# WHAT'S NEW IN EXTREME LOADING ® FOR STRUCTURES 6.0?

This new version of Extreme Loading® for Structures (ELS) reinforces the fact that ELS is a practical easy-to-use structural analysis software program created for practicing engineers who have complex projects with non-traditional load cases. The majority of the new features provided in ELS Version 6 are advanced modeling and postprocessing 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.

ELS Modeler Features:

1. **Group Trim/Extend -** In many cases, small changes in the cross section of columns and girders cause integrity issues with the model. In other words, some components may have small overlap or small gap due to changes in the Styles that are made by the user. With this option, the user can select these components and rather than cutting and creating irregularly shaped/size elements, they can adjust the objects ending points by either extending or shrinking them to maintain connectivity between components.

2. **Array Along Path option in array dialog box -** This feature helps generating components along a pre-defined path.

3. **Automated Model Reports -** This feature is a great new feature for documenting your model. The Automated Report Generator allows the user to 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. These reports are of great help when submitting a model for review or as a reference when the model needs to be checked. The user has the ability to customize views in the report. The report is generated in HTML format and can be printed easily into a PDF file.

4. **Export STL Extension -** This feature helps when exchanging the ELS file with other software using STL format.

5. **Search/Select Object by Name -** When the user generates a report, it shows all screen shots of the used components, loading, etc. All components are referred to by their name. This feature helps the reviewer find a specific component in their ELS model searching the model by its name when he is reviewing its report.

6. **Convert to Hollow in Implicit Plate Dialog -** This feature helps generating hollow implicit steel sections.

7. **Link Spring Area (Lattice Method) -** Some users may need to use the Long Spring feature to solve structures using the Lattice Method. This is possible with all previous versions of ELS. The main obstacle was that the long spring area is connected to styles directly. This means that a problem with 1,000 lattice element requires the user to manually add 1,000 style objects with different areas. To fix this issue, the Spring Area was added as a property to each spring. Now the user has the option to select group of elements and convert them into a Lattice.

8. **Edit by Object -** This feature helps the user greatly. Assume that the user selected two or more objects. The Properties panel shows only the common properties. In some cases, the user wants to edit the select properties one by one. Clicking on this check box allows the user to show the list of selected objects and edit them one by one. This makes the process much easier compared to previous version.

9. **Import & Export Filtration Queries -** In many cases, the user is looking for specific objects using the Filter. To do this, they have to build the search criteria which is a combination between and/or statements to look for the objects. Now the user can export these criteria for use of other ELS files saving the time needed to create them

10. **Filter by Interface ID -** This helps the user find objects with a specific Interface ID.

11. **Filter by Pre-stress Criteria -** This feature helps when the user is looking for objects where pre-stressing is applied.

12. **Right Click Menu for Objects -** Show Selected only & Invert Selection.

13. **Mesh Elements Option in the Additional Task Menu -** The user has the ability to refine the mesh of selected elements. This help making some segments of the mesh finer when stresses are concentrated.

14. **Convert Selected Elements to Lines in the Additional task menu -** This feature helps when the user wants to refine the mesh at certain segments of the model and to make use if the lines generated at the elements boundaries to snap for the new generated elements

15. **Steel Section Styles -** Added UK-Europe Library.

16. **Added "Select Elements inside Region" to Right Fly Menu -** In some cases, elements surrounded by a specific region needs to be selected to apply pressure loads. The easiest way is to create the region and with this feature, the elements will be automatically selected.

17. **Adding check box of generate separate per segment in extrusion -** This helps when the user wants to modify the properties and reinforcement and/or material for each segment separately.
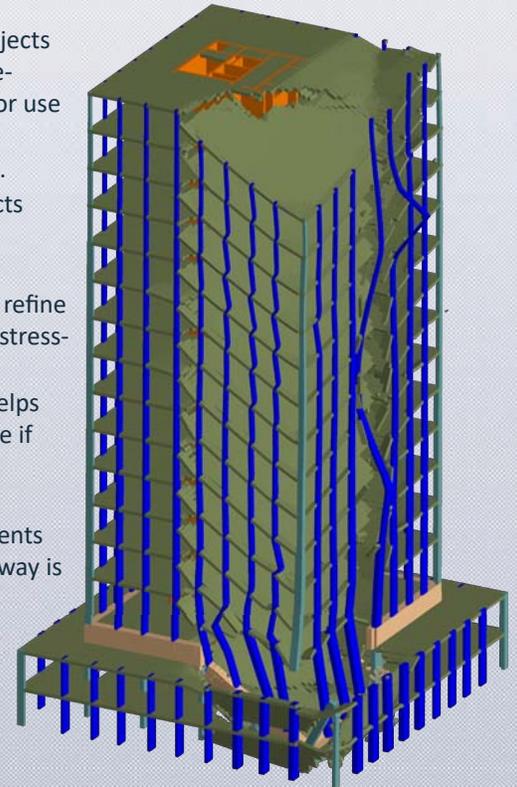
Solver Features:

1. **Custom Blast Loading -** Allows the user control over the blast waves applied to different parts of the building. In many cases, the user may not model non-structural components. This affects the accuracy of the Automatic Blast Loading option available in ELS versions 5.0 and older. With this new Custom Blast loading type, the user can decide front and side surfaces, as well as select elements where the pressure is applied. Several examples are added to the Verification Samples that compare with the UFC code showing good agreement.

