). Traditionally, engineers would extract forces from the analysis and perform calculations by hand or spreadsheets to check capacities. Today's FEM software often incorporate code check modules to automate this process and seamlessly tie it into the analysis.

Integrated design modules: Programs like MIDAS Civil and CSI Bridge include built-in design code libraries. For instance, MIDAS Civil has the AASHTO LRFD 2017 specifications (and others) implemented so that it can directly evaluate if a steel or concrete member passes the code criteria. The software knows the formulas and limits from the code (e.g., for a steel I-girder, the bending stress limit = $0.55 \cdot F_y$ for LRFD Service I limit state, etc.) and can compare the analysis results to these limits. Figure 1 below illustrates a sample output from MIDAS Civil's code check for a curved I-girder bridge: the model of the bridge is shown with its FEM mesh, and a portion of the design report is visible with section properties and calculated stresses vs. allowable values for an AASHTO LRFD load combination.

Figure 1. Example of an FEM model of a curved steel bridge and an automated design check report. The output (from MIDAS Civil) shows an AASHTO LRFD code check for a steel I-girder under a positive flexure condition, listing section properties, computed stresses, and code-allowed values. Such integrated tools help engineers quickly verify that the bridge components meet code requirements.

The advantage of these integrated checks is efficiency and reduction of human error. The software will flag if any member is overstressed or if a section is too small. For concrete bridges, code checks might include capacity calculations for moment and shear, as well as crack width checks or prestressing force checks. For steel bridges, checks include classification of sections, lateral torsional buckling checks, plate buckling in plate girders, fatigue life evaluation, etc. Many of these calculations are tedious to do manually for every member and load combination, so automation is a big help.

Load Rating: In addition to design for new bridges, FEM software can perform load rating for existing bridges. Load rating involves determining the bridge's capacity under standard rating vehicles. Using the analysis results (often from a model that simulates the existing condition of the bridge) and factoring in resistances with condition factors, the software computes rating factors. CSI Bridge and MIDAS Civil both have load rating features integrated. This is essentially another code compliance check, but for evaluation rather than design, and often uses a slightly different safety factor format (like HL-93 design vs. HL-93 rating, or specific trucks for state ratings). The ability to do this in the same environment means engineers can update a model (say, change a section property to represent section loss from corrosion) and immediately re-run rating to see how the rating factor is affected.

Serviceability and deflection: Codes also require checking deflections, vibrations, etc. FEM naturally provides deflected shapes and frequencies. Software may highlight if deflection limits are exceeded (e.g., $L/800$ for a pedestrian bridge vibration comfort, or a specific camber requirement). Some programs have utilities to perform *vibration serviceability* checks – for example, checking if a footbridge's frequency is above the threshold to avoid resonance with walking, or if a road bridge's deck acceleration under a truck is within comfort limits. These are often not mandated by code but are best-practice checks that software can assist with (sometimes via additional plugins or scripts).

One should note that while software can automate code compliance, the engineer must ensure the software is using the correct parameters. For example, in steel design, the unbraced length for lateral torsional buckling must be set correctly in the model for the code check to be valid. Similarly, the resistance factors or partial safety factors should match the edition of the code being used. It's not

uncommon for engineers to do a few manual spot-checks on critical members to validate the software's design module is applying formulas correctly.

In summary, FEM software with integrated **design code compliance checks and rating features** greatly streamline the bridge design process. They ensure that the transition from analysis results to design decisions is smooth and error-free. As a result, engineers can iterate quickly – if a member fails the check, they can adjust the design (e.g., choose a deeper beam) and re-run, all within the same software. This tight integration of analysis and design is a major advantage of modern bridge FEM programs over older methods. As one user review noted, *having various bridge design codes incorporated in the software, with quick and reliable checks and detailed calculation reports, boosts confidence and saves time.*

Construction Stage Simulation

Bridges are unique structures in that the sequence of construction can significantly influence the forces and stresses that develop. Many bridges (especially large ones) are built in stages: segments are added one by one, supports (like scaffolding or cables) are reconfigured, and time-dependent effects like concrete creep occur between stages. Simulating the **construction stage** is therefore an important part of bridge analysis that FEM software often facilitates.

