GTSTRUDL 29.1, 29.2, 30, and Beyond
Current and New Features and Enhancements

Leroy Z. Emkin , Ph.D., P.E., F.ASCE
Founder and Co-Director
Computer Aided Structural Engineering Center
Georgia Institute of Technology
Atlanta, Georgia 30332-0355 U.S.A.
Tel: +1-404-8942260
Fax: +1-404-8948014
email: casec@ce.gatech.edu
Web Page: www.gtstrudl.gatech.edu

Presentation Outline

- Version 29.1 – (released September 2007)
- Version 29.2 – (available upon request)
- Enhancements and demonstration of new features in Version 30
- Future enhancements

Version 29.1
New Features

- The GTSES sparse equation solver, first implemented under the STIFFNESS ANALYSIS GTSES command in Version 29, has been implemented as a stand-alone program. When executed as such, the GTSES sparse equation solver is able to allocate all available virtual memory to its own execution process, thereby increasing the efficiency of the equation solution beyond the improvements already made by the STIFFNESS ANALYSIS GTSES command.

  The STIFFNESS ANALYSIS GTSES command also stores the results of the analysis (joint displacements, member and finite element forces, finite element stresses and strains, reactions and resultant joint loads) in files in the current working directory, further increasing the size of static analysis models that can be solved and the efficiency with which they are solved.

Version 29.1
New Features

- An example of a large model execution which completed in Version 29.1 but ran out of memory in Version 29 is shown below:

Number of Joints	40042
Number of Members	3055
Number of Elements	41332
Number of Loadings	50
Number of Loading Combinations	48
Average Bandwidth + Standard Deviation	584

Time to solve using GTSES for 240,252 degrees of freedom = 197 seconds
Total STIFFNESS ANALYSIS GTSES time = 767 seconds

Version 29.1
New Features

- The AREA LOAD command has improved geometrical error detection and reporting. In addition, the total area and applied load are now printed as an additional verification tool.
- The output from the DESIGN SLAB command was modified to display a full listing of elements selected and used in the computation of the total moment acting on a cut section, for both the Wood & Armer and element force algorithms. The DESIGN SLAB command remains a prerelease feature in Version 29.1.

Version 29.1
Error Corrections

- Error corrections as documented in the release guide

Version 29.2

- The modified features and error corrections implemented in Version 29.2 are described on the next few slides.
- This version is available only as a download upon request. You must have Version 29.1 installed as Version 29.2 is installed in the Version 29.1 installation directory.

Version 29.2
General

- A warning message is now issued when Member Releases have been specified for a member but no member release data has been given as shown in the following example:

MEMBER RELEASES
1 2
(Note that no release data has been given)

Version 29.2
GTMenu

- Contours are now plotted correctly for the case when all contour values are either all positive or all negative.
- An abort will no longer occur during Redraw Solid for models which contain a mixture of 3D solid elements and 2D elements.

Version 29.2
GTSTRUDL Output Window

- The datasheets for displacements, member end forces, and section forces will now work when such results have been computed using the GTSES solver.
- The paths to the DXF and CIS/2 Import and Export functions available under the File pulldown have been corrected.

Version 29.2
Nonlinear Analysis

- Nonlinear analysis will no longer abort in the second of multiple load analyses when rigid bodies are incorrectly specified as having nonlinear geometric effects.
- An abort will no longer occur if the strain exceeds the limits of the piecewise linear stress-strain curve in a custom plastic hinge.
- Pushover analysis displacement control will no longer abort when inactive or deleted joints are present.

Version 29.2
Offshore

- The computation of chord SCF values for fatigue analysis has been improved. In earlier versions, excessively conservative chord SCF values were computed by requiring that any chord SCF had to be no less than the maximum corresponding brace SCF from all braces connected to the chord. This requirement is not necessary and has been removed.

Version 29.2
Offshore

- The ASSEMBLE FOR FATIGUE command no longer aborts when wave loads have been read from multiple wave load files created by GTSelos and each of the wave load files contains member loads that reflect multiple duplicate wave heights and wave periods.
- The error that may cause the incorrect computation of Kuang and Smedley chord-side stress concentration factors and thus possibly incorrect fatigue damages by the COMPUTE FATIGUE LIFE and PERFORM FATIGUE ANALYSIS commands has been corrected.

Version 29.2
Steel Design

- The deflection parameters DefLimLo, DefLimYL, and DefLimZL are now printed correctly.
- The Check and Select commands for a deflection check or design will no longer abort when the new GTSES solver has been used.

Version 29.2
Steel Design

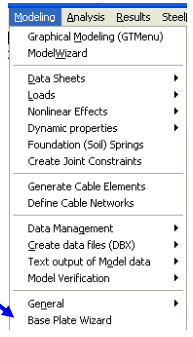
- When a value of -99.0 is displayed for provision 'H1-1 COM' of the ASD9 code or provision 'EQ1.6-1A' of the 78AISC or 69AISC codes, the actual/allowable ratio indicates an incomplete combined stress computation. This generally means that the value of *fa/Fe* is greater than or equal to 1.0 which causes the denominator of equation H1-1 of the 1989 AISC ASD Ninth Edition or equation 1.6-1a of the 1978 and 1969 AISC codes to be less than or equal to zero [$(1 - fa/Fe) \leq 0.0$]. When *fa/Fe* is greater than 1.0, the stress due to axial force is larger than the Euler buckling stress.

