GT STRUDL ®

Version 31

Release Guide

Volume 1 of 2

August 2010

Computer-Aided Structural Engineering Center
School of Civil & Environmental Engineering
Georgia Institute of Technology
Atlanta, Georgia 30332-0355
U.S.A.

Telephone: (404) 894-2260
Fax: (404) 894-8014
e-mail: casec@ce.gatech.edu

NOTICES

This GTSTRUDL® Release Guide is applicable to Version 31, with a release date in the GTSTRUDL title block of August 2010.

The GTSTRUDL® computer program is proprietary to, and a trade secret of the Georgia Tech Research Corporation, Atlanta, Georgia, U.S.A.

DISCLAIMER

NEITHER GEORGIA TECH RESEARCH CORPORATION NOR GEORGIA INSTITUTE OF TECHNOLOGY MAKE ANY WARRANTY EXPRESSED OR IMPLIED AS TO THE DOCUMENTATION, FUNCTION, OR PERFORMANCE OF THE PROGRAM DESCRIBED HEREIN, AND THE USER OF THE PROGRAM IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE PROGRAM IN THEIR OWN ENVIRONMENT.

Commercial Software Rights Legend

Any use, duplication, or disclosure of this software by or for the U.S. Government shall be restricted to the terms of a license agreement in accordance with the clause at DFARS 227.7202-3 (June 2005).

This material may be reproduced by or for the U.S. Government pursuant to the copyright license under the clause at DFARS 252.227-7013, September 1989.

GTSTRUDL® is a registered service mark of the Georgia Tech Research Corporation, Atlanta, Georgia USA.

Windows 7®, Windows Vista®, Windows XP® and Windows 2000® are registered trademarks of Microsoft Corporation in the United States and/or other countries.

Intel® Core™2 Duo and Intel® Core™2 Quad are registered trademarks of Intel Corporation in the United States and other countries.

Table of Contents

Chapter **Page**

Chapter 1

Introduction

Version 31 covers GTSTRUDL operating on PC's under the Windows 7, Windows Vista, Windows XP and Windows 2000 operating systems. Chapter 2 presents the new features and enhancements which have been added since the release of Version 30. Chapter 3 provides you with details regarding error corrections that have been made since the Version 30 release. Chapter 4 describes known problems with Version 31. Chapter 5 describes prerelease features -- new features which have been developed and subjected to limited testing, or features for which the user documentation has not been added to the GTSTRUDL User Reference Manual. The command formats and functionality of the prerelease features may change before they become supported features based on additional testing and feedback from users.

The Prerelease features are subdivided into Design, Analysis, and General categories. The features in these categories and their sections numbers in Chapter 5 are shown below:

5.2 Design Prerelease Features

- 5.2.1 EC3-2005 Steel Design Code
- 5.2.2 API Twenty-First Edition steel design code for Basic Hydrostatic Pressure and Punching Shear stress checks
- 5.2.3 LRFD3 Steel Design Code
- 5.2.4 ACI Concrete Code 318-99
- 5.2.5 Rectangular and Circular Concrete Cross Section Tables
- 5.2.6 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

5.3 Analysis Prerelease Features

- 5.3.1 Calculate Error Estimate Command

We encourage you to experiment with these prerelease features and provide us with suggestions to improve these features as well as other GTSTRUDL capabilities.

Chapter 2

New Features in Version 31

This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GTSTRUDL in Version 31. This release guide is also available online upon execution of GTSTRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 DBX

1. The WRITE ELEMENTS ATTRIBUTES command now reports the number of elements not included in the generated DBX file due to missing element properties. Previously, only the total number of complete element attributes written was reported.

2.2 Dynamics

1. The LIST DYNAMIC PARTICIPATION FACTORS Command now has the ability to output the mass participation factors in a tabulated form sorted by mode, by one of the global directions, or by each one of the global directions. By default, the output will only show active modes. However, the user can specify if the output should include all modes. The command can also output only totals in each global direction. The revised command is described in Section 2.4.6.4 of Volume 3 of the Reference Manuals.

2. The ACTIVE/INACTIVE MODES Command has been enhanced to automatically Activate/ Inactivate modes according to minimum target of mass participation factors given for each global direction. The revised command is described in Section 2.4.5.6 of Volume 3 of the Reference Manuals.

   An example of output using the revised LIST DYNAMIC PARTICIPATION FACTORS and ACTIVE/INACTIVE MODES commands is shown below:

```
{  535} > LIST DYNAMIC PARTICIPATION FACTORS ORDER BY EACH ALL

    ****************************
    *RESULTS OF LATEST ANALYSES*
    ****************************

     PROBLEM -           TITLE - NONE GIVEN

     ACTIVE UNITS  FEET KIP  DEG  DEGF SEC

    **********************************
    *   MASS PARTICIPATION FACTORS   *
    *        ALL MODES SHOWN         *
    **********************************

    * ORDER IN EACH GLOBAL DIRECTION *

     Order By X mass %          Order By Y mass %          Order By Z mass %
Mode# X mass %  Cum. Sum.  Mode# Y mass %  Cum. Sum.  Mode# Z mass %  Cum. Sum.
---  ---------  ---------  ---  ---------  ---------  ---  ---------  ---------
  3  83.408910  83.408910    9   0.000000   0.000000    1  87.491623  87.491623
  8   0.247280  83.656191   10   0.000000   0.000000    5   8.995125  96.486749
  9   0.000000  83.656191    8   0.000000   0.000000    9   0.036113  96.522862
 10   0.000000  83.656191    7   0.000000   0.000000    6   0.007008  96.529870
  7   0.000000  83.656191    6   0.000000   0.000000   10   0.000000  96.529870
  6   0.000000  83.656191    5   0.000000   0.000000    8   0.000000  96.529870
  2   0.000000  83.656191    4   0.000000   0.000000    7   0.000000  96.529870
  4   0.000000  83.656191    3   0.000000   0.000000    2   0.000000  96.529870
  5   0.000000  83.656191    1   0.000000   0.000000    4   0.000000  96.529870
  1   0.000000  83.656191    2   0.000000   0.000000    3   0.000000  96.529870
---  ---------  ---------  ---  ---------  ---------  ---  ---------  ---------
Total %         83.656191              0.000000                        96.529870
Active%         83.656191              0.000000                        96.529870

{  536} >
{  537} > ACTIVE MODES MINIMUM TARGET MASS PARTICIPATION FACTORS XMIN 83.5 XGE 5. -
{  538} >_YMIN 0. YGE 1. ZMIN 80. ZGE 8.

*** ACTIVE MODE STATUS ***

    Active modes  :           4
    Inactive modes:           6
    Total         :          10

  ACTIVE MODES ALL BUT  2 4 6 7 9 10
```

3. The viscous damper element (dash pot) has been moved to release status. The viscous damper element may be used for linear and nonlinear dynamic analysis by the direct integration procedure. The commands related to the viscous damper element are shown below and described in Section 2.4.3.7 of Volume 3 of the Reference Manuals:

   DAMPER ELEMENT DATA, described in Section 2.4.3.7.1.
   PRINT DAMPER ELEMENT DATA, described in Section 2.4.3.7.2.
   DELETE DAMPER ELEMENT DATA, described in Section 2.4.3.7.3.

2.3 General

1. The efficiency of the LIST MAXIMUM JOINT DISPLACEMENTS and LIST MAXIMUM REACTIONS commands has been dramatically improved when the analysis results are stored on external save files, which occurs when the GTSES/GT64M sparse equation solvers are used for the linear and nonlinear static and dynamic analyses. For jobs consisting of large models with tens and even hundreds of loading conditions, the time to execute these commands has decreased from an hour or more for the largest jobs to a few minutes at most.

2. The MATERIAL command now respects the EXISTING MEMBERS ONLY and the EXISTING ELEMENTS ONLY options for a member list. Previously, the ONLY specification was ignored and the material properties (E, DENSITY, etc.) were assigned to <u>all</u> members and elements in the EXISTING list.

   <u>Examples</u>:

   MATERIAL CONCRETE ELEMENTS EXISTING ELEMENTS ONLY

   Only finite elements are assigned concrete material properties.

   MATERIAL STEEL MEMBERS EXISTING MEMBERS ONLY

   Only members are assigned steel material properties

3. The EXISTING option for lists now works with load names. Previously, only LOAD LIST would recognize EXISTING, although the options of BUT and PLUS were not recognized, but no other load name list recognized the EXISTING option.
   Example:
   PRINT LOAD DATA LOADS EXISTING BUT 100

4. The CALCULATE PRESSURE command has been improved by the addition of support for analysis results stored in files, created by the execution of linear and nonlinear static analyses using the GTSES/GT64M sparse equation solvers.

5. The COUTPUT command will now assume COUTPUT standard if an invalid file name is specified. An invalid file name can be due to illegal characters in the name, or restrictive permissions on the folder where the file is attempting to open.

2.4 GTMenu

1. The Split Member option now allows a member list to be built using all settings in the Mode Bar. Previously, only the Hit and List modes were available.

2. When entering GTMenu, the structure is now drawn only once. For large models, multiple drawings of the structure slowed down the entry into GTMenu.

3. A Current Group must now be selected before Current Group options are made active.

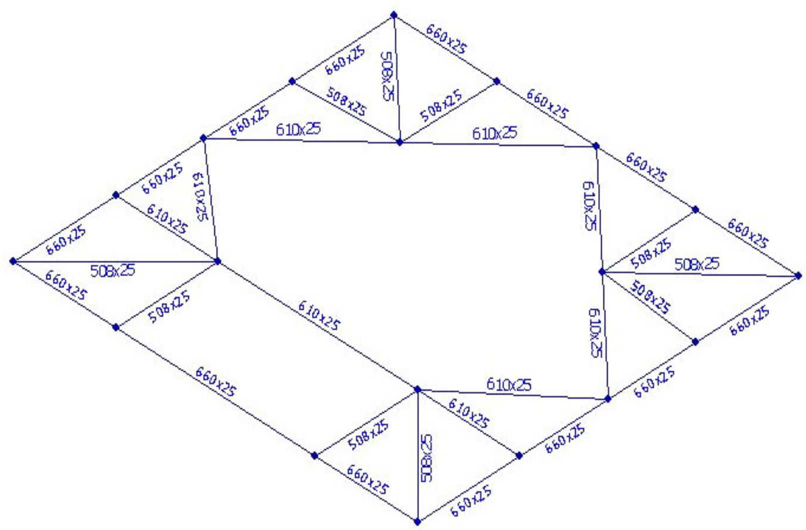
4. A message is now given when Check Model results are written to a file.

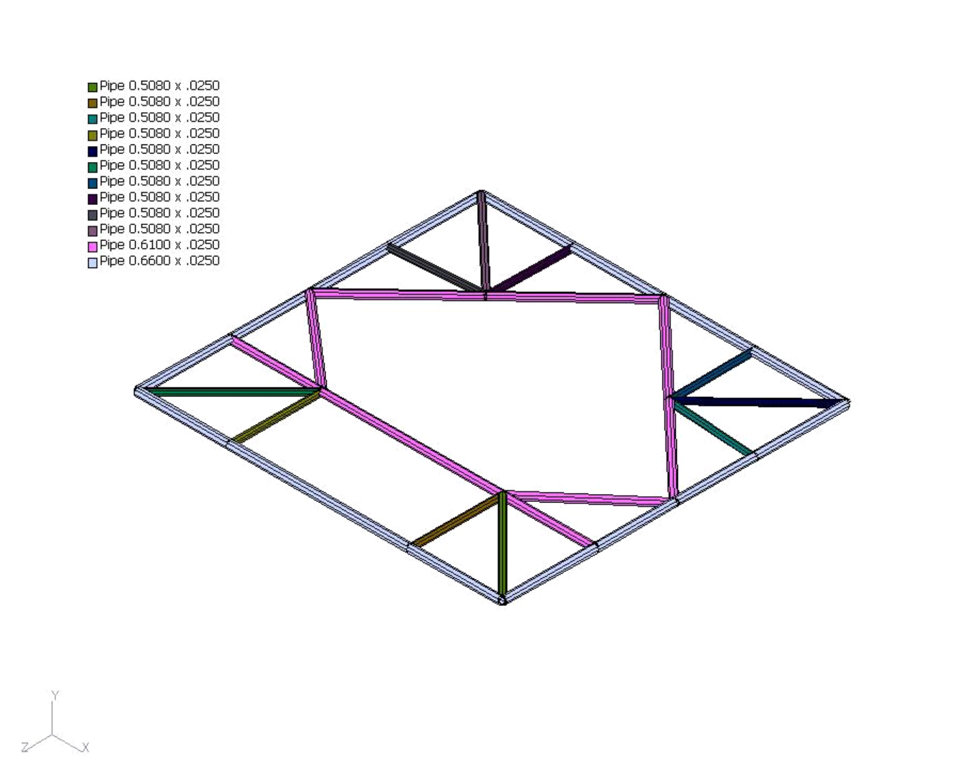
5. The efficiency of Color by Section has been improved for large models.

6. The cursor now displays as an hourglass when the structure is being redrawn.

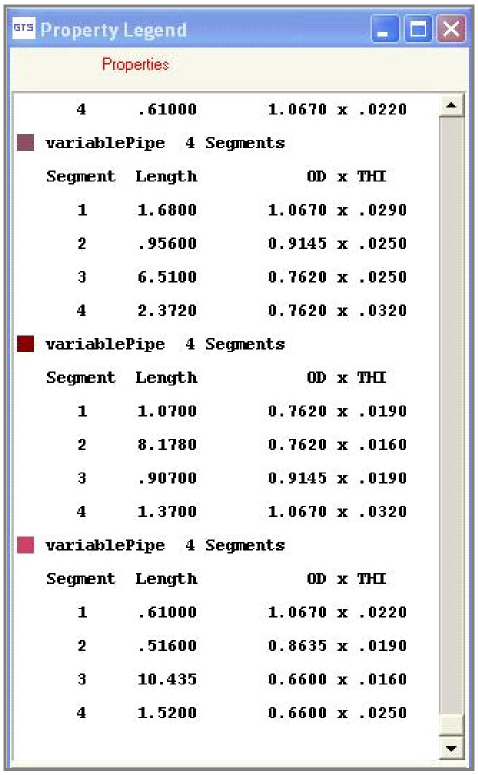
7. When entering GTMenu from command mode with members which have pipe properties, the property group for a pipe section is now named according to the PIPE dimensions: Outer Diameter x THIckness. The name is generated in inches if the units are English, and the name is generated in millimeters if the units are Metric.

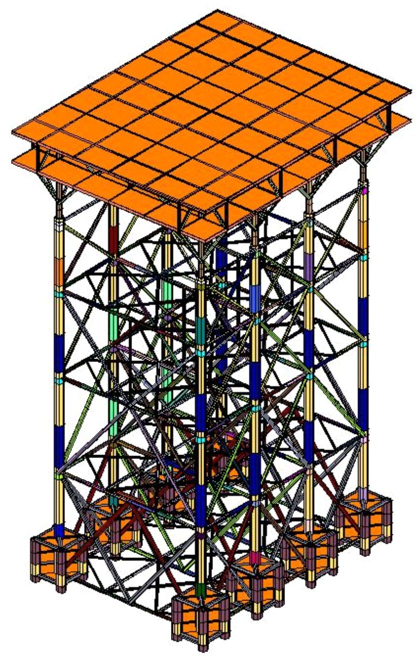
8. Also for the Display Model Section Names, a pipe descriptor will be used such as 10x1.0 for a pipe with a 10 inch OD and a 1.0 inch wall thickness. The Display Model Color by Section will use the same descriptor. Examples of the output from Display Model Section Names and Color by Section using the new pipe descriptor are shown on the next page.


660x25
508x25
610x25


Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.5080 x .0250
Pipe 0.6100 x .0250
Pipe 0.6600 x .0250
Y
Z
X

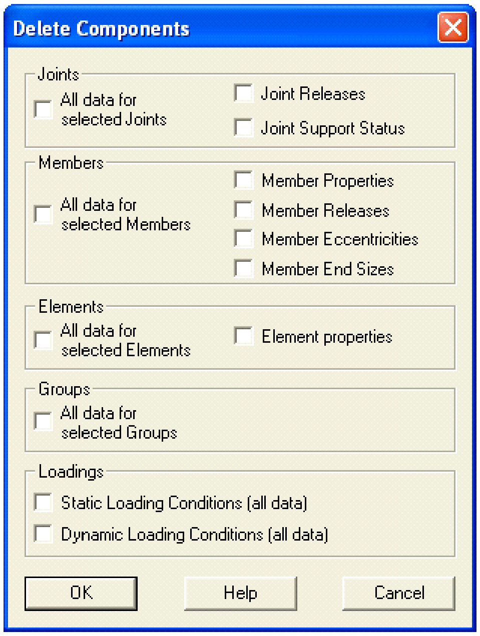
9. The Color by Section Legends (both the scrolling box and the graphics screen legend) have been enhanced to fully display the specifications of a variable section: that is, the length and type of each segment (in current units where applicable) are given. An example is shown below:



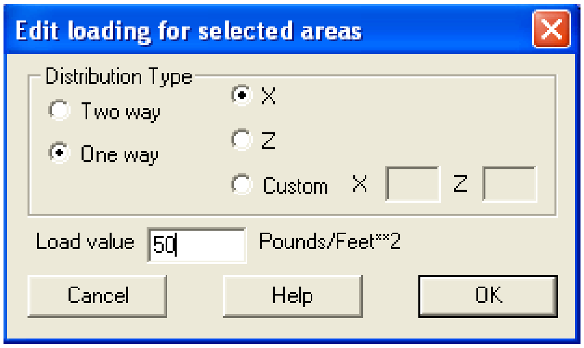


2.5 GTSTRUDL Output Window

1. The Deletions dialog (Modeling – Data Management – Deletions) has been expanded to include finite elements and groups. The new dialog is shown below.

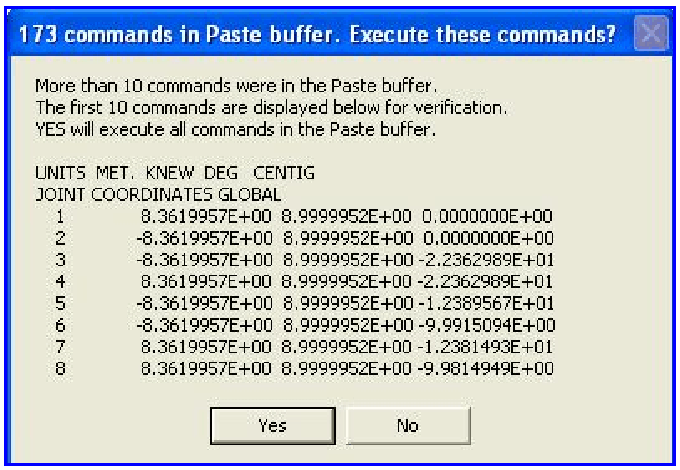


2. In the Area Load dialog, Individual buttons for 'One way X' and 'One way Z' have been added. The Custom option is the old method, where the user was required to enter at least one value in the X or Z box. Both the Standard load settings in the Elevation dialog and the Edit loading dialog (shown below) use the new method. **Note**: The direction labels X and Z are appropriate for loadings perpendicular to global Y. If a different loading axis is chosen, the direction labels will be changed appropriately.



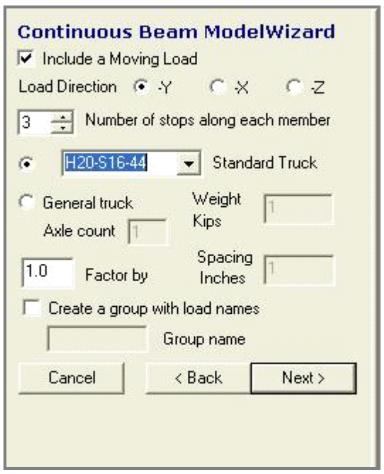
3. The 'All' option has been removed from the File -> Save menu. The 'All' option activated the first three Save options only, which was confusing to some users. Now, click on each option separately if more than one type of Save is desired.

