GT STRUDL ®

Version 32

Release Guide

Volume 1 of 2

NOTICES

This GTSTRUDL_® Release Guide is applicable to Version 32, with a release date in the GTSTRUDL title block of June 2012.

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.

© Copyright 2012
Georgia Tech Research Corporation
Atlanta, Georgia 30332-0355
U.S.A.

ALL RIGHTS RESERVED

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

This page intentionally left blank.

Chapter 1

Introduction

Version 32 covers GTSTRUDL operating on PC's under the Windows 7, Windows Vista and Windows XP operating systems. Chapter 2 presents the new features and enhancements which have been added since the release of Version 31. Chapter 3 provides you with details regarding error corrections that have been made since the Version 31 release. Chapter 4 describes known problems with Version 32. 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   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.

    5.2.2   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.

    5.2.3   Rectangular and Circular Concrete Cross Section Tables

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

    5.2.5   ISO 19902:2007(E) code for fixed steel offshore structures.

    5.2.6   ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.

5.3   Analysis Prerelease Features

    5.3.1   Calculate Error Estimate Command

5.4   General Prerelease Features

     5.4.1   Rotate Load Command

     5.4.2   Reference Coordinate System Command

     5.4.3   GTMenu Point Coordinates and Line Incidences Commands

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 32

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 32. This release guide is also available online upon execution of GTSTRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1   DBX

1.   WRITE CODE CHECK RESULTS has been added. This DBX file contains the same data as found in the output from LIST CODE CHECK RESULTS. ASCII80 and BINARY sequential formats are supported.

Syntax:

WRITE CODE (CHECK) (RESULTS) (MEMBERS list)

Examples:

WRITE CODE ALL MEMBERS
WRITE CODE RES MEMBERS EXISTING 1 TO 1000

Documentation:

"Section 2.1.14.4.3 The LIST CODE CHECK RESULTS Command", Volume 1, GTSTRUDL Reference Manual.
"Summary of DBX Commands," "The WRITE Command," Volume 5, GTSTRUDL Reference Manual. "Table 1-30," Volume 5, GTSTRUDL Reference Manual.

2.   The WRITE CABLE FORCES command has been added to the DBX feature. This DBX file contains the normal stress, the corresponding normal force, and the three global element nodal reaction components at each of the two to six cable element nodes for all active static loading conditions, computed from the latest nonlinear analysis. All documented file formats and access modes are supported.

Syntax:

WRITE CABLE FORCES (MEMBERS list)

Example:

WRITE CABLE FORCES ALL MEMBERS

WRITE CABLE FORCES MEMBERS EXISTING 1 TO 1000

Note that only the cable elements among the specified list of members are processed.

Documentation:

"Summary of DBX Commands."   "The WRITE Command," Volume 5, GTSTRUDL Reference Manual. "Table 1-31," Volume 5, GTSTRUDL Reference Manual.

2.2   Dynamics

1.   The GT64MLANCZOS eigenvalue analysis solution method has been implemented, extending high-performance sparse-equation eigenvalue analysis to 64-bit computer platforms. GT64MLANCZOS is the most powerful version of the GTLANCZOS family of eigenvalue analysis solution methods (GTLANCZOS, GTSELANCZOS, and now GT64MLANCZOS) presently available in GTSTRUDL and incorporates the following features:

   1)   The Lanczos iteration process employs an in-core, single processor version of the GT64M sparse equation solver. Multiple processors and out-of-core processing are not available as of this initial implementation for Version 32.

   2)   All computations associated with the Lanczos iteration process take advantage of 64-bit addressing on 64-bit platforms, greatly increasing the number of degrees of freedom that can be treated and the number of modes that can be computed when compared to the GTLANCZOS and GTSELANCZOS methods.

   The GT64MLANCZOS method is activated when the GT64M option is specified by the ACTIVE SOLVER command:

         ACTIVE SOLVER GT64M

or when the GT64MLANCZOS method is specified in the EIGENSOLUTION PARAMETERS command:

        EIGENSOLUTION PARAMETERS
            SOLVE USING GT64MLANCZOS
                .
                .
                .
        END

Note that the GT64M solver suite is a premium feature and requires a separate fee, license, and password.

Documentation

        Section 2.1