Staged construction analysis allows the model to be “built” virtually the same way as the actual bridge. In a staged analysis, the structure is initially empty or partially built, and elements are activated step by step. Loads can be applied or removed at specific steps to represent casting of concrete, tensioning of cables, removal of formwork, etc. MIDAS Civil and CSI Bridge both support comprehensive construction stage analysis modules.

Consider a concrete segmental bridge built by cantilevering out from a pier: at each stage a new segment is poured (which introduces weight on the cantilever) and the previously poured concrete continues to undergo creep and shrinkage. A staged FEM analysis will activate the new segment element with its weight, apply the prestress force if any new tendon is stressed, and then advance time to allow creep/shrinkage effects to accumulate. By the end of construction, the analysis provides the locked-in forces and deflections. This is critical because when the two cantilever arms meet at mid-span closure, any small difference in deflection or rotation will cause misalignment. Engineers use FEM to predict the camber (pre-shape) needed so that the bridge will line up correctly during construction and settle into the desired profile once all dead loads are applied.

For steel bridges erected in pieces, staged analysis can account for the fact that not all members are present to share loads initially. Temporary supports (like falsework or hold cranes) can be modeled and then removed in a later stage (simulating their dismantling). The software can automatically carry over the internal forces from one stage to the next, so the residual stresses from construction are included in the final state.

Another example is the construction of a suspension bridge: the cables are strung and saddled, then the deck sections are hung one by one. The tension in the main cable and back stays evolves as each section is added, and the towers bend slightly under the asymmetric loading until the bridge is completed and symmetric. Staged FEM analysis can replicate this process, and it often includes *tension control* features – adjusting hanger forces to desired values as each stage completes. LUSAS and MIDAS have specific capabilities for cable tensioning sequences where the program can iterate to achieve target forces or geometry at key stages.

Time-dependent effects: Creep and shrinkage of concrete and relaxation of prestressing steel are critical in post-tensioned concrete bridges. Over months and years, concrete will creep under sustained load, causing deformations and redistribution of stress. FEM software can incorporate standard creep/shrinkage models (such as CEB or AASHTO models) to simulate these. In a staged analysis, after a concrete segment is placed, the program will compute creep effects on that segment as time advances. This affects long-term deflections and prestress losses. Without FEM, engineers would have to do separate hand calculations for creep and shrinkage (which is very cumbersome for complex structures). With FEM, these effects can be included in the same run, leading to a more accurate prediction of long-term behavior (e.g., sag in a long span after 10 years).

Geometry control: A practical outcome of construction stage analysis is determining pre-camber or pre-stressing. Engineers often run an “inverse” analysis where they specify the desired final geometry and let the software determine the initial elevations required for formwork at each stage. This ensures that when deflections occur during construction, the end result is the correct roadway profile.

Software like MIDAS Civil outputs detailed stage-by-stage results: for each segment or component, you can get the stress history and deflection at each stage. This is useful to check, for example, that during construction no element is overstressed (perhaps a cantilever moment gets too large before closure pour – which might require a temporary stay or support to alleviate).

In steel bridges, staged analysis can predict locked-in forces due to welding sequences or bolting up members. For instance, if two girders are connected at a splice in the air, their own weight causes rotations that when fixed by the splice, introduce residual moments. Staged analysis could capture that by “activating” the connection at a certain step.

Modern FEM tools have made **construction stage simulation** relatively user-friendly: one defines a series of construction steps, assigns element activation/deactivation and load application to each, and the software handles the bookkeeping of internal force carry-over. The result is a realistic picture of the bridge’s evolution. Many iconic bridge projects credit construction stage FEM analysis for ensuring things went smoothly. For example, the Millau Viaduct in France (a cable-stayed bridge with multiple pier column segments and deck launching) was analyzed in stages to confirm that all segments would fit and that piers would not be overly stressed during the asymmetric construction.