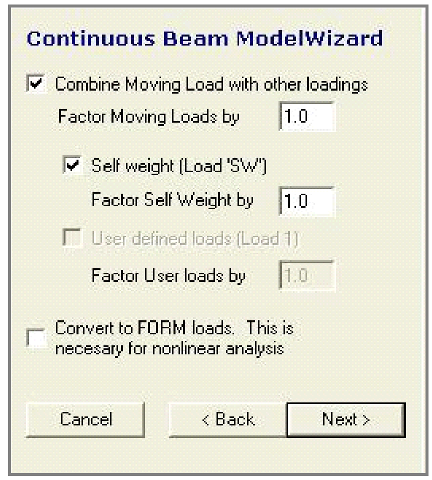
4. In the GTSTRUDL Output Window (Text Mode), when the Paste buffer is more than 10 lines, only the first 10 lines of the buffer are now displayed for verification. Previously, all lines in the Paste buffer were displayed, which could result in a dialog where the Yes and No buttons were off the screen for very large Paste buffers. An example of the new pop-up dialog that appears when pasting into the GTSTRUDL Output Window is shown below:



2.6 Model Wizard

1. The Continuous Beam Wizard has been enhanced to include a Moving Load page and an Automatic Load Combination page. In addition, the default support condition at X = 0.0 will be pinned, instead of fixed. Examples of the new dialogs are shown below:

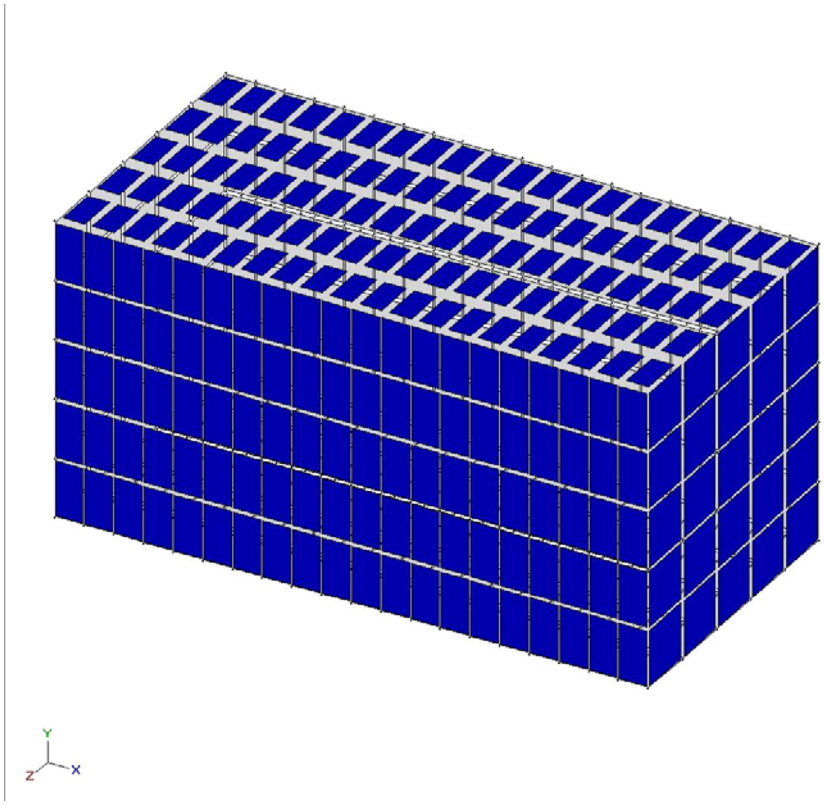




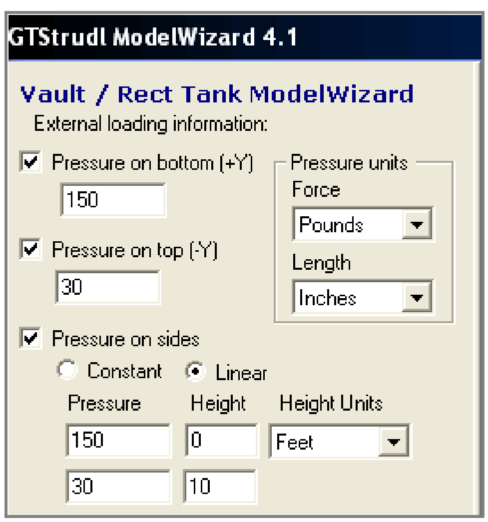
2. The Model Wizard now adds SET ELEMENTS HASHED to all models with more than 300 joints. Previously only the Vault and Space Frame Wizards added this command. SET ELEMENTS HASHED speeds up the processing of commands for large input files.

3. The Vault/Rectangular Tank Wizard now includes the option to add a top as shown in the dialog below:

An example of a rectangular tank with a top created using the Wizard is shown below:



You can now add pressure to the top and the sides of the tank using the new dialog shown below. The side pressure allows you to model a surcharge loading.



4. The Plane Frame Wizard now includes new bracing types:

Inverted Chevron Braces with offsets and eccentricities

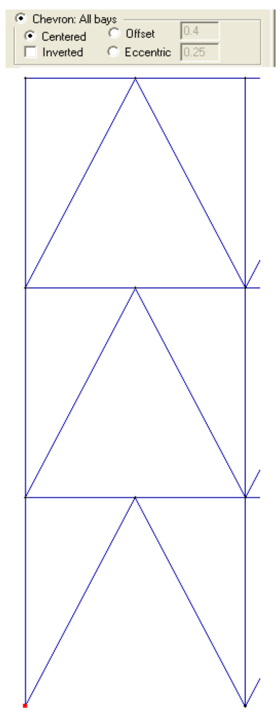
Left and right Diagonal braces with eccentricities

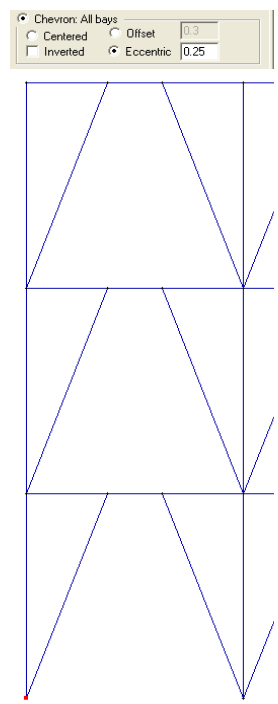
Knee braces

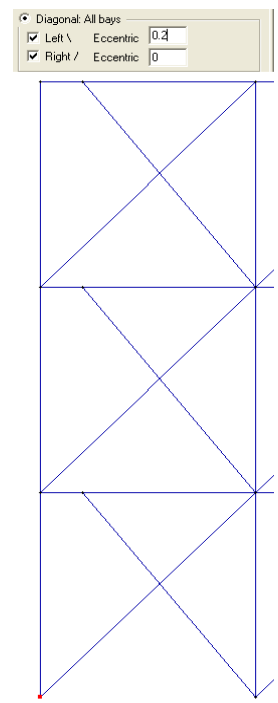
The new bracing dialog is shown below which includes the ability to specify bracing offsets and eccentricities as a fraction of the width of a bay:

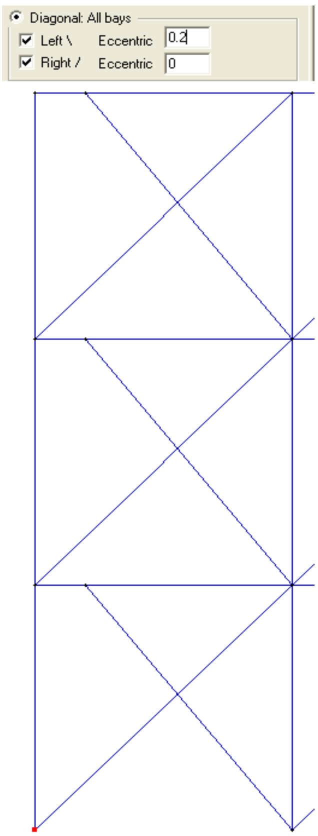
The Variable option shown in the above dialog can be used to specify bracing by bay. In addition, different properties can be specified for the link beams when eccentric braces are used.

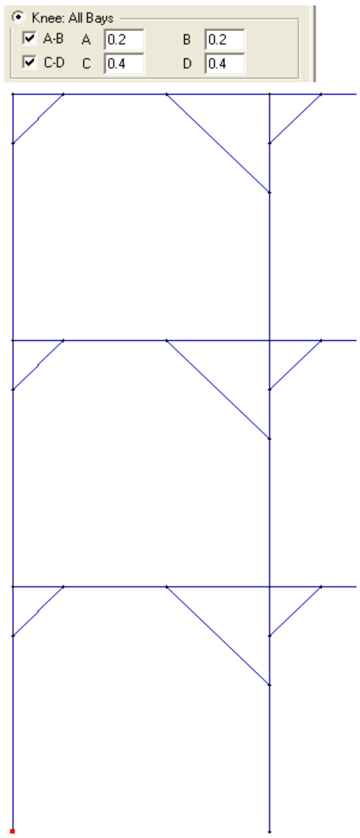
Examples of these new bracing types are shown on the next two pages:


Chevron: All bays
Centered
Offset
0.4
Inverted
Eccentric
0.25


Chevron: All bays
Centered
Offset
0.3
Inverted
Eccentric
0.25


Diagonal: All bays
Left \
Eccentric
0.2
Right /
Eccentric
0


Diagonal: All bays
Left \ Eccentric 0.2
Right / Eccentric 0


Knee: All Bays
A-B A 0.2 B 0.2
C-D C 0.4 D 0.4

2.7 Nonlinear

1. Support for nonlinear geometric behavior has been implemented for the SBHQ, SBHQ6, SBHT, and SBHT6 plate finite elements. The nonlinear geometric behavior capability is designated for these elements by the NONLINEAR EFFECTS command as it is done for frame and truss members. This capability is only available for the GTSES solver option. More details about this new feature are found in Section 2.5.2.1 of Volume 3 of the GTSTRUDL User Reference Manual.

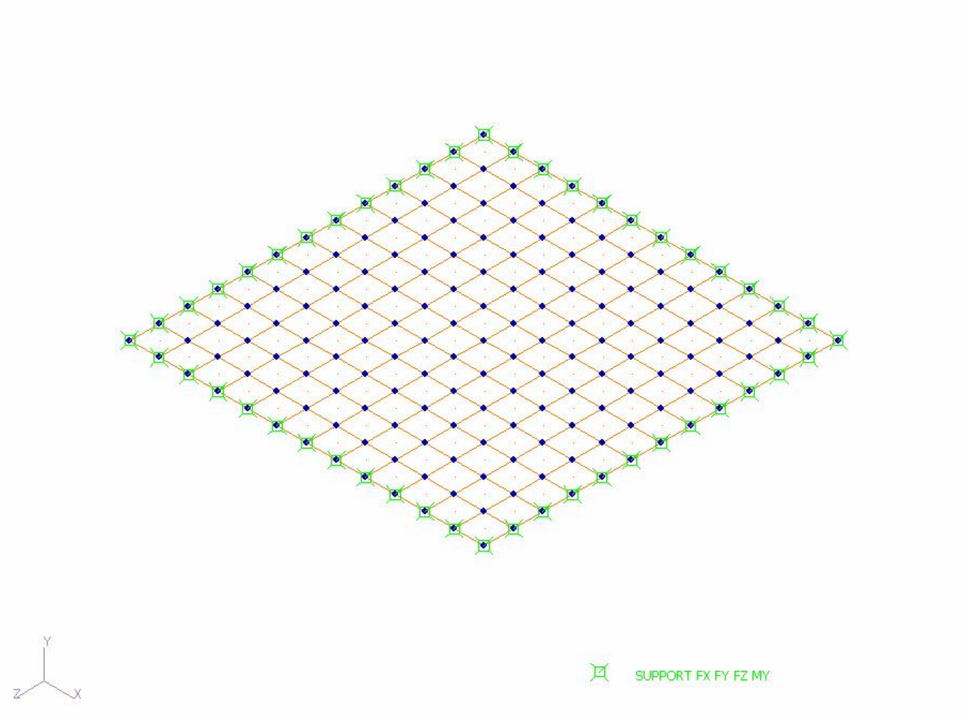
   An example of accounting for the geometric nonlinear behavior using these elements in a pushover analysis on a square simply supported plate with a uniform load is shown below:

   Nonlinear Effects

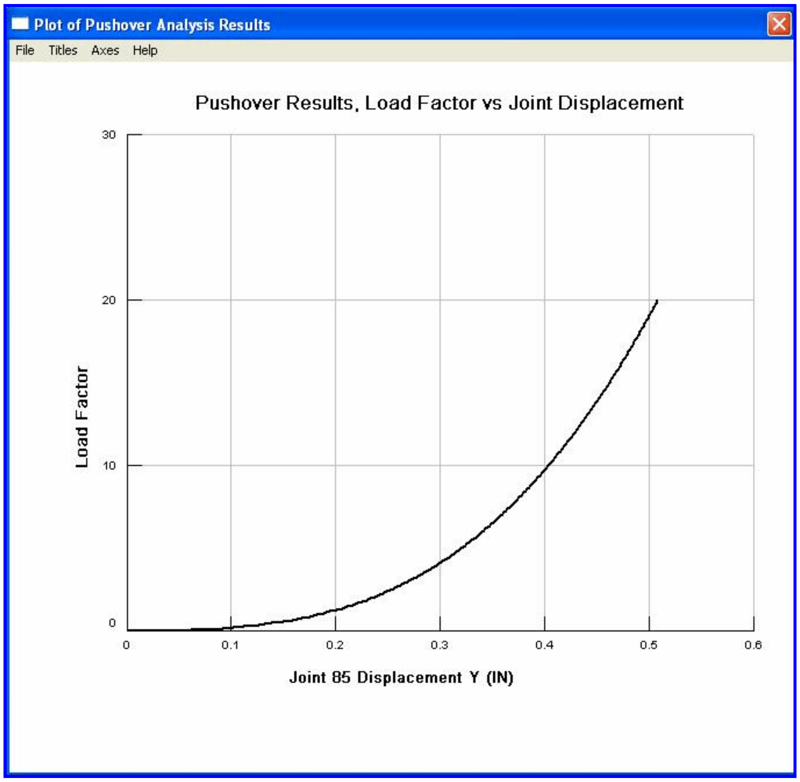
   Geometry elements 1 to 144

   End

   Active Solver GTSES



A plot of the y-displacement at the center of the plate versus the load factor is shown below:



2. The PERFORM PUSHOVER ANALYSIS command (Section 2.5.5, Volume 3) has been enhanced with the addition of the GTSES option, which indicates that the GTSES sparse equation solver shall be used for the execution of the nonlinear analysis steps of the pushover analysis procedure. The PERFORM PUSHOVER ANALYSIS command also uses the GTSES solver when the ACTIVE SOLVER GTSES command (Section 2.1.1.13.4, Volume 1) is given prior to the PERFORM PUSHOVER ANALYSIS command.

3. The PERFORM CABLE PRESTRESS ANALYSIS command (Section 2.6.3, Volume 3) has been enhanced with the addition of the GTSES option, which indicates that the GTSES sparse equation solver shall be used for the execution of the nonlinear analysis steps of the cable prestress analysis procedure. The PERFORM CABLE PRESTRESS ANALYSIS command also uses the GTSES solver when the ACTIVE SOLVER GTSES command

(Section 2.1.1.13.4, Volume 1) is given prior to the PERFORM CABLE PRESTRESS ANALYSIS command.

4. Nonlinear analysis has been extended to support the GTSES sparse equation solver. The GTSES solver is executed for nonlinear static analysis when the NONLINEAR ANALYSIS GTSES command is given or when the ACTIVE SOLVER GTSES/GT64M command is given prior to any NONLINEAR ANALYSIS commands. When nonlinear analysis uses the GTSES solver, it also stores the analysis results into files, rather than in memory. The GTSES solver in conjunction with the storage of results in files provides nonlinear static analysis with the ability to solve significantly larger problems with much greater speed and efficiency.

   An example of the performance improvement using the GTSES solver is shown below:

   Industrial Building Model

# of joints	=	2900
# of members	=	6300
# of active loads	=	73
# of DOFs	=	18000

Time for nonlinear analysis, GTSES	=	14.0 min.
Average Time/Loading	=	11.5 sec.
Time for nonlinear analysis, Standard (V30)	=	240.0 min.
Average Time/Loading	=	197.0 sec.

2.8 Offshore

1. A new *API Recommended Practice 2A-WSD (RP 2A-WSD) Twenty-First Edition* steel design code for *Basic Hydrostatic Pressure and Punching Shear stress* checks has been implemented as a prerelease feature. This new code, APIWSD21, may be used to select or check Circular Hollow Sections (Pipes). The Prerelease documentation for the APIWSD21 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Offshore Loading, Analysis and Design , and APIWSD21: API RP 2A-WSD, 21^{st} Edition in the GTSTRUDL Output Window.

2.9 Reinforced Concrete

1. The ACI 318-05 reinforced concrete code has been added. The command format is

   METHOD ACI 318-05 (SEISMIC)

   The ACI 318-05 code supports all options included with 318-89 including special moment frame detailing according to Chapter 21 if requested. The default assumptions for 318-05 are listed in Volume 4, Table 2.4-1.

2.10 Steel Design

1. The steel design code AISC13, which is based on the *Steel Construction Manual AISC Thirteenth Edition*, is moved to released status. The documentation for the AISC13 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, AISC13 in the GTSTRUDL Output Window.

2. A new ANSI/AISC N690, *American National Standard Specification for the Design, Fabrication, and Erection of Steel Safety-Related Structures for Nuclear Facilities*, 1994 Edition, has been implemented as a released feature. This new code, N690-94, may be used to select or check any of the following shapes:

Design for axial force and bi-axial bending:

I shapes

Channels

Single Angles

Tees

Double Angles

Round Bars

Pipes

Square and Rectangular Bars

Structural Tubing

The documentation for the N690-94 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and N690-94 in the GTSTRUDL Output Window.

3. New ASME, *Boiler and Pressure Vessel Code, Section III, Rules for Construction of Nuclear Power Plant Components. Division I - Subsection NF Component Supports,* 2004 and 2007 Editions with their respective addenda, have been implemented as released features. These new codes, NF-2004 and NF-2007, may be used to select or check any of the following shapes:

Design for axial force and bi-axial bending:

I shapes

Channels

Single Angles

Tees

Double Angles

Round Bars

Pipes

Square and Rectangular Bars

Structural Tubing

The documentation for the NF-2004 and NF-2007 codes may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and "NF-2004 and NF-2007" in the GTSTRUDL Output Window.

4. Results of a steel code check or select using the MAXIMUM ENVELOPE option of the CHECK and SELECT commands are now available in the code check results datasheet, the view code check results in GTMenu, and the LIST CODE CHECK RESULTS command.

5. A new feature has been added to the steel design CHECK and SELECT commands to check the user specified parameters to see if they are valid for the requested design code (i.e., AISC13, ASD9, EC3-2005, N690-94, NF-2007, etc.). This new feature has been implemented due to anticipating that a user may take an existing model based on the ASD9 code and now modifying the model to perform a code check based on the AISC13 code. This new feature starts checking user specified parameters for the requested code at the beginning of the CHECK or SELECT command. If there are any user specified parameters that do not apply to the specified design code, a warning message with a list of parameters that are not applicable is printed.

6. A new *Eurocode 3, EN 1993-1-1: 2005 (E)* steel design code has been implemented as a prerelease feature. This new code, EC3-2005, may be used to select or check any of the following shapes:

   Design for axial force and bi-axial bending:

      I shapes

      Circular Hollow Sections (Pipes)

      Rectangular Hollow Sections (Structural Tube)

      Solid Round Bars

   Design for axial force only:

      Single Angles

      Double Angles

   The prerelease documentation for the EC3-2005 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and EC3-2005 in the GTSTRUDL Output Window.

7. A new option has been added to ASD9 and AISC13 codes to check the slenderness ratio and compute the number of bolts according to the transmission tower code. When a value of YES has been specified for parameter TowerCK, transmission tower provisions are checked in addition to the provisions of the AISC (ASD9 and AISC13) codes. Applicable cross-sections are single and double angles.

