WHITEPAPER

Three Ways to Solve Your Structural Engineering Challenges

STAAD.Pro®

Bentley®
Advancing Infrastructure

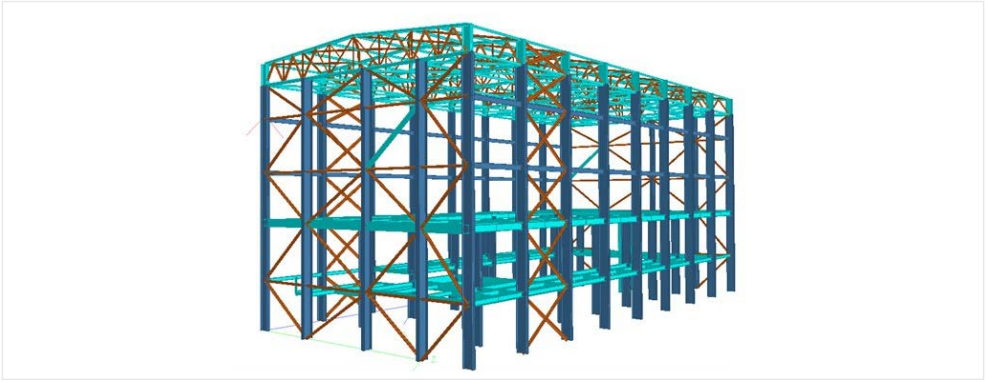
CONTENTS

CHOOSING THE BEST MODELING APPROACHES, ANALYSIS METHODS, AND ANALYTICAL RESULTS FOR YOUR PROJECT

The roles and responsibilities of a structural engineer can vary greatly. However, at the heart of the position is the need to ensure that the structures they design are safe and appropriate for their stated purpose. The design requirements for a simple hay barn and those of a nuclear containment structure will clearly be different. The engineer is responsible for ensuring that their proposals meet the specifications of their specific project.

Analysis and design requirements vary widely, but fundamentally, any project will require validation to a set of specifications that will be prescribed by interested parties, which can range from government bodies, local authorities, or clients. The days when verification could be done with a few hand calculations is long gone, even for a hay barn.

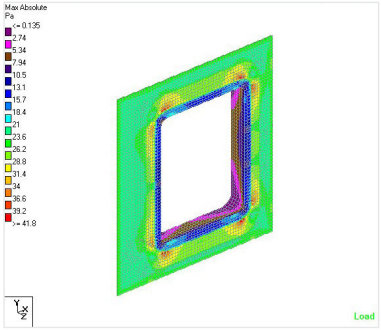
With the development of computerized systems and digital workflows, one of the first stages of any structural design is to create one or more suitably accurate digital representations of the project. These representations can be used to determine the structure's expected behavior at events that would occur during its working life. These events can be represented by loading, either static or varying, and resulting in deformation, internal and external forces, stresses, and vibrational responses.



Example of a 3D-rendered view in STAAD.Pro.

The degree of complexity of the model used in a structural analysis can range widely – from a rudimentary representation of large parts of the structure with simple elements, to highly detailed models that include the stiffening effects of secondary structures. The truth is that there is never only one answer as to how complex or detailed the model should be. The structural engineer should always ask the question, "What is this model intended to tell me?" and have in mind as to the cost/benefit any model would provide.[1]

- If you produce a model that is too simple, then you will fail to capture much of the complexity of its behavior, and it will not measure, for example, the stresses in the bolts of a key connection.
- If you produce a digital model that is of the highest accuracy, such as a finite element model of every square centimeter or inch, then any analysis could take days or months to complete. The design team would typically not have time for this detailed analysis, especially as it is not uncommon for designs to frequently iterate during the design process, which might require performing new analyses.



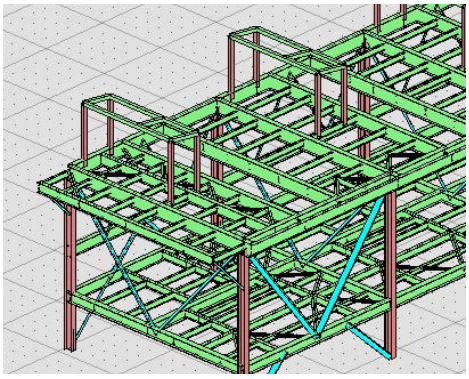
Example of a finite element stress contour plot.

MODELING APPROACHES

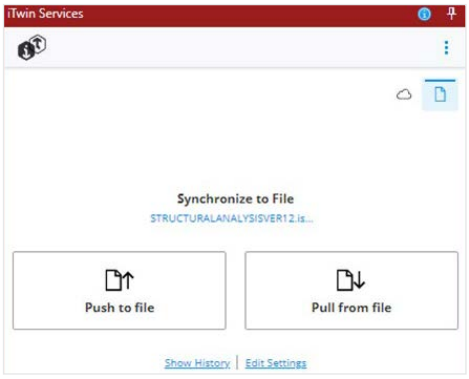
If you are looking for the right place to begin the process and create the best model for the specific analyses you need to undertake, STAAD.Pro, a structural analysis application from Bentley Systems, provides four modeling approaches.