2. **New Iterative Solver -** In previous versions of ELS there was only an Exact Solver. The Exact Solver is optimum in static analysis and dynamic analysis with a relatively large time step, like 0.01 or 0.001 seconds. The Iterative Solver is 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). The Iterative Solver has the following advantages:

    a. In many cases will be faster than exact analysis.

    b. User control over may reach Exact Solver accuracy.

    c. In Exact Analysis, the Exact Solver may need up to 70 GB of RAM in cases of 1,000,000 elements. This is down to about 3-4 GB of RAM in cases using the Iterative Solver, which is a big advantage.

3. **Added Contact for Static Loads -** In all versions before 6.0, Contact Analysis was only performed for Dynamic Analysis. In ELS 6.0, Contact Analysis is now calculated during Static Analysis. This has various applications such as:

    a. Adding weakening in columns for static loads where bearing from contact springs replaces the matrix springs.

    b. In some static analysis cases when compression failure occurs, the structure loses all its strength due to lack of compression resistance. With this feature, contact helps by adding post-failure resistance, allowing the analysis to continue for larger deformations instead of giving instability.

4. **Blast Limit Error Message -** A new error message creates a blast error text file at output folder when blast limits are exceeded during the blast check.

5. **Shear Stress Weight Added to Steel Model -** In several cases in versions before 6.0, the steel model will not reach its full ductility due to contribution of shear stresses into the failure criteria. In some cases, like impact problems and with very thin sections, the steel tends to bend rather than cut in Shear. The user now has the ability to control the Shear Stress Weight. Setting his value to 1.0 results in Shear having full weight (like in previous versions of ELS). When the Shear Stress Weight is set to 0.1, the Steel Springs are not allowed to be cut if the Shear Stress contribution to Ultimate Strength is greater than 10%. This helps the user control the ductility of the steel model. A default value of 0.1 is recommended for most of practical cases.

6. **Irregular Ground -** Several bug fixes were made in irregular ground. The user can draft irregular ground geometry and import it directly into the model. This helps when the ground is sloped or irregular and saves the user the effort and analysis time needed to add elements to represent the ground.
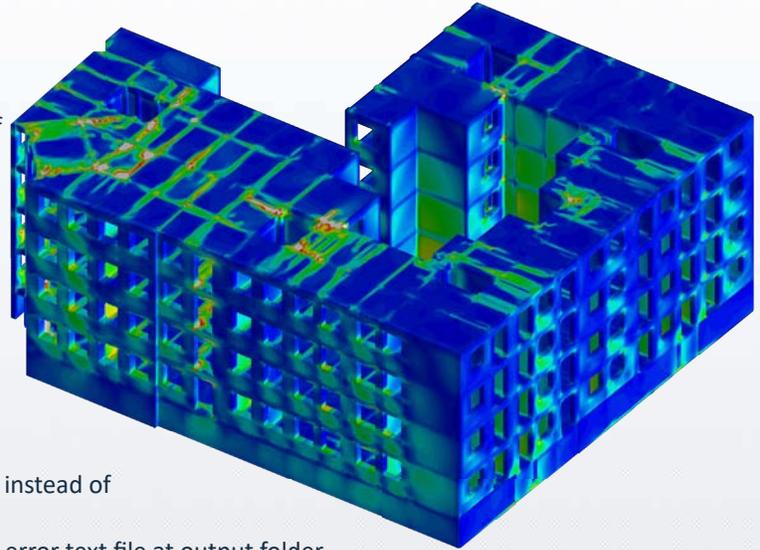
ELS Viewer Features:

1. **New Open GL 3.3 - Makes viewing output much faster! -** Considerable speed enhancement achieved by upgrading to the new Open GL 3.3 resulting in much faster navigation, rotation, and walk through. This is very important for big problems especially when element details and/or springs are shown.

2. **Select by Coordinates -** In many cases, the user wants to select a specific element ID at a specific coordinate. For example, when comparing results of two cases where ID's are different while the user wants to get the element ID at a specific point.

3. **View Custom Components Option to Internal Forces Menu -** This helps defining the used custom components to assign new ones or modify existing ones.

4. **Differentiation & Integration Options to Charts -** This feature helps tremendously to differentiate or integrate charts with respect to X or Y. For example, differentiating the X- displacement of each floor with respect to Z coordinate results in story drift. Differentiating Load-Displacement or stress-strain results in stiffness. Integrating Load-Displacement results in energy. So, instead of relying on other software to do it after exporting the chart data, the user can do it internally.

5. **Export STL extension -** This feature helps exporting the output for one frame from the ELS Viewer to other software tools using STL format.

6. **Select Custom Component Elements in Add Task of Object -** This feature helps if the user wants to review the elements associated with a specific custom component or edit them for other operations in the ELS Viewer.

7. **Import & Export Queries in Filtration Dialog -** The user can use this feature to exchange filters between different ELS files. For example, if the user is using the filtration option to find objects of specific criteria, like elements at top floor with rotation greater than specific value. Instead of regenerating the filters for each file, the user can exchange it between files easily saving time.

8. **Mass Participation in Eigen Mode -** Especially helpful in seismic analysis, the user can use this feature to obtain the Mass Participation Factor in each mode.