   The prerelease documentation may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and Transmission Tower code modifications to AISC codes in the GTSTRUDL Output Window.

2.11 Steel Tables and GTTABLE

1. The following new Metric Tables from AISC Thirteenth Edition have been added to GTSTRUDL:

WAISC13M	W shapes from Table 1-1 of the AISC 13th Edition
MSHP-13M	M, S, and HP shapes from Tables 1-2, 1-3, and 1-4 of the AISC 13th Edition
CAISC13M	Channel C and MC shapes from Tables 1-5 and 1-6 of the AISC 13th Edition
L-ALL13M	Single angles from Table 1-7 of the AISC 13th Edition
L-EQ-13M	Equal leg single angles from Table 1-7 of the AISC 13th Edition
L-UN-13M	Unequal leg single angles from Table 1-7 of the AISC 13th Edition
WTAIS13M	Tee WT, MT, and ST shapes from Tables 1-8, 1-9, and 1-10 of the AISC 13th Edition
ReHSS13M	Rectangular and square HSS from Tables 1-11 and 1-12 of the AISC 13th Edition
RdHSS13M	Round HSS from Table 1-13 of the AISC 13th Edition
2LALL13M	Double angles from Table 1-15 of the AISC 13th Edition

2L-EQ13M	Equal legs double angles from Table 1-15 of the AISC 13th Edition
2L-LL13M	Long legs back-to-back double angles from Table 1-15 of the AISC 13th Edition
2L-SL13M	Short legs back-to-back double angles from Table 1-15 of the AISC 13th Edition
WBEAM13M	W shapes commonly used as beams from Table 3-6 of the AISC 13th Edition
WCOL-13M	W shapes commonly used as columns from Table 4-1 of the AISC 13th Edition

2. The GTTABLE commands have been modified to accept long profile names. Also, the GTTABLE output has been modified to display both long and short profile names. The affected GTTABLE commands are shown below:

MODIFY ITEMS

TRANSFER ITEMS

DELETE ITEMS

OUTPUT

PRINT ITEMS

PRINT ITEM NAMES

The above commands are documented in the GTTABLE User Manual. The output generated by any of the GTTABLE commands now displays the long and short profile names except the PRINT ITEM NAMES command. The PRINT ITEM NAMES command prints the long profile names only. If the short profile names are desired, specify the PRINT ITEM NAMES SHORT NAMES command. The user should keep in mind that in order to specify commands with the long profile names, table conversion files must exist. The Table conversion files are located at “<installation_folder>\Table Conversion Files.” For help on how to create a table conversion file, see “<installation_folder>\Table Conversion Files\User\README.txt.”

The only command in GTTABLE that does not accept long profile names is the ADD ITEMS command. When you are creating a new table, profile names specified by the ADD ITEMS command are limited to a maximum of 8 characters. You can then create a table conversion file which must contain the long profile name (maximum of 24 characters) and it's short profile name (maximum of 8 characters) equivalent which had been specified in the ADD ITEMS command. See "<installation_folder>\Table Conversion Files\User\README.txt" for more help.

2.12 Utility Programs

1. GTS2ACAD (GTSTRUDL to AutoCAD) has been upgraded to version 4.2. Two-dimensional elements with mid-side nodes are now processed. Previously the translation would not include those elements, with an error message: "Element not yet implemented... will be ignored".

   Elements with mid-side nodes now processed are:

   Triangles: LST, UTLQ1

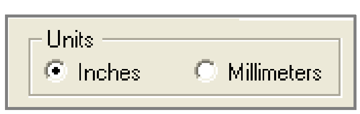
   Quads: IPQQ, IPCQ, IPQLQ1, IPQLQ2, IPQLQ2B, IPQLQ3, IPQLQ4, IPBQQ

2. ReadCIS2 and WriteCIS2 have been upgraded to version 2.0 with the addition of the following tables: AISC 13th Edition 2005, AISC 13th Edition 2005 Metric and LRFD 3rd Edition.

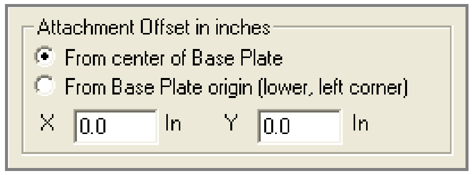
3. The window size and placement of the Scope Editor is now saved and used the next time it is opened. Also, the Paragraph font is maintained when the Paragraph is edited.

2.13 Base Plate

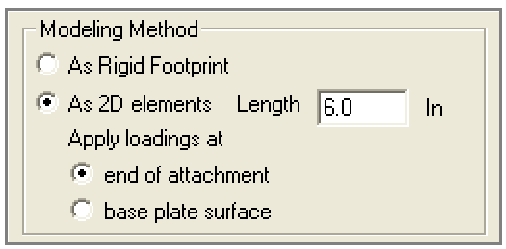
1. The Metric unit of "Millimeters" has been added to the Plate tab as shown below:



2. The Attachment placement offset can now be specified from the plate origin as well as the plate center in the Attachment dialog shown below:



3. Attachments modeled as FE extensions may be loaded at the extension end or at the plate surface. This feature was added at the request of users since results from a frame analysis are usually output at the top of the base plate's surface. The modified Add Attachment dialog with the new option to Apply loadings at the end of attachment or the base plate surface is shown below:



4. Comments in the generated .gti file have been expanded.

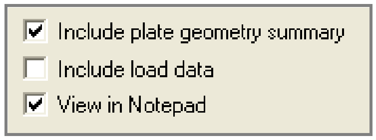
5. The Base Plate Wizard now prompts to save a .gtbp file if changes to the plate have been made.

6. Current unit display has been added to dialogs that previously only had a bare edit box. Current units display has also been added to the results datasheets.

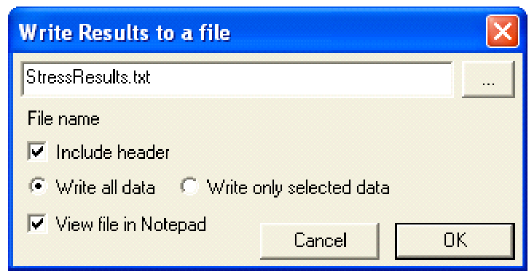
7. The AISC 9th Edition metric profile names have been added as available attachments.

8. When a .gtbp file is read or saved, its name is added to the Base Plate Wizard window title and the file name is used as the default .gti file in the GTSTRUDL Input File options dialog.

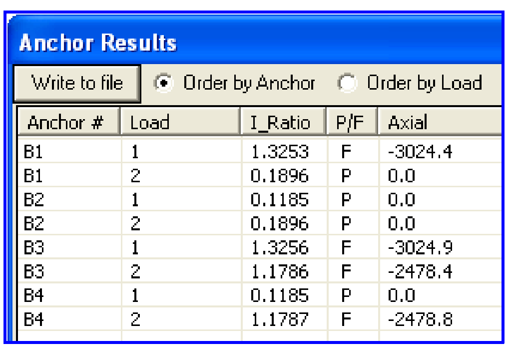
9. An ‘Include load data’ option has been added to the Results Summary dialog as shown below:



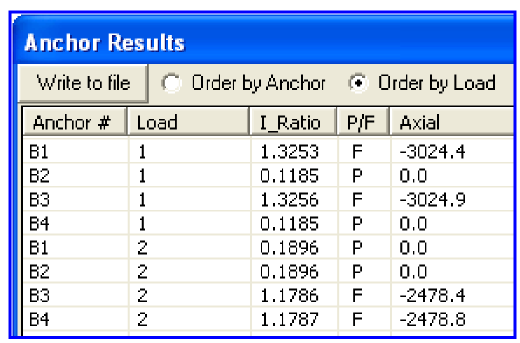
10. A ‘Write to File’ option has been added to the individual result datasheets.



11. The Anchor Results datasheet now includes options to organize anchor results by anchor or by load as shown in the two examples below:



Anchor #	Load	I_Ratio	P/F	Axial
B1	1	1.3253	F	-3024.4
B1	2	0.1896	P	0.0
B2	1	0.1185	P	0.0
B2	2	0.1896	P	0.0
B3	1	1.3256	F	-3024.9
B3	2	1.1786	F	-2478.4
B4	1	0.1185	P	0.0
B4	2	1.1787	F	-2478.8



Anchor #	Load	I_Ratio	P/F	Axial
B1	1	1.3253	F	-3024.4
B2	1	0.1185	P	0.0
B3	1	1.3256	F	-3024.9
B4	1	0.1185	P	0.0
B1	2	0.1896	P	0.0
B2	2	0.1896	P	0.0
B3	2	1.1786	F	-2478.4
B4	2	1.1787	F	-2478.8

12. The Plate Displacements datasheet now offers two sorting options, by ‘Lift off’ (+Z displacement) or by ‘Depress’ (-Z displacement) as shown below:

CHAPTER 3

ERROR CORRECTIONS

This chapter describes changes that have been made to GTSTRUDL to correct errors. These errors may have produced aborts, incorrect results, or restricted use of a feature in previous versions of GTSTRUDL. The error corrections are discussed by the primary feature areas of GTSTRUDL.

3.1 General

1. Previously, when <u>two</u> LIMITS were specified in an ELEVATION command, an error message was generated and the ELEVATION command was ignored. A single LIMITS specification did not cause a problem. A sample error message is shown below:

```
ELEVATION 30.0 LIMITS X 0.0 170.0 Z -90.0 0.0 -
EXCEPT LIMITS X 30.0 110.0 Z -71.50 -38.50 -
VALUE 0.30 ONE WAY X

****ERROR_ARLD03 - Invalid LIMITS specified for direction Y.
                   This ELEVATION command is ignored.
```

This problem has been corrected and the second LIMITS (used in the EXCEPT clause) is now processed correctly.

The following two ELEVATION commands, although similar to the problem above, did not cause a problem because only one LIMITS specification is used:

```
ELEVATION 30.0 LIMITS X 0.0 170.0 Z -90.0 0.0 -
VALUE 0.30 ONE WAY X

ELEVATION 30.0 -
EXCEPT LIMITS X 30.0 110.0 Z -71.50 -38.50 -
VALUE 0.30 ONE WAY X
```

(No GPRF issued)

2. The DEAD LOAD command can now process "MEMBERS EXISTING ELEMENTS ONLY". Previously, the "ELEMENTS ONLY" would be ignored and generate the following error message:

```
**** ERROR_STGNDL -- Member list missing or no valid members in the list.
                     DEAD LOADS not generated and SCAN mode entered.

     CI-w-cmdnpro, ERROR: The following symbols were not processed.
     ONLY
```

   Example:

```
DEAD LOAD 1 'Elements only' -Y INCLUDE FINITE ELEMENTS -
MEMBERS EXISTING ELEMENTS ONLY
```

   The above command will load all the existing finite elements, but none of the members. (No GPRF issued)

3. The SUMMARY FINAL ONLY option for LIST SECTION FORCES and LIST SECTION STRESSES now works correctly. Previously, the summaries for each individual member were printed, plus an error message stating that "FINAL" was not processed. Now, only the final summary with the max and min for all members in the list is printed. (No GPRF issued)

4. The abort of the FORM NOTIONAL LOAD command caused by a specified gravity loading condition that contains only applied member moment loads has been corrected. (GPRF 2009.03)

5. The LOCATE DUPLICATE MEMBER command now correctly reports duplicate members after member deletions. Previously, under some circumstances, the report could contain incorrect duplicate names. (GPRF 2009.06)

6. The QUERY command previously included rigid bodies in the Members count as well as the Rigid Bodies count. Now, rigid bodies are included only in the Rigid Bodies count. (No GPRF issued)

3.2 GTMenu

(*GPRF's are not issued for GTMenu unless specifically noted below*)

1. In GTMenu, EXTRUDE using the ID increments option no longer aborts when you click Done.

2. Nonlinear springs no longer cause GTMenu to abort when you have selected Weight and Center of Gravity in the Check Model dialog. The nonlinear springs are now ignored when calculating the weight and center of gravity.

3. The global axis used to be drawn twice and at different locations after creating an input file. This no longer occurs.

4. An abort no longer occurs when displaying Parameters for models which contain rigid bodies.

5. Previously, only 6 characters of a Parameter value were labeled. Now, all 8 characters are labeled.

6. Previously, if members which had Parameters were split, an abort could occur when creating an input file. This no longer occurs.

7. When splitting members, Undo now undoes all of the splits since the last Store. Previously, only the very last member that was split would be undone.

8. An abort will no longer occur when splitting members and you select No in response to the following question in the pop-up dialog:

   "Some of the Split Members Are Loaded. Do You Want to Keep the Loads?"

9. When entering GTMenu, the structure is now drawn only once. For large models, this previously slowed down the entry into GTMenu.

10. GTMenu no longer aborts when exiting GTMenu and returning to command mode after displaying temperature loads on members which have been created by performing a Copy Model.

11. An abort will no longer occur when refining a finite element mesh if the elements have joint temperature loads.

12. The Material Group is now retained when refining a finite element mesh.

13. An abort no longer occurs when entering GTMenu with models which contain superelements which consist only of members with more than 21 boundary nodes in the superelement.

14. Plastic hinge results are no longer reversed at the ends of some members.

15. Points may now be defined in command mode without Lines/Curves. They may also be defined in clusters in the input file - they no longer have to be defined all in one location in the input file.

16. Negative coordinates may now be specified when Projecting an Axis onto a line.

17. Labels no longer disappear after performing a Save (in the File pulldown).

18. Rigid Bodies can now be viewed in an orientation where they are superimposed.

3.3 GTStrudl Output Window

(*GPRF's are not issued for the GTStrudl Output Window unless specifically noted below*)

1. The Area Load dialog now respects the Plane Tolerance specified in the "New Loading" dialog. Previously, the plane tolerance for determining which joints lay at the specified elevation was always 2.0 in the current units. For example, if the current units were METERS, the plane tolerance would be 2.0 meters, which could result in unexpected area determination. The AREA LOAD command was unaffected by this problem.

2. Previously, a program abort was possible when using the joint displacement Datasheet selection from the Results menu when the External File Solver is active, i.e. when analysis results are stored in external files. This problem has been corrected.

3. You will no longer get the following error message if you have specified the Parameter Code as AISC13 prior to entering the Parameters dialog under the Steel Design pulldown: "Specified code AISC13 is not a valid code name"

3.4 Model Wizard

(*GPRF's are not issued for the Model Wizard unless specifically noted below*)

1. The Finite Element Mesh Wizard now sets edge support conditions properly for meshes with 2 elements per side. Previously, some interior nodes would be declared as supports.

2. The Space Frame Wizard now correctly processes the CONRECT table. Previously, profiles selected from the CONRECT table would generate an invalid MEMBER PROPERTIES command.

3. The Vault/Rectangular Tank Wizard can now correctly process small element sizes. Previously, when the size of an element approached a value of 0.25 in the current geometry units, element incidence errors could occur along the edges of the tank.

3.5 Nonlinear Analysis

1. Nonlinear dynamic analysis does not support the presence of joint constraints in the model as documented in Section 2.6.5, Volume 3 of the Reference Manual. However, when all joint constraints have been deleted, a subsequent request to execute a nonlinear dynamic analysis would report that joint constraints are still present, causing the analysis to be terminated without completions. The has been corrected so that now, under these conditions, nonlinear dynamic analysis will execute to completion. (No GPRF issued)

3.6 Offshore

1. In versions prior to and including Version 30, the READ WAVE LOADS FOR FATIGUE NEW command would drop wave joint load components when the wave loads were created by GTSelos from a mode superposition dynamic wave load analysis and during processing, both the joint and wave loading names changed simultaneously. This error has been corrected.

   (No GPRF issued)

2. The quality and consistency of the convergence of fatigue analysis load-dependent Local Joint Flexibility (LJF) iterations has been improved. In Version 30, which is the first version in which the fatigue LJF analysis appeared, the load-dependent LJF iterations do not consistently converge to the expected load-dependent LJF member stiffnesses when such stiffnesses do not change between the fatigue loads for which the fatigue analysis is executed. (No GPRF issued)

3.7 Reinforced Concrete

1. After the MATERIAL REINFORCED CONCRETE command is given and when the constant E is specified for a list of members, the reinforced concrete constant EC will be set to the same value for all the members in the list. Previously, EC was set to the new E value for only the last member in the list. (GPRF 2009.04)

3.8 Utility Program (Scope Editor)

(GPRF's are *not* issued for Utility Programs unless specifically noted below)

1. Discs and Circles (representing joints and points) handling has been improved. They are now scaled accurately, instead of being a fixed size.
2. Centering graphics from the Dynamic dialog plots now works properly on all computers. Previously, some computers (especially 64-bit computers) would not display the diagrams.

CHAPTER 4

KNOWN DEFICIENCIES

This chapter describes known problems or deficiencies in Version 31. These deficiencies have been evaluated and based on our experience, they are seldom encountered or there are workarounds. The following sections describe the known problems or deficiencies by functional area.

4.1 External File Solver

1. When the EXTERNAL FILE SOLVER is active for any sequence of static and/or dynamic analyses, extra care must be exercised when using ADDITIONS and DELETIONS operations on joints, members, and finite elements in order to avoid inconsistencies between analysis results stored in the external results files and the topological characteristics of the model, such as numbers of joints, members, finite elements, and the order in which they were created and deleted. Such inconsistencies can result in erroneous computation of other results such as load combination results and results errors in the reports created by LIST commands (LIST DISPLACEMENTS, LIST FORCES, etc.), or the display of such results in GTMenu.

   In particular, the inconsistencies indicated above will occur when DELETIONS operations, followed by ADDITIONS operations, are performed between two successive analyses. After the second analysis, any results computed and stored by the first analysis but prior to the DELETIONS and ADDITIONS operations are erroneous, being no longer consistent with the state of the model following the DELETIONS and ADDITIONS operations and the second analysis. Such results, if used following the DELETIONS and ADDITIONS operations and the second analysis, will produce erroneous reports and graphical results displays. Consider the following sequence of commands:

```
ACTIVE SOLVER GTSES   $ Turns on the GTSES solver for all analyses
                      $ with analysis results stored in external files
```

   create model

```
LOAD LIST 1
STIFFNESS ANALYSIS
```

perform DELETIONS, then ADDITIONS of joints, members, finite elements

LOAD LIST 2
STIFFNESS ANALYSIS

CREATE LOAD COMBINATION ‘LC1' SPECS 1 1.0 2 1.0

Note that the analysis results stored in the external files for loading 1 are inconsistent with the state of the model following the second stiffness analysis and are erroneous. Therefore, the results computed for load combination LC1 by the above CREATE LOAD COMBINATION command are erroneous.

The best procedure to follow in order to ensure the calculation of consistent and correct results under the conditions described above is to create the model in full prior the first analysis. ADDITIONS and DELETIONS operations on joints, members, and finite elements may be safely used in any order at that time. If any joints, members, and finite elements must be “added” and “deleted” between any two successive analyses, the ACTIVE and INACTIVE functions can be used to do this safely. The following sequence of commands produces and maintains consistent and correct analysis results:

```
ACTIVE SOLVER GTSES    $ Turns on the GTSES solver for all analyses
                       $ with analysis results stored in external files
```

create model, including all joints members, and finite elements both immediately and eventually required

perform INACTIVE operations on desired joints, members, and finite elements

LOAD LIST 1
STIFFNESS ANALYSIS

perform ACTIVE and INACTIVE operations on desired joints, members, finite elements

LOAD LIST 2
STIFFNESS ANALYSIS
CREATE LOAD COMBINATION ‘LC1' SPECS 1 1.0 2 1.0

(GPRF 2010.05)

4.2 Finite Elements

1. The ELEMENT LOAD command documentation indicates that header information such as type and load specs are allowed. If information is given in the header and an attempt is made to override the header information, a message is output indicating an invalid command or incorrect information is stored. (GPRF 90.06)

2. Incorrect results (displacements, stresses, reactions, frequencies, ... etc.) will result if a RIGIDITY MATRIX is used to specify the material properties for the IPSL, IPSQ, and TRANS3D elements. (GPRF 93.09)

3. The CALCULATE RESULTANT command may either abort or print out an erroneous error message for cuts that appear to be parallel to the Planar Y axis. (GPRF 94.21)

4. If a superelement is given the same name as a member or finite element, an abort will occur in the DEVELOP STATIC PROPERTIES command. (GPRF 95.08)

5. The curved elements, TYPE 'SCURV' and 'PCURV' will produce incorrect results for tangential member loads (FORCE X). An example of the loading command which will produce this problem is shown below:

   ```
   LOADING 1
   MEMBER LOADS
   1  FORCE  X  UNIFORM   W  -10
   ```

   where member (element) 1 is a 'SCURV' or 'PCURV' element.