In summary, construction stage analysis with FEM is an indispensable technique for bridge engineers. It captures effects that are invisible in a completed-structure analysis but very real in practice. By using this tool, engineers can design **construction strategies** (like where to add temporary supports or how much to jack at a bearing) to mitigate unwanted effects. It also helps in designing the bridge for not just its final state, but also its **construction state**, which can sometimes govern the required strength of components.

Real-World Applications and Case Studies

To illustrate the use of FEM software in bridge design, this section highlights a few real-world bridge projects where finite element analysis played a key role. These examples demonstrate the range of applications – from designing record-breaking new structures to assessing existing bridges – and how the tools and techniques described above come together in practice.

- **Millau Viaduct (France, 2004)** – *Cable-Stayed Bridge, 342 m tall piers*: The Millau Viaduct is one of the tallest bridges in the world and required meticulous analysis for both the construction process and the final design. Engineers built detailed FEM models in software including SAP2000 and BRIGADE/Plus to perform dynamic nonlinear analysis of the structure. Special attention was paid to wind dynamics due to the bridge’s slenderness and high exposure. The FEM model was used to predict the bridge’s mode shapes and frequencies and to ensure that the design would not experience aerodynamic instability. During construction, FEM analysis was used to plan the launching of the deck and the sequential erection of cantilever segments from the piers. The consistency of results between different FEM packages (SAP2000 vs. Brigade) and with measured data (from ambient vibration tests) gave confidence in the design. The Millau Viaduct stands as a testament to how FEM enabled pushing the limits of span and height by providing insight into complex behaviors (like pier flexibility under wind and the interaction between multiple cable-stayed spans).
- **Akashi Kaikyo Bridge (Japan, 1998)** – *Suspension Bridge, 1991 m main span*: Designing the longest suspension bridge in the world required advanced analysis to address wind and seismic forces. A commercial FEM software (ANSYS) was employed as part of extensive studies on the bridge’s aerodynamic stability. The FEM model included the main cables, hangers, stiffening girder, and towers. Engineers performed modal analysis to obtain natural frequencies and mode shapes, then conducted coupled flutter analysis by applying wind load patterns to the model. The

critical flutter wind speed predicted by the FEM model was compared with wind tunnel tests. Additionally, seismic analyses were performed – a finite element model helped simulate the bridge's response to large earthquakes, which was crucial since during construction the infamous Kobe earthquake struck nearby (necessitating a retrofit in the design). The FEM analysis confirmed the robustness of the Akashi Kaikyo's design and guided the development of its tuned mass dampers and other mitigation devices. In essence, without FEM, it would have been nearly impossible to accurately assess the behavior of such a long and flexible structure under dynamic loads.

- **Veterans Memorial Bridge (Mississippi, USA, re-evaluation)** – *Truss Bridge Load Rating*: In some cases, FEM is used on older bridges to assess their capacity. For a century-old truss bridge requiring higher load postings, engineers built a detailed finite element model of the entire truss using SAP2000. The model incorporated the riveted connections and floor system. By applying modern truck load patterns to the model, they determined the internal force distribution more accurately than hand methods (which often overly simplify load paths in complex trusses). The analysis revealed some reserve capacity due to load-sharing not accounted for in simpler methods. As a result, the bridge's load rating was improved, avoiding unnecessary strengthening. This case highlights how FEM software can be used not only for design of new bridges but also for forensic analysis and life extension of existing structures by better understanding their true behavior.
- **Golden Gate Bridge Digital Twin (USA, ongoing)** – *Suspension Bridge Health Monitoring*: A very modern application of FEM is in creating **digital twins** for structural health monitoring. Engineers at Ozen Engineering developed a digital twin of the Golden Gate Bridge using Ansys Twin Builder, linking an FEM model with real sensor data. They used a reduced-order FEM model (simplified representation capturing the key dynamics) to calculate how the bridge responds to wind loading in real time. This digital model runs in parallel with the physical bridge, predicting forces in critical members as measured winds or traffic loads vary. The FEM-based twin can alert if measured responses deviate from expected (which could indicate damage or degradation). This project demonstrates the future of bridge maintenance: continuously updated FEM models that serve as virtual replicas of the structure. By combining FEM with machine learning (to speed up calculations), the Golden Gate Bridge twin can analyze data faster than real-time, providing immediate insights into structural performance [ansys.com](https://www.ansys.com). The success of this case study is influencing many other bridge owners to consider digital twins for long-span bridges, leveraging the rich predictive power of FEM outside of just the design stage.