  Instead of printing -99.0 for the default code check results in Version 29.2, the combined axial and bending stress values based on equation H1-3 of the 1989 AISC ASD Ninth Edition or equation 1.6-2 of the 1978 and 1969 editions of AISC are printed. The error messages that the denominator of equations H1-1 or 1.6-1a is a less than or equal to zero are still printed, but in the code check default output, the value for the combined axial and bending stress (equation H1-3 or 1.6-2) also will be displayed. If you request summarized results, you will see a value of -99.0 for provision 'H1-1 COM' or 'EQ1.6-1A', which indicates the problem described above when computing the Euler buckling stress and the interaction equation.

Version 30 Enhancements

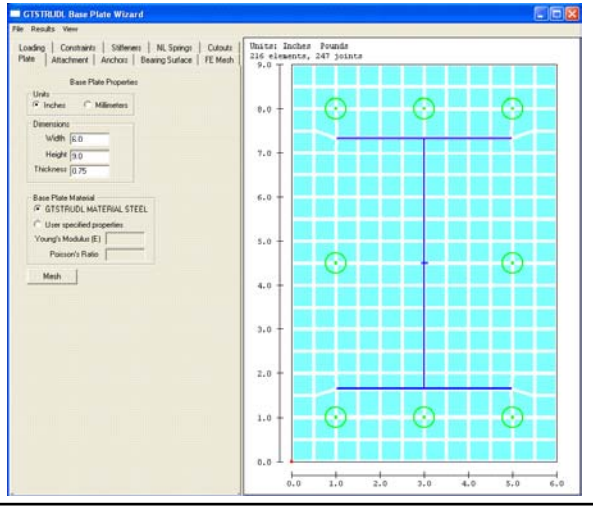
Version 30
New Base Plate Wizard

- A new Base Plate Wizard is available under the Modeling pulldown:

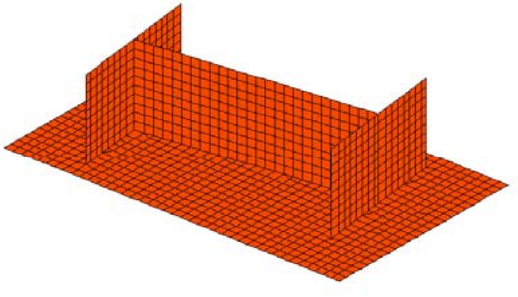


Version 30
New Base Plate Wizard

- A base plate model consists of the plate, attachments, anchors, bearing surface and stiffeners. The model may be adjusted with constraints and cutouts.
- Up to 10 attachments per plate
  - AISC Wide Flanges, Tubes, Channels, Tees, Angles, and Pipes are included
  - User defined shapes can be used
  - Attachments may be rotated in-plane
  - Attachments may be modeled as in-plane RIGID SOLID or as FE extension
- Anchors may be modeled as elastic or NL springs, or as frame members.
- Bearing surface stiffness may be specified as Fc', E, or F/L**3
- Stiffeners may be rectangular, trapezoidal or triangular
- Constraints allow extra boundary conditions or specified node locations
- Cutouts allow modeling of notches and skewed plates
- Base plate models may be saved and recovered using .gtbp format
- Results (anchor forces, bearing surface pressure, plate stresses and displacements) may be viewed in the BPW



Version 30
New Base Plate Wizard



Version 30
New Features - Dynamics

- The CREATE TIME HISTORY command has been extended. You may now specify *time segment specs*, which allow you to create a new time history file by extracting a segment of an existing time history file and factoring an initial portion of that segment with a ramp function.

  This feature allows you to isolate a much shorter, perhaps critical, portion of a longer kinematic or force/moment time history function in order to perform, for example, a preliminary transient analysis.

  The modified command is shown on the next slide.

Version 30
New Features - Dynamics

CREATE TIME (HISTORY) FILE 'filename$_{new}$' (FROM) { joint specs | time segment specs }

where,

joint specs = JOINT { 'a$_1$' | i$_1$ } TRANSLATION { X | Y | Z } LOADING { 'a$_2$' | i$_2$ }

time segment specs =

TIME SEGMENT (OF) FILE 'filename$_{src}$' START (TIME) v_S -

END (TIME) v_E RAMP (FUNCTION) { LINEAR | COSINE } BUILD (TIME) v_B

Version 30
New Features - Dynamics

- Dynamic analysis external file solver support, as specified by the DYNAMIC PARAMETERS command, has been added to harmonic analysis. Harmonic analysis with the external file solver enabled is now able to solve significantly larger jobs at improved speeds over the existing harmonic analysis.

Version 30
New Features – Dynamic Analysis

- Modifications will be made to the RESPONSE SPECTRUM LOAD command and the procedures for the computation of response spectrum mode combinations (the *Gupta Method*) in order to implement the *Complete Solution for Response Spectrum Analysis* pursuant to US NRC Regulatory Guide 1.92, Rev. 2 Section 1.5.