   (GPRF 99.13)

4.3 General Input/Output

1. An infinite loop may occur if a GENERATE MEMBERS or GENERATE ELEMENTS command is followed by a REPEAT command with an incorrect format. An example of an incorrect REPEAT command is shown below by the underlined portion of the REPEAT Command:

   GENERATE 5 MEM ID 1 INC 1 FROM 1 INC 1 TO 2 INC 1
   REPEAT 2 TIMES ID 5 FROM <u>7 INC 1 TO 8 INC 1</u>

   Only the increment may be specified on the REPEAT command. (GPRF 93.22)

2. Rigid body elements cannot be deleted or inactivated as conventional finite elements. The specification of rigid body elements as conventional finite elements in the INACTIVE command or in DELETIONS mode will cause an abort in a subsequent stiffness, nonlinear, or dynamic analysis. (GPRF 97.21)

3. The path plus file name on a SAVE or RESTORE is limited to 256 characters. If the limitation is exceeded, the path plus file name will be truncated to 256 characters. This is a Windows limitation on the file name including the path. (No GPRF issued)

4. Object groups, created by the DEFINE OBJECT command, may not be used in a GROUP LIST as part of a list. If the OBJECT group is the last group in the list, processing will be correct. However, if individual components follow the OBJECT group, they will fail. Also, you can not copy members or joints from the OBJECT group into a new group. (GPRF 99.26)

5. Numerical precision problems will occur if joint coordinate values are specified in the JOINT COORDINATES command with more than a total of seven digits. Similar precision problems will occur for joint coordinate data specified in automatic generation commands. (GPRF 2000.16)

6. Internal member results will be incorrect under the following conditions:

   1. Dynamic analysis is performed (response spectra or time history)
   2. Pseudo Static Loadings are created
   3. Buckling Analysis is Performed
   4. Internal member results are output or used in a subsequent steel design after the Buckling Analysis.

In addition, the eigenvalues and eigenvectors from the Dynamic Analysis are overwritten by the eigenvalues and eigenvectors from the Buckling Analysis.

We consider this problem to be very rare since we had never encountered a job which contained both a Dynamic Analysis and a Buckling Analysis prior to this error report.

Workaround:

Execute the Buckling Analysis in a separate run which does not contain a dynamic analysis.

Alternatively, execute the Buckling Analysis before the Dynamic Analysis and output the Buckling results and then perform a Dynamic Analysis. The Dynamic Analysis results will then overwrite the buckling multiplier and mode shape which is acceptable since the buckling results have been output and are not used in any subsequent calculations in GTSTRUDL.

(GPRF 2004.14)

4.4 GTMenu

1. Gravity loads and Self-Weight loads are generated incorrectly for the TRANS3D element.

   Workaround: Specify the self-weight using Body Forces under Element Loads. ELEMENT LOADS command is described in Section 2.3.5.4.1 of Volume 3 of the GTSTRUDL Reference Manual.

   (GPRF 95.18)

2. The Copy Model feature under Edit in the Menu Bar will generate an incorrect model if the model contains the TRANS3D element.

   Workaround: Use the DEFINE OBJECT and COPY OBJECT commands in Command Mode as described in Section 2.1.6.7.1. and 2.1.6.7.5 of Volume 1 of the GTSTRUDL Reference Manual.

   (GPRF 95.21)

3. The Load Summations option available under CHECK MODEL will produce incorrect load summations for line, edge, and body loads on all finite elements. The Load Summations are also incorrect for projected loads on finite elements. The load summations for line and edge loadings should be divided by the thickness of the loaded elements. The body force summations should be multiplied by the thickness of the loaded elements for two-dimensional elements.

   Workaround: You can check the load summation by specifying the LIST SUM REACTIONS command after STIFFNESS ANALYSIS.

   (No GPRF issued)

4. Projected element loads will be displayed incorrectly when they are created or when they are displayed using Display Model → Loads.

   Workaround: Verify that the loads are correct in the GTSTRUDL Output Window using the PRINT LOAD DATA command or by checking the reactions using LIST SUM REACTIONS.

   (No GPRF issued)

4.5 Scope Environment

1. OVERLAY DIAGRAM in the Plotter Environment produces diagrams that are much smaller relative to the plot size than the Scope environment does. This is because the structure plot is magnified to fill the Plotter graphics area, but the height of the diagram is not increased. As a work-around, use the PLOT FORMAT SCALE command to decrease the scale factor, which will increase the size of the diagram. The current value is printed with a Scope Environment OVERLAY DIAGRAM. The value printed with a Plotter Environment OVERLAY DIAGRAM is incorrect. For example, if a Moment Z diagram is OVERLAYed with a scale factor of 100.0 on the Scope, the command PLOT FORMAT SCALE MOMENT Z 50. would scale a reasonable OVERLAY DIAGRAM for the Plotter. (GPRF 96.19)

CHAPTER 5

PRERELEASE FEATURES

5.1 Introduction

This chapter describes new features that have been added to GTSTRUDL but are classified as prerelease features due to one or more of the following reasons:

1. The feature has undergone only limited testing. This limited testing produced satisfactory results. However, more extensive testing is required before the feature will be included as a released feature and documented in the GTSTRUDL User Reference Manual.
2. The command formats may change in response to user feedback
3. The functionality of the feature may be enhanced in response to user feedback.

The Prerelease features in Version 30 are subdivided into Design, Analysis, and General categories. The features in these categories are shown below:

5.2 Design Prerelease Features

5.2.1 EC3-2005 Steel Design Code

5.2.2 API Twentieth-First Edition steel design code for Basic Hydrostatic Pressure and Punching Shear stress checks

5.2.3 LRFD3 Steel Design Code. Rather than use the LRFD3 code which is a prerelease feature, users should use the AISC13 code which is now a released feature.

5.2.4 ACI Concrete Code 318-99. Rather than use the 318-99 code which is a prerelease feature, users should use the 318-05 code which is now a released feature.

5.2.5 Rectangular and Circular Concrete Cross Section Tables

5.2.6 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

We encourage you to experiment with these prerelease features and provide us with suggestions to improve these features as well as other GTSTRUDL capabilities.

5.2 Design Prerelease Features

5.2.1 EC3-2005 Steel Design Code

A new *Eurocode 3, EN 1993-1-1: 2005 (E)* steel design code has been implemented as a prerelease feature. This new code, EC3-2005, may be used to select or check any of the following shapes:

Design for axial force and bi-axial bending:

I shapes

Circular Hollow Sections (Pipes)

Rectangular Hollow Sections (Structural Tube)

Solid Round Bars

Design for axial force only:

Single Angles

Double Angles

The prerelease documentation for the EC3-2005 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and EC3-2005 in the GTSTRUDL Output Window.

5.2.2 API Twenty-First Edition Hydrostatic Pressure and Punching Shear Checks

A new *API Recommended Practice 2A-WSD (RP 2A-WSD), Twenty-First Edition* steel design code for *Basic Hydrostatic Pressure and Punching Shear stress* checks has been implemented as a prerelease feature. This new code, APIWSD21, may be used to select or check Circular Hollow Sections (Pipes). The Prerelease documentation for the APIWSD21 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Offshore Loading, Analysis and Design , and APIWSD21: API RP 2A-WSD, 21^{st} Edition in the GTSTRUDL Output Window.

Also see the following Sections 4.3.1, 4.3.2 and 4.4 for Punching Shear Chord Classification and Check Punching Shear commands. These sections are numbered as they will appear when added to Volume 8 of the GTSTRUDL User Reference Manual.

4.3 Punching Shear Classification

The classification of each joint to be checked may be done explicitly or automatically by GTSTRUDL. Section 4.3.1 discusses the explicit joint classification and Section 4.3.2 discusses the automatic joint classification.

4.3.1 CHORDS FOR PUNCHING SHEAR Command: Explicit Joint Classification

Simple form:

CHORDS (FOR) (PUNCHING SHEAR)

(CHORD) c_1 (JOINT) j_1 (joint configuration) –

(BRACES) $\left\{ \begin{matrix} \text{list}_1 \\ \underline{\text{ALL}} \end{matrix} \right\}$

•   •   •
•   •   •
•   •   •

END (OF CHORD DATA)

where:

$$(\text{joint configuration}) = \left\{ \begin{matrix} \underline{\text{GEO}}\text{METRY} & \left\{ \begin{matrix} \rightarrow \ '\underline{\text{K}}' \\ '\underline{\text{T}}' \\ '\underline{\text{Y}}' \\ '\underline{\text{X}}' \\ '\underline{\text{CROSS}}' \\ '\underline{\text{DOUBLE-Y}}' \end{matrix} \right\} \\ \underline{\text{OVE}}\text{RLAP} & \left\{ \begin{matrix} \rightarrow \ -1 \\ v_1 \end{matrix} \right\} \\ \underline{\text{DIA}}\text{PHRAGM} & \\ \underline{\text{L}} & v_2 \\ \underline{\text{L1}} & v_3 \\ \underline{\text{L2}} & v_4 \\ \underline{\text{TW}} & v_5 \\ \underline{\text{VW}} & v_6 \end{matrix} \right\}^{*}$$

General form:

$$\underline{\text{CHO}}\text{RDS } (\underline{\text{FOR}})\ (\underline{\text{PUN}}\text{CHING } \underline{\text{SHE}}\text{AR})$$

$$(\underline{\text{CHO}}\text{RD})\ c_1\ (\underline{\text{JOI}}\text{NT})\ j_1$$

$$\underline{\text{BRA}}\text{CE}\ \ b_1 \left(\underline{\text{LOA}}\text{D} \left\{ \begin{array}{l} \ \ \text{list}_2 \\ \rightarrow \underline{\text{ALL}} \\ \ \ \ \underline{\text{ALL}}\ \ \underline{\text{BUT}}\ \ \text{list}_2 \end{array} \right\} \right) \quad \langle\ \text{joint configuration}\ \rangle$$

$$\begin{array}{ccccc} \vdots & \vdots & \vdots & \vdots & \vdots \end{array}$$

$$\underline{\text{END}}\ (\underline{\text{OF}}\ \underline{\text{CHO}}\text{RD}\ \underline{\text{DAT}}\text{A})$$

where,

$$\langle\ \text{joint configuration}\ \rangle \ = \ {}^{*}\left\{ \begin{array}{l}
\underline{\text{GEO}}\text{METRY} \left\{ \begin{array}{l}
\rightarrow\ '\underline{\text{K}}' \\
\quad '\underline{\text{T}}' \\
\quad '\underline{\text{Y}}' \\
\quad '\underline{\text{X}}' \\
\quad '\underline{\text{CROSS}}' \\
\quad '\underline{\text{DOUBLE-Y}}' \\
\quad \underline{\text{PAR}}\text{T} \left\{ \begin{array}{l}
'\underline{\text{K}}'\ \ r_1\ \ '\underline{\text{T}}' \\
'\underline{\text{K}}'\ \ r_1\ \ '\underline{\text{Y}}' \\
'\underline{\text{K}}'\ \ r_1\ \ '\underline{\text{CROSS}}' \\
'\underline{\text{T}}'\ \ r_1\ \ '\underline{\text{CROSS}}' \\
'\underline{\text{Y}}'\ \ r_1\ \ '\underline{\text{CROSS}}' \\
'\underline{\text{K}}'\ \ r_1\ \ '\underline{\text{T}}'\ \ r_2\ \ '\underline{\text{CROSS}}' \\
'\underline{\text{K}}'\ \ r_1\ \ '\underline{\text{Y}}'\ \ r_2\ \ '\underline{\text{CROSS}}'
\end{array} \right\}
\end{array} \right\} \\
\underline{\text{OVE}}\text{RLAP} \left\{ \begin{array}{l} \rightarrow\ -1 \\ \quad\ v_1 \end{array} \right\} \\
\underline{\text{DIA}}\text{PHRAGM} \\
\underline{\text{L}} \qquad\quad v_2 \\
\underline{\text{L1}} \qquad\ \ v_3 \\
\underline{\text{L2}} \qquad\ \ v_4 \\
\underline{\text{TW}} \qquad\ \ v_5 \\
\underline{\text{VW}} \qquad\ \ v_6
\end{array} \right\}$$

Elements:

c_i = integer or alphanumeric id of the member that is designated as a chord member at joint j_i.

j_i = integer or alphanumeric id of the joint for which classification data are specified.

$list_1$ = integer or alphanumeric id of the members designated as brace members at joint j_i.

b_i = integer or alphanumeric id of the member designated as a brace member at joint j_i.

$list_2$ = list of integer/alphanumeric loading names.

v_1 = the gap between the two brace members where they intersect the chord applicable to the K joint configuration only.

r_1, r_2 = fractional factor ranging from 0.0 to 1.0 which defines the percentage of different configurations acting at the joint j_i.

v_2 = circumference of brace contact with chord, neglecting presence of overlap.

v_3 = circumference of that portion of the brace which contacts the chord (actual length).

v_4 = the projected chord length (one side) of the overlapping weld, measured perpendicular to the chord.

v_5 = the lesser of the weld throat thickness or the thickness of the thinner brace.

v_6 = AISC allowable shear stress for the weld between braces. Default value is 18.0 ksi.

Explanation:

The user must completely describe the geometry of the braced joints when using the explicit method of joint classification. The chord member, the chord joint, and the brace members framing into the joint must be identified. The joints, chords, and the braces framing into the joints must be identified exactly as specified under JOINT COORDINATES and MEMBER INCIDENCES, respectively. Normally, this information is placed before the PARAMETERS command.

GEOMETRY:

The input values r_1 and r_2 are the ratios (expressed as decimal values) used to assign the various classes in a partial classification.

The API code classifies joint geometries into three categories: K, T&Y, and CROSS joints (Figure 4.3.1-1). The joint configuration GEOMETRY 'T' or 'Y' of CHORDS FOR PUNCHING SHEAR command is the T&Y geometry classification of the API Code. Valid designations for the 1983 version of the Norwegian Petroleum Directorate code are: K, T or Y, CROSS or X, and DOUBLE-Y.

Example:

```
BRACE 1 LOAD 1 GEOMETRY PART 'K' 0.5 'Y' 0.3 'CROSS'
```

The above example under load 1 shows the geometry for brace 1 is 50 percent K, 30 percent Y and 20 percent CROSS. GTSTRUDL assigns the ratio equal to $1.0\text{-}r_1\text{-}r_2$ = 1.0 - 0.5 - 0.3 = 0.2 to the CROSS classification.

OVERLAP:

The OVERLAP designation specifies the minimum spacing between brace members where they intersect the chord (Figures 4.3.1-1 and 4.3.1-2). In terms of the API Punching Shear Check, the overlap status of a joint configuration is relevant for 'K' geometries only. The OVERLAP PARAMETER should be omitted for 'T', 'Y', 'CROSS' or 'X', and 'Double-Y' geometries. The integer -1 means that some of the brace members are overlapped. In this case the 'K' geometry has overlap condition. The factor which accounts for the effects of type of loading and geometry (Q_q) is assumed equal to 1.8, and the allowable axial load component perpendicular to the chord is checked (overlapping joints).

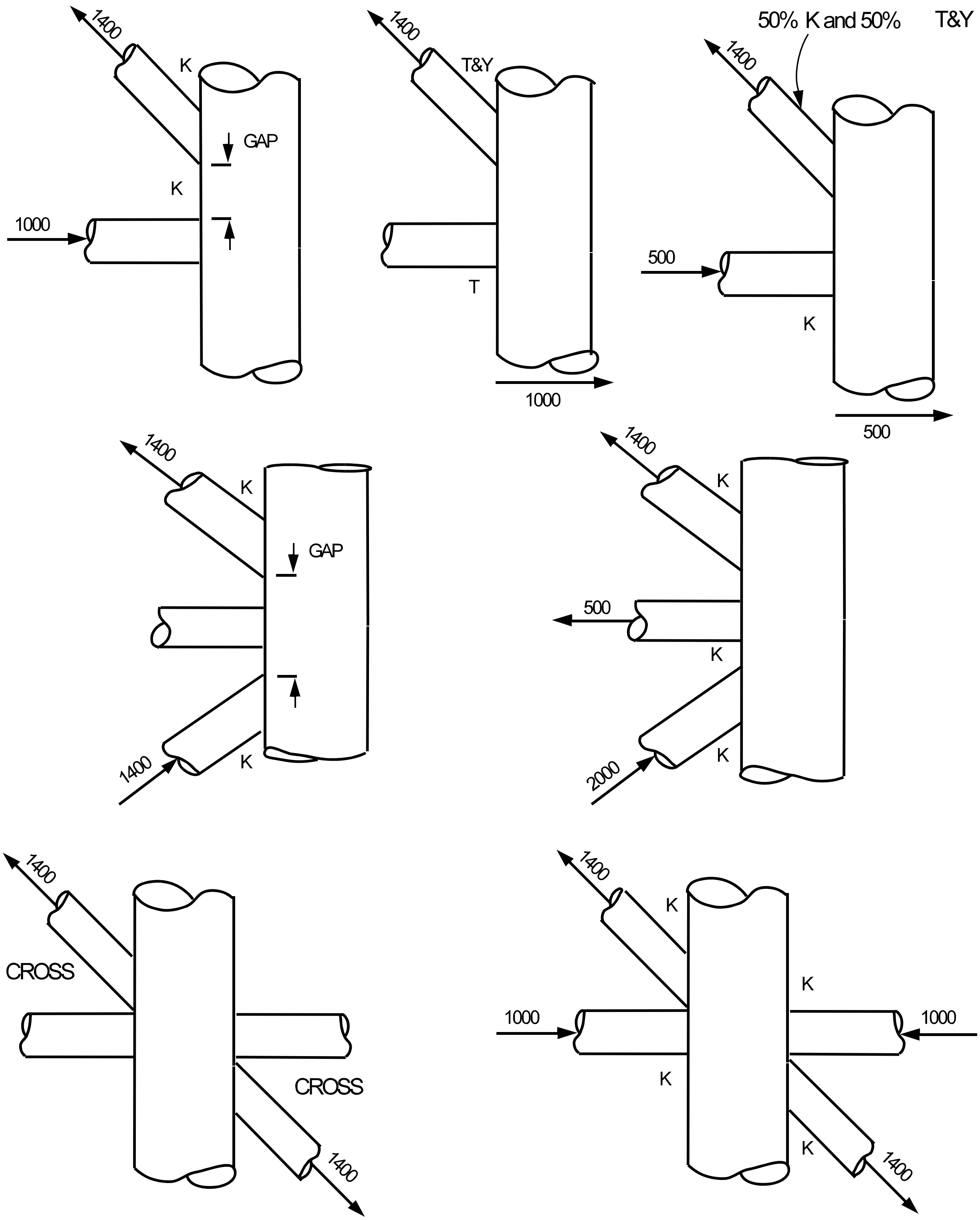


Figure 4.3.1-1Examples of Joint Classification

A value of 0 indicates the 'K' geometry with a gap. An integer 0 initiates a computation of the minimum brace separation distance based on the input chord/brace geometry. When explicit classification is used and the chord data are input with the general form, the value 0 may not be input. Brace separation values can be input directly by the user entering a positive decimal number. Negative decimal values will result in the joint configuration being treated as overlapped. The procedure used by GTSTRUDL to compute the minimum brace separation distance assumes that all brace members and the chord member lie in the same plane. For automatic classification, the default tolerance for braces considered in one plane is ±10 degrees by default. Furthermore, the braces are assumed to lie on the same side of the chord. Therefore, if you choose to let the system determine the overlap status of a joint and you are classifying joints explicitly, be certain to group the brace list so that the above criteria are met.