1. **Existing Digital Model**
   STAAD.Pro enables you to leverage data from an existing digital model that another team member developed, even if third-party applications were used.

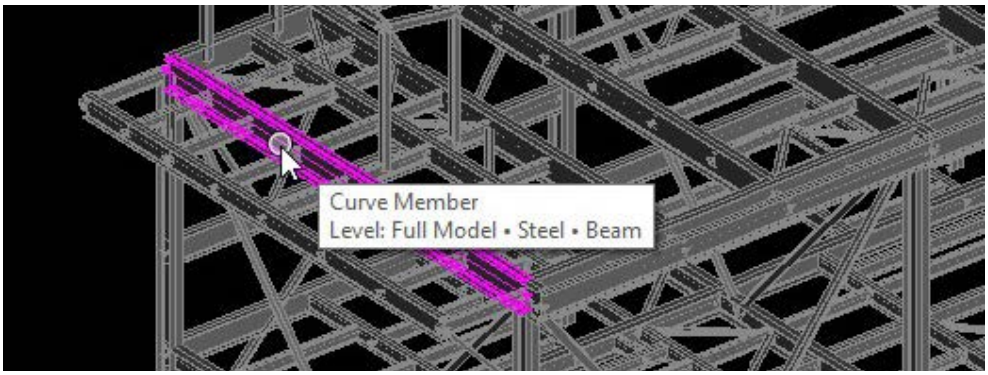
   - iTwin® Services included in the STAAD.Pro Physical Model Workflow provide integrated structural model (ISM) capabilities, allowing you to share structural engineering project information among modeling, analysis, design, drafting, and detailing software applications, which, in turn, enables easy model creation.



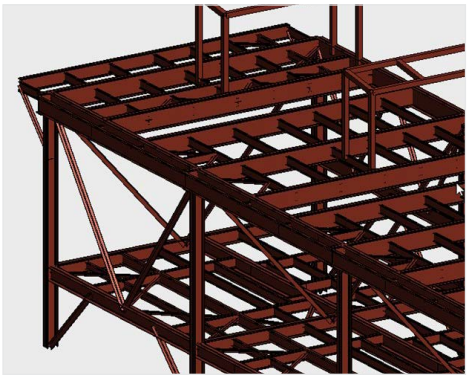
A model in OpenBuildings®.



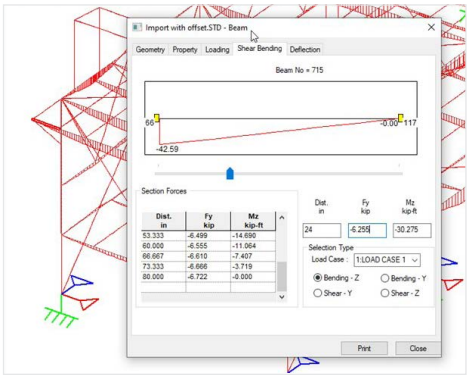
iTwin Services shown in the STAAD.Pro Physical Modeler.



An ISM repository displayed in the Analytical Synchronizer.



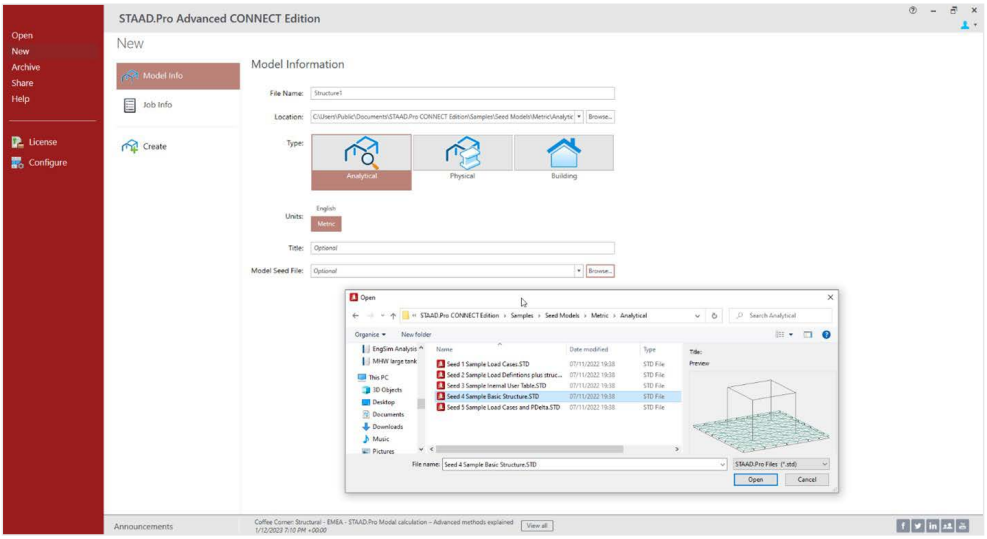
A model imported into the STAAD.Pro Physical Workflow.



