View from beneath a new bridge under construction, showing pre-stressed concrete beams and deck at the abutments. The weight of the bridge itself, as well as traffic, wind, water, temperature, corrosion, and time, all contribute stresses and deformations that influence the performance and service life of the structure.

Improving **Bridge Performance** with Finite Element Analysis Software

Historically, most large bridges are "overdesigned" with substantial margins of safety built in to compensate for unknown forces that could affect their integrity over time. For the reliability, maintenance, and economic viability of the bridges of the future, better performance from the ground up is critical.

Associate Professor Daniel Linzell and his research group in the department of Civil and Environmental Engineering at Penn State University employ advanced Finite Element Analysis (FEA) to create computer models for studying the structural behavior of bridges. Using this technology, Linzell and his graduate students are able to focus in on potential trouble spots in individual bridges, helping civil engineers anticipate problems and make adjustments before construction begins. The simulation results can also be used to make decisions about maintenance requirements.

Modeling Real-World Stresses

"Civil engineering has until very recently relied on linearly elastic, small deflection FEA methods used in software tools as the backbone of bridge analysis. However, this method is, in many instances, an approximation and doesn't capture the full range of real-world nonlinear responses in these increasingly larger and more complicated structures," says Linzell. "Higher-order methods found in advanced FEA software are becoming more commonplace in the industry because they provide the capability to incorporate nonlinearities to account for realistic stresses and deformations that will influence the performance and service life of a bridge."

Aspects that can be incorporated into an advanced FEA model in addition to material and geometric nonlinearities to assess the structural integrity of a bridge include the response of concrete or steel to the weight of the bridge itself, as well as to traffic, wind, water, temperature fluctuation, corrosion, and even time (both concrete and steel "creep," resulting in long-term deformation).

Economic Factors

Economics is another driving force behind the need for more sophisticated analysis tools, Linzell says. "Extreme overdesigning has become too expensive. There's now a strong push to minimize material costs and simplify design to reduce labor."

Lighter, stronger materials are being developed: steel that is available now has yield stresses of 100 ksi (100,000 psi), almost three times what it was just 10–15 years ago. But while stronger steel allows builders to use smaller sections to support the same bridge loads, the new materials may also be more flexible. "You need higher-order tools to better predict such nonlinear geometric deformation," Linzell points out.

Helping the Bridge Building Industry Communicate

Linzell's group is concerned about a long tradition of separation between designers and constructors in the bridge industry. "Designers need to understand that a bridge doesn't arrive in one piece and get dropped into place; on-site construction decisions can influence bridge life and functionality. You have to think about how it's going to be built, because that in turn influences how it's going to perform in the end," he says.

"What we're trying to do is come up with guidelines or tools that will bring designers and constructors closer together. Those are the intentions with what we're doing with our FEA models."