The following items describe the specific implications that various OVERLAP option values have on the gap factor, Q_g, which is computed only when the punching shear check is based on the APIWSD21 code:

1. When an exact value -1 (default) is specified for the OVERLAP option, the gap factor, Q_g, is set equal to 1.0. This will result in a conservative punching shear check.

2. An exact value of 0 for the OVERLAP parameter initiates the gap computation for the ‘K’ joint geometry, as indicated above, which is valid only when the CHORDS FOR PUNCHING SHEAR AUTOMATIC command (Section 4.3.2) is specified. According to this procedure, if an actual overlap is detected between two braces, the overlap distance is not computed, a value of -1 is automatically set for the OVERLAP option, and Q_g is computed according to Item 1 above. If an actual gap, or positive overlap, is detected, the gap distance is computed and used directly in the computation of Q_g.

3. A positive or negative decimal value directly specified for the OVERLAP option indicates an actual gap or overlap distance respectively, which is used directly in the computation of Q_g.

4. Note that the exact values of 0 and -1 are reserved options of the OVERLAP parameter as described in items 1 and 2 above.

a. If an exact decimal value of 0.0 is required to be specified for the OVERLAP parameter, the user may specify a small value like 0.001 or -0.001 for the overlap distance.

b. If an exact decimal value of -1.0 is required to be specified for the OVERLAP parameter, the user may specify a value of -1.001 for the overlap distance.

DIAPHRAGM:

The DIAPHRAGM modifier indicates the presence of a stiffening diaphragm in the chord for 'CROSS' joints configurations.

L/L1/L2/TW/VW:

The input quantities L, L1, L2, TW and VW describe the weld that connects the brace to the chord member. The dimension L is the circumference of the brace contact with the chord, neglecting the presence of the overlap. L1 and L2 are weld length dimensions shown in the Figure 4.3.1-4

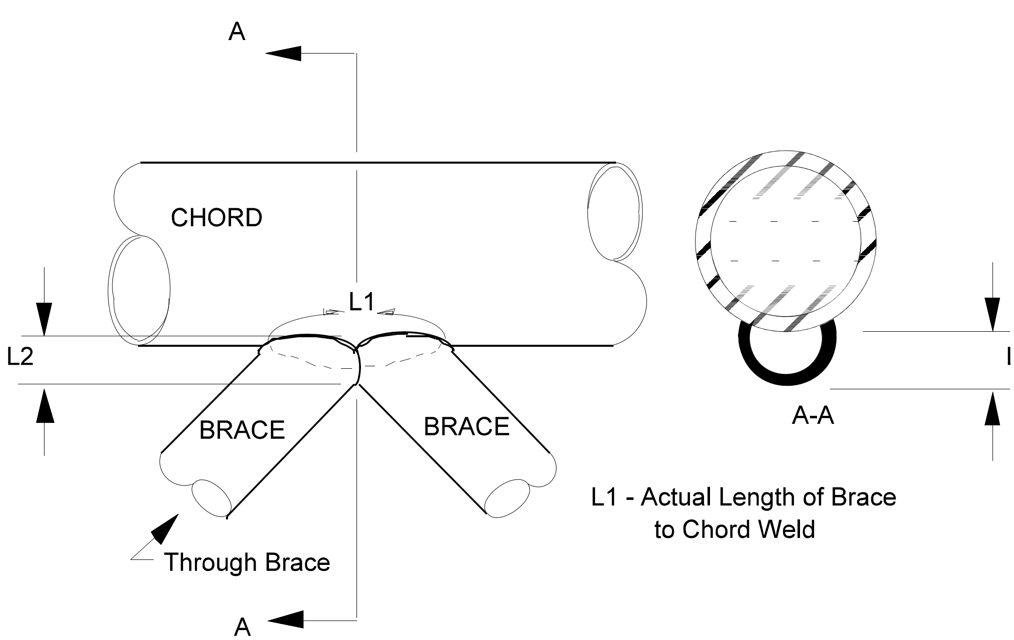


Figure 4.3.1-4 Stiffening Diaphragm

TW is the weld thickness and VW is the allowable weld stress in force units per square length unit (i.e., psi, ksi). The default values of these weld properties are listed in the table on the next page. Welds are assumed as partial to full penetration groove welds.

Example:

The correct user input for the K joint configuration in Figure 4.3.1-3 shown as:

Correct:

```
CHORDS FOR PUNCHING SHEAR
   CHORD 1   JOINT 50   BRACES 2 3 4
   CHORD 1   JOINT 50   BRACES 6 7
   CHORD 1   JOINT 50   BRACES 5 8
   CHORD 9   JOINT 50   BRACES 2 3 4
   CHORD 9   JOINT 50   BRACES 6 7
   CHORD 9   JOINT 50   BRACES 5 8
```

Incorrect:

```
CHORDS FOR PUNCHING SHEAR
   CHORD 1   JOINT 50   BRACES 2 TO 8
   CHORD 9   JOINT 50   BRACES ALL
```

The incorrect coding presumes that all the braces and the chord lie in a common plane and that the braces are all located on the same side of the chord. The computation of brace separation distance will include any brace member eccentricities specified at the chord joint. The automatic classification will define the chords as in the correct case and classify the joint according to the brace loads.

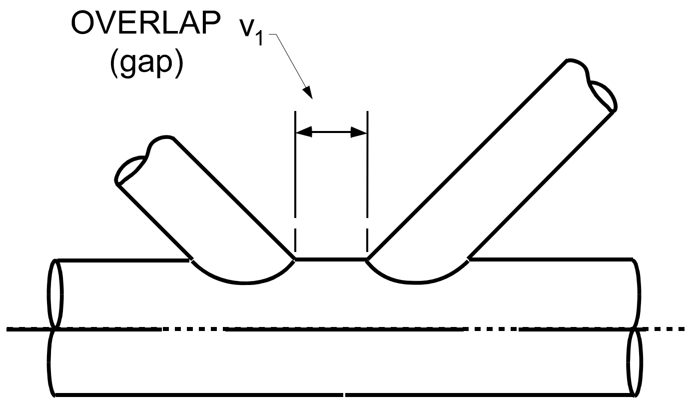


Figure 4.3.1-2 The OVERLAP Parameter

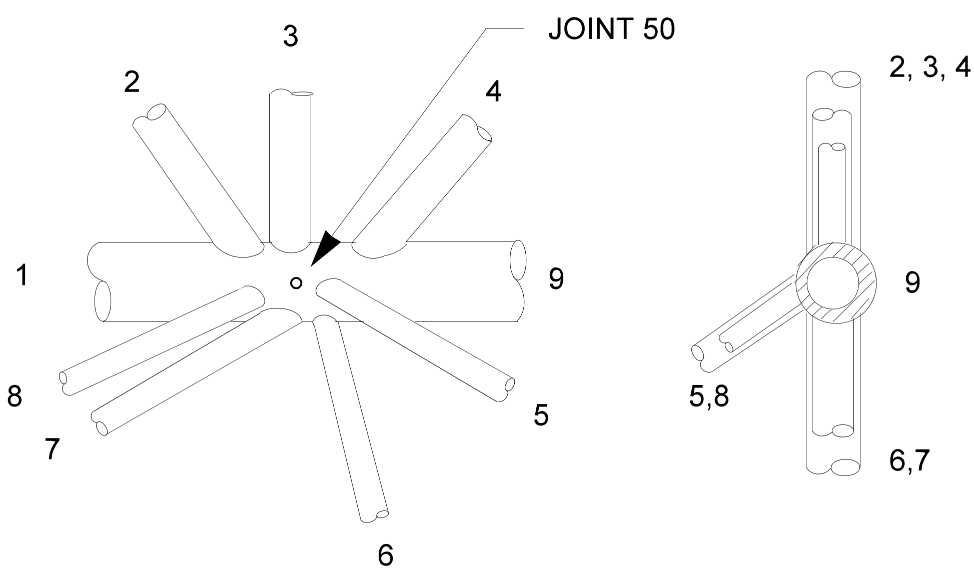


Figure 4.3.1-3 Example for Automatic Joint Classification

Table 4.3.1-1

Weld Properties

Quantity	Symbol	Default Value
Single Brace Weld Length	L	$\pi \times$ Brace Diameter/Sin Θ where Θ = brace-chord angle
Actual Weld Length	L1	$\pi \times$ Brace Diameter/Sin Θ where Θ = brace-chord angle
Brace Offset	L2	0.0
Weld Thickness	TW	Brace wall thickness.
Weld Allowable	VW	18.0 ksi

If several brace members form a K joint, conservative results will be obtained from the code check if minimum weld properties are specified.

4.3.2 CHORDS FOR PUNCHING SHEAR AUTOMATIC Command: Automatic Joint Classification

General form:

$$\underline{\text{CHO}}\text{RDS}\ (\ \underline{\text{FOR}}\ )\ (\ \underline{\text{PUN}}\text{CHING}\ \underline{\text{SHE}}\text{AR}\ )\ \underline{\text{AUT}}\text{OMATIC}\ -$$

$$\left(\left\{\begin{array}{l} \rightarrow\underline{\text{APIWSD21}} \\ \ \ \underline{\text{APILRFD1}} \\ \ \ \underline{\text{APIWSD20}} \\ \ \ \underline{\text{APIAPR87}} \\ \ \ \underline{\text{APIOCT84}} \end{array}\right\}\right)\ (\ \underline{\text{CLA}}\text{SSIFICATION}\ )\ -$$

$$\left(\underline{\text{JOI}}\text{NTS}\ \left\{\begin{array}{l} \text{list}_1 \\ \rightarrow\underline{\text{ALL}} \\ \underline{\text{ALL}}\ \underline{\text{BUT}}\ \text{list}_1 \end{array}\right\}\right)\qquad\left(\underline{\text{LOA}}\text{DS}\ \left\{\begin{array}{l} \text{list}_2 \\ \rightarrow\underline{\text{ALL}} \\ \underline{\text{ALL}}\ \underline{\text{BUT}}\ \text{list}_2 \end{array}\right\}\right)\ -$$

$$(\ \underline{\text{LIS}}\text{T}\ \underline{\text{CHO}}\text{RDS}\ )\quad(\ \text{joint configuration}\ )$$

$$(\text{joint configuration}) = \left\{\begin{array}{ll} \underline{\text{OVE}}\text{RLAP} & \left\{\begin{array}{l} \rightarrow -1 \\ \ \ 0 \\ \ \ v_1 \end{array}\right\} \\ \underline{\text{DIA}}\text{PHRAGM} & \\ \underline{\text{L}} & v_2 \\ \underline{\text{L1}} & v_3 \\ \underline{\text{L2}} & v_4 \\ \underline{\text{TW}} & v_5 \\ \underline{\text{VW}} & v_6 \end{array}\right\}^{*}$$

Elements:

list_1 = integer or alphanumeric id of the joint which automatic classification is specified.

list_2 = integer or alphanumeric id of the load which computations for classification is performed.

For joint configuration elements, see Section 4.3.1, Volume 8 of the GTSTRUDL User Reference Manual.

Explanation:

The automatic classification option will automatically identify the chords framing into the joints given in the list and will classify the joint according to the brace loads for each active load according to API RP 2A-LRFD 1st, 2A-WSD 21st, 20th, 17th, or 15th Edition. (External loads are ignored thus producing the effect of shear in the chords.)

The chords at a joint are identified and defined according to the following criteria:

- The largest continuous diameter member pair framing into a joint (independent of wall thickness), or,

- If several members of the same size frame into a common joint, the member that has the greatest wall thickness.

- If all members are of the same diameter and wall thickness, or the chord cannot be determined because more than one chord could exist by the preceding, then a warning message instructing the user to classify the joint using the explicit CHORD command. No check is performed for this joint.

- Under all circumstances, if the chord is made up of the ends of two connecting members of equal diameter making the chord continuous through the joint, i.e., two members form the chord, one from the left and one from the right, they must lie on the same vector.

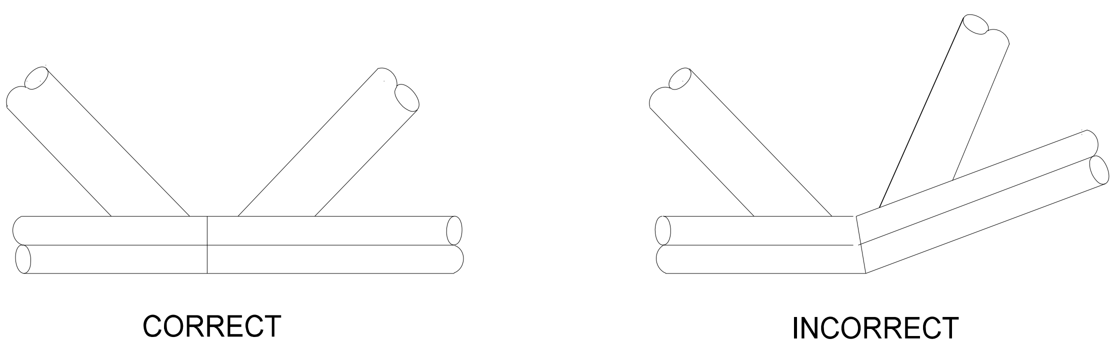


Figure 4.3.2-1 Angle between the continuous chords

- If an angle between the two chord segments continuing through a joint is greater than a tolerance value, a warning message is given instructing the user to classify the joint using explicit joint classification and no check is performed for the joint. The tolerance for members considered continuous or in one plane is ±10 degrees, which can be overridden by the parameter TOLPLANE up to a maximum of ±20 degrees. This parameter should be specified before automatic joint classification commands.

The possible automatic joint classifications are K, T, Y, CROSS, PART K CROSS, PART K T, PART K Y, PART CROSS T, PART CROSS Y, PART K CROSS T, and PART K CROSS Y. The classification is made by systematically balancing brace forces of opposite sign in half planes on each side of the chord. The process begins with the two highest brace forces of opposite sign, continuing the process in each half plane until no more members are left for balancing. If only one brace is found in a half plane, the brace is either T or Y, depending solely on geometry. Refer to Appendix B.2, Volume 8 of the GTSTRUDL User Reference Manual for the procedure description.

As voluminous computations and output could result from the different possible classifications for different loads, you are cautioned to specify only loads representative of the different conditions. For example, identify only selected wave loads from the different directions, rather than all wave loads or all loads.

The parameter GRPLOAD may be used to limit the computations for classification while still allowing proper checks for all active loads. By your identifying a load case representative of numerous similar loads, GTSTRUDL computes the classification for the one load identified and assigns the same classification for all of the loads in the LOAD list. This parameter should be specified before CHORDS FOR PUNCHING SHEAR Commands.

Example:

```
PARAMETER
GRPLOAD    1    LOADS 20 TO 30
```

Only the load identified as 1 for parameter GRPLOAD will be used for automatic classification. All members for LOADS 20 to 30 each will be assigned the same classification as assigned for load 1. Note that load 1 must also be identified in the LOAD portion of the automatic classification options.

When the automatic classification is used, the chords and braces can be listed by GTSTRUDL by specifying the LIST CHORDS option. The chord output is similar to the explicit identification for chords and braces. For example, referring to the Figure 4.3.1-3, for joint 50, the GTSTRUDL output listing the chords is as follows:

```
CHORD 1 JOINT 50
        BRACE 2 LOAD 1 GEOMETRY 'K' OVERLAP 6.62
        BRACE 3 LOAD 1 GEOMETRY PART 'K' 0.6 'T'
          OVERLAP 6.62
        BRACE 4 LOAD 1 GEOMETRY PART 'K' 0.3 'CROSS' -
          OVERLAP 6.62
        BRACE 5 LOAD 1 GEOMETRY 'K' OVERLAP 4.37
        BRACE 6 LOAD 1 GEOMETRY 'K' OVERLAP 5.18
        BRACE 7 LOAD 1 GEOMETRY PART 'K' 0.5 'CROSS' 0.34 -
          'Y' OVERLAP 5.18
        BRACE 8 LOAD 1 GEOMETRY 'K' OVERLAP 4.37

CHORD 9 JOINT 50
        BRACE 2 LOAD 1 GEOMETRY 'K' OVERLAP 6.62
        BRACE 3 LOAD 1 GEOMETRY PART 'K' 0.6 'T'
          OVERLAP 6.62
        BRACE 4 LOAD 1 GEOMETRY PART 'K' 0.3 'CROSS' -
          OVERLAP 6.62
        BRACE 5 LOAD 1 GEOMETRY 'K' OVERLAP 4.37
        BRACE 6 LOAD 1 GEOMETRY 'K' OVERLAP 5.18
        BRACE 7 LOAD 1 GEOMETRY PART 'K' 0.5 'CROSS' 0.34 -
          'Y' OVERLAP 5.18
        BRACE 8 LOAD 1 GEOMETRY 'K' OVERLAP 4.37
```

4.4 CHECK PUNCHING SHEAR Command

General form:

$$\underline{\text{CHE}}\text{CK} \left\{ \begin{array}{c} \underline{\text{APIWSD21}} \\ \underline{\text{APILRFD1}} \\ \underline{\text{APIWSD20}} \\ \underline{\text{APIAPR87}} \\ \underline{\text{APIOCT84}} \\ \underline{\text{NPD83}} \end{array} \right\} (\ \underline{\text{PUN}}\text{CHING})\ (\ \underline{\text{SHE}}\text{AR}\ )\ (\ \text{class data}\ )\ -$$

$$(\text{summary spec}) \left\{ \begin{array}{l} \underline{\text{ALL}} \\ (\ \underline{\text{MEM}}\text{BER}\ )\ \text{list}_1 \end{array} \right\}$$

where

$$(\text{class data}) = \underline{\text{AUT}}\text{OMATIC}\ (\left\{ \begin{array}{c} \rightarrow\underline{\text{APIWSD21}} \\ \underline{\text{APILRFD1}} \\ \underline{\text{APIWSD20}} \\ \underline{\text{APIAPR87}} \\ \underline{\text{APIOCT84}} \end{array} \right\})\ (\underline{\text{CLA}}\text{SSIFICATION})\ -$$

$$\left(\underline{\text{JOI}}\text{NTS} \left\{ \begin{array}{l} \text{list}_2 \\ \rightarrow \underline{\text{ALL}} \\ \underline{\text{ALL}}\ \underline{\text{BUT}}\ \text{list}_2 \end{array} \right\} \right) \quad \left(\underline{\text{LOA}}\text{DS} \left\{ \begin{array}{l} \text{list}_3 \\ \rightarrow \underline{\text{ALL}} \\ \underline{\text{ALL}}\ \underline{\text{BUT}}\ \text{list}_3 \end{array} \right\} \right)\ -$$

$$(\ \underline{\text{LIS}}\text{T}\ \underline{\text{CHO}}\text{RDS}\ )\quad(\ \text{joint configuration}\ )$$

$$(\text{joint configuration}) = {}^{*}\left\{ \begin{array}{ll} \underline{\text{OVE}}\text{RLAP} \left\{ \begin{array}{c} \rightarrow -1 \\ 0 \\ v_1 \end{array} \right\} & \\ \underline{\text{DIA}}\text{PHRAGM} & \\ \underline{\text{L}} & v_2 \\ \underline{\text{L1}} & v_3 \\ \underline{\text{L2}} & v_4 \\ \underline{\text{TW}} & v_5 \\ \underline{\text{VW}} & v_6 \end{array} \right\}$$

$$(\text{summary spec}) = \left(\begin{Bmatrix} \underline{\text{WI}}\text{TH } \underline{\text{SUM}}\text{MARIZE } (\underline{\text{RES}}\text{ULTS}) \\ \underline{\text{SUM}}\text{MARIZE } (\underline{\text{RES}}\text{ULTS}) \ \underline{\text{ONL}}\text{Y} \end{Bmatrix} \right)$$

Elements:

$list_1$ = integer or alphanumeric id of the member which check for punching shear is specified.