Typical analytical results.

2. **Existing Analytical Model**
When the starting point of multiple structures shares many attributes, such as load cases, combinations, materials, and profiles, then employing a template containing those attributes can save significant time.

- STAAD.Pro incorporates seed files, through which an existing model can be used as templates for new models. Seed files preserve the integrity of the original file and ensures that any new model has a head start.

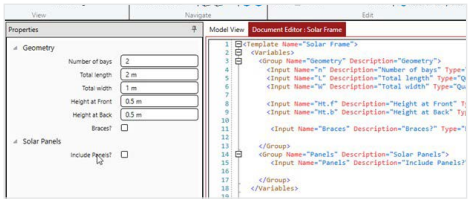


A new model setup with a sample model acting as the seed file.

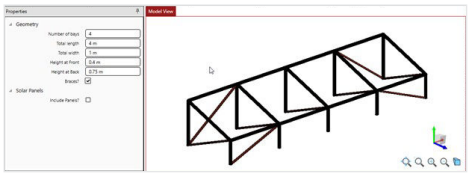
3. **Standard Structures**
When you need to resolve a commonly occurring design problem, standard structures can help. For example, some engineering teams will design, construct, and deliver variations of a single type of structure that can be customized for each project. Defining the basic framework in a parameterized format can solve multiple specific design issues with minimal modification.

- Structure Wizards are provided by STAAD.Pro for physical and analytical models. However, the system provided with the physical model workflow has a distinct advantage, because the module is a fully customizable environment that allows any regular structure to be defined using a rudimentary script.

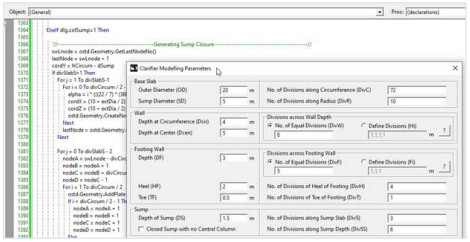
- When the requirements are more complex, you might need to pull in data from multiple sources into the analytical data format. By deploying a more generic system that can be coded using Python or VBA, the rapid creation of standard models is enabled when entering key data only.



Scripting for the Physical Structure Wizard.



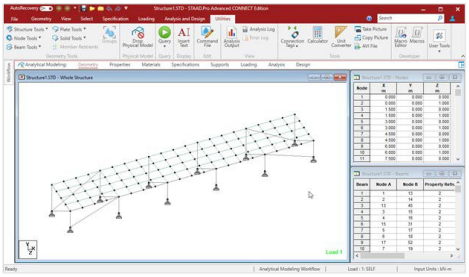
An example of a frame structure created using the Physical Structure Wizard.



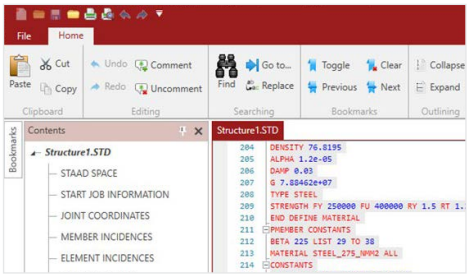
A sample OpenSTAAD macro.

4. **Clean Slate**
   When there is no other option, start with a blank slate. If you only have a description of the analysis problem as a basic sketch or a written description, then having a simple, well-designed system to construct the digital model is important.

   - You can use a basic graphical user interface (GUI) table control or the standard data syntax entered in an editor environment, such as the one provided with STAAD.Pro, or Windows Notepad.