Version 30
New Features - General

- A new feature, CALCULATE PRESSURE, has been added to report a calculated pressure for each joint incident to the specified finite elements, based on spring forces at the joint and the tributary area of the joint. This command approximates the pressure felt by a continuous supporting material, such as soil or concrete under a foundation.

  The new command is shown on the next slide.

Version 30
New Features - General

- New Calculate Pressure command:

CALCULATE PRESSURE PLANE $\left\{\begin{matrix}\text{X}\\\text{Y}\\\text{Z}\end{matrix}\right\}$ (EQUAL) v_1 –

((PLANE) TOLERANCE v_2) ((PLANE) ANGLE (TOLERANCE) v_3) –

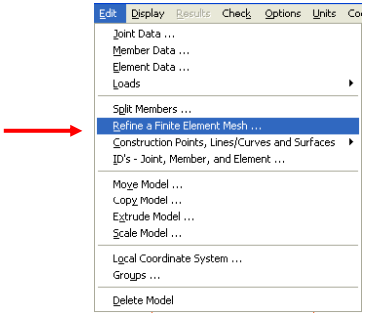
(ELEMENTS *list*) ((SHOW) CALCS) (SUMMARY (ONLY))

Version 30
New Features - GTMenu

- A new option to the Edit pull-down has been added which will allow you to Refine a Finite Element mesh by
  - Subdividing the elements uniformly or non-uniformly
  - Changing the elements to a higher order element (i.e. a 4 node element automatically changed to an 8 node element)

Version 30
New Features - GTMenu

- A new Refine Finite Element Mesh feature has been added to the Edit pulldown



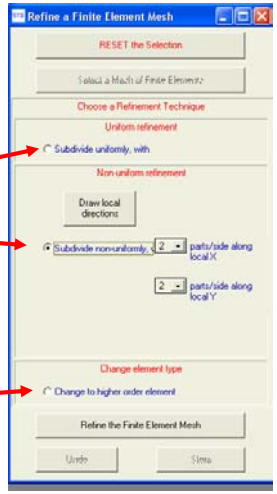
New Refine FE Mesh dialog

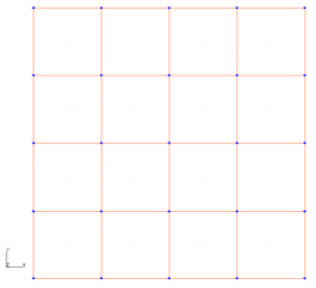
Uniform Subdivision (up to breaking each element into a 10x10 mesh)

Non-uniform subdivision

OR

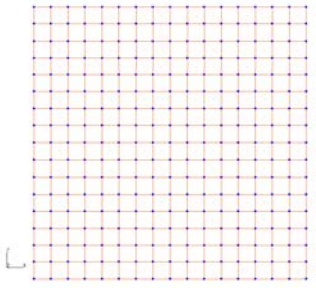
Change to a higher order element





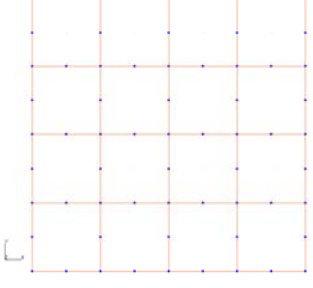
Mesh before refinement

(Original Mesh)

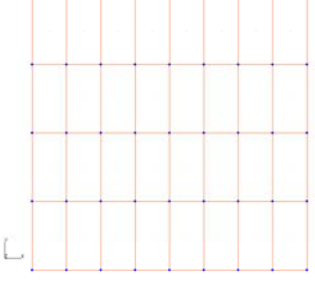


Mesh after refinement

(each element subdivided into a 4x4 grid)

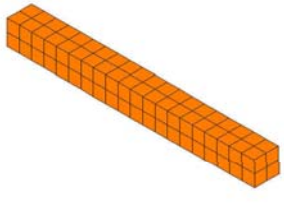


Refinement by changing to a Higher Order (8 node) element



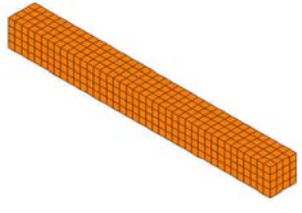
Refinement by subdividing non-uniformly

(2 divisions in local x and 1 division in local y)



Mesh before refinement

(Original Mesh)

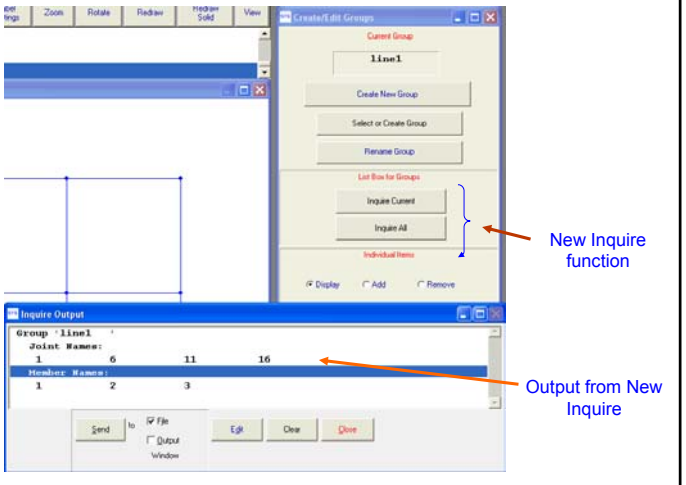


Mesh after refinement

(each element divided into 2x2x2)

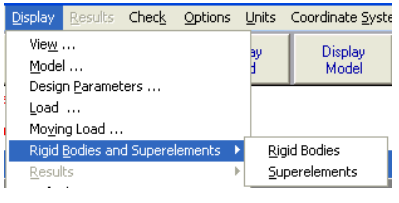
Version 30
New Features - GTMenu

- The contents of a selected Group or All Groups may now be output to the Inquire Output Window as shown on the next slide. This feature was requested by the Users Group.