$list_2$ = integer or alphanumeric id of the joint which automatic classification is specified.

$list_3$ = integer or alphanumeric id of the load which computations for classification is performed.

For joint configuration elements, see Section 4.3.1.

Explanation:

The designations shown in the closed bracket immediately after the CHECK command are the valid code names for the punching shear check. Table 4.4-1 lists the available GTSTRUDL punching shear codes.

The members given in this command refer to those members identified as chords in the CHORDS FOR PUNCHING SHEAR command (Section 4.3).

The AUTOMATIC option specifying the joints ALL to be classified refers to all joints in the structure not classified by explicit chord identifiers. Note that regardless of the code identified immediately following the CHECK command, the chord/brace classification is always as defined by APIWSD21, APILRFD1, APIWSD20, APIAPR87, or APIOCT84 guidelines. The default classification is for all loads. For efficient computer use, you should utilize the LOAD LIST option or the GRPLOAD parameter to limit the loads used for classification and checks.

The CHECK PUNCHING SHEAR command follows the PARAMETERS command; it initiates action on the information previously specified. Consistent with other GTSTRUDL "design options", you will, upon successful execution, receive the results of the checks made without having to ask for any output specifically. The amount and format of this output is dependent upon the parameter TRACE that is active for the CHORD MEMBERS listed. TRACE equal to 4 produces the default output which summarizes the values for the checks made as well as a limited amount of input information related to each chord.

APIWSD21, APILRFD1, and APIWSD20 output:

The default TRACE output for the APIWSD21 and APIWSD20 punching shear check shows the joint and the geometry classification name, the partial classification percentages, the chord and the brace name, the load name, SIN(ANG) (the Sin of the angle between the chord and the brace) and the chord area, the outside diameter and the moment of inertia in the Y direction of the chord, the thickness and the section modulus in the Y direction of the chord, the brace axial force, shear force in the Y direction, and the shear force in the Z direction at the chord connection, the brace torsional moment and the bending moment in the Y and Z direction at the chord connection, the actual punching shear stress due to the brace axial force and the allowable stress, the actual punching shear stress due to the brace in-plane moment and the allowable stress, the actual punching shear stress due to the brace out-of-plane moment and the allowable stress, the actual axial load component perpendicular to the chord and the allowable, the highest actual/allowable ratio and the provision name. The output for APIWSD21, APILRFD1, and APIWSD20 Codes is in the active units.

APIAPR87 and APIOCT84 output:

The default TRACE output for the APIAPR87 and APIOCT84 punching shear check shows the joint and the geometry classification name, the chord and the brace name, the load name and SIN(ANG) (the SIN of the angle between the chord and the brace), the outside diameter and the thickness of the chord, the chord area and the brace axial force at the chord connection, the chord moment of inertia in the Y direction and the brace shear force in the Y direction at the chord connection, the chord section modulus in the Y direction and the shear force in the Z direction at the chord connection, the chord's weight in pound per foot and the brace torsional moment at the chord connection, the brace moments in the Y and Z directions at the chord connection, the actual punching shear stress due to the brace axial force and the allowable stress, the actual punching shear stress due to the brace in-plane moment and the allowable stress, the actual punching shear stress due to the brace out-of-plane moment and the allowable stress, the actual axial load component perpendicular to the chord and the allowable, the highest unity stress check (actual/allowable) ratio and the provision name.

Table 4.4-1

GTSTRUDL Punching Shear Codes

Code Name	Parameter Table	Application
APIWSD21	1.5-1	Based on the Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms - Working Stress Design. API Recommended Practice 2A-WSD (RP 2A-WSD) Twenty-First Edition, December 2000, with Errata and Supplement 1, 2, and 3 (16).
APILRFD1	4.2-1	Based on the Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms Load and Resistance Factor Design. API Recommended Practice 2A-LRFD (RP 2A-LRFD) First Edition, July 1, 1993.
APIWSD20	4.2-2	Based on the Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms - Working Stress Design. API Recommended Practice 2A-WSD (RP 2A-WSD) Twentieth Edition, July 1, 1993 (11).
APIAPR87	E.1-1	Based on the API recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms, Seventeenth Edition, dated April 1, 1987 (2).
APIOCT84	E.1-1	Based on the API recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms, Fifteenth Edition, dated October 22, 1984 (1).
NPD83	E.1-2	Based on the Norwegian Petroleum Directorate (NPD), Guidelines on Design and Analysis of Steel Structures in the Petroleum Activity, dated 1983 (9).

NPD output:

The default TRACE output for the NPD punching shear check shows the joint, the chord name, the outside diameter, the thickness, the chord area, the moment of inertia in the Y direction at the chord connection and SIN(ANG) (the SIN of the angle between the chord and the brace), the chord section modulus in the Y direction and the brace axial force at the chord connection, the chord's weight in pound per foot and the brace shear force in the Y direction at the chord connection, the allowable punching shear stress and the brace shear force in the Z direction at the chord connection, the actual punching shear stress and the torsional moment, the actual/allowable punching shear stress ratio, the brace name and the moment of inertia in the Y direction, the load name and the moment of inertia in the Z direction.

WITH SUMMARIZE RESULTS option indicates that the standard TRACE 4 punching shear output should be printed and then print a summary of the highest actual/allowable punching shear code check ratio. This option is applicable to APIWSD21 and APIWSD20 punching shear code check only. Figure 4.4-1 shows output for this option.

SUMMARIZE RESULTS ONLY option indicates that only print a summary of the highest actual/allowable punching shear code check ratio. Standard TRACE 4 output option is not printed and only summarize results is printed. This option is applicable to APIWSD21 and APIWSD20 punching shear code check only. Figure 4.4-2 shows output for this option.

TRACE 3 provides the user with a full report of the checks made for each CHORD specified in the TRACE member list. Although TRACE 3 output is valuable to your understanding and verification of GTSTRUDL checks, TRACE 3 can be very expensive and counterproductive when specified indiscriminately. SUMMARIZE Command is not available for PUNCHING SHEAR code check command.

```
{  154} >
{  155} > CHECK  APIWSD20  PUNCHING  SHEAR  WITH SUMMARIZE RESULTS  MEMBER  1


        *****************************************************
        *  STRUDL CODE CHECK   OPERATIONAL UNITS  FEET KIP   *
        *****************************************************

  JOINT    CHORD    LOAD       FINAL----SIZE       FX    TORSION    Vp-AX  Vp-IPB   Vp-OPB   ACT-NF   ACTUAL/ALLOWABLE
  GEOMETRY BRACE    SIN(ANG)   OD       TK         FY       MY     Vpa-AX  Vpa-IPB  Vpa-OPB  ALL-NF   PROVISION NAME
  PART               AX        IY       SY         FZ       MZ
=======================================================================================================================

  2        1        1          0.99     0.030     1.93     0.05     0.29     0.30     0.03             0.215      0.825
  T        5        1.0000                        2.49     0.31     5.48     1.20     0.82             ASD F4-1   4.1-1
                    0.092      0.011    0.021    -0.06     2.69

  2        1        1          0.99     0.030    -0.62    -0.33     0.11     0.96     0.18             0.682      0.767
  T        8        1.0000                        3.04    -1.08     5.77     1.20     0.91             ASD F4-1   4.1-1
                    0.092      0.011    0.021    -0.17    -5.93

  2        1        2          0.99     0.030    -2.21     0.11     0.34     0.36     0.06             0.259      0.825
  T        5        1.0000                        0.48     0.58     5.50     1.20     0.83             ASD F4-1   4.1-1
                    0.092      0.011    0.021    -0.11    -3.22

  2        1        2          0.99     0.030    -0.22    -0.45     0.04     0.45     0.35             0.367      0.767
  T        8        1.0000                        1.49    -2.14     5.79     1.20     0.91             4.3.1-3b   4.1-1
                    0.092      0.011    0.021    -0.34    -2.77

  2        1        3          0.99     0.030     0.61    -0.09     0.09     0.34     0.04             0.202      0.825
  T        5        1.0000                        1.02    -0.40     5.48     1.20     0.83             ASD F4-1   4.1-1
                    0.092      0.011    0.021     0.08    -3.07

  2        1        3          0.99     0.030    -0.59     0.27     0.11     0.77     0.26             0.513      0.767
  T        8        1.0000                        2.33     1.58     5.78     1.20     0.91             ASD F4-1   4.1-1
                    0.092      0.011    0.021     0.25    -4.74

=======================================================================================================================

Maximum Punching Shear Summary Results
-------------------------------------

  Joint      Chord      Brace      Loading    Geometry   Provision    Unity Ratio   Provision    Unity Ratio   Status
  -----      -----      -----      -------    --------   ---------    -----------   ---------    -----------   ------

  2          1          5          2          T          ASD F4-1       0.259       4.1-1          0.825      Passed
  2          1          8          1          T          ASD F4-1       0.682       4.1-1          0.767      Passed
```

Figure 4.4-1 GTSTRUDL output for punching shear With Summarize Results

```
{  156} >
{  157} > CHECK  APIWSD20  PUNCHING  SHEAR  SUMMARIZE RESULTS ONLY  MEMBER  1


        *****************************************************
        *  STRUDL CODE CHECK   OPERATIONAL UNITS  FEET KIP   *
        *****************************************************


Maximum Punching Shear Summary Results
-------------------------------------

  Joint      Chord      Brace      Loading    Geometry   Provision    Unity Ratio   Provision    Unity Ratio   Status
  -----      -----      -----      -------    --------   ---------    -----------   ---------    -----------   ------

  2          1          5          2          T          ASD F4-1       0.259       4.1-1          0.825      Passed
  2          1          8          1          T          ASD F4-1       0.682       4.1-1          0.767      Passed
```

Figure 4.4-2 GTSTRUDL output for punching shear Summarize Results Only

This page intentionally left blank.

5.2.3 LRFD3 Steel Design Code

The LRFD3 code is primarily based on the AISC "Load and Resistance Factor Design Specification for Structural Steel Buildings" adopted December 27, 1999 with errata incorporated as of September 4, 2001. The Specification is contained in the Third Edition of the AISC Manual of Steel Construction, Load and Resistance Factor Design (96). The LRFD3 code utilizes the Load and Resistance Factor design techniques of the AISC Specification. Rather than use the LRFD3 code which is a prerelease feature, users should use the AISC13 code which is now a released feature.

The LRFD3 code of GTSTRUDL may be used to select or check any of the following shapes:

Design for bi-axial bending and axial forces:

I shapes	Round Bars
Channels	Square Bars
Single Angles	Rectangular Bars
Tees	Plate Girders
Double Angles	

Design for bi-axial bending, axial, and torsional forces:

Round HSS (Pipes)
Rectangular and Square HSS (Structural Tubes)

The documentation for the LRFD3 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and LRFD3 in the GTSTRUDL Output Window.

5.2.4 ACI Code 318-99

Design of beams and columns by the 1999 ACI code has been added. Only members designated as TYPE BEAM or TYPE COLUMN in a DESIGN DATA command can be PROPORTIONed when the METHOD is set to ACI318-99. When you specify ACI318-99, you will be reminded that it is a pre-release feature by a message (see the Example below).

Note that CHECK is not available for codes after ACI318-77, including ACI318-99.

$$\underline{\text{METH}}\text{OD} \; (\left\{ \begin{array}{l} \rightarrow \; \underline{\text{ULT}}\text{IMATE} \; (\underline{\text{STR}}\text{ENGTH}) \\ \quad \underline{\text{WORK}}\text{ING} \; (\underline{\text{STR}}\text{ESS}) \end{array} \right\}) \left\{ \begin{array}{l} \underline{\text{ACI318-99}} \\ \underline{\text{ACI318-89}} \\ \underline{\text{ACI318-83}} \\ \underline{\text{ACI318-77}} \\ \underline{\text{ACI318-63}} \\ \underline{\text{(BSI)}} \; \underline{\text{CP110-72}} \\ \underline{\text{(BSI)}} \; \underline{\text{BS8110}} \end{array} \right\} -$$

$$(\; \underline{\text{BARS}} \left\{ \begin{array}{l} \rightarrow \; \underline{\text{ASTM}} \\ \quad \underline{\text{CAN}}\text{ADIAN} \; (\underline{\text{STA}}\text{NDARD}) \\ \quad \underline{\text{UNE}}\text{SCO} \\ \quad \underline{\text{KOR}}\text{EAN} \; (\underline{\text{S}}\text{TANDARD}) \end{array} \right\}) \; (\left\{ \begin{array}{l} \rightarrow \; \underline{\text{NONS}}\text{EISMIC} \\ \quad \underline{\text{SEIS}}\text{MIC} \\ \quad \underline{\text{MODE}}\text{RATE} \; \underline{\text{SEIS}}\text{MIC} \end{array} \right\})$$

Example:

```
METHOD ACI318-99
  ****INFO_MET – 318-99 is a pre-release feature.

          DESIGN DATA FOR MEMBER 1
            TYPE BEAM RECT
          PROPORTION MEMBER 1
                  ....
          ACTIVE CODE  =  ACI 318-99
                  ....
                  (the rest of the output is the same format as previous codes)
```

The table of CONSTANTS and assumed values for ACI 318-99 is shown below:

TABLE 2.4-1. CONSTANTS and Assumed Values for ACI 318-99

CONSTANT	Explanation	ACI 318-99	Assumed Value
FCP	Compressive strength of concrete, f'_c		4000 psi
FY	Yield strength of reinforcement, f_y		60000 psi
WC	Unit weight of plain concrete		145 pcf
DENSITY	Unit weight of reinforced concrete (1)		150 pcf
FC	Allow compr. stress in concrete, F_c	A.3.1	0.45(FCP)
VU	Ult. shear stress in beam with web reinf. (2)	11.5.6.9	$(8\sqrt{FCP}+v_c)$ (5)
V	Allow. shear stress in beam with web reinf.	A.3.1(b)	$(5.5\sqrt{FCP})$
RFSP	Splitting ratio, $f_{ct}/(\sqrt{f'_c})$ (3)	9.5.2.3	6.7
FYST	Yield strength of stirrups		60000 psi
FYSP	Yield strength of spiral		60000 psi
FS	Allowable tension stress in primary reinf.		20000 psi for
FSC	Allowable compressive stress in column reinf. (4)	A.3.2	Grades 40, 50
FV	Allowable tension stress in stirrups (5)		24000 psi for
			Grade 60
PHIFL	Flexure capacity reduction factor	9.3.2	0.9
PHISH	Shear capacity reduction factor	9.3.2	0.85
PHIBO	Bond capacity reduction factor	9.3.2	0.85
PHITO	Torsion capacity reduction factor	9.3.2	0.85
PHISP	Spiral column capacity reduction factor	9.3.2	0.75
PHITI	Tied column capacity reduction factor	9.3.2	0.7
BLFR	Ratio of max p, $(p - p')$ or $(p_w - p_f)$ to p_{bal}	10.3.3	0.75
PMAXCO	Maximum allowable reinforced ratio in columns	10.9.1	0.08
PMINCO	Minimum allowable reinforced ratio in columns	10.9.1	0.01
PMINFL	Minimum allowable reinforced ratio in flexural members	10.5.1	200/FY
ES	Modulus of elasticity for reinf. steel	8.5.2	29×10^6 psi
EC	Modulus of elasticity for concrete	8.5.1	$33(WC)^{1.5}\sqrt{FCP}$
EU	Ult. strain in concrete at extreme comp. fiber	10.2.3	0.003

Notes:

1. The constant 'DENSITY' is the GTSTRUDL constant of the same name which has been set to a value of 150 pcf for reinforced concrete.

2. VU is multiplied by PHISH internally.

3. Calculations for V_c and T_c are modified by replacing $\sqrt{f_c'}$ with RFSP/6.7($\sqrt{f_c'}$) as per Section 11.2.1.1.

4. The assumed value of FSC is also limited to 30,000 psi maximum.

5. This value is defined only at the time of stirrup design.

5.2.5 Rectangular and Circular Concrete Cross-Section Tables

New tables have been added for rectangular and circular concrete cross sections. The new table for rectangular sections is called CONRECT and the new table for circular sections is called CONCIR. These tables are added to facilitate the modeling and analysis of concrete cross sections but may not be used in the design of concrete cross sections. In order to design concrete sections, the MEMBER DIMENSION command must be used (see Section 2.5 of Volume 4 of the GTSTRUDL User Reference Manual).

The profiles in the CONCIR table are shown below where the name CIRxx indicates a circular cross section and xx is the diameter in inches. Thus, CIR12 is a 12 inch diameter circular cross section.

CIR12	CIR24
CIR14	CIR26
CIR16	CIR28
CIR18	CIR30
CIR20	CIR32
CIR22	CIR34
	CIR36

The profiles in the CONRECT table are shown below where the name RECYYXZZ indicates a rectangular cross section with a width of YY inches and a depth of ZZ inches. Thus, REC16X24 is 16 inch wide and 24 inch deep rectangular cross section.

REC6X12	REC8X12	REC10X12	REC12X12	REC14X12	REC16X12
REC6X14	REC8X14	REC10X14	REC12X14	REC14X14	REC16X14
REC6X16	REC8X16	REC10X16	REC12X16	REC14X16	REC16X16
REC6X18	REC8X18	REC10X18	REC12X18	REC14X18	REC16X18
REC6X20	REC8X20	REC10X20	REC12X20	REC14X20	REC16X20
REC6X22	REC8X22	REC10X22	REC12X22	REC14X22	REC16X22
REC6X24	REC8X24	REC10X24	REC12X24	REC14X24	REC16X24
REC6X26	REC8X26	REC10X26	REC12X26	REC14X26	REC16X26
REC6X28	REC8X28	REC10X28	REC12X28	REC14X28	REC16X28
REC6X30	REC8X30	REC10X30	REC12X30	REC14X30	REC16X30
REC6X32	REC8X32	REC10X32	REC12X32	REC14X32	REC16X32
REC6X34	REC8X34	REC10X34	REC12X34	REC14X34	REC16X34
REC6X36	REC8X36	REC10X36	REC12X36	REC14X36	REC16X36

REC18X12	REC20X12	REC22X12	REC24X12	REC26X12	REC28X12
REC18X14	REC20X14	REC22X14	REC24X14	REC26X14	REC28X14
REC18X16	REC20X16	REC22X16	REC24X16	REC26X16	REC28X16
REC18X18	REC20X18	REC22X18	REC24X18	REC26X18	REC28X18
REC18X20	REC20X20	REC22X20	REC24X20	REC26X20	REC28X20
REC18X22	REC20X22	REC22X22	REC24X22	REC26X22	REC28X22
REC18X24	REC20X24	REC22X24	REC24X24	REC26X24	REC28X24
REC18X26	REC20X26	REC22X26	REC24X26	REC26X26	REC28X26
REC18X28	REC20X28	REC22X28	REC24X28	REC26X28	REC28X28
REC18X30	REC20X30	REC22X30	REC24X30	REC26X30	REC28X30
REC18X32	REC20X32	REC22X32	REC24X32	REC26X32	REC28X32
REC18X34	REC20X34	REC22X34	REC24X34	REC26X34	REC28X34
REC18X36	REC20X36	REC22X36	REC24X36	REC26X36	REC28X36

REC30X12	REC32X12	REC34X12	REC36X12
REC30X14	REC32X14	REC34X14	REC36X14
REC30X16	REC32X16	REC34X16	REC36X16
REC30X18	REC32X18	REC34X18	REC36X18
REC30X20	REC32X20	REC34X20	REC36X20
REC30X22	REC32X22	REC34X22	REC36X22
REC30X24	REC32X24	REC34X24	REC36X24
REC30X26	REC32X26	REC34X26	REC36X26
REC30X28	REC32X28	REC34X28	REC36X28
REC30X30	REC32X30	REC34X30	REC36X30
REC30X32	REC32X32	REC34X32	REC36X32
REC30X34	REC32X34	REC34X34	REC36X34
REC30X36	REC32X36	REC34X36	REC36X36

5.2.6 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

The goal of the DESIGN SLAB command is to select reinforcing steel for concrete flat plate systems using finite elements as a tool for the determination of design moments.