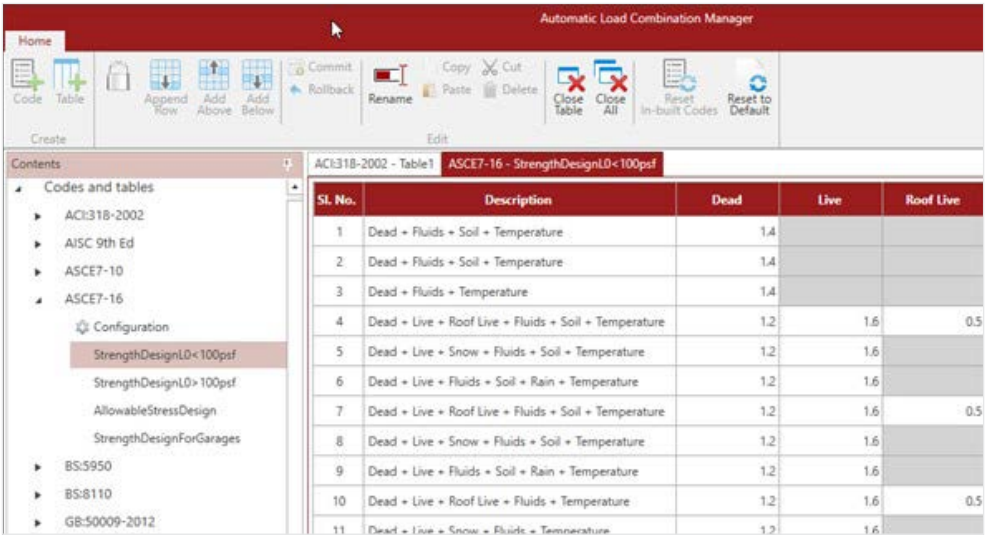


A basic graphical user interface.



An editor with IntelliSense.

You also might need to consider numerous loading conditions. These separate conditions could occur simultaneously in various combinations, leading to the consideration of probability. Many designs will include safety factors, which are modified by the probability of their occurrence when combined with other effects. Think of how likely it is to experience a significant earthquake combined with the most significant wind loading. STAAD.Pro helps define rules for specifying the factors when combining load cases.



The Combination Generator manager.

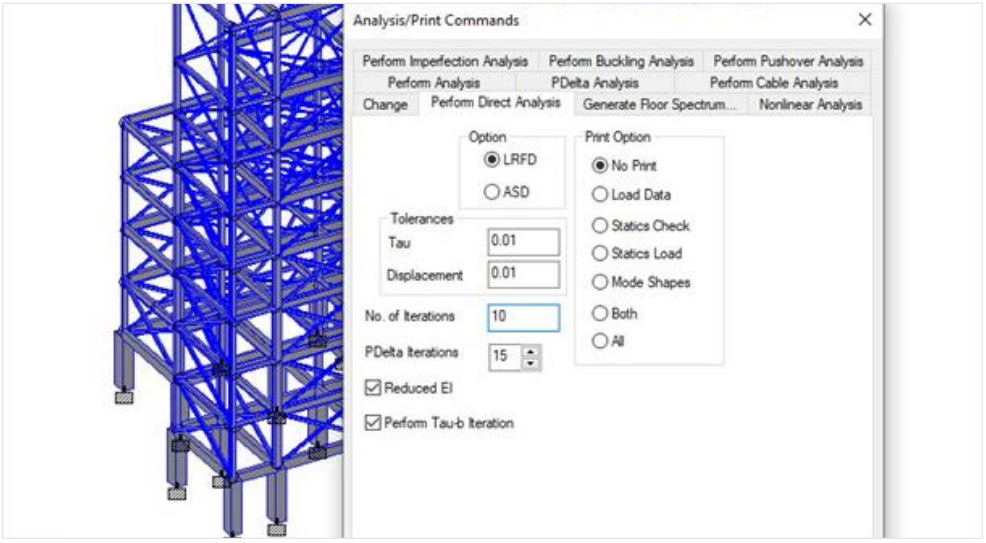
Additionally, loading conditions may occur in different directions. Think of how the effect of wind in one direction can differ when it originates from a different direction, due to protection from other structures or the landscape.

ANALYSIS METHODS

Now that modeling approaches have been covered, the next question is which type of analysis should be considered?

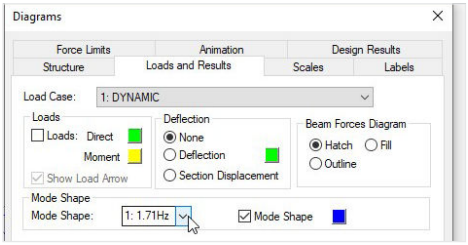
1. **STAAD.Pro: Nonlinear Effects and Dynamic Behavior Analysis**
   Most construction projects are designed to have lifespans that can be considered as continuous. Therefore, the goal is to ensure that under the loading the structure is typically subjected to, it remains within the capacity of the materials being used. That is, steel does not yield or fracture, and any cracking that will develop in reinforced concrete does not cause failure.[2]

   Under most situations, structures are expected to be static and unmoving so that we can admire them from a distance. They do flex, even though we do not necessarily perceive it at a macro scale. In this case, we can perform static-based analysis using traditional elastic analysis methods. However, more design specifications will lead engineers to investigate the performance of structures when subject to conditions in excess of the expected environment and by introducing safety factors on the loading, pushing the models to the limit of their capacity. Finding their limits starts by introducing **nonlinear effects**, which can be captured using analysis methods such as P-Delta, in which the effect of deformation itself adds additional loading into the system, or the Direct Analysis method,[3] in which significant loading and forces in the structure reduce its capacity to accommodate forces that it would otherwise have available.