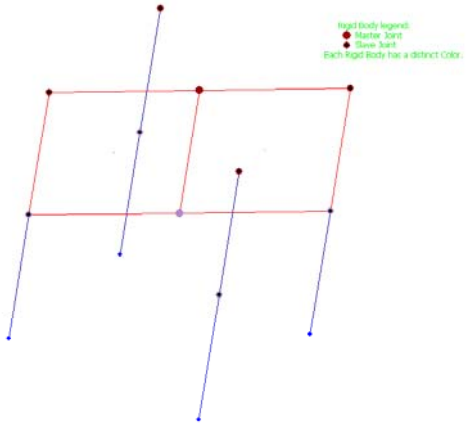
Version 30
New Features - GTMenu

- You may now output and display information regarding Rigid Bodies and Superelements



Version 30
New Features - GTMenu

- New Rigid Body Display dialog options:
  - Inquire function to output master and slave joint information
  - Display function to display rigid body information

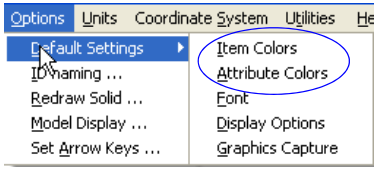


Rigid Body Display with Legend on Screen

Indicating Master and Slave Joints

Version 30
New Features - GTMenu

- A separate dialog has been created for setting Attribute colors. The old Colors dialog has been renamed as setting Item Colors as shown below.

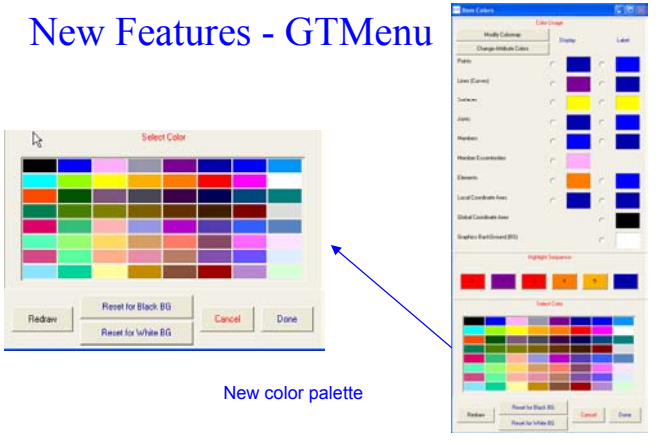


Version 30
New Features - GTMenu

- The Item Colors dialog is shown on the next slide. Note that the Color Palette has been expanded to now contain 64 colors.

  Also note that buttons now exist which will reset the colors for a black or white background in the Graphics Window.

Version 30
New Features - GTMenu



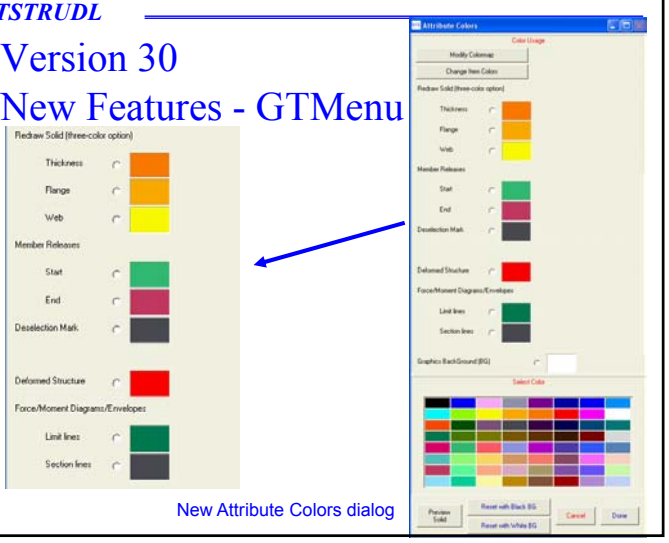
New color palette

Version 30
New Features - GTMenu

- The new Attributes Colors dialog has options which now allow you to set the color for the following functions:
  - Three color option of Redraw Solid
  - Member Releases annotation
  - Deselection mark
  - Deformed Structure
  - Force/Moment Diagram and Envelopes

  The new dialog is shown on the next slide.