Instead of dealing with results on an element-by-element basis, the user will be able to design the reinforcing steel for slab systems based on cuts. Here, the term *cut* refers to the cross-section of a strip at a particular location to be designed. A cut is defined by two nodes identifying the start and end of the cut, and by an element in the plane of the cut.

Once the definition of the cut has been determined, the resultant forces along the cut are computed using either moment resultants (otherwise known as the Wood and Armer method) or element force results (using the CALCULATE RESULTANT command, as described in Section 2.3.7.3 of Volume 3 of the GTSTRUDL User Reference Manual). The final design moment is determined by computing the resultant moment acting on the cut for each loading condition, and reducing these moments to a design envelope.

Once the design envelope is computed, the cross-section is designed according to ACI 318-05 either using default design parameter or with certain user specified design parameters such as the bar size or spacing.

An important distinction is to note that each cut is designed independently from all other cuts. That is, a cut specified in one region is independent with respect to a design in another region. As such, if the user wishes to use the same bar size over multiple adjacent cuts, this information must be specified for each cut.

The form of the command is as follows:

$$\underline{\text{DES}}\text{IGN }\underline{\text{SLAB}}\text{ (}\underline{\text{REIN}}\text{FORCEMENT) (}\underline{\text{US}}\text{ING)} -$$

$$\left\{\begin{array}{l}\underline{\text{WOOD}}\text{ (}\underline{\text{AND}}\text{) (}\underline{\text{ARM}}\text{ER)}\left\{\begin{array}{l}\rightarrow \underline{\text{AVE}}\text{RAGE} \\ \quad\underline{\text{MAX}}\text{IMUM}\end{array}\right\} \\ \underline{\text{CALC}}\text{ULATE (}\underline{\text{RES}}\text{ULTANT) (}\underline{\text{ELE}}\text{MENT) (}\underline{\text{FOR}}\text{CES)}\end{array}\right\}\text{ (}\underline{\text{AL}}\text{ONG)} -$$

$$\text{(}\underline{\text{CUT}}\left\{\begin{array}{c}\text{'a'} \\ i_1\end{array}\right\}\text{)}\left\{\begin{array}{l}\underline{\text{JOI}}\text{NTS} \\ \underline{\text{NODE}}\text{S}\end{array}\right\}\text{ list}_1\ \underline{\text{ELE}}\text{MENT list}_2\text{ (}\underline{\text{TABLE}}\left\{\begin{array}{l}\rightarrow \underline{\text{ASTM}} \\ \quad\underline{\text{UNESCO}}\end{array}\right\}\text{)} -$$

$$^{*}\left\{\begin{array}{l}\underline{\text{TOP}}\text{ (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S } i_2\text{) (}\underline{\text{SPA}}\text{CING } v_1\text{)} \\ \underline{\text{BOT}}\text{TOM (}\underline{\text{FACE}}\text{) (}\underline{\text{BAR}}\text{S } i_3\text{) (}\underline{\text{SPA}}\text{CING } v_2\text{)} \\ \underline{\text{BOTH}}\text{ (}\underline{\text{FACE}}\text{S) (}\underline{\text{BAR}}\text{S } i_4\text{) (}\underline{\text{SPA}}\text{CING } v_3\text{)}\end{array}\right\} -$$

$$\left\{\begin{array}{l}\rightarrow \underline{\text{IN}}\text{NER (}\underline{\text{LAY}}\text{ER)} \\ \quad\underline{\text{OUT}}\text{ER (}\underline{\text{LAY}}\text{ER)}\end{array}\right\}\text{ (}\underline{\text{COV}}\text{ER } v_4\text{) (}\underline{\text{LIN}}\text{EAR (}\underline{\text{TOL}}\text{ERANCE) } v_5\text{)} -$$

$$\text{(}\underline{\text{TOR}}\text{SIONAL (}\underline{\text{MOM}}\text{ENT) (}\underline{\text{WARN}}\text{ING) } v_6\text{)}$$

where,

‘a’ or i_1 refer to an optional alphanumeric or integer cut name

list_1	=	list containing ID’s of the start and end node of the cut
list_2	=	list containing the ID of an element in the plane of the cut
i_2	=	bar size to be used for bars on the top surface of the slab
i_3	=	bar size to be used for bars on the bottom surface of the slab
i_4	=	bar size to be used for both the top and bottom surfaces of the slab
v_1	=	reinforcing bar spacing to be used on the top surface of the slab
v_2	=	reinforcing bar spacing to be used on the bottom surface of the slab
v_3	=	reinforcing bar spacing to be used on both surfaces of the slab
v_4	=	optional user-specified cover distance for reinforcing bars
v_5	=	linear tolerance used in element selection rules for moment computation
v_6	=	optional ratio of torsion to bending moment allowed on the cross-section
TOP	=	element surface with +Z PLANAR coordinate
BOTTOM	=	element surface with -Z PLANAR coordinate

Explanation:

The DESIGN SLAB command allows the user to communicate all data necessary for the reinforcing steel design. This information is processed and a design is calculated based on the input. The command is designed to provide varying levels of control for the user so as to make the command as broadly applicable as possible.

The user must first define the cut. A cut is defined by a start and end node ID, and an element ID in the plane of the cut. The user has the option of giving each cut an alphanumeric name for organizational purposes. The purpose of the required element ID is to determine the appropriate plane to design in the event that multiple planes of finite elements intersect along the cut, as defined by the start and end node. An example where this might occur is the intersection of a slab with a shear wall. In this case, a misleading design could be generated if the slab was designed using the forces in the shear wall. The cut definition constitutes all information required to compute the resultant forces acting along the cut.

The total moment acting on a cut cross-section is computed using one of two methods. The use of moment resultants, also known as the Wood and Armer method, is implemented as the default method. In this method, the moment resultants MXX, MYY, and MXY are resolved on a per node basis along the cut, and either the average effect or the maximum effect on the cut is applied to the entire cross-section.

The other option for moment computation is based on the use of element forces. In this method, the total resultant moment acting on the cross-section is computed using the CALCULATE RESULTANT command, and the element force nodal moments are resolved for each node of each element adjacent to the cut.

Once the cut has been defined, the user may indicate parameters to be used to design the system. The user may constrain the bar size or spacing to a certain value, either for the top face, bottom face, or for both faces. In this case, the final design will utilize the information provided. If the bar size is constrained, the appropriate spacing of bars is determined. If the bar spacing is constrained, the appropriate bar size is determined. In the case that the user supplies a bar size and spacing for the cut, the application will simply check the strength of the cross-section against the computed design envelope according to ACI 318. If the user specifies no design constraints, the application assumes a bar size and designs the section to satisfy ACI 318. As such, the user maintains explicit control over the function of the application.

The user may also specify which layer of bars to be designed, using the modifier INNER or OUTER. These refer to the location of reinforcing bars on each surface. At most slab locations, reinforcement is placed in two perpendicular directions on both surfaces of the slab. Since each layer of reinforcement cannot occupy the same space, one layer must be placed on top of the other. OUTER refers to the layer closest to the surface, while INNER refers to the layer nearest the center of the slab.

All user-specified constraints, such as concrete compressive strength, yield strength, cover, and spacing are checked against ACI minimum/maximum values, as specified in ACI 318-02. The thickness of the cross-section is determined internally based on the modeled thickness of the user-specified element.

With respect to the interpretation of results, “top” always refers to the face of the slab on the +Z PLANAR side of the element, and “bottom” always refers to the face of the slab on the -Z PLANAR side of the element. “Positive bending” refers to bending that produces tension on the bottom face of the slab and compression on the top face, as defined previously. “Negative bending” produces tension on the top face and compression on the bottom face, as defined previously.

Requirements:

The MATERIAL REINFORCED CONCRETE command must be specified before the DESIGN SLAB. The MATERIAL REINFORCED CONCRETE command initializes the RC capabilities of GT STRUDL and sets the relevant material and design quantities to their default values for design. At this point, the user can issue the CONSTANTS command to modify any material properties to be used in the design. The default values are:

ECU	=	0.003
ES	=	29,000,000 psi
FCP	=	4000 psi
FY	=	60,000 psi
PHIFL	=	0.9

The STIFFNESS command must be issued prior to the DESIGN SLAB command. The STIFFNESS command solves the global equilibrium equation and computes the quantities required for the determination of the bending moments that the DESIGN SLAB command uses.

Only elements known to appropriately model the behavior of slab systems are included in the computation of design forces. For a flat plate system, only plate bending and plate elements are used. Thus, if the user models the system using plane stress / plane strain elements, and then issues the DESIGN SLAB command, a warning message is output and the command is ignored.

Plate bending elements supported include the BPHT, BPR, BPHQ, CPT, and IPBQQ finite elements. General plate elements supported include the SBCT, SBCR, SBHQ, SBHQCSH, SBHT, SBHT6, and SBHQ6 finite elements.

Usage:

Studies have shown that the CALCULATE RESULTANT ELEMENT FORCE option of the DESIGN SLAB command is only applicable in regions where the cut orientation is generally orthogonal to the directions of principle bending. If the geometry of a region dictates that a cut be oriented non-orthogonally to the principal bending directions, a significant torsional effect may occur. In this case, the Wood and Armer method must be employed due to its ability to correctly compute the ultimate moment in a strong torsion field. In the DESIGN SLAB command, the user is warned if the element force implementation computes a resultant torsion greater than 10% of the resultant bending moment on a particular cross-section. The user may modify the torsion warning threshold via the modifiers TORSIONAL MOMENT WARNING. If there is any question of the orientation of the cut with respect to the directions of principal bending, the user should investigate the behavior in the finite element results section of GTMENU.

Usage Example: Description of Example Structure

The example structure is a rectangular slab system, shown in Figure 5.2.5-1. The clear span of the structure is thirty feet, and the slab strip has a width of ten feet. The two ends of the slab are fully fixed, while the thirty foot sides are free, resembling a fixed-fixed beam. The slab is one foot thick and constructed of normal strength concrete with FCP = 4000 psi. The example structure can be idealized as a subset of a larger slab system, perhaps the design strip running between two column faces in an interior region. The structure is loaded with a distributed surface pressure of 150 psf over the entire surface of the slab.

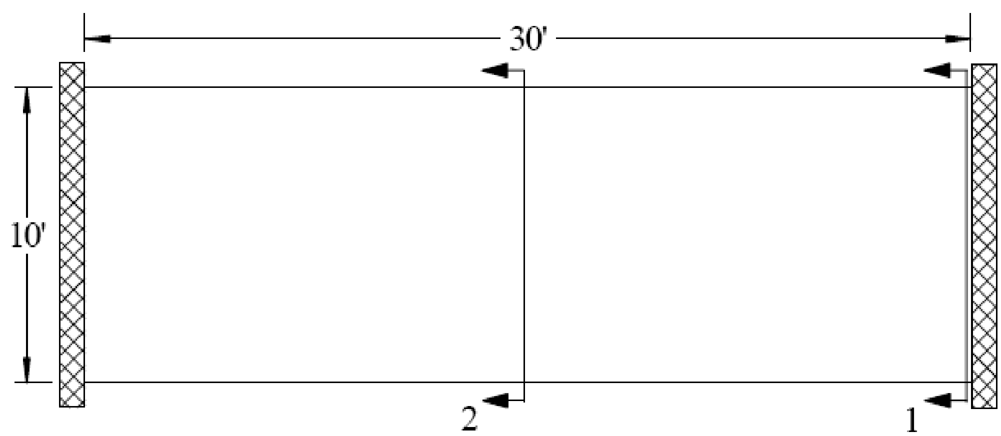
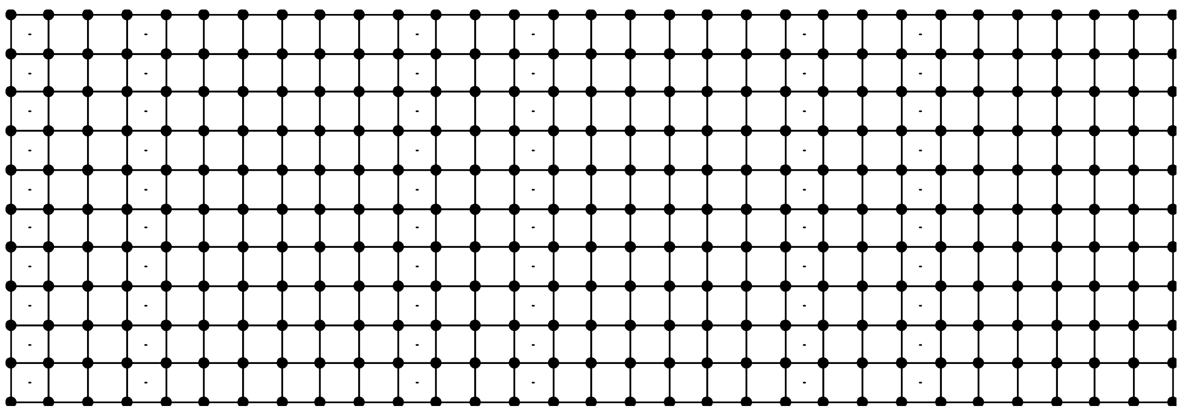


Figure 5.2.5-1 Example Flat Plate Structure (PLAN)

GT STRUDL Finite Element Model

The example structure was modeled in GT STRUDL using PLATE BENDING finite elements. The BPHQ element was utilized, and the configuration modeled corresponded to a mesh of ten elements by thirty elements. The model contained 300 finite elements and 341 nodes. The material properties were the default values associated with the MATERIAL REINFORCED CONCRETE command. All 6 degrees of freedom were restrained at each node along the supported ends of the slab system. Each element was loaded with a surface pressure of 150 psf, resulting in a confirmed summation of vertical reaction of 45,000 lb.

Figure 5.2.5-2 Example Finite Element Model



Definition of Cut Cross-Sections

Two “cuts” are considered for the verification example, as shown in Figure 5.2.5-1.

Cut 1-1:

The cross-section Cut 1-1 is defined along the fixed support at the end of the slab strip and represents the maximum “negative moment” section in the slab where top reinforcing steel would be required. Cut 1-1 originates at node #1 and terminates at node #11. The elements along Cut 1-1 are elements #1-#10. The command given for Cut 1-1 is:

“DESIGN SLAB USING CALCULATE RESULTANT JOI 1 11 ELE 1 TOP BAR 5"

In this case, the user requests that a slab cross-section beginning at node #1, ending at node #11, and in the plane of element #1 be reinforced according to the section moment computed using the CALCULATE RESULTANT command. The user has specified that #5 bars are to be used on the top surface, indicating that spacing is to be computed. The results of the DESIGN SLAB command are shown in the following table.

Calculation	Surface	Bar	Spacing	Area Prov.	Moment Strength	Moment Required
		#	in	sq. in.	lb-in	lb-in
DESIGN SLAB	Top	5	13.0	2.862	1561006.4	1354381.5
DESIGN SLAB	Bottom	NA	NA	NA	NA	NA

The GTSTRUDL output for this example is as follows:

```
** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

     PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS


     RELEVANT ACTIVE UNITS:  INCH LB

     NUMBER OF ACTIVE LOADINGS:     1

     REINFORCEMENT ORIENTATION PERPENDICULAR TO A CUT BEGINNING AT NODE 1
       AND TERMINATING AT NODE 11       AND IN THE PLANE OF ELEMENT 1


** ELEMENT FORCE IMPLEMENTATION **


** DESIGN MOMENT ENVELOPE **

     NEGATIVE MOMENT =      -1354381.48   DUE TO LOAD   150psf
     POSITIVE MOMENT =             0.00   DUE TO LOAD   (none)

     NOTE:
      - Negative moment produces tension on the positive PLANAR Z surface, requiring TOP
        bars.
      - Positive moment produces compression on the positive PLANAR Z surface, requiring
        BOTTOM bars.


** SLAB CROSS-SECTION **

     Width        Depth          FCP           FY         Cover        Layer
    ________________________________________________________________________

    120.00       12.00        4000.00       60000.00      0.750        Inner


** DESIGN RESULTS (per ACI 318-05) **

     Face       Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS
__________________________________________________________________________________________

     TOP        # 5    13.000      2.862        1561006.4280    1354381.4844    PASSES

     BOTTOM            ( Reinforcement Not Required )
```

Cut 2-2:

The cross-section Cut 2-2 is defined along the center line in the middle region of the slab strip and represents the maximum “positive moment” section in the slab where bottom reinforcing steel would be required. Cut 2-2 originates at node #166 and terminates at node #176. The elements along Cut 2-2 are elements #141-#150 on one side and #151-#160 on the other side. The command given for Cut 2-2 Case 1 is:

“DESIGN SLAB WOOD AND ARMER JOI 166 176 ELE 141 TABLE UNESCO BOTTOM SPACING 10 OUTER LAYER"

In this case, the user requests that a slab cross-section beginning at node #166, ending at node #176, and in the plane of element #141 be reinforced according to the average effect produced by the Wood and Armer method. The user has specified that UNESCO metric reinforcing bars are to be used. The bottom reinforcement spacing has been constrained to 10 inches, and the reinforcement to be designed is located in the outer layer. The results of the DESIGN SLAB command are shown in the following table:

Calculation	Surface	Bar	Spacing	Area Prov.	Moment Strength	Moment Required
		#	in	sq. in.	lb-in	lb-in
DESIGN SLAB	Bottom	M14	10.0	2.864	1664920.7	671358.2
DESIGN SLAB	Top	NA	NA	NA	NA	NA

The GTSTRUDL output for this example is as follows:

```
  ** FLAT PLATE SLAB DESIGN BASED ON THE RESULTS OF FINITE ELEMENT ANALYSIS **

       PROBLEM - VFE103    TITLE - DESIGN SLAB VERIFICATION - VERIFY DESIGN CALCULATIONS


       RELEVANT ACTIVE UNITS:  INCH LB

       NUMBER OF ACTIVE LOADINGS:     1

       REINFORCEMENT ORIENTATION PERPENDICULAR TO A CUT BEGINNING AT NODE 166
         AND TERMINATING AT NODE 176      AND IN THE PLANE OF ELEMENT 141


  ** WOOD & ARMER IMPLEMENTATION **

       Design using average result acting on section.


  ** DESIGN MOMENT ENVELOPE **

       NEGATIVE MOMENT =            0.00   DUE TO LOAD   150psf
       POSITIVE MOMENT =       671358.19   DUE TO LOAD   150psf


NOTE:
       - Negative moment produces tension on the positive PLANAR Z surface, requiring TOP
bars.
       - Positive moment produces compression on the positive PLANAR Z surface, requiring
BOTTOM bars.


  ** SLAB CROSS-SECTION **

       Width        Depth          FCP          FY        Cover        Layer
      _______________________________________________________________________

      120.00       12.00        4000.00      60000.00     0.750        Outer


  ** DESIGN RESULTS (per ACI 318-05) **

       Face       Bar    Spacing   AS PROV'D    MOMENT STRENGTH    MOMENT REQ'D    STATUS

__________________________________________________________________________________________

       TOP               ( Reinforcement Not Required )

       BOTTOM     M14    10.000      2.864        1664920.7190     671358.1875     PASSES
```

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

$$\underline{\text{CAL}}\text{CULATE } \underline{\text{ERR}}\text{OR } (\underline{\text{EST}}\text{IMATE}) \ (\underline{\text{BAS}}\text{ED } \underline{\text{O}}\text{N}) \ -$$

$$* \left\{ \begin{array}{l} \underline{\text{ENERGY}} \ (\underline{\text{NORM}}) \\ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \\ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{PERCENT}} \ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \\ \underline{\text{NORM}}\text{ALIZED} \ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFF}}\text{ERENCE} \\ \underline{\text{NORM}}\text{ALIZED} \ \underline{\text{PERCENT}} \ \underline{\text{DIFF}}\text{ERENCE} \ \underline{\text{FROM}} \ \underline{\text{AVER}}\text{AGE} \end{array} \right\} \ -$$

$$(\underline{\text{AT}}) \ {}^{*}\left\{ \begin{array}{l} \underline{\text{TOP}} \\ \underline{\text{MID}}\text{DLE} \\ \underline{\text{BOT}}\text{TOM} \end{array} \right\} (\underline{\text{SUR}}\text{FACES}) \ (\underline{\text{FOR}}) \ \left\{ \begin{array}{l} \rightarrow \underline{\text{ALL}} \\ \\ \underline{\text{ELE}}\text{MENT list} \end{array} \right\}$$

The results from this command provide an estimate of the errors in the finite element discretization of the problem. Energy norm (L_2 norm) and nodal error estimates are available.

