

COST EFFECTIVE · NONLINEAR DYNAMIC · PERFORMANCE BASED DESIGN

EXTREME LOADING 6

FOR STRUCTURES

THE PRACTICING ENGINEER'S SOLUTION FOR
ADVANCED NONLINEAR DYNAMIC ANALYSIS

WWW.EXTREMELOADING.COM



WWW.APPLIEDSCIENCEINT.COM

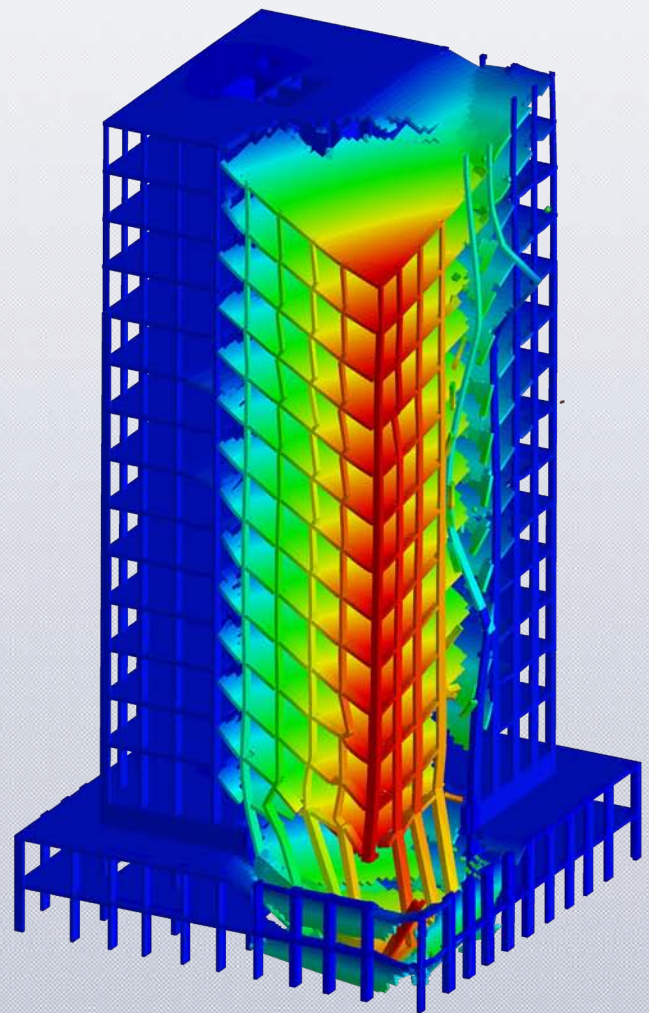
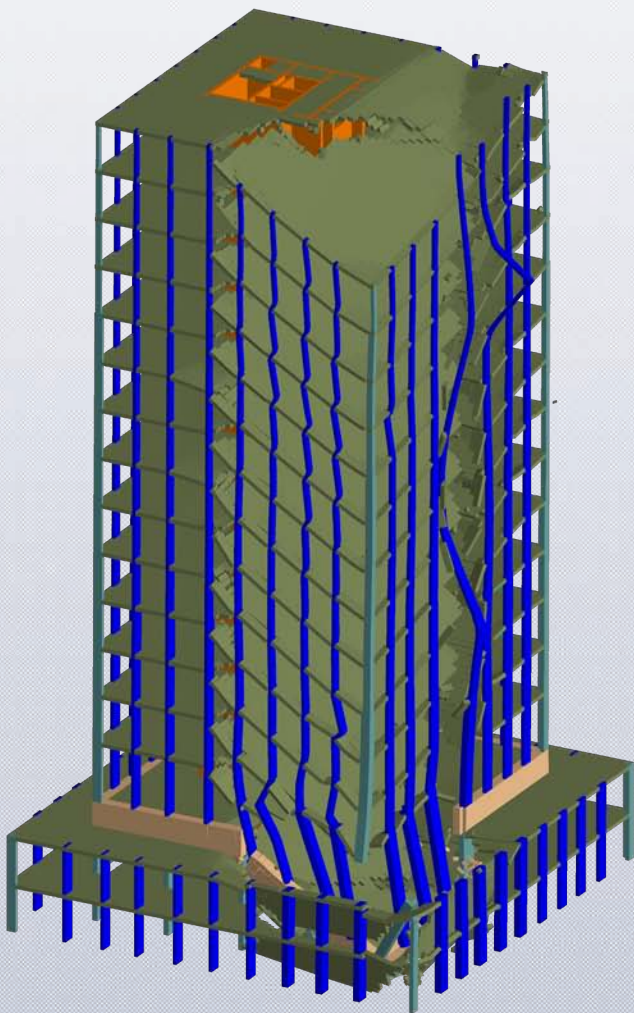
ABOUT EXTREME LOADING FOR STRUCTURES

ASI's Extreme Loading® for Structures (ELS) software is the ideal fully nonlinear structural analysis tool for structural engineers to study the behavior of steel, reinforced concrete, and masonry structures against corrosion, seismic, wind, progressive collapse, and blast. ELS will allow users to easily perform static or dynamic analysis with full nonlinear material and geometric behavior, from the elastic to plastic stages. This includes automated cracking, buckling, post-buckling, P-Delta effects, contact, complete element separation, collision, and effects of falling debris.