Version 30
New Features - GTMenu



New Attribute Colors dialog

Version 30
New Features - GTMenu

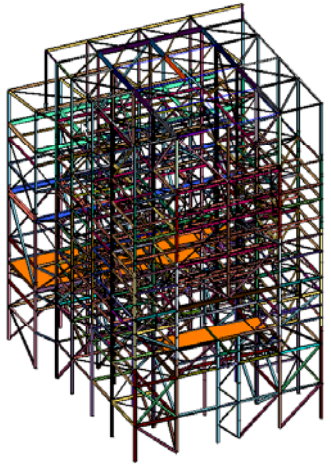
- A new option has been added to the Display Model dialog which will draw each Table section using a different color (up to 64 different colors). These colors are assigned arbitrarily by default. The revised Display Model dialog is shown on the next slide.

Version 30
New Features - GTMenu

- New Display Model dialog

New Color by Section option

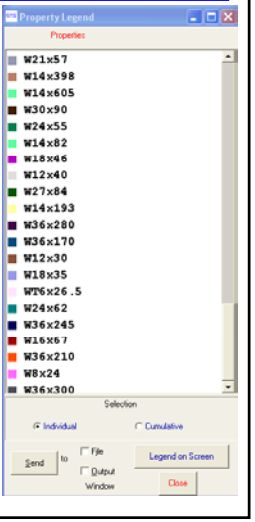
An example is shown on the next slide

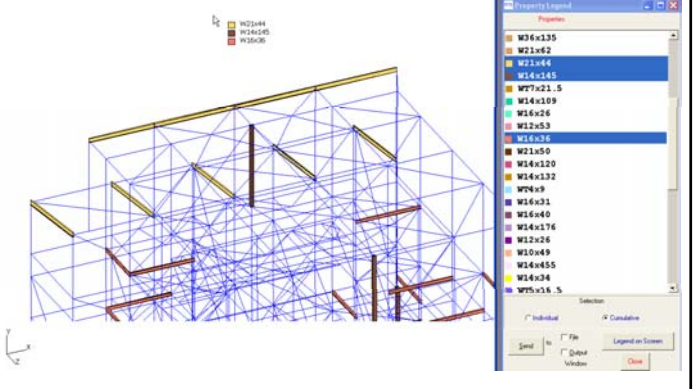


Color by Section Example with each section assigned a color.

Version 30
New Features - GTMenu

- A Property Legend appears in a separate dialog box. You may place the legend on the screen. You may also select individual sections and they will be displayed in color with an Individual or Cumulative legend on the screen as shown on the next slide.



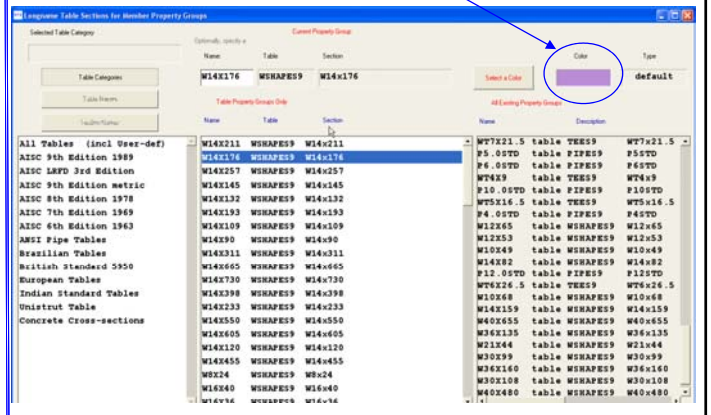


Cumulative Selection of Profiles to be displayed with legend on screen

Version 30
New Features - GTMenu

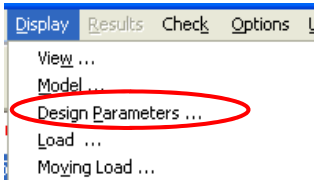
- You may also assign a specific color to a profile when you Create the Table Member Property group. An example of the new Table Member Property dialog is shown on the next slide.

New Color option



Version 30
New Features - GTMenu

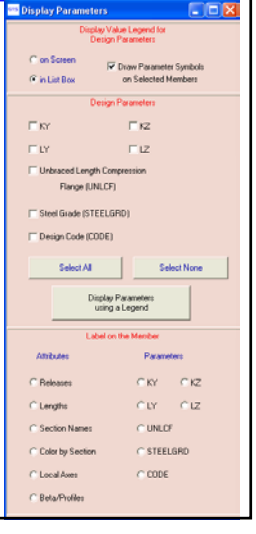
- A new option has been added to display steel design parameters. This new option is available under the Display pulldown as shown below:

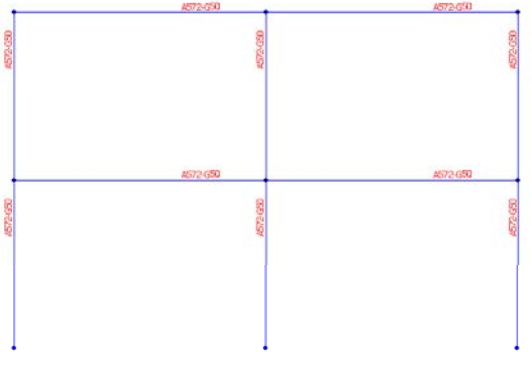


The new Display Parameters dialog is shown on the next slide and an example is then shown on another slide.

