Brief Summary of Expected New Features in GTSTRUDL 27

Among other new features and error corrections, GTSTRUDL 27 is expected to include the following new features:

- ■ General Feature Enhancements

  - A new "Large Problem Resource Reservation" option under the "Modeling/General" menu option in the command/text window (and the command "LARGE PROBLEM n") has been added that may dramatically decrease processing time for large analysis and design jobs.
  - The data base RESTORE processing time for large data bases has been dramatically decreased. For example, RESTORE processing times have been seen to be reduced by factors of 10x or more for large data base save (*.gts) files .
  - The LIST SECTION FORCES and LIST SECTION STRESSES commands include an option to output a summary of maximum and minimum section values for each selected member and maximum/minimum values over the entire list of selected members.
  - A "ROTATE LOAD" command has been added. This command creates a new independent static loading condition by rotating the global joint and member load components through a specified set of rotation angles.
  - Viscous damper element (dash pot) for linear and nonlinear dynamic analysis.
  - The new NLS4PH (nonlinear spring 4-parameter hysteretic) general purpose spring element with up to six uncoupled degrees-of-freedom for nonlinear static and dynamic analysis which can be modeled at support joints and between any pair of joints. Employs a 4-parameter, bi-linear force-displacement/moment-rotation curve including parallel stiffness hysteretic behavior.
  - Hysteretic friction element nonlinear effect that can be assigned to truss members. Applicable for nonlinear static and dynamic analysis.
  - Stress-strain curves for plastic hinges can now be defined in piece-wise linear form using a table of stress-strain data points.
  - A new and far more robust next generation nonlinear finite length plastic hinge segment model that provides more flexibility in the modeling of plastic hinge behavior
  - The "LIST PLASTIC HINGE DUCTILITY RATIOS" and "LIST PUSHOVER DUCTILITY RATIO" commands are enhanced by including the direct specification of a fiber material yield strain, which provides more flexibility in the definition of plastic hinge fiber yielding for the determination of displacements corresponding to the onset of yielding.
  - The "LIST PUSHOVER LIMIT STRAIN LOAD" command lists the pushover analysis incremental loads that correspond to the onset of a user-defined fiber limit strain throughout the structural model. This is very useful for evaluating ductility-based limit states for frames using the plastic hinge model or new plastic segment model.
  - The cable prestress analysis computational procedure has been enhanced to provide for the specification of the prestress convergence tolerance parameter separately for each cable network. In previous versions, it was only possible to specify a single prestress analysis convergence tolerance that applied uniformly to all cable networks.

- The "JOINT RELEASES" spring stiffness damping specification has been expanded to allow the specification of a separate damping factor for each spring degree-of-freedom. The purpose of this enhancement is to improve and broaden the composite modal damping computation.
- The "LIST RESPONSE SPECTRUM BASE SHEAR" command. This command is used to print base shear and story shear results for response spectrum loads. Both modal results (i.e., on a mode-by-mode basis) and mode combination results can be output.
- The "CREATE RESPONSE SPECTRUM" command has been extended to provide for the automatic creation of response spectrum curves as functions of PERIOD in addition to functions of FREQUENCY.
- The output from the "LIST TRANSIENT MAXIMUM" command has been improved to include the time point sequence number as well as the actual time corresponding to the maximum response value.
- A "FORM UBC97 LOAD" command has been added. This command computes an equivalent static lateral load pursuant to the provisions of Section 1630 of the 1997 Edition of the UBC code.