(The above case studies show just a few instances; many other bridges, such as the **Tsing Ma Bridge** in Hong Kong, the **Øresund Bridge** between Denmark and Sweden, and the **Stonecutters Bridge** in Hong Kong, have all relied on extensive FEM analysis during design. Whether it's determining cable tensions, investigating a peculiar vibration, or optimizing a structural detail, FEM has been the engineer's microscope into the behavior of these monumental structures.)

Advantages and Limitations of FEM for Bridges

The finite element method offers numerous advantages for bridge design, but it also comes with limitations and challenges. Recognizing both is important for effective use of FEM in practice.

Advantages:

- **Ability to handle complex geometry**: Bridges with irregular or non-prismatic geometry (curved alignments, varying cross-sections, unique arch shapes, etc.) can be accurately modeled. Unlike simplified analytical methods, FEM does not require oversimplifying the shape – the mesh can closely follow the real geometry, capturing effects of curvature or eccentric loading accurately. This leads to an *accurate representation of complex bridge geometries* in the analysis.
- **Multi-material and composite action**: Many bridges use composite action (e.g. concrete deck on steel girders) or have different materials in different parts. FEM easily incorporates *dissimilar materials* with their distinct properties in one model. For example, one can combine steel

elements for a truss with concrete elements for a deck slab and even soil elements for foundations, all in the same analysis, something that closed-form solutions struggle to do.

- **Comprehensive load representation:** FEM allows the application of loads in a very detailed manner – point loads, distributed loads, moving convoys, thermal gradients, wind pressure distributions, and inertia forces from earthquakes can all be applied simultaneously. This *holistic load simulation* means the combined effects are accounted for naturally (while in hand calculations, combinations are often linear superpositions with assumptions). Local effects, such as wheel loads on a particular stringer, are captured and can be traced through the structure, which is a big advantage in design refinement.
- **Local effect capture:** Because of the discretization, FEM can capture stress concentrations or local deformations that global methods average out. For instance, FEM might show higher stress around a diaphragm connection or a notch in a girder, alerting the engineer to reinforce that area. This ability to *capture local effects* improves the safety and performance of the design, as details can be addressed before construction.
- **Iterative design and optimization:** With FEM software, engineers can quickly test “what-if” scenarios – e.g., what if I use one more girder, or a deeper section, or a different cable diameter? The model can be adjusted and rerun, and integrated design checks updated. This makes FEM a tool not just for verification but for **optimization**. AI and optimization algorithms can interface with FEM to iterate on designs and find materially efficient solutions that still meet all criteria. Some bridges have achieved significant material savings because FEM helped identify where material could be removed without compromising strength (for example, optimizing the thickness of stiffeners or the layout of post-tensioning tendons).
- **Visualization and insight:** FEM results are visual. Engineers can see deflected shapes, mode shapes (vibration patterns), stress contours, influence lines, etc., in graphical form. This greatly aids understanding. Complex phenomena like buckling mode or torsional behavior of a girder become clear when visualized. It improves communication as well – plots from FEM are often used in reports to explain why certain design decisions were made.
- **Integration with BIM/Digital workflows:** Modern FEM software can import detailed geometry from Building Information Modeling (BIM) tools or export analysis results back to design models. This reduces duplication of work and errors. For bridges, this means a roadway alignment from a civil CAD program can directly form the basis of the FEM mesh, and conversely, reactions from FEM can be sent to foundation design software. This interconnectedness speeds up the design process.

In summary, FEM empowers engineers to **thoroughly evaluate bridge designs under realistic conditions**, thereby increasing confidence in safety and performance. It often reveals efficiencies (or potential problems) that simpler methods cannot, leading to better optimized and more reliable structures.