The L_2 norm is given by:

$$\left\| e_\sigma \right\|_{L2} = \left(\int_\Omega (e_\sigma)^T (e_\sigma) \, d\Omega \right)^{1/2}$$

where e_σ is the error in stress and Ω is the domain of the element. The error stress is the difference between the average stress, σ^*, and element stress at the nodes, σ. The stress norm is obtained by using the shape functions used for displacements, thus,

$$\| e_\sigma \|_{L2} = \left(\int_\Omega (\sigma^* - \sigma)^T N^T \cdot N (\sigma^* - \sigma) \, d\Omega \right)^{1/2}$$

where N is the shape functions used for the assumed displacement field of the element.

The stress norm uses the average stresses and is given by:

$$\|\sigma\|_{L2} = \left(\int_{\Omega} \left(\sigma^*\right)^T N^T \cdot N\left(\sigma^*\right) d\Omega \right)^{1/2}$$

The relative percentage error which is output for each element is given by:

$$\eta = \frac{\|e_\sigma\|}{\|\sigma\| + \|e_\sigma\|} \times 100$$

The nodal error estimates estimate the accuracy of the data in a selected nodal output vector. Six nodal error estimation methods are available:

- Maximum Difference.
- Difference from Average.
- Percent Maximum Difference.
- Percent Difference from Average.
- Normalized Percent Maximum Difference.
- Normalized percent Difference from Average.

These error estimates look at the variations in stresses at the nodes. An error estimate of nodal output data will be based on the gradients that data produces in each element. That is, how the data varies across that node based on the different data values from the elements connected at that node. The calculation of error estimates for nodal output is fairly straightforward, the values at each node connected at an element are simply compared. The six nodal error measures are outlined in more detail below:

Maximum Difference Method

$$\left| \text{Value}_{\text{Max}} - \text{Value}_{\text{Min}} \right|$$

Difference from Average Method

$$\text{MAX}\ (\left| \text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}} \right|, \left| \text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}} \right|)$$

Percent Maximum Difference Method

$$\left| \frac{\text{Value}_{\text{Max}} - \text{Value}_{\text{Min}}}{\text{Value}_{\text{Avg}}} \right| \times 100\%$$

Percent Difference from Average Method

$$\frac{\text{MAX}\left(\left| \text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}} \right|, \left| \text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}} \right| \right)}{\left| \text{Value}_{\text{Avg}} \right|} \times 100\%$$

Normalized Percent Maximum Difference

$$\left| \frac{\text{Value}_{\text{Max}} - \text{Value}_{\text{Min}}}{\text{Value}_{\text{VectorMax}}} \right| \times 100\%$$

Normalized Percent Difference from Average Method

$$\frac{\text{MAX}\left(\left| \text{Value}_{\text{Max}} - \text{Value}_{\text{Avg}} \right|, \left| \text{Value}_{\text{Min}} - \text{Value}_{\text{Avg}} \right| \right)}{\left| \text{Value}_{\text{VectorMax}} \right|} \times 100\%$$

In each of these calculations, the “Min”, “Max”, and “Avg” values refer to the minimum, maximum, and average output values at the node. The “Vector Max” values refer to the maximum value for all nodes in the output vector. All error estimates are either zero or positive, since all use the absolute value of the various factors.

The choice of an appropriate error estimation method largely depends on the conditions in the model. As many error estimates as required may be calculated. In general, the Max Difference method is good at pointing out the largest gradients in the portions of your model with the largest output values. The Difference from Average Method will also identify areas with the largest output values. In this case however, areas where only one or a few values are significantly different will be accentuated. The Max Difference method will identify the steepest gradients in the most critical portions of your model. The Difference from Average Method will identify just the steepest non-uniform gradients, the ones that vary in only a single direction. The two percentage methods identify the same type of gradients, but do not make any distinction between large and small output values. These methods are to be used only if the magnitude of the output is less important than the changes in output. The two percentage methods estimate the error as a percent of the average stress. However, at nodes where there is a change in sign of the stress, the average stress becomes very small and often close to zero. As a result, the value of the error becomes enormous. In order to quantify this error, the error at such nodes is given a value of 1,000 percent. The final two normalized percentage methods are usually the best at quantifying overall errors in area with peak stress values.

The results produced by the CALCULATE ERROR ESTIMATE command may also be contoured in GTMenu. To produce a contour of the error estimate in GTMenu, follow the steps below after performing a STIFFNESS ANALYSIS for a static loading:

1. Enter GTMenu.

2. Select Results, Finite Element Contours, and then Energy & Stress Error Estimates.

3. Select the Estimate Method including Value, Surface, and Stress Component.

4. Select the Loading.

5. Select Display (solid colors or lines) to produce a contour of the error estimate.

6. Select Legend to place a legend on the screen indicating the type of error estimate, loading, and surface.

5.4 General Prerelease Features

5.4.1 ROTATE LOAD Command

The ROTATE LOAD command will rotate an existing loading and create a new loading condition in order to model a different orientation of the structure or the loading. The ROTATE command is described below and is numbered as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.11.4.6 The ROTATE LOAD Command

General form:

$$\underline{\text{ROT}}\text{ATE}\ \ \underline{\text{LOA}}\text{DING}\ \left\{\begin{matrix} i_R \\ \\ 'a_R\ ' \end{matrix}\right\}\ (\ \underline{\text{ANG}}\text{LES}\ )\left[\underline{\text{T1}}\right] r_1 \left[\ \underline{\text{T2}}\ \right] r_2 \left[\ \underline{\text{T3}}\ \right] r_3$$

Elements:

i_R/'a_R' = integer or alphanumeric name of the existing independent loading condition whose global components are to be rotated.

r_1, r_2, r_3 = values in current angle units of the load component rotation angles θ_1, θ_2, θ_3 as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

In many instances, loading conditions are defined for a structure having a given orientation in space, but then the same structure may need to be analyzed for different additional orientations. Applied loading components that are defined with respect to local member or element coordinate systems remain unchanged regardless of the structure's orientation. However, loading components that are defined with respect to the global coordinate system may need to be rotated in order to properly reflect a new orientation for the structure. This is particularly true for self-weight loads, buoyancy loads, etc.

The ROTATE LOADING command is used to take the global applied loading components from an existing loading condition, rotate them through a set of rotation angles, and copy the new rotated global components to a new or modified different destination loading condition. The existing independent loading condition, the ROTATE load, from which the rotated global load components are computed is specified by the loading name i_R/'a_R'. The angles of rotation are specified by the values r_1, r_2, r_3. These rotation angles are defined according to the same conventions as those that define the local support release directions in the JOINT RELEASE command described in Section 2.1.7.2, Volume 1 of the GTSTRUDL User Reference Manual, and illustrated in Figure 2.1.7-1.

The ROTATE LOADING command is always used in conjunction with one of the following loading definition commands: LOADING, DEAD LOAD, and FORM LOAD. These commands will define the name (and title) of a new or existing destination loading condition into which the ROTATE LOADING results are copied. The ROTATE LOADING command may be given with any additional applied loading commands such as JOINT LOADS, MEMBER LOADS, ELEMENT LOADS, etc.

Taking the specified loading i_R/'a_R', the ROTATE LOADING command performs the following operations and copies the results into the destination loading condition:

1. Rotate all joint loads, including applied joint support displacements.
2. Rotate all member force and moment loads defined with respect to the global coordinate system. Member force and moment loads defined with respect to the member local coordinate system are simply copied without rotation.
3. Rotate all element force loads defined with respect to the global coordinate system. Element force loads defined with respect to any applicable local or planar coordinate systems are copied without rotation.
4. All other types of loads such as member temperature loads, member distortions, joint temperatures, etc. are copied without changes.

Examples:

1.
```
UNITS DEGREES
LOADING 2 'ROTATED LOADING'
MEMBER DISTORTIONS
      1 TO 10 UNIFORM FR LA 0.0 LB 1.0 DISPL X 0.001
ROTATE LOADING 1 ANGLES T1 45.0
```

The applied loads from previously defined loading 1 will be processed according to Steps 1 to 4 above and copied into the new destination loading 2, which includes the specified member distortion loads applied to members 1 to 10.

2.
```
UNITS DEGREES
CHANGES
LOADING 3
ADDITIONS
ROTATE LOAD 4 ANGLES T2 -30.0
```

Previously defined loading 3 is specified in CHANGES mode, followed by a return to ADDITIONS mode. The ROTATE LOAD command is then given to add the components of load 4, including appropriate rotations, to loading 3.

Error Messages:

Incorrect data given in the ROTATE LOADING command will cause the following error conditions to be identified and error messages printed:

1. The following error message is printed if the ROTATE loading name is identical to the name of the destination load. An example of the commands that produce this error are also included:

```
{   114} > LOADING 201
{   115} > ROTATE LOAD 201 T1 45.0

  **** ERROR_STROLO -  The ROTATE loading is illegally the same as the
                       destination loading.
                       Command ignored.
```

Loading 201 is illegally named as both the destination load and the loading whose components are rotated.

2. In the following error example, loading 51 is undefined.

```
{   111} > LOADING 201
{   112} > ROTATE LOAD 51 T1 45.0

**** ERROR_STROLO -    Loading to be rotated undefined.
                       Command ignored.
```

3. The following error message is produced because loading 4, specified as the ROTATE load, is a load combination, or dependent loading condition. The ROTATE load must be an independent loading condition.

```
{   141} > LOADING 108
{   142} > ROTATE LOADING 4 T3 45.0

**** ERROR_STROLO -    Rotated Loading 4 is an illegal dependent load.
                       Command ignored.
```

4. This error condition and message is caused by the fact that the destination load 108 is defined as a loading combination.

```
{   144} > LOAD COMB 108 'ALL' COMBINE 1 1.5 2 1.0 3 1.0
{   145} > ROTATE LOADING  1 T3 45.0

**** ERROR_STROLO -    Destination independent loading not defined.
                       Rotated load components not computed.
```

5.4.2 Reference Coordinate System Command

General form:

$$\underline{\text{REF}}\text{ERENCE }(\underline{\text{COO}}\text{RDINATE}) \ (\underline{\text{SYS}}\text{TEM}) \begin{Bmatrix} i_1 \\ 'a_1' \end{Bmatrix} -$$

$$\begin{Bmatrix} (\underline{\text{ORI}}\text{GIN } [\underline{X}] \ v_x \ [\underline{Y}] \ v_y \ [\underline{Z}] \ v_z) \ (\underline{\text{ROT}}\text{ATION } [\underline{\text{R1}}] \ v_1 \ [\underline{\text{R2}}] \ v_2 \ [\underline{\text{R3}}] \ v_3) \\ \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X} \ v_4 \ \underline{Y} \ v_5 \ \underline{Z} \ v_6 \end{Bmatrix} \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X} \ v_4 \ \underline{Y} \ v_5 \ \underline{Z} \ v_6 \end{Bmatrix} \begin{Bmatrix} \underline{\text{JOI}}\text{NT} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X} \ v_4 \ \underline{Y} \ v_5 \ \underline{Z} \ v_6 \end{Bmatrix} \end{Bmatrix}$$

Explanation:

The REFERENCE COORDINATE SYSTEM is a right-handed three-dimensional Cartesian coordinate system. The Reference Coordinate System's origin may be shifted from the origin (X=0.0, Y=0.0, Z=0.0) of the overall global coordinate system. The Reference Coordinate System axes may also be rotated from the corresponding orthogonal axes of the overall global coordinate system.

At the present time, this command is used to specify additional coordinate systems which may be used in GTMenu (see Volume 2 of the GTSTRUDL Release Guide) to facilitate the creation of the structural model. Reference Coordinate systems created using the above command will be available as Local systems in GTMenu. In a future release, the user will be able to output results such as joint displacements and reactions in a Reference Coordinate System.

There are two optional means of specifying a Reference Coordinate System:

(1) Define the origin and rotation of coordinate axes of the reference system with respect to the global coordinate system, and

(2) define three joints or the coordinates of three points in space.

In either case, i_1 or 'a_1' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x, v_y, and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x, v_y, and v_z, are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive directions of these axes; v_1, v_2, and v_3 are the rotation angles R_1, R_2, and R_3 in active angular units between the orthogonal axes of this system and the axes of the overall global coordinate system. The description of these angles is the same as given in Section 2.1.7.2 of Volume 1 of the GTSTRUDL User Reference Manual for rotated joint releases (θ_1, θ_2, and θ_3).

In the second case, three joints are required. Each of the three joints may be defined either by a joint identifier using the JOINT option of the command or by its global X, Y, and Z coordinates. If the joint identifier option is used, however, the coordinates of the joint must be specified previously by the JOINT COORDINATES command. The first time (i_2 or 'a_2' or v_4, v_5, and v_6) defines the origin of the reference system; the X-axis of the reference system is determined by the first and second joints (i_3 or 'a_3' or v_7, v_8, and v_9). The positive X-axis is directed from the first to the second joint. The third joint (i_4 or 'a_4' or v_{10}, v_{11}, and v_{12}) is used to define the XY-plane of the reference system. The positive Y-axis is directed toward the third joint. The Z-axis then is determined by the right-hand rule.

Only one reference system can be specified in one command, but the command may be used any number of times.

Modifications of Reference Systems:

In the changes mode, the translations of the origin and/or the rotations of the axes of the reference system from those of the overall global system can be changed. Only that information supplied in the command is altered. The other data that might be supplied in the command remains unchanged. The CHANGES mode, however, does not work for the second option discussed above (i.e., define a reference coordinate system by three joints or the coordinate of three points in space). The reason is that data for these joints are not stored permanently in GTSTRUDL. When this option is used, a reference system is created and its definitions of the system origin, rotation angles, as well as the transformation matrix between the global coordinate system and the reference system are generated and stored as would be for the first option. Therefore, if any of the coordinates for the joints used to specify a reference system is changed after the REFERENCE COORDINATE SYSTEM command has been given, the definition of the reference system remains unchanged. For this reason, care must be taken in using the three joints option in conjunction with the changes of joint coordinates. The reference system should be deleted first if any of the coordinates of the joints used to define the reference system are to be changed. Under the DELETIONS mode, the complete definition of the reference coordinate system is destroyed.

Examples:

a) UNITS DEGREES
REFERENCE COORDINATE SYSTEM ‘FLOOR2’ -
    ORIGIN 0.0 15.0 0.0 R1 30.

This command creates a Reference Coordinate System called FLOOR2 at Y=15 with the axes rotated 30 degrees about global Z.

b) REF COO 1 -
    X 120 Y 120 Z -120 -
    X 120 Y 240 Z 0 -
    X -120 Y 120 Z 0

This command creates Reference Coordinate System 1 with its origin at 120, 120, -120 and its X-axis from this origin to 120, 240, 0 and its Y axis is the plane defined by the two previous coordinates and the third coordinate, -120, 120, 0, with the positive Y-axis directed toward the third coordinate.

c) REFERENCE COORDINATE SYSTEM 2 -
    JOINT 10 JOINT 20 JOINT 25

This command creates Reference Coordinate System 2 with its origin located at Joint 10 and its X-axis directed from Joint 10 toward Joint 20. The XY plane is defined by Joints 10, 20, and 25 with the positive Y-axis directed toward Joint 25.

d) CHANGES
REFERENCE COORDINATE SYSTEM ‘FLOOR2’ -
    ORIGIN 10 20 30
ADDITIONS

The above commands change the origin of the Reference System FLOOR2 defined in a) above. The rotation RI = 30 remains unchanged.

e) DELETIONS
REFERENCE SYSTEM 2
ADDITIONS

The above command deletes Reference System 2.

5.4.2-1 Printing Reference Coordinate System Command

General form:

$$\underline{\text{PRI}}\text{NT }\underline{\text{REF}}\text{ERENCE }(\underline{\text{COO}}\text{RDINATE})\ \ (\underline{\text{SYS}}\text{TEM})\left\{\begin{matrix}\rightarrow \underline{\text{ALL}}\\ \text{list}\end{matrix}\right\}$$

Explanation:

The PRINT REFERENCE COORDINATE SYSTEM command will output the Reference Systems. The origin and rotation angles will be output.

5.4.3 GTMenu Point and Line Incidences Commands

GTMenu can now write construction geometry commands to an input file, which can be read later into GTSTRUDL in order to initialize the construction geometry of GTMenu. The two commands written are "GTMenu POINT COORDINATES" and "GTMenu LINE INCIDENCES".

(1) **GTMenu POINT COORDINATES**

General Form:

GTMenu POINT COORDINATES

$$\left\{ \begin{matrix} i_1 \\ 'a_1' \end{matrix} \right\} \text{coordinate-specs}_1$$

.
.
.

$$\left\{ \begin{matrix} i_n \\ 'a_n' \end{matrix} \right\} \text{coordinate-specs}_n$$

Elements:

coordinate-specs = [X] v_1 [Y] v_2 [Z] v_3

Where,

$i_1, i_2, ..., i_n$ = unsigned integer Point identifiers.

$'a_1', 'a_2', ..., 'a_n'$ = 1 to 8 character alphanumeric Point identifiers.

v_1, v_2, v_3 = Cartesian Point coordinates (integer or real).

(2) **GTMenu LINE INCIDENCES**

General Form:

<u>GTMenu</u> <u>LIN</u>E <u>INC</u>IDENCES

$$\left\{ \begin{matrix} i_1 \\ 'a_1' \end{matrix} \right\} \text{type}_1 \ \text{incidence-specs}_1$$

.
.
.

$$\left\{ \begin{matrix} i_n \\ 'a_n' \end{matrix} \right\} \text{type}_n \ \text{incidence-specs}_n$$

Elements:

$$\text{type} = \left\{ \begin{matrix} \rightarrow \underline{\text{LIN}}\text{E} \\ \underline{\text{POL}}\text{YNOMINAL } (\underline{\text{CUR}}\text{VE}) \\ \underline{\text{ARC}} \ (\underline{\text{TEM}}\text{PLATE}) \\ \underline{\text{CEN}}\text{TERED } (\underline{\text{ARC}}) \ \underline{\text{PER}}\text{CENT } v_1 \\ \underline{\text{BEZ}}\text{IER } (\underline{\text{CUR}}\text{VE}) \\ \underline{\text{SPL}}\text{INE } (\underline{\text{CUR}}\text{VE}) \ (\underline{\text{ORD}}\text{ER } k_2) \end{matrix} \right\}$$

$$\text{incidence-specs} = \left\{ \begin{matrix} i_1 \\ 'a_1' \end{matrix} \right\} \left\{ \begin{matrix} i_2 \\ 'a_2' \end{matrix} \right\} \ldots \left\{ \begin{matrix} i_p \\ 'a_p' \end{matrix} \right\}$$

Where,

$i_1, i_2, ..., i_n$	=	unsigned integer Line/Curve identifiers.
‘a_1’, ‘a_2’, ..., ‘a_n’	=	1 to 8 character alphanumeric Line/Curve identifiers.
$i_1, i_2, ..., i_p$	=	unsigned integer Point identifiers used.
‘a_1’, ‘a_2’, ..., ‘a_p’	=	1 to 8 character alphanumeric Point identifiers.
v_1	=	positive number (integer or real).
k_2	=	integer between 2 and the number of incidences.
1, 2, ...,p	=	Point subscripts for a Line/Curve. The following table gives the number of Points used to specify different types of Line/Curve:

type	number of incidences
LINE	2 - 500
POLYNOMIAL CURVE	2 - 10
ARC TEMPLATE	3
CENTERED ARC	3
BEZIER CURVE	2 - 10
SPLINE CURVE	2 - 10

End of Document