Display Parameters using Symbols on Screen with Legend either on Screen or in List Box

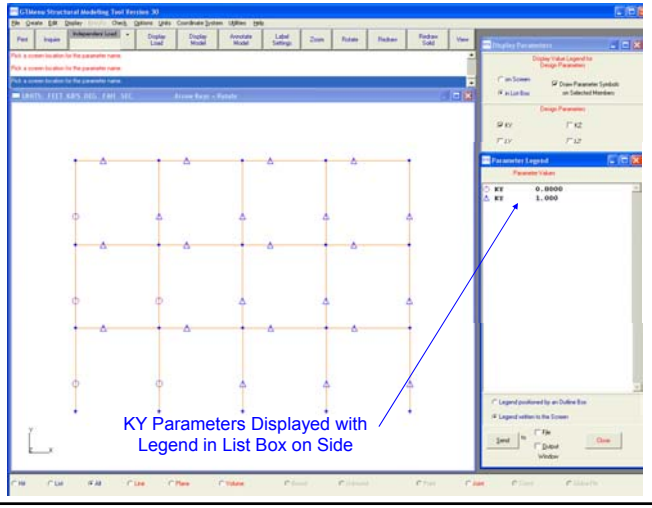
Label Attributes or Parameter Values on Screen





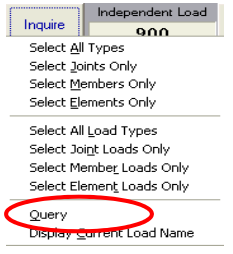
Steel Grade Design Parameters

Label on the Members Option



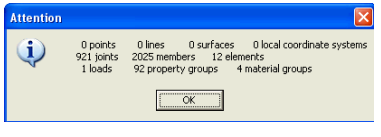
Version 30
New Features - GTMenu

- A new Inquire Query function has been added to Display Model information such as the number of joints, members, elements, etc:



Version 30
New Features - GTMenu

- Pop-up dialog output from Inquire - Query



- You can also output this information in message area using Hotkey ctrl+q

Version 30
New Features - GTMenu

- Member releases are now positioned away from the ends of the member.
- The legend for support springs now identifies the spring directions.
- When an item is deleted, its name is now removed from any group lists.
- Redraw Solid now colors the solid shapes using colors that have been assigned by the user in the Create Property Groups dialog.

Version 30
New Features - GTMenu

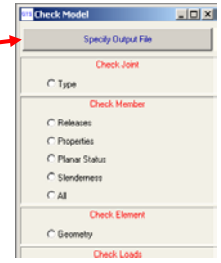
- When Ending GTMenu without changes and then reentering it, the view rotation is now retained.
- Fewer redraws occur which improves the performance for large structures.
- The cursor is now changed to an Hourglass while generating an input file.
- When displaying results, the load listing has been changed to indicate whether or not loads have results.

Version 30
New Features - GTMenu

- A filename can now be specified for Check Model output for large models.

  The default name is "GTMenu_Check.txt".

  This file is then used when the output is greater than 1000 lines.



Version 30
New Features - GTMenu

- The spacing of text in the Message Area, the Inquire Output pop-up, Create Table Property Groups dialog, and other areas has been improved.
- A CABLE element's midside node is not recalculated even if it has a value of 0.0.
- GTMenu now ignores inactive items.

Version 30
New Features - GTMenu

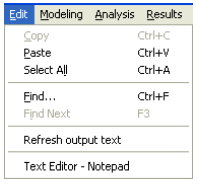
- Steel results are now available for display in GTMenu when parameter TRACE values of 1,2, or 3 have been specified for a steel design Code Check or Select command.

Version 30
New Features – GTSTRUDL Output Window

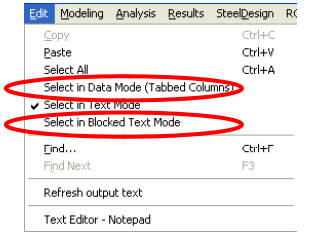
- Two additional selection modes have been added to the Text Output window. The Edit menu has been changed to reflect these new modes. The selection modes are further described on the next slides. Once a selection has been made, the selection mode can be changed with the Edit menu without losing the selection.

Version 30
New Features – GTSTRUDL Output Window

Version 29.1



Version 30



Version 30
New Features – GTSTRUDL Output Window

- The Data Mode with Tabbed Columns makes it much easier for copying selected output columns and insertion in spreadsheets. The Blocked Text Mode allows you to select a block of text for insertion in other documents respecting the spaces in the output.

Version 30
New Features – GTSTRUDL Output Window

- An example of selecting in Data Mode with tabbed columns is shown below:

RESULTANT JOINT DISPLACEMENTS FREE JOINTS

JOINT		DISPLACEMENT			
		X DISP.	Y DISP.	Z DISP.	X ROT.
6	GLOBAL	0.0318012	0.0008872	0.0000000	0.0000000
7	GLOBAL	0.0309290	-0.0000969	0.0000000	0.0000000
8	GLOBAL	0.0302791	-0.0000040	0.0000000	0.0000000
9	GLOBAL	0.0298399	0.0000063	0.0000000	0.0000000
10	GLOBAL	0.0295679	-0.0000727	0.0000000	0.0000000
11	GLOBAL	0.0628776	0.0013150	0.0000000	0.0000000
12	GLOBAL	0.0618584	-0.0001218	0.0000000	0.0000000
13	GLOBAL	0.0611176	-0.0000042	0.0000000	0.0000000
14	GLOBAL	0.0606400	0.0001129	0.0000000	0.0000000
15	GLOBAL	0.0604421	-0.0013018	0.0000000	0.0000000
16	GLOBAL	0.0901274	0.0014401	0.0000000	0.0000000
17	GLOBAL	0.0791742	-0.0001269	0.0000000	0.0000000
18	GLOBAL	0.0784186	-0.0000042	0.0000000	0.0000000
19	GLOBAL	0.0779684	0.0001183	0.0000000	0.0000000
20	GLOBAL	0.0778165	-0.0014273	0.0000000	0.0000000