Limitations:

Despite its strengths, FEM is not a panacea and has several limitations that must be acknowledged:

- **Modeling assumptions and idealizations:** The adage “all models are wrong, but some are useful” applies. An FEM model is only an approximation of reality. It relies on assumptions about boundary conditions, connectivity, and element behavior. If an engineer misrepresents something (say, assuming a fixed support where the actual support has some give), the results will deviate from reality. Users need to *understand the theories, assumptions, and limitations of numerical modeling*, as well as the limits of the software’s element formulations. For instance, an Euler-Bernoulli beam element in an FEM program assumes no shear deformation – which might be fine for a long slender beam, but for a deep pier cap (short and thick), shear deformation could be significant and a Timoshenko beam (shear-flexible) element would be more

appropriate. If the wrong type is used, the model might overly stiffen or soften the predicted response.

- **Garbage in, garbage out:** FEM will dutifully solve whatever problem it is given, even if that problem doesn't make sense. The accuracy of results depends on the accuracy of input data – geometry, material properties, and loads. Material properties, for example, can be uncertain (especially for existing bridges where material may have deteriorated). An assumption like “concrete compressive strength = 30 MPa” might be wrong if the actual concrete is weaker, leading to unconservative results. Similarly, loads like wind or earthquake are inherently variable, and an FEM analysis is only as good as the scenarios tested. Engineers must use judgment to ensure they have enveloped the critical scenarios.
- **Computational demands:** Large, detailed FEM models can become computationally intensive. A 3D solid model of a long span bridge could have hundreds of thousands (or millions) of degrees of freedom. Solving such a system, especially for nonlinear or dynamic analysis, can tax computer resources and take significant time. While computing power has grown, so have ambitions for model detail. Real-time analysis is generally not possible for very fine models – which is why surrogate models or reduced models are explored for digital twins [mdpi.com](https://www.mdpi.com). If analysis takes too long, it can impede the iterative design process. Engineers sometimes need to simplify models to get results in a reasonable timeframe, which is a trade-off between accuracy and practicality.
- **Results interpretation:** FEM outputs a lot of data – perhaps too much. It requires skill to interpret the results correctly. For example, stress “hot spots” at singularities (like where a point load is applied or at a sharp re-entrant corner) might appear alarming (high stresses) but are often artifacts of the idealized model (in reality, load would distribute or the corner would be filleted). An inexperienced user might misinterpret such results and over-design a fix for a problem that doesn't actually exist in the physical structure. Conversely, one must know where to look: FEM might show that overall a girder is fine, but perhaps one specific stiffener is overloaded – the software won't necessarily flag that explicitly unless it's checked via code module. Human engineering judgment is needed to identify what matters.
- **Software limitations and bugs:** FEM programs, as complex software, may have bugs or limitations. Certain analysis types (e.g., a specific combination of moving load with staged construction and frequency analysis) might not have been fully tested by the vendor, leading to errors. Users must validate important results through independent methods or simpler models. Additionally, some programs enforce limits (like maximum number of modes, or they may neglect small mass in dynamic analysis, etc.), which if not understood can limit the scope of results. That's why verification (comparing with known solutions for a simple case, or cross-checking between two different programs) is a healthy practice.
- **Cost and training:** High-end FEM software can be expensive to license. Moreover, they require significant training to use effectively. Bridge engineers must invest time to learn how to model complex features and to stay updated with software changes. A limitation in practice is that not every firm or department has the budget or expertise to fully utilize FEM for every project, especially smaller ones. So there's a threshold of complexity above which FEM is routinely used, but below which it might be seen as overkill.
- **Overconfidence in output:** There is a subtle risk that detailed colorful output (stress contours, etc.) can give a false sense of security – an engineer might accept the FEM results at face value without questioning them as they might a hand calc. This cognitive bias can be dangerous if the model had an error. Thus, a limitation is that FEM requires disciplined skepticism of its own results, and cross-verification with hand calculations or fundamental principles. As the Bridge Handbook suggests, it's *strongly recommended* that users study FEM theory and textbooks and not treat the software as a black box.