ELS delivers high-end structural analysis found only in "scientific" software tools in an efficient engineer-friendly package by eliminating the need for transition elements, smearing of reinforcement, manual placement of plastic hinges, and other time consuming processes and assumptions associated with FEM software of the past. This results in a more detailed analysis and more accurate results in an economical time-frame.

WHAT'S NEW IN ELS 6?

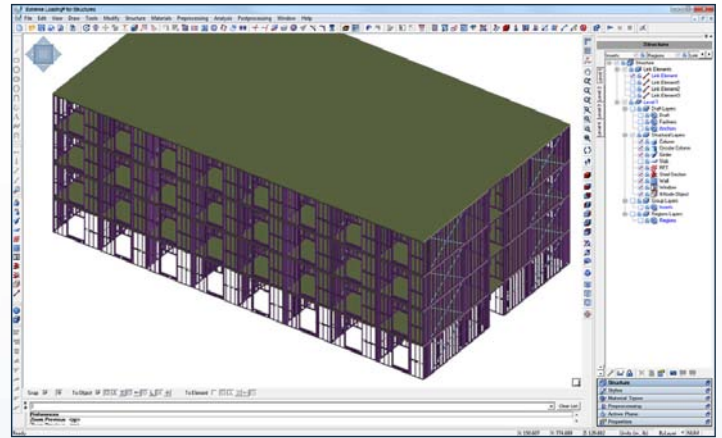
This new version of ELS reinforces the fact that it is a practical, easy-to-use structural analysis software program created for practicing Civil/Structural Engineers who have complex projects with non-traditional load cases and wish to provide their customers with the added value of performance based design. The majority of the new features provided in ELS Version 6 are advanced modeling and post-processing features proposed by software users who wanted additional tools to help them create and solve more complex models. Additional features have been added to provide ease-of-use and versatility for everyday projects.



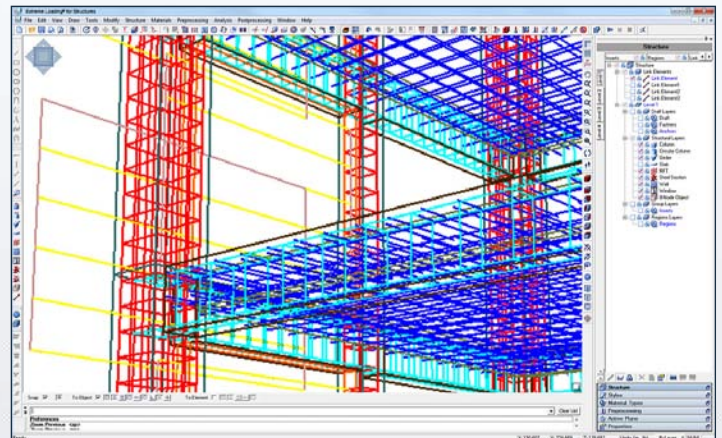
MODELING FEATURES

The Modeler with ELS 6 provides Engineers with an intuitive style-based modeling interface that allows users to quickly and easily create structures in both 2-D and 3-D modes.

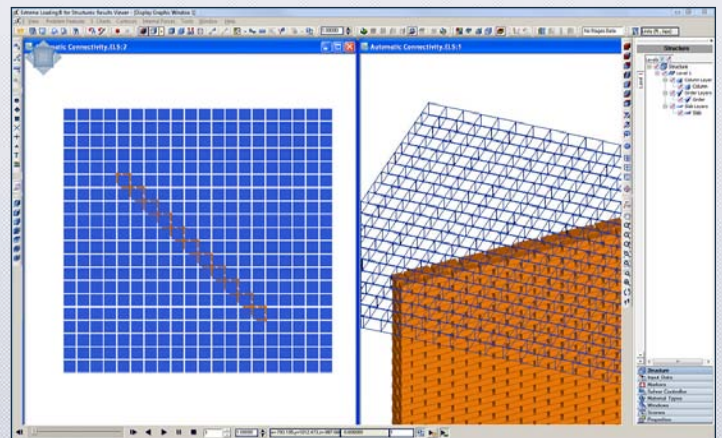
- **Full 3-D Modeling:** build and modify your models in a full 3-D environment.
- **Style Based Modeling (Building Components):** saves time by allowing the user to use standard building components or edit and create new ones for streamlined modeling.
 - **Steel Section Library:** complete library of ASIC & UK/ European metric and imperial steel sections.
 - **Cold-Formed Steel Section Library:** complete library of light steel framing studs and track provided courtesy of The Steel Network, Inc.*
 - **Columns:** create standard and circular columns and customize dimensions, mesh type, material, and embedded reinforcement or steel section style.
 - **Girders:** customize girders by depth, material, and embedded reinforcement or steel section style.
 - **Slabs:** customize slabs by depth, material, and embedded reinforcement or steel section style.
 - **Masonry Walls:** customize masonry walls by thickness, separate brick and mortar materials, with a normal mesh or staggered brick pattern.
 - **Multilayer Windows:** create models for single or multi-layer glazing sections with frames and mullions.
 - **Reinforcement:** model custom reinforcement bars, variably spaced stirrups, and steel sections. Create custom names for reinforcement bar styles and import/export databases for bar sizes.
 - **8-Node Objects:** create custom 8-node objects.
- **Structure Templates:** create models for multi-story structures, frames, trusses, towers, domes, light steel framing walls, hollow core slabs, corrugated sheet, stairs, and segmental bridges.
- **Advanced Modeling Features:**
 - **Convert Draft Objects:** draft objects and convert into girder, masonry wall, RFT, steel section, link members, column & girder structure, steel section component, or polyline.
 - **Enhanced Modeling of Steel Bars:** Automatically convert reinforcement bars inside concrete sections from springs to physical elements for enhanced modeling of steel bars.
 - **Create Hinges:** The ability to create intermediate hinges inside and between columns and girders.
 - **Interface Materials:** Specify custom-created interface materials and specify a custom name using the enhanced Interface Materials dialog.
 - **Pre-stressing:** apply pre-stressing in reinforcement bars and link elements.
- **Filtration Queries:** create custom filtration queries based on complex criteria (Object Type, Material, Interface ID, Pre-Stressing, Loading, etc.). Import & export queries for common tasks.
- **Automated Model Reports:** create a document that includes screen shots of the model, material model, reinforcement bars, styles, loading stages, and all data used to build the model. The user has the ability to customize views. The report is generated in HTML format and can be printed easily into PDF.