Version 30
New Features – Offshore

- The fatigue analysis type options of the Perform Fatigue Analysis command have been expanded to include a Deterministic method in addition to the Discrete and PSD wave spectrum methods.
- The DNV Local Joint Flexibility brace member stiffness modifications have been added for fatigue analysis.

Version 30
New Features - Reinforced Concrete Design

- Add ACI 318-05 design for beams and columns with specified dimensions.
  (GIVEN B and H).
- The DESIGN SLAB command will be brought to release status.

Version 30
New Features – Static Analysis

- The GTSES external file solver has been rewritten and implemented on 64 bit XP and Vista operating systems. This solver takes advantage of multiple cores as well as the increased virtual address space.

Version 30
New Features – Static Analysis

- An example of this solver executing on a quad core computer running Vista 64 will now be shown as a video clip.

Version 30
New solver statistics

- How long do you think that it will take to solve the equations for the following problem?

  55,003 joints (all with 6 dof) – over 330,000 dof

  56,020 elements

  1,785 half-band

  19 independent loadings

Solution is being conducted on an HP quad core computer with 8 GB memory.

Version 30
New Features – Steel Design

- Torsion and Warping design have been added to the ASD9 code. Applicable cross sections are shown below:
  - I shapes
  - Channels
  - Single Angles
  - Structural Tubes

  New parameters have also been added.

Version 30
New Features – Steel Design

- Efficiency improvements have been implemented into the steel design SELECT and CHECK commands. The time to perform code checks and design has been reduced dramatically as shown on the next slide.

Version 30
New Features – Steel Design

- Example
  11,181 joints
  20,982 members
  31 analysis and design loading conditions

	Version 29	Version 30
Steel Design (19,086 members, 3 sections per member)	4 hrs 7.4 min	30 min

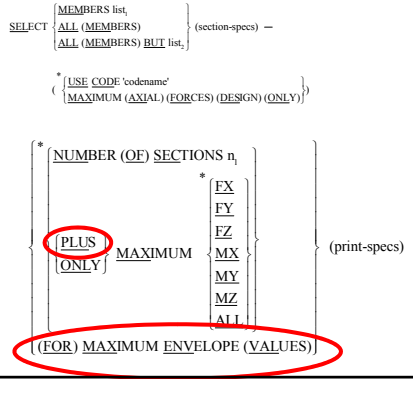
Version 30
New Features – Steel Design

- The Code Check Datasheet is now available when parameter TRACE values of 1, 2, or 3 have been specified.
- New error checking has been added to the steel design Parameter command. User specified values are checked for correctness. Alphanumeric values are check against accepted values.

Version 30
New Features – Steel Design

- A new section specification has been added to the CHECK and SELECT commands. This option allows the user to specify the maximum forces or moments section locations to be added to the user specified section locations. This new section option also allows the user to create a single section point which contains the maximum axial, shears, torsion, and bending moments (envelope option).

Version 30
New Features – Steel Design



Version 30
New Features – Steel Design

- The AISC 13th Edition code, AISC13, has been implemented. Supported cross sections are
  - I shapes
  - Channels
  - Single Angles
  - Tees
  - Double Angles
  - Square and Rectangular HSS (Structural Tubes)
  - Round HSS (Pipes)
  - Solid Round Bars
  - Solid Rectangular and Square Bars

Version 30
New Features – Steel Design

- **Steel Design Dialogs**
- *Parameter Dialog* – Add structural tubes and pipes steel grades into the parameter dialog for ASD9 code.
- *Parameter Dialog* – Add new Torsion and Warping parameters to parameter dialog.
- *Parameter Dialog* – Add parameters for AISC13 code.

Interfaces to Other Programs

- AutoCAD – interfaces via DXF converter
- Intergraph's Frameworks
- Intergraph's SmartPlant 3D – CIS/2
- Structural Desktop by Structural Desktop, Inc
- Tekla Structures
- Zentech's PADDS (offshore platforms)
- Other CAD programs which support D XF or CIS/2

Future Enhancements
(GTSTRUDL Version 31+)

Version 31
New Features – Dynamic Analysis

- The GTSELANCZOS sparse equation Eigenvalue analysis procedure will be enhanced as a stand-alone program similar to the stand-alone GTSES static analysis equation solver in Version 29.1. This will enable considerably larger Eigenvalue analysis jobs to be solved even more efficiently than the present implementation of the GTSELanczos Eigenvalue analysis solver.
  In conjunction with this enhancement, the entire mode superposition method of analysis will be extended to support the external file strategy for storing and managing dynamic analysis results.
- Computation of minimum seismic load according to latest IBC provisions.