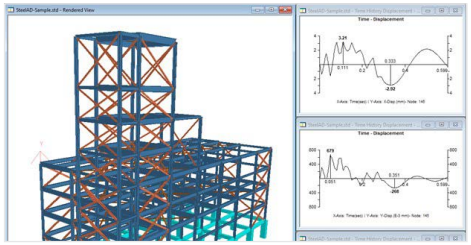


Analysis methods in STAAD.Pro.

Additionally, it is now common for engineers to gain improved understanding of the dynamic behavior of their structures by looking at their **vibrational characteristics**, which are sometimes referred to as its natural frequencies. In other words, the engineer will identify the frequencies that the structure will be sensitive to if it were subject to vibration. That might be a seismic event in which the structure is vibrated through its foundations, or by having a machine supported somewhere in the structure vibrating at a given frequency.



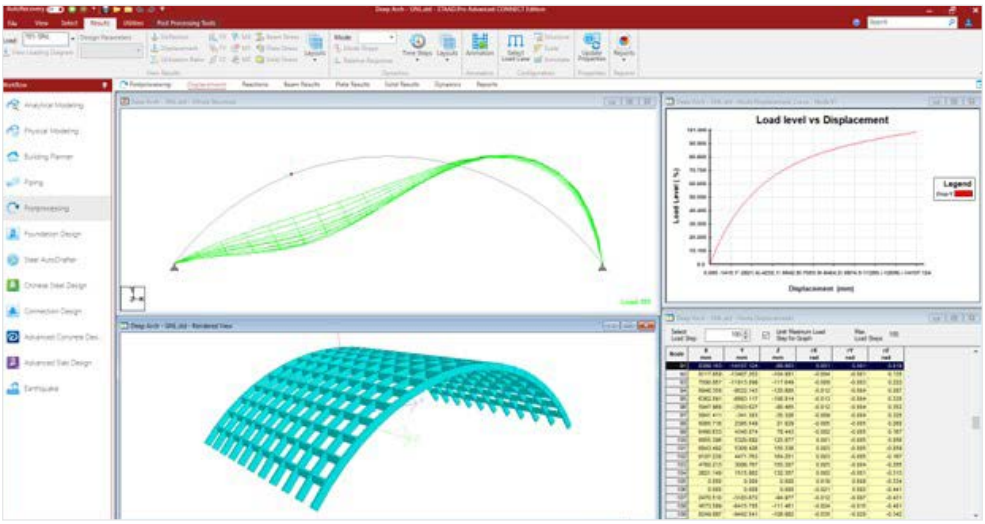
Structure diagrams and results selector.



Time history analysis results.

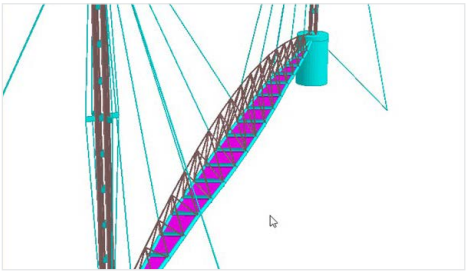
2. **STAAD.Pro Advanced: GNL Analysis, Advanced Cable Analysis, Steady State Analysis, and Eigen Buckling Analysis**

   What was purely academic only a few years ago is becoming more mainstream, and now informs the design of today's infrastructure projects. STAAD.Pro Advanced offers additional analysis capabilities, including **geometric nonlinear (GNL) analysis**. This analysis is helpful when you need to determine a more accurate reflection of the behavior of their model under loading that is producing significant deformations. These deformations will induce secondary effects that traditional, first-order forms of analysis tend to ignore or account for with a simplified modification to the loading vector.[4] The use of a GNL analysis enables the engineer to visualize how the growth of a defined loading pattern affects specific locations in their model, and can help determine whether or not action needs to be taken, or if this loading is within tolerance for their needs.

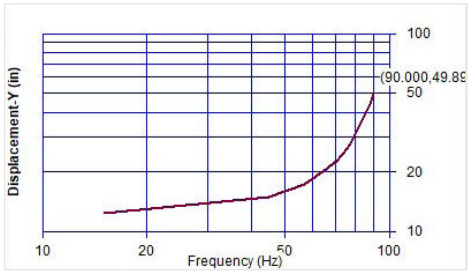


Geometric nonlinear analysis deformation.

Other advanced analysis methods available include an advanced cable analysis, which is designed when the structural model includes slender elements that will undergo significant sagging under their own weight.



A cable-stayed bridge.