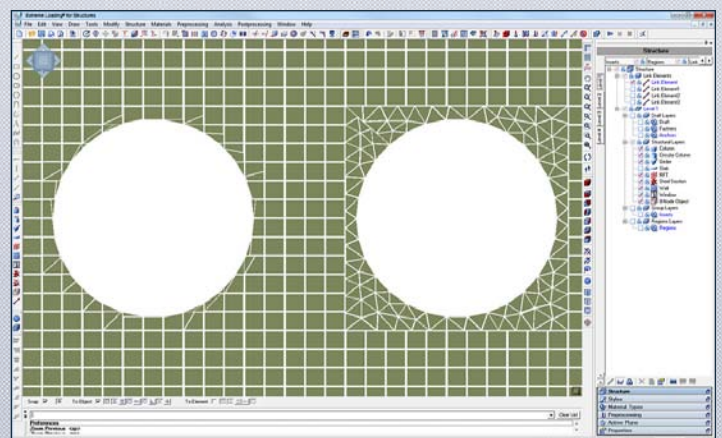
Cold-Formed Steel Structure



Custom Reinforcement



No Re-Meshing Required for Connectivity



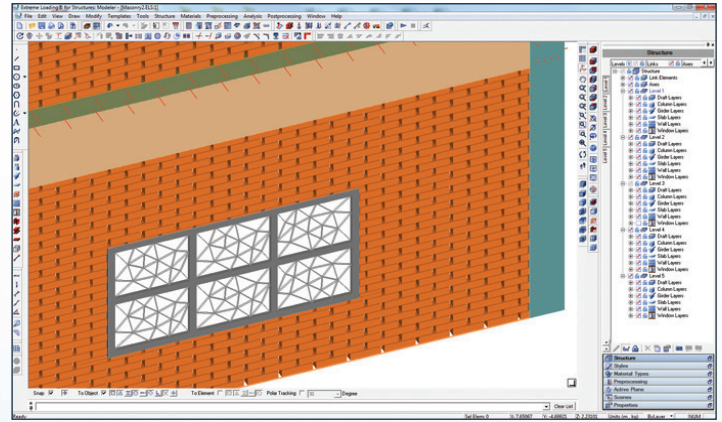
Slab with Hole Before & After Re-meshing

- **Modeling Tools & Navigation:**

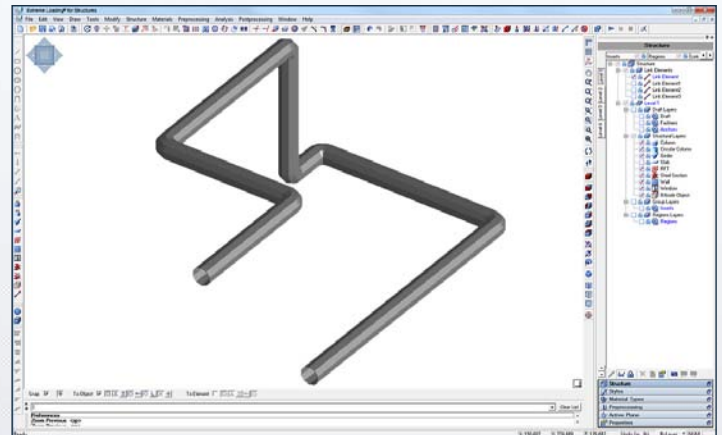
- **Active Planes:** Specify active planes to simplify creating 3-D models.
- **Enhanced Window Options:** Create multiple windows with cascade and tile options or dynamically view your model using snippet view.
- **Enhanced Show/Hide:** select, show, and hide objects using multiple filtering criteria.
- **Pick Command:** specify the coordinates of any point you need by simply selecting it using the pick command available in all dialog boxes.
- **Copy Format:** copy structural objects with assigned loads and boundary conditions.
- **Multiple Level Arrays:** generate multiple copies of levels.
- **Snap Options:** helps you build your model quickly and accurately.
- **Smart Recognition:** helps the user convert lines and shapes either imported or drawn into the modeling window to three-dimensional structural components.
- **Model Pile Interface:** Ability to model pile interface with surrounding soil without the need to model the ground. This enables the soil-structure interaction to be considered without the need to include huge number of elements for soil.
- **Import/Export ELS Files:** allow users to import and export all or part of an ELS model from one ELS file to another.
- **Import Draft Objects:** allows user to import draft objects for reference and conversion into objects.
- **Command Line Settings:** define custom command line shortcuts.