- **Additional Dialogs and Menu Options in the GTSTRUDL Text Output Window Have Been Added Including:**

  - A "View Current Command History" menu option has been added to the "File" menu which will display comments and commands from the current GTSTRUDL text window to a text editing window from which editing and saving of the history of commands may be performed.
  - A "Import - GTSTRUDL Output File" menu option has been added to the "File" menu which allows the creation of a GTSTRUDL text command file from an existing text output file.
  - A "Notes" menu option has been added to the "Modeling/General" menu which permits a file of user created notes to be maintained within the GTSTRUDL data base, and to be output by the new PRINT NOTES command and by the "Generate STRUDL Input File" menu option in GTMenu.
  - A "Set Startup Units" menu option has been added to the "Modeling/General" menu so that a user may change the default startup units of GTSTRUDL from Inches, Pounds, Radian, Degree Fahrenheit, and Seconds to any other recognized set of GTSTRUDL units.
  - An "Output File Control" menu option has been added to the "Modeling/Text Output of Model Data" menu which permits the user to change the destination of the PRINT command type of output from the text window to a specified file (in either Append or Replace mode), and back to the text window (the menu option for the COUTPUT command).
  - A "Model Verification" menu option has been added to the "Modeling" menu which enable the user to request a variety of model check options including CONSISTENCY CHECK, LOCATE INTERFERENCE JOINTS, LOCATE DUPLICATE JOINTS, and REMOVE DUPLICATE JOINTS. Of course, additional model checking is available by a variety of GTSTRUDL commands as well as various options provided within the GTMenu graphical interface.

- A "Foundation (Soil) Springs" menu option has been added to the "Modeling" menu which will cause the *automatic* calculation of elastic support conditions (i.e., linear elastic spring constants) under a user specified finite element mesh (e.g., under a model of a reinforced concrete mat foundation, or under a steel base plate), and based on a user specified soil Modulus of Subgrade Reaction.
- A "CREATE Load Combinations" menu option has been added to the "Modeling/Loads" menu to cause loading combinations to be performed by Algebraic, RMS, and Absolute Value summations following one or more analyses (the menu option for the CREATE LOAD COMBINATION command).
- The command "PRINT TIME" has been added to output the current date and time.
- The "WRITE MEMBER PROPERTIES LONG NAME" Data Base Exchange (DBX) function now uses a caching technique to speed up the file writing process. For large numbers of members, the speedup can exceed 80%, for example, from more than 5 seconds to less than 1 second. The caching technique is most effective for models with large numbers of members with the same long names.
- A "Section Stresses" menu option has been added to the "Results/Text Output" menu.
- An "Output File Control" menu option has been added to the "Results/Text Output" menu which permits the user to change the destination of the LIST command type of output from the text window to a specified file (in either Append or Replace mode), and back to the text window (the menu option for the COUTPUT command).
- The previous "Design" menu option has been changed to "SteelDesign".
- A new "RC_Design" menu has been added with dialogs for MATERIAL REINFORCED CONCRETE, METHOD, MEMBER DIMENSIONS, CONSTANTS, DESIGN DATA, PROPORTION, PRINT REINFORCEMENT, SCHEDULE and QUANTITY TAKEOFF.

■ Additional Menu Options and Dialogs Have Also Been Added That Permit the User to Easily:

- Create Equivalent Static Earthquake Loads using structural modal mass distribution and/or Response Spectrum loading data (based on Section 3.3.3.2.C of the NEHRP Guidelines for the Seismic Rehabilitation of Buildings - FEMA Publication 273)
- Create UBC '97 statically equivalent earthquake loads
- Specify dynamic degrees of freedom and general dynamic parameters for the control o f linear and nonlinear transient dynamic analysis
- Execute linear physical analysis (direct integration)
- Execute nonlinear transient dynamic analysis
- Create X-Y graphical plots of Pushover Analysis Results
- Output nonlinear analysis plastic hinge stresses, displacements, ductility ratios, and hinge status
- Output pushover analysis results including strain limit loads, global ductility ratio, and plastic hinge ductility ratios

- ◼ New GTMenu Graphical Interface Features:

  - Information in the Inquire output window can now be routed to the command text window for printed output.
  - Expanded the INQUIRE menu option to include load inquiry.
  - Joint, member, and finite element loads may be far more easily edited for the currently active load using the "Control-Key/Right-Click" operation on a joint, member, or finite element.
  - Display member and finite element temperature loads, member distortion loads, and joint displacement loads.
  - A new ACTIVE LOAD option added to the button bar
  - Added hot keys (see Hot Key Help menu) to perform the function of stepping backward and forward through views previously created by rotating, panning, windowing, zooming, etc.
  - Deformed shape of members (i.e., bending deformations) may now be displayed in a static position and animated.
  - Table shape long names are displayed in the Create Member Properties Table dialog.
  - Added the ability to zoom on an image containing both an undeformed and overlaid deformed graphical display.]
  - Increased to 1.0 million the number of joints, members, and finite elements that can be displayed within GTMenu

- ◼ Additional Steel Design Features Including:

  - AISC LRFD Third Edition code, LRFD3.
    - , I shapes
    - , Single Angles (axial force only)
    - , Double Angles (axial force only)
  - British Standard, BS5950 code based on the BS 5950-1:2000
    - , I shapes
    - , Single Angles (axial load only)
  - Brazilian Tables from ABNT, NBR 5884.
    - , CS Table – Welded plate girder shapes for columns (I shapes)
    - , CVS Table – Welded plate gird shapes for beams and columns (I shapes)
    - , VS Table – Welded plate gird shapes for beams (I shapes)
  - Automatic computation of the AISC effective length factor from the Compute k-factor command no longer requires the specification of the Column Line command.
  - Computed values of k-factors are now displayed by the SELECT and CODE CHECK commands.
  - Member deflection code check information has been improved.
  - For offshore jackets, and for the hydrostatic collapse check under the API working stress design (APIWSD20) and API load and resistance factor design (APILRFD1) codes, the steel design Parameter Z (depth below the water surface of some section along the member at which the hydrostatic collapse check is being performed) no longer needs to be specified for each member. Rather, the Parameter Z is now automatically calculated for each such section along each member.

- Reorganized steel design pull-down menu and new steel design dialogs and menu options in the GTSTRUDL text output window including:

  , A new option has been added to the Physical Member dialog to automatically create physical member names.
  , Since the Column Line command is no longer required for automatic k-factor calculations, the shortcut to the Column Line command has been removed from the Steel Design Wizard.
  , New "Loading Combination" and "Edit Load Combination" dialogs have been added to the Steel Design Wizard.
  , The load display page in the Steel Design Wizard for Stiffness Analysis is now optional. The default is not to show the stiffness analysis loads and to assume all currently defined independent and load combination static loads are active for the next stiffness analysis. The user can click on the check box on the Stiffness Analysis page to activate the load display page and select which loads are to be active and inactive for the next stiffness analysis.
  , A new selection dialog for the parameter TBLNAM has been added into the Parameter dialog which displays the GTSTRUDL Table Category that corresponds to the selected CODE value.
  , In the PARAMETER dialog, a text box is now displayed to make the editing of a member list easier to accomplish.

- Updated Steel Design Documentation

  1. IS800 (Indian) Steel Code Documentation.
     Section IS800.1   -   IS800 Code
     Table IS800.1-1   -   IS800 Code Parameters
     Table IS800.1-2   -   GTSTRUDL Indian Standard Codes
     Table IS800.1-3   -   GTSTRUDL Profile Tables for the Design based on the IS800 Code
     Section IS800.2   -   Properties Used by IS800
     Section IS800.3   -   Parameters Used by IS800
     Section IS800.4   -   Provisions of IS800
     Section IS800.4.1   -   General Nomenclature for IS800
     Section IS800.4.2   -   IS800 Provisions for I shapes
     Section IS800.4.3   -   IS800 Provisions for Channels
     Section IS800.4.4   -   IS800 Provisions for Single Angles
     Section IS800.4.5   -   IS800 Provisions for Tees
     Section IS800.4.6   -   IS800 Provisions for Double Angles
     Section IS800.4.7   -   IS800 Provisions for Round Bars
     Section IS800.4.8   -   IS800 Provisions for Pipes
     Section IS800.4.9   -   IS800 Provisions for Square and Rectangular Bars
     Section IS800.4.10   -   IS800 Provisions for Structural Tubing

, <u>LRFD3 code documentation</u>.

■   Other very useful and powerful features