In conclusion, the **advantages of FEM** in bridge design are numerous – it enables safer, more efficient, and innovative designs by providing deep insight into structural behavior. However, its

limitations remind us that engineering judgment and experience remain paramount. FEM is a tool, not a replacement for understanding. The best outcomes arise when engineers use FEM judiciously: leveraging its strengths (comprehensive analysis) while mitigating its weaknesses (model uncertainty and computational issues) through validation and sound judgment. When used in this manner, FEM has proven to be an overwhelmingly positive force in advancing bridge engineering.

Recent Developments and Future Directions

The field of bridge engineering is continuously evolving, and FEM software is evolving with it. In recent years, several trends have emerged that are shaping how bridges are analyzed and designed. Two of the most significant are the integration of **artificial intelligence (AI) and machine learning techniques** with structural analysis, and the development of **digital twin technology** for bridges. Additionally, improvements in computation (cloud computing, parallel processing) and better data integration are influencing future FEM workflows.

AI Integration and Machine Learning:

Artificial intelligence is making inroads in structural engineering in a few ways. One area is **AI-driven design optimization**. Instead of manually iterating a design, engineers can use algorithms (like genetic algorithms or neural networks) to search for the optimal design that meets all criteria with minimum cost or weight. These algorithms rely on evaluating many design variants, which is where they connect with FEM – each variant needs analysis to judge its performance. Traditionally, running thousands of FEM analyses would be too time-consuming, but machine learning offers a solution: train a surrogate model to approximate FEM results. For example, a neural network could be trained on a sample of FEM simulations of a bridge girder with varying dimensions to predict bending stress and deflection. Once trained, the AI can explore the space much faster than real-time FEM. According to industry reports, firms using AI for design optimization have achieved material savings on the order of 18–25%. In one reported case in Pennsylvania, AI-generated bridge designs required **20% less material while maintaining the same capacity**, by finding shapes that distributed forces more efficiently. These outcomes are highly promising in terms of sustainability and cost.

Generative design algorithms can propose unconventional but efficient bridge forms that a human may not immediately think of. These algorithms employ techniques like topology optimization (which has been used to shape bridge ribs or truss patterns optimally) and can be guided by machine learning. However, AI suggestions must always be validated by high-fidelity FEM – typically, the workflow is AI narrows down candidates, and those candidates are then rigorously tested with detailed FEM and checked against all code provisions. This symbiosis of AI and FEM can significantly speed up the design process and lead to innovative solutions.

Another application of AI is in **structural health monitoring and predictive maintenance** for bridges. Machine learning models can be trained on simulation data (from FEM) and sensor data to detect patterns indicative of damage. For example, by using thousands of FEM simulations of various damage scenarios on a bridge, an AI could learn to identify, from measured vibration characteristics, which member might be damaged. AI can thus serve as a bridge between vast sensor data and actionable insights, something that traditionally would require manual analysis of data and perhaps manual FEM model updating.

Digital Twins:

A digital twin of a bridge is a virtual replica that mirrors the state of the physical bridge in near real-time by integrating sensor data with analytical models. FEM plays a central role in building these twins as the analytical engine. Recent developments show that combining FEM with machine learning can overcome one of the challenges of digital twins: real-time computation. High-fidelity FEM models are often too slow for real-time use, as noted earlier, but researchers have developed surrogate models (reduced-order models) that are trained on FEM data to respond instantaneously. In the study by Jayasinghe et al. (2023), a digital twin was created for a laboratory bridge where an inverse analysis ML model, trained on FEM results, could predict global structural behavior from a few sensor readings in real time. They found tree-based machine learning models (like random forest

and XGBoost) to be particularly effective surrogates for FEM, outperforming neural networks in accuracy for that application.