- **Streamlined Meshing:**

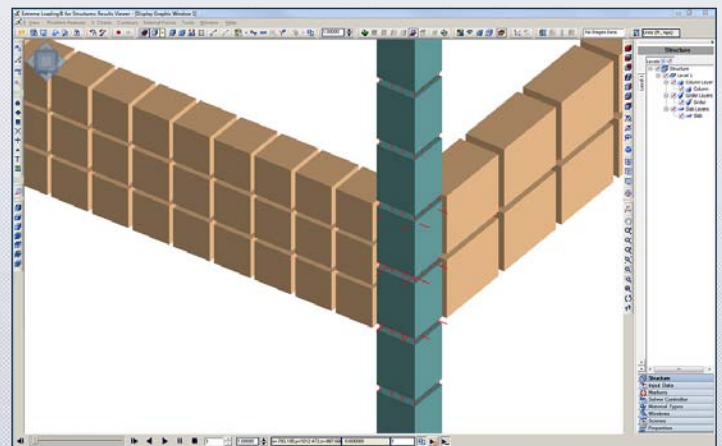
- **Re-meshing:** re-mesh element selections into square or Voronoi (random triangle) shaped elements to enhance and optimize element size and shape.
- **Automatic Mesh Adjustment & Element Connectivity:** element connectivity is generated automatically by the program without user-intervention. Re-meshing to create transition elements is not required. This saves precious time typically lost due to complex connectivity issues.
- **Automatic Formation of Plastic Hinges During Static & Dynamic Loading:** the user is not required to make assumptions about which areas are prone to failure as this is determined automatically by the program. This saves time typically lost during trial and error processes.



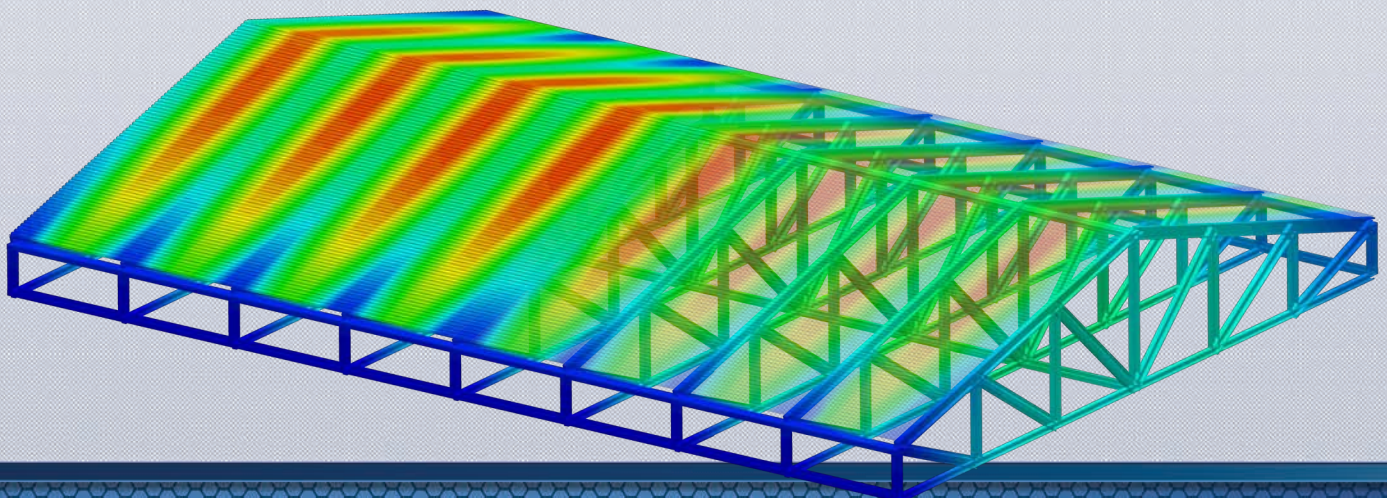
Easily Model Masonry



Pipe Created with Follow-Me Option



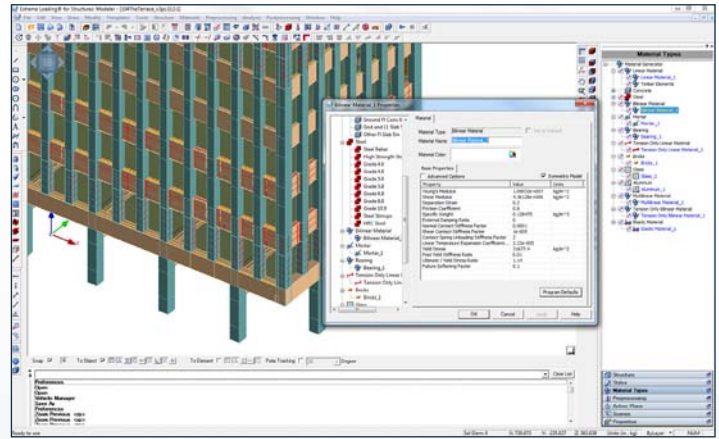
Automatic Mesh Adjustment & Element Connectivity