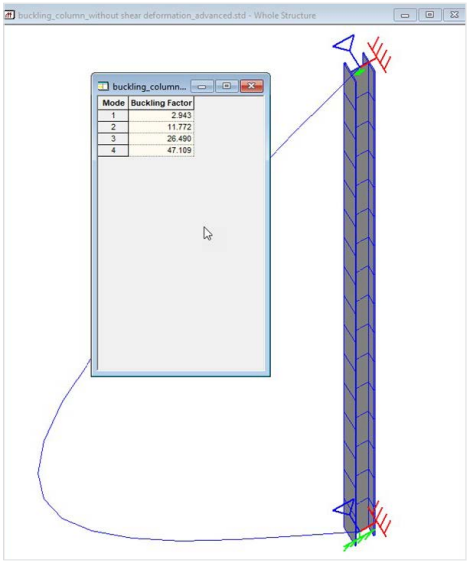


Steady-state response.

Steady-state analysis is used when it is important to capture the effects of the initialization of the dynamic analysis once the transients have been eliminated. These effects are not limited to just a single forcing frequency, as steady-state analysis also provides a method to subject the model to multiple forcing frequencies.

It is possible to determine some form of limits to the capabilities of the structure to withstand a loading pattern by invoking a **buckling analysis**. A rudimentary method offered by the regular STAAD.Pro offering is supplemented with STAAD.Pro Advanced. That application can provide additional insights into the nature of buckling by determining the mode shapes to which the structure is sensitive, in a similar way to the modes of vibration identified when looking at a dynamic analysis.

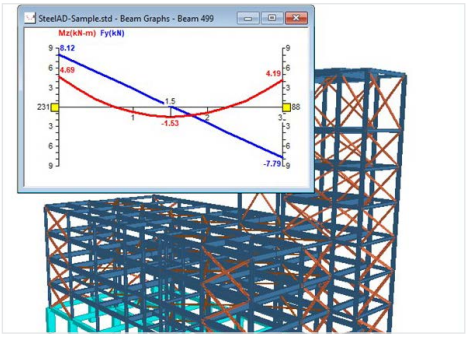
3. **ADINA: Advanced Analysis, Nonlinear Time History Analysis**
   When the methods afforded by STAAD.Pro Advanced do not meet analytical needs, the STAAD.Pro model can still be used through new interoperability with ADINA.[5] There, you can make use of its comprehensive analysis methods, in particular with a nonlinear time history method.



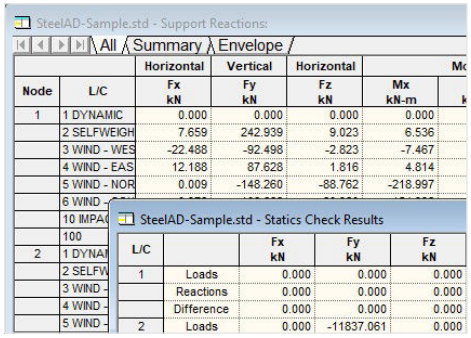
An eigen column buckling solution.

ANALYTICAL RESULTS

1. **Nodes and Beams**
   No analysis and design solution would be complete without access to the results in a tabular or graphical format. Peak deformations for each loading case are provided at each defined node point and can be extracted for discrete locations along members.

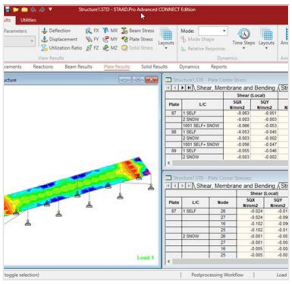


An example of a bending moment and shear force graph.

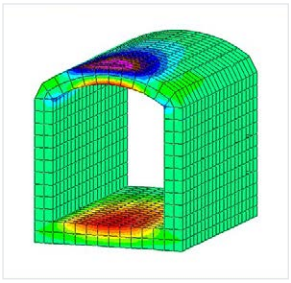


A sample support reaction and static check.

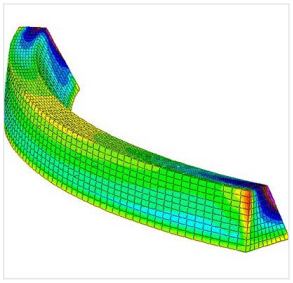
2. **Plates and Solids**
   Determining the results on complex finite elements models can be daunting, but STAAD.Pro gives you access to the data through a range of standard controls.



A typical plate stress contour map.