Version 31
New Features – Finite Elements

- New eight node flat six degree of freedom stretching and bending element for thin and moderately thick plates
  - Surface, temperature, and body forces
  - Lumped and Consistent Mass matrix for dynamic analysis
  - Standard finite element output – LIST, CALCULATE AVERAGE
  - Integration into GTMenu (geometry, loads, input file, contour results)

Version 31
New Features - General

- LOCATE DUPLICATE MEMBERS (*order*) –
  ((AND) REMOVE ({ ADD LOADS }))
  { FORCE (REMOVAL) }

  FORCE

  Remove duplicates (honoring RESPECT / DISREGARD) despite non-similarity of releases, etc.

  Cannot use with ADD LOADS.

  ADD LOADS

  Add loadings from duplicates to master. Member added inertia and dynamic loads are lost.

  Cannot use with FORCE.

Version 31
New Features - General

- LOCATE INTERFERENCE JOINTS *list*

  Add finite elements to the search for interference joints. Joints that lie within a tolerance of a finite element boundary edge or are interior to a finite element (2D or 3D) will be reported.

  Allow a *list* of members and/or elements. This allows you check limited areas for interference joints, for example, floor by floor.

Version 31
New Features - General

- LOCATE FLOATING JOINTS ((AND) - REMOVE)

  Only joints that are not attached to any member, finite element, nonlinear spring or rigid body will be considered as possible "floating" joints. If a joint is used as a BETA REFERENCE JOINT it will not be considered "floating".

Version 31
New Features - General

- LOCATE DUPLICATE ELEMENTS ((AND) REMOVE (ADD LOADS))
  Similar to LOCATE DUPLICATE MEMBERS.

- LOCATE INTERSECTING MEMBERS *list* TOL *v*
  Detect members within a tolerance (TOL) of each other to locate "crossing" members that may look correct graphically but should actually have a common joint at their intersection.

Version 31
New Features - General

- AREA LOAD enhancements

  Check for interference joints and intersecting members in plane before attempting to locate bounded areas.
  - Improve error reporting:
  - List of 0.0 length members
  - Start joint for "illegal configuration" to help with debugging.

  Add IGNORE NONORTHOGONAL MEMBERS option.

  Ignore members not within specified angle of global axis so bracing doesn't need to be inactivated.

  PLOT option - create a Scope Editor file like the dialog display with shaded bounded areas.

Version 31
New Features – Nonlinear Analysis

- The nonlinear member end connection and plastic hinge models will be supported by nonlinear dynamic analysis.

- Extend the GTSES external file solver in Version 29.1 to nonlinear static analysis.

Version 31
New Features – Steel Design

- Displacement Constraint Design Procedure will be available to select members to satisfy joint displacement constraints. The new GTSES external file solver will be added as an option.
- Torsion and Warping design will be implemented for the AISC13 code. Implementation would be for I shapes, Channels, Single Angles, Round HSS, Rectangular and Square HSS.

Version 31
New Features – Steel Design

- Add new EC3-2005 code

Future Enhancements

- Multiple windows to allow for result display in a separate window than the model display.
- Graphically specify joints and elements to be used with LIST SUM FORCES command.
- List the contents of a Group in an Inquire box. Currently the contents can only be displayed graphically.
- Graphically specify Area Loads in GTMenu.

Future Enhancements (cont)

- Add a Display Options capability to the Principal Vector display for principal stress vectors similar to the one shown here for contouring.

Future Enhancements (cont)

- Add the option to Split Members to interpolate eccentricities between the start and end of the split members.
- Ability to click on a member and have a graphical summary of the results on just that member in a separate Window that can be printed.

Future Enhancements (cont)

- Addition of the following items to the input file created by GTMenu:
  - Dynamic Loadings
  - Eigen Parameters
  - Dynamic Modal Damping Data
  - Nonlinear Effects
  - Nonlinear Spring Properties
  - Nonlinear Solution Parameters
  - Cable Network Data

Future Enhancements (cont)

- Continue expanding the model data and results which are available in Datasheets. Use the new datasheets which allow printing and facilitate exporting to Excel.

Future Enhancements (cont)

- ASCE 7 equivalent static earthquake load.
- Hysteretic behavior for plastic hinges and nonlinear member end connections.
- Add elasto-plastic behavior to a limited group of elements.
- Graphically display other pushover information.

Future Enhancements (cont)

- Extend standalone solvers for dynamic and nonlinear analysis to take advantage of 64 bit operating systems.

Future Enhancements (cont)

- Incremental nonlinear analysis to facilitate the use of nonlinear analysis to solve construction sequence problems.
- Add models for multi circular spirals in rectangular columns for pushover analysis.
- Add options for users to type in rebar coordinates or automatic equal spacing for pushover analysis.

Future Enhancements (cont)

- Generalizing nonlinear frame member model to support large finite rotation behavior.
- Center of stiffness computation in dynamic analysis.
- New eight node solid element with incompatible modes which will model bending behavior more accurately than existing eight node isoparametric elements.

Future Enhancements (cont) Scope Editor

- Automatic insertion of items from a template
  - File name
- Library of symbols to use for copy and paste
- Multi-page document
- Implement “Undo” function

As always, your assistance is needed to help us improve GTSTRUDL in your Committee meetings.