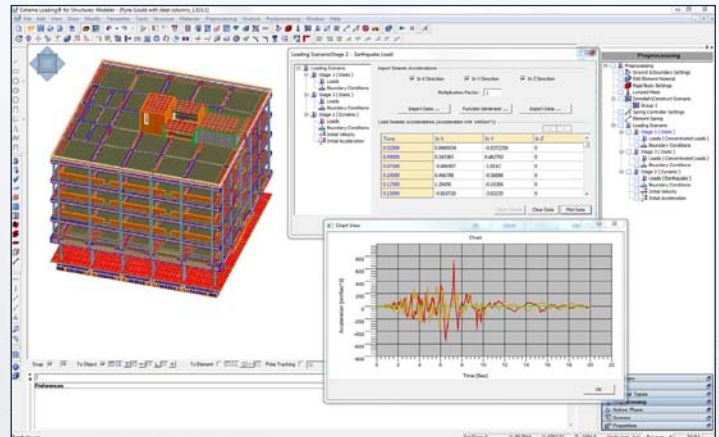
LOADING FEATURES

A wide variety of loading scenarios including multiple successive hazards can be implemented in The Extreme Loading® for Structures 6 utilizing its multi-stage sequential loading:

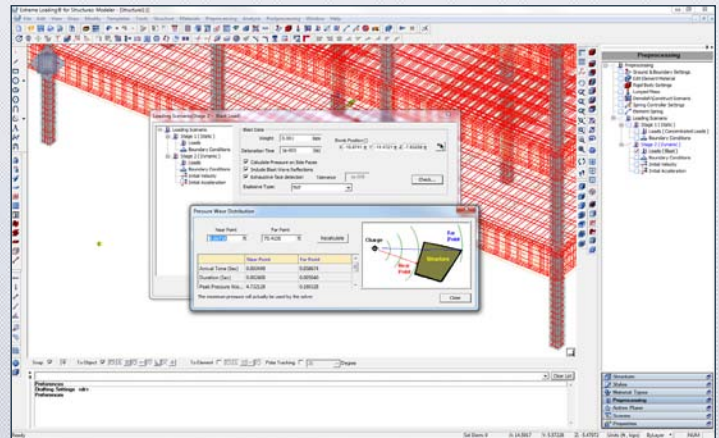
- **Material Modeling:**
 - **Linear Material Models:** for concrete, steel, masonry, mortar, glass, aluminum, bearing, linear elastic, and tension only linear elastic.
 - **Nonlinear Materials Models:** for concrete, steel, aluminum, brick, glass, tension only, and elastic, Bilinear, Multilinear, Tension Only Bilinear, Elastic Material, and Tension-Compression.
 - **Material Regions:** edit the material of all elements enclosed in a certain region.
 - **Spring Filter:** filter springs based on the spring direction. This helps modeling anisotropic materials by assigning different properties for different directions.
- **Loading:**
 - **Static Loading:** apply concentrated, displacement, hydrostatic pressure, uniform pressure, moving, line, and custom loads.
 - **Dynamic Loading:** apply concentrated, displacement, earthquake, custom or automated blast, uniform pressure, pressure sources, vehicle load, moving, or custom loads.
 - **Staged construction and deconstruction:** Activate or remove structural components at different times or stages.
 - **Initial velocity and acceleration:** apply initial velocity and/or acceleration to elements or structural component.
 - **Corrosion Effects:** consider the effects of corrosion on all types of objects and materials to determine the failure capacity of corroded structures as well as failure capacity after retrofitting.
 - **Spring Controller:** gives users control over the properties of springs enclosed in a specified region based on Time or Stage; allowing the user to Change Materials, Add/Remove Springs, Apply Cracks, Apply Temperature Loading, Apply Corrosion, Apply Pre-stressing.
- **Analysis Settings:**
 - **Kill Boundaries:** eliminates unwanted debris and decreases simulation time based on “killing” boundary conditions set by the user.
 - **Physical Data Conflict Dialogue:** automatically detects conflicts in loads and boundary conditions, allowing the user to remove them.