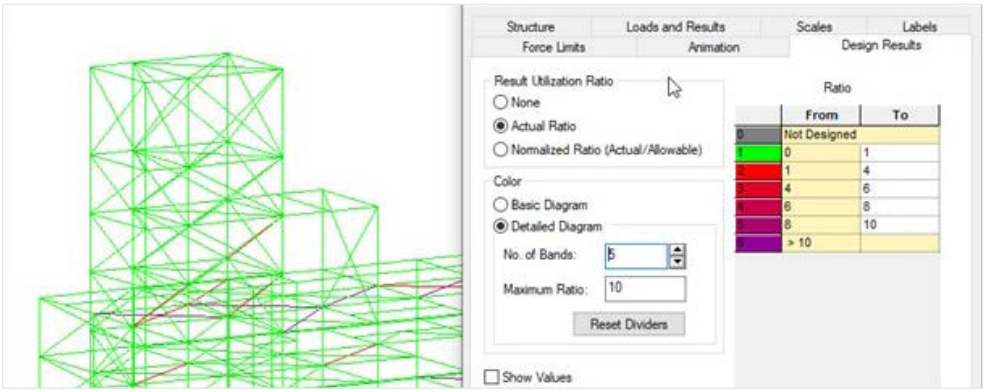
Stresses in a simple tunnel segment.



Stresses in a simple dam.

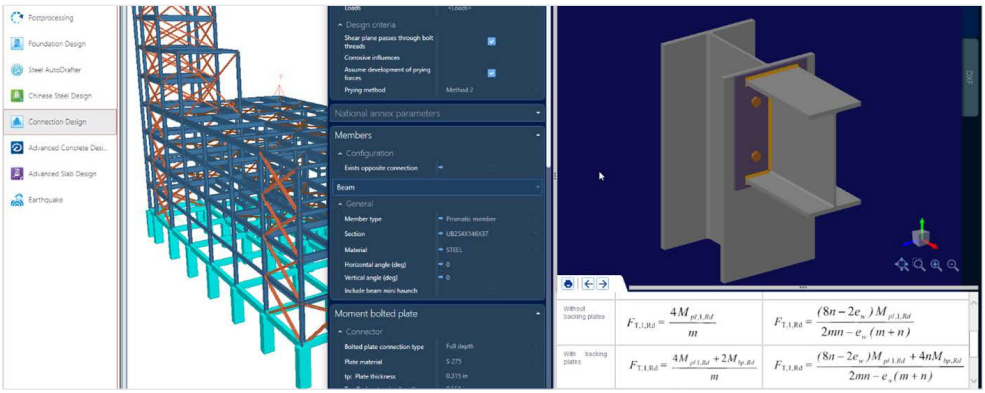
3. **Design**

Understanding the patterns and distribution of deformation and stresses around the structure provide the platform for determining the acceptance criteria for structural elements. For steel members, STAAD.Pro offers checking capabilities to identify locations that need further investigation, or to recommend a profile that might meet the current specifications.



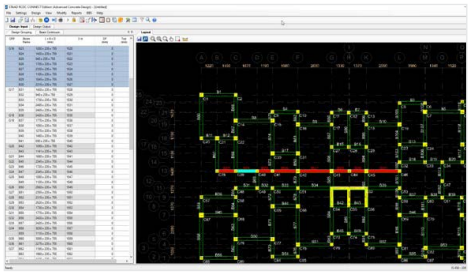
Color-coded members to indicate design compliance.

Working in conjunction with RAM® Connection[6], which has been integrated into STAAD.Pro as the Steel Connection Design Workflow, not only can you check the members for compliance, you can also analyze the connection to the rest of the structure, be it welded, bolted, wide flange, angle, or tubular.

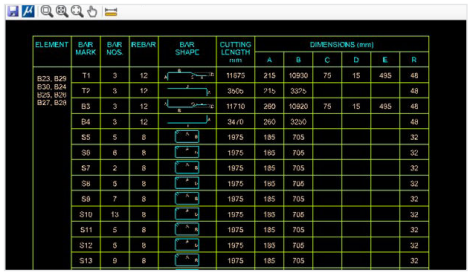


A typical beam-column flange connection.

With concrete structures, design is a more physical-based operation. While a range of element design capabilities are provided as a batch-style solution, the real power of STAAD.Pro becomes apparent when used in conjunction with the STAAD Advanced Concrete workflow.[7] The workflow offers the designer the option to coordinate reinforcement cages for similar structural components, which might be subject to significantly different loadings to ensure a practical construction. Drawings produced in STAAD Advanced Concrete are CAD-agnostic and can be pushed into your CAD package of choice.



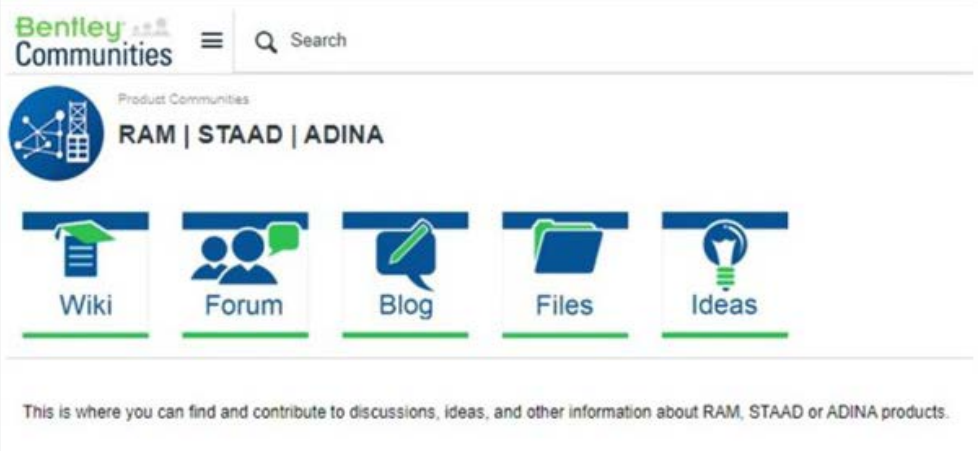
A beam design using STAAD Advanced Concrete Design.