What this means for the future is that large bridges could be continuously monitored by their FEM twin. The twin receives input like measured strains, temperatures, traffic loads, etc., and then computes the stresses and even remaining fatigue life of various components instantly. If something deviates (say one hanger in a suspension bridge shows lower tension than predicted, hinting it might have yielded or slipped), the system would flag it. Owners can then target inspections to those areas. Digital twins thus enhance *predictive maintenance and decision-making* by integrating real-time data with FEM analysis. They essentially allow an operating bridge to be “analyzed” 24/7. This is a big shift from the current paradigm of periodic inspections and occasional load ratings.

Several pilot projects are underway globally on digital twins for bridges. Aside from the Golden Gate example, another is the Futurist project in the UK, applying a digital twin to the Forth Road Bridge, and projects in China combining BIM, IoT sensors, and FEM for long-span bridges. As sensor costs go down and data analytics improve, this approach is expected to become common for important infrastructure.

Improved Computational Tools:

The future of FEM in bridge design will also be influenced by computational advancements. Cloud computing and parallel processing are increasingly being used to run very large models or many model variants in parallel. Some FEM software vendors offer cloud solving services where you can offload a heavy analysis to a cluster of servers. This will make high-fidelity analysis more accessible on demand, without requiring local high-end hardware.

Moreover, **integrations with parametric modeling environments** (like Rhino/Grasshopper, Python scripting) are improving. This means engineers can more easily create parametric bridge models – change a few parameters and automatically regenerate the FEM model and run it. This is excellent for exploring design spaces and also for **sensitivity analysis** (checking how sensitive results are to changes in materials, geometry, etc.). In an AI context, parametric models are what AI algorithms often need to interface with to loop through design options.

Another future direction is **enhanced user feedback in FEM software using AI**. We may see intelligent assistants in the software that warn, for example, “your mesh is coarse in a region with high stress gradient, refine for better accuracy” or “mode 3 and mode 4 are closely spaced, consider more precision or check for potential coupling.” These could be based on machine learning trained on many past models and results.

BIM and Digital Workflow:

Bridges are now often delivered with full 3D BIM models for construction. We can expect FEM analysis to tie in more with these workflows. Already, some software can take a BIM model (which might be very detailed, down to rebar) and automatically generate an analysis model (simplified stick model) from it. Future developments may allow the analysis to write back results into the BIM (for example, color-coding members in the BIM model by utilization ratio). This closes the loop between design and documentation.

AI for Construction Control:

AI could also assist during construction through FEM. For instance, using real-time data from sensors and comparing to a construction stage FEM model, an AI could adjust the tensioning schedule or predict if a certain stage will cause issues. This is akin to a digital twin but specifically for the construction phase.

In summary, the future of bridge design using FEM software is moving towards greater *integration* – integration with AI for smarter design and analysis, integration with real-time data for living models of structures, and integration with other digital tools for a seamless design-construction-maintenance continuum. Bridges of the future might be designed with the aid of AI suggesting optimal forms, analyzed to fine detail with powerful FEM in the cloud, and then outfitted with sensors that keep an FEM-based twin running for its lifetime. This convergence of technologies

promises not only safer and more economical bridge designs but also extends the service life of bridges through proactive management. Importantly, though, engineers will still be at the helm – interpreting results, making judgments, and ensuring that these advanced tools are used responsibly and effectively to serve society's infrastructure needs.

Conclusion

Finite element method software has revolutionized bridge design and analysis, enabling engineers to tackle projects of unprecedented scale and complexity with confidence. This paper has reviewed how FEM principles are applied in bridge engineering, from the basics of discretizing a structure into elements to the use of sophisticated software platforms that integrate modeling, analysis, and design checks. We have seen that leading software like MIDAS Civil, SAP2000, ANSYS, LUSAS, and CSI Bridge each contribute unique strengths, whether it be user-friendly bridge templates or advanced nonlinear analysis capabilities. By examining real-world case studies such as the Millau Viaduct and Akashi Kaikyo Bridge, we highlighted that FEM is not just a theoretical tool but a practical necessity for designing safe and efficient bridges in the modern era.

Crucial modeling techniques – choosing appropriate element types, simulating loads accurately, checking code compliance, and modeling construction sequences – form the backbone of reliable FEM analysis. When done correctly, the FEM results align well with physical testing and observations, as evidenced by many successful bridge projects. The advantages of using FEM software are clear: the ability to capture complex behaviors, optimize designs, and ensure thorough safety checks. At the same time, engineers must remain vigilant about the limitations: the quality of an FEM analysis depends on the quality of the input and assumptions, and computational results must be tempered with engineering judgment.

Looking ahead, the integration of AI and the rise of digital twins represent an exciting frontier in bridge engineering. These technologies, built on the foundation of FEM analysis, promise to make design processes more exploratory and bridges themselves more “intelligent” and well-monitored throughout their life. An AI-augmented FEM workflow could unlock design innovations and material savings, while FEM-based digital twins could change how we maintain bridges – shifting from scheduled inspections to continuous, condition-based maintenance with instant analytical feedback.

In conclusion, FEM software has become an everyday “partner” for bridge engineers – much like the slide rule was in the past, the FEM model is now an extension of the engineer's mind for understanding how a bridge will behave. As we incorporate new tools and address future challenges (longer spans, new materials, climate impacts), the core principles remain: a sound understanding of structural mechanics and careful validation of results. Bridges are ultimately public safety structures, and FEM is a powerful means to ensure they perform as intended. By combining the power of computational tools with the creativity and caution of human engineers, we will continue to push the boundaries of bridge design safely and efficiently. The bridges of the future will stand as proof of what this synergy can achieve – smartly designed, thoroughly analyzed, and robustly monitored structures that serve our societies for generations.

References

- Computers and Structures, Inc. (2022). *SAP2000 (Version 24)* [Computer software]. <https://www.csiamerica.com/products/sap2000>
- MIDAS Information Technology Co., Ltd. (2022). *MIDAS Civil (Version 2022)* [Computer software]. <https://www.midasbridge.com/>
- LUSAS. (2022). *LUSAS Bridge (Version 20)* [Computer software]. <https://www.lusas.com/products/bridge/>
- Computers and Structures, Inc. (2022). *CSiBridge (Version 24)* [Computer software]. <https://www.csiamerica.com/products/csibridge>
- ANSYS, Inc. (2022). *ANSYS Mechanical (Version 2022 R2)* [Computer software]. <https://www.ansys.com/products/structures/ansys-mechanical>
- Dassault Systèmes. (2022). *ABAQUS*

- Unified FEA (Version 2022)* [Computer software]. <https://www.3ds.com/products-services/simulia/products/abaqus/>
- Bentley Systems, Inc. (2022). *STAAD.Pro CONNECT Edition (Version 22)* [Computer software]. <https://www.bentley.com/software/staad/>
- SOFiSTiK AG. (2022). *SOFiSTiK Bridge Design Suite* [Computer software]. <https://www.sofistik.com/products/bridge-design>
- Dlubal Software GmbH. (2022). *RFEM 6* [Computer software]. <https://www.dlubal.com/en/products/rfem-6>
- Paudel, Gaurab, and Mohamed Sorour. "Load Sharing in Transversely Post-Tensioned Pre-Cast Box Girder Skew Bridges." *Engrxiv*, 17 Apr. 2025, <https://doi.org/10.31224/4528>.
- Saber, Qahtan Adnan, et al. "Structural Finite Element Analysis of Bridge Piers with Consideration of Hydrodynamic Forces and Earthquake Effects for a Sustainable Approach." *Mathematical Modelling of Engineering Problems* 12.3 (2025).
- Abbozzo, Alessia, et al. "Bridge Management Systems: Digital Twins as the Link between Bridge Information Modelling, Finite Element Analysis and Structural Health Monitoring." *Finite Element Analysis and Structural Health Monitoring*.
- Lee, Seungjun, and Young-Joo Lee. "New Computational Platform of Finite Element Reliability Analysis for Efficient Safety Assessment of Bridges Employing MIDAS/Civil." *KSCE Journal of Civil Engineering* (2025): 100163.

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.