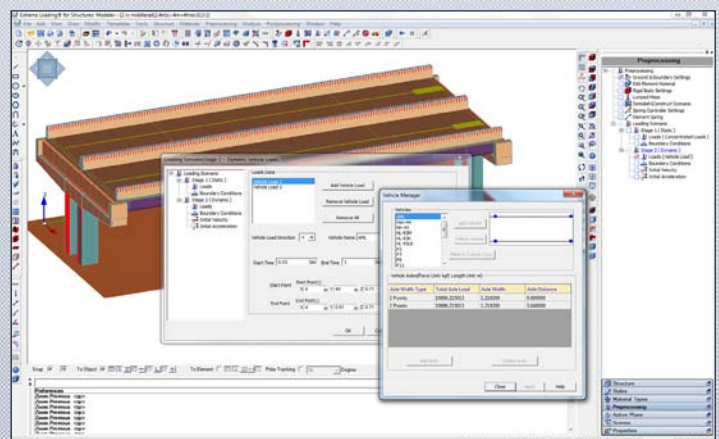
Material Loading



Time-History Earthquake Loading



Blast Loading



Vehicle Loading

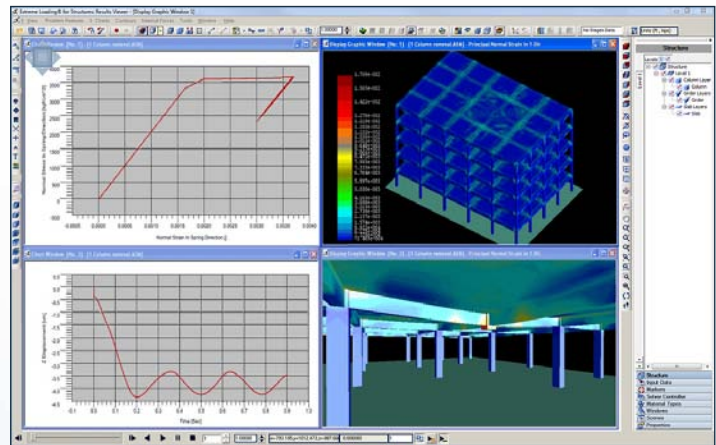
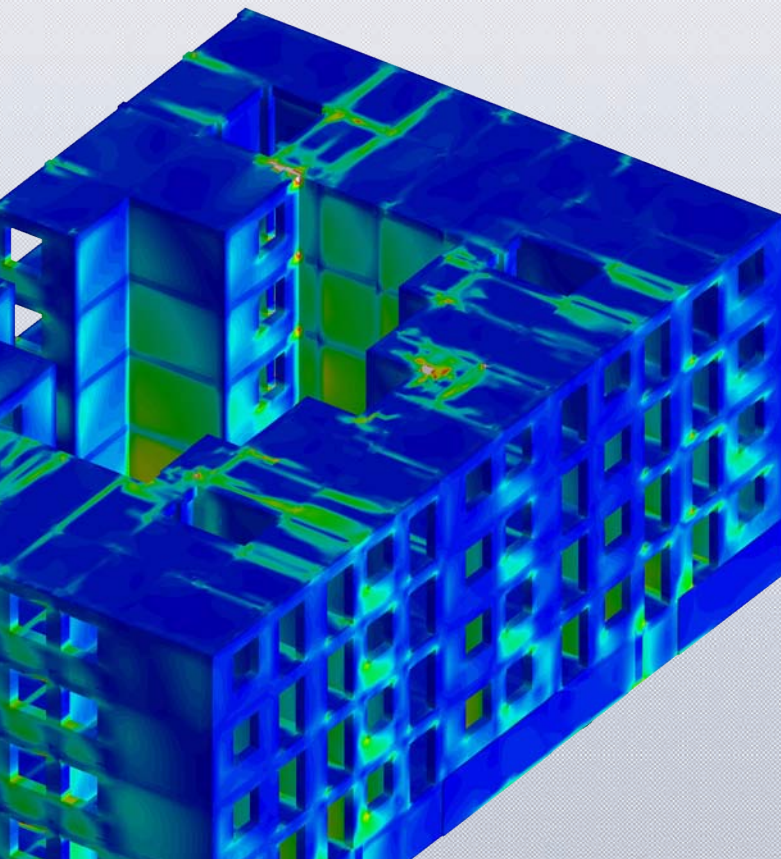
ANALYSIS FEATURES

The Applied Element Method (AEM) based solver in Extreme Loading® for Structures 6 is a derivative of the Finite Element Method (FEM) and the Discrete Element Method (DEM). AEM is capable of performing both linear and nonlinear analysis that follows the behavior of structures through separation, collision and collapse while automatically taking into consideration:

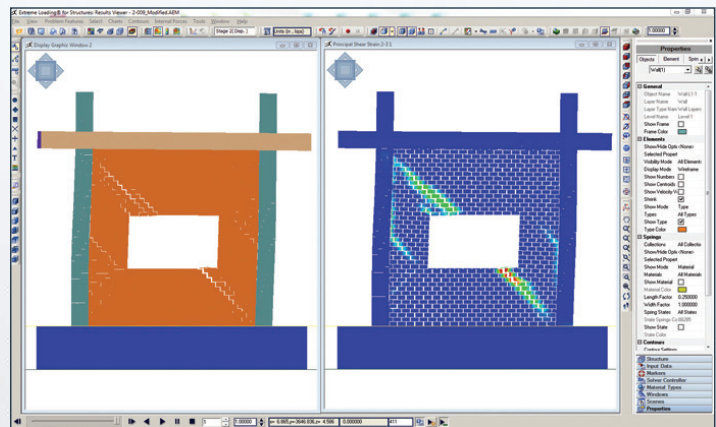
- **Yielding of Reinforcement:** automatically calculates material strain from elastic to plastic deformation.
- **Plastic Hinge Formation:** automatically places plastic hinges.
- **Buckling and Post-Buckling:** automatically calculates elastic and plastic bending under compressive loads.
- **Crack Propagation:** automatically calculates the location and propagation of cracks.
- **Membrane Action & P-Delta Effect (P- Δ):** automatically calculates the dynamic force and displacement caused by Membrane Action and the P-Delta Effect.
- **Separation of Elements:** automatically separates elements based on nonlinear material properties.
- **Collision and Collapse:** automatically calculates the collision and collapse of separated elements in both static & dynamic analysis.

There are two solver options available depending on the type of analysis you are performing and the time you have to solve.

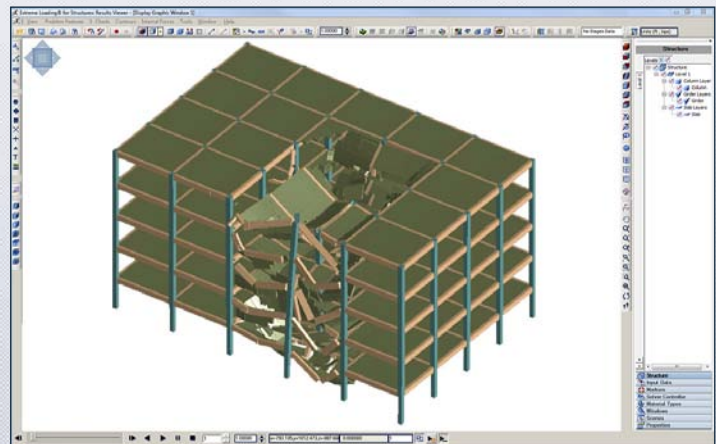
- **Exact Solver:** optimal in static and dynamic analysis with a relatively large time step (0.01 or 0.001 sec).
- **Iterative Solver:** optimal in cases where the time step used is very small, like in penetration or blast analysis problems (recommended time step is less than 0.0001 sec). This results in a significant reduction in RAM required to process the problem.