A bar-bending schedule for a sample member in RCDC.

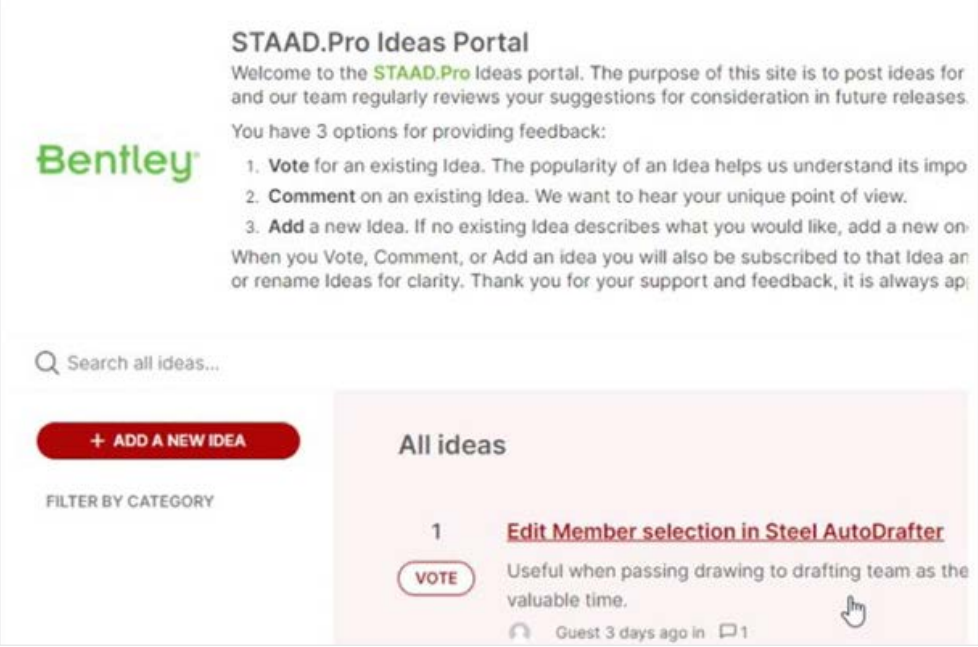
JOIN THE COMMUNITY

This e-book only skims the surface of the most popular benefits that STAAD.Pro can offer. However, as analysis solutions like STAAD.Pro grow ever more sophisticated, an easy way for engineers to keep informed and share ideas is through the **Bentley Communities RAM/STAAD/ADINA Forum**. The forum gives you the opportunity to discuss any topic relating to your structural analysis software with insights from literally thousands of participants and experienced engineers from around the world. In addition, the platform is monitored and administered by Bentley experts who have many years of structural engineering and/or STAAD experience, ensuring feedback is both rapid and relevant.



Bentley Communities.

We want to hear from you! We actively support engineers who have suggestions for improving Bentley software by encouraging them to log ideas in a dedicated forum – the **Ideas Portal**. Other engineers can comment and vote to support these ideas so that the Bentley development team can evaluate what topics are most valuable to the engineers who are working with STAAD.Pro.



The STAAD.Pro Ideas portal.

REFERENCES

[1] It is not uncommon for larger projects to result in engineers developing multiple models of differing complexity for the same structure. Perhaps a simpler frame model could determine overall deformation characteristics of a structure, down to highly detailed finite element models of components such as a structural connection. In the larger model, this intersection may be represented as a simple rigid connection.

[2] A known behavior of concrete is that it cracks. Clever arrangement of reinforcing bars in concrete structures controls cracking, making it a controlled behavior.

[3] Direct Analysis is defined in the publication AISC 360-16.

[4] Geometric nonlinear analysis does not account for large deformation or snap.

[5] ADINA analysis solutions are offered in different bundles. ADINA Structural is needed in this case and would need to be purchased separately in addition to any STAAD.Pro package.

[6] Full access to RAM Connection requires a separate license, but is included with Structural WorkSuite.

[7] Access to STAAD Advanced Concrete requires a separate license, but is included with STAAD.Pro Advanced as well as Structural WorkSuite.