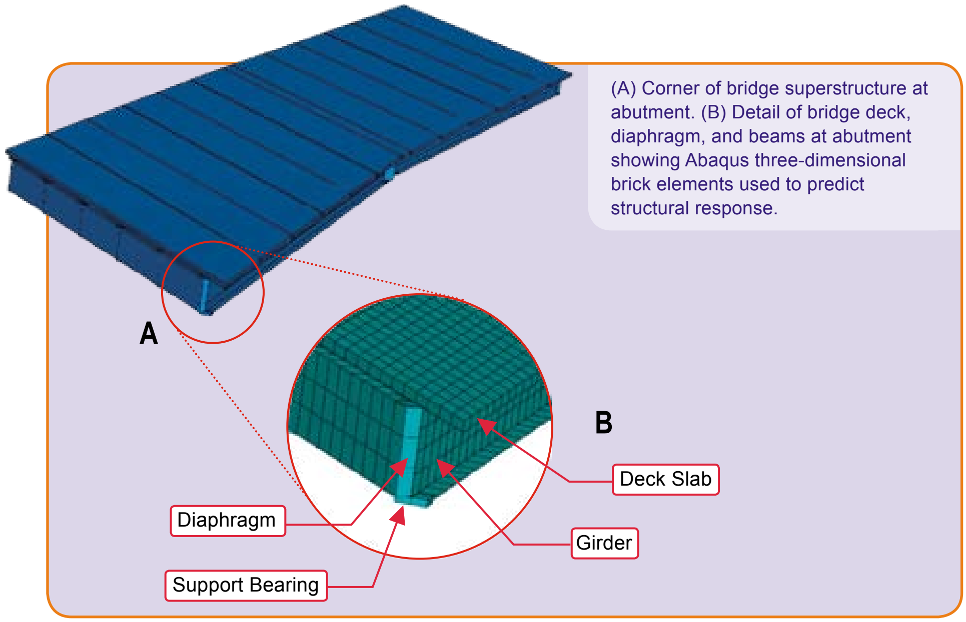
How to Build a Virtual Bridge

Linzell's group uses Abaqus/Standard to create these virtual bridges. First, they build a numerical model based on an existing design. (They can work with designs that are still on paper or in CAD (computer-aided design) format, or they can use as-built measurements taken from a structure already under construction.)

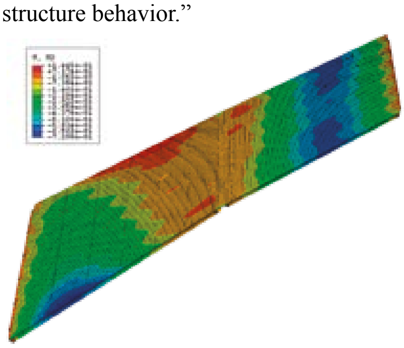
They next select elements (these are the tiny geometric shapes, mathematically representing physical units, that are linked by nodes to form a numerical model) and then material models. "You choose elements depending on available material (constitutive) models and geometry, then select what's best for the materials being used, such as concrete or steel."

Next they set up the boundary conditions for the model. "We use Abaqus a lot in this stage," he says. "We select how the bridge is going to be restrained, whether we are going to utilize a contact condition or a discrete restraint, and how friction will be represented, for example."

Finally Linzell's group applies various loads to the parts of the bridge, such as the bridge deck to represent vehicle loads or onto the beam faces to represent wind loads. (Stresses can be determined either at nodes or in the elements themselves.) "This is a fairly prescribed process, but it depends on what you are looking at—such as traffic loads, or the weight of the structure itself," he says. The Abaqus creep module is used for time-dependent factors. "Creep is a big issue with concrete, and similar time-dependent effects influence steel behavior as well. Thermal loads are important, too: We've taken data from bridges where there was a 50–60 degree temperature change during construction that certainly affected structure behavior."



(A) Corner of bridge superstructure at abutment. (B) Detail of bridge deck, diaphragm, and beams at abutment showing Abaqus three-dimensional brick elements used to predict structural response.



Abaqus 3D finite element analysis (FEA) model of vertical displacement contour of a bridge deck, representing how the entire bridge moves under its own weight.

Depending on where loads prove to be excessive, the bridge model—and, ultimately, the performance of the actual bridge—can be modified. The process can be repeated until the optimum configuration for the bridge is reached. "There's a lot that goes into modeling how a massive, highly-indeterminate structure like a bridge is going to respond," Linzell notes.

"Abaqus helps us get our bridge models as accurate as possible."

Real-World Applications

The software has other application potential in the field of bridge analysis besides designing and testing new structures, according to SIMULIA senior engineer Deepak Datye. "Abaqus can be used to evaluate the residual life of a damaged structure that is still standing but may be cracked. And it can also be used for forensic purposes, to help pinpoint the reason for a collapse."

Linzell sees nonlinear FEA playing an increasingly important role in building better bridges for the future. He is part of a group of researchers, practitioners, and Department of Transportation (DOT) engineers who are collecting questionnaires from fellow bridge-building professionals related to their current use of numerical tools.

"We are hoping to come up with unified FEA guidelines for bridges because our industry really doesn't have a unified publication yet," he points out. "Other disciplines like aerospace engineering, and to some extent mechanical engineering, already do, so we're trying to initiate that process."

For More Information

simulia.com/solutions/architecture
www.engr.psu.edu/ce