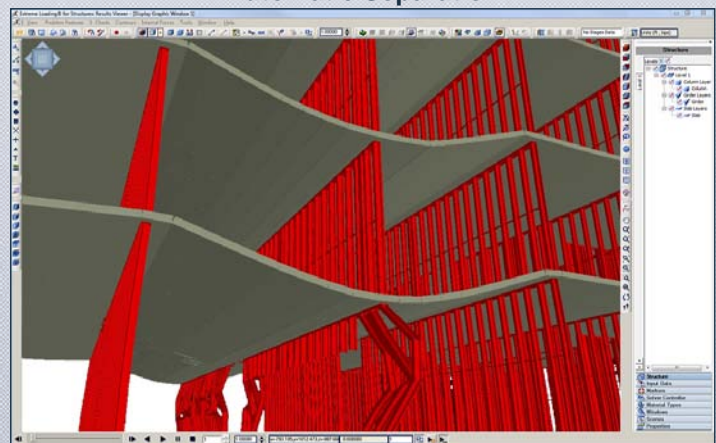
Automatic Plastic Hinge Formation



Automatic Cracking



Automatic Separation

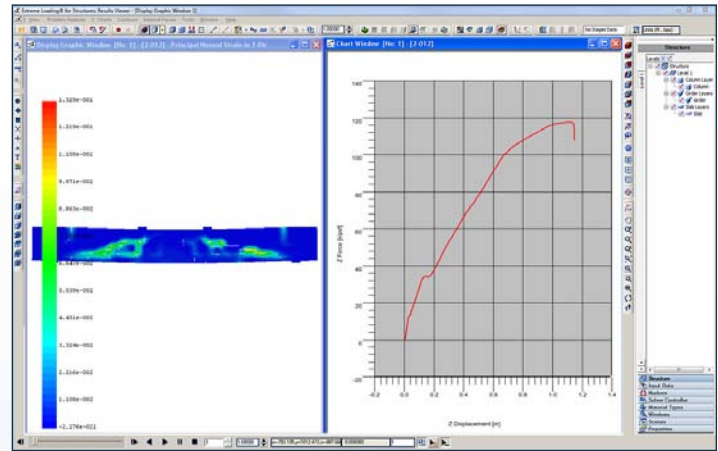


Automatic Membrane Action & P- Δ Effect

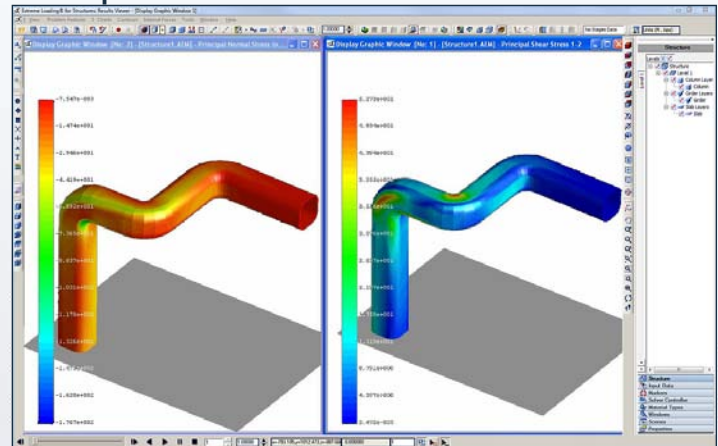
OUTPUT VIEWER FEATURES

The frame by frame Extreme Loading® for Structures Results Viewer allows the user to view, analyze and export presentation materials from the following options:

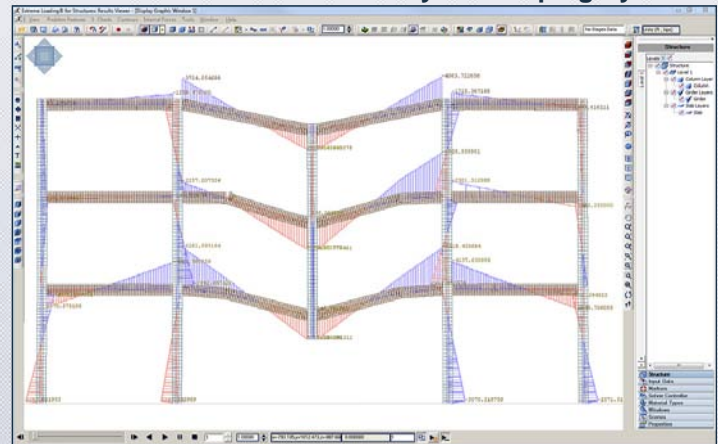
- **Open GL 3.3 Enabled:** Makes viewing and navigating expansive output for large problems especially smooth.
- **Show Status:** Show status of all springs (cracked, yielded, failed in tension or compression, separated, etc.).
- **Create Report:** Create a report on the maximum internal forces in each object separately
- **Filter Elements:** Ability to filter elements or springs based on output values. This enables the user to get elements of specific speed, displacements or acceleration in addition to springs with maximum stress, strain, etc.
- **Problem Features:** gives users several viewing options for interpreting results.
 - Velocity vectors
 - Contact points
 - Blast surfaces
 - Crack locations
- **Enhanced Display Controls:** allow users more control over output display and selection of elements, springs, sections, components, or levels.
- **Enhanced Filters:** display objects by material or component type.
- **Filtration Queries:** create custom filtration queries based on complex criteria (Object Type, Material, Interface ID, Pre-Stressing, Loading, Velocity, Rotation, Displacement, etc.). Import & export queries for common tasks.
- **Eigen Modes:** view animated periods, frequencies, and mass participation factor for all mode shapes of the model.
- **Custom Internal Force Diagrams:** create animated internal force diagrams for components, levels, or the whole structure taking into consideration cracks, rebar, yielding and other phenomena that occur during loading.
 - Bending moment
 - Shear
 - Normal
 - Torsion
- **Contour Diagrams:** create animated stress, strain, kinematics, and blast contour diagrams for components, levels, or the whole structure filtered by the component type or material type.
- **Charts:** allow users more control over all aspects of charts including Differentiation & Integration Options, and the ability to animate charts to follow the steps of the analysis.
 - Load
 - Displacement
 - Stress
 - Strain
 - Time
 - Velocity
 - Acceleration
- **Movie Generation:** add text and markers to *.avi and *.bmp.



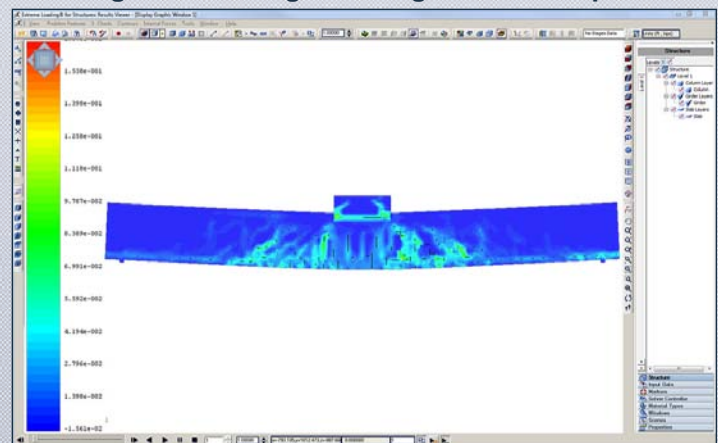
Displacement Chart: Reinforced Concrete Beam



Stress Contours: Seismic Analysis of Piping System



Bending Moment Diagram: Progressive Collapse Case



Stress Contour Diagram: Reinforced Concrete Beam

UPDATED SAMPLES

Samples are problems solved using ELS that discuss static, dynamic, blast, and progressive collapse cases. Most cases document a comparison of analysis performed using ELS with experimental results or analytical solution using other analysis methods or tools. Documentation and an ELS file are available explaining each case. You can use the ELS file to run the sample and view its output.

UPDATED MANUALS & TUTORIALS

Each ELS Subscription includes a complete set of manuals (Quick Start Guide, Blast Manual, Viewer Manual, Theoretical Manual) and more than 20 video tutorials that take users through all of the steps required to create and analyze various models and analysis cases. These manuals and video tutorials are updated on a regular basis and as new features are added.

ASI PORTAL

From the new ASI User Portal, Licensed ELS Users have 24-7 access to Software Updates, Tutorials, Quick Start Guide, Modeler Manual, Blast Manual, Viewer Manual, Theoretical Manual, Technical Reference Manual and the ability to upgrade or renew their license.

ELS CERTIFIED TRAINING

In addition to the startup materials that come with each license of Extreme Loading® for Structures Software (ELS), users will benefit from completing the Extreme Loading® for Structures Certification program. Training modules include:

- Structural Vulnerability Assessment
- Blast Analysis
- Progressive Collapse Analysis
- Seismic Analysis
- Forensic Engineering
- Performance Based Design

For more information on ELS Certified Training visit <https://www.extremeloading.com/customer-services-support/els-certified-training>

PROJECT SUPPORT

ASI offers ELS users various levels of project support. From modeling assistance, to peer review, to custom materials and loading, to remote simulation, to full project consultation services, ASI is here for you! For more information visit: <https://www.extremeloading.com/customer-services-support/project-support>

ADDITIONAL INFORMATION OR PURCHASE

For more information or a quote for Extreme Loading® for Structures visit: <https://www.extremeloading.com> or call us at 1-919-645-5090.

SYSTEM REQUIREMENTS

Minimum:

- Operating System:** Windows 7, 8, 8.1, & 10 (All 64 bit only)
- Processor:** Intel® Pentium® 64-bit 2 GHz
- Hard Disk:** 1 GB for Installation and 10GB for analysis output
- Memory:** 2 GB RAM
- Processor:** 64bit-Processor
- Graphics Card:** With OpenGL 3.3 Support, 1GB Memory

Recommended:

- Operating System:** Windows 7, 8, 8.1, & 10 (All 64 bit only)
- Processor:** Intel® Core™ i7 64-bit 3.6 GHz or faster
- Hard Disk:** 1 GB for Installation, additional space for analysis output depending on analysis options
- Memory:** 32 GB RAM or more
- Graphics Card:** With OpenGL 3.3 Support, Memory 2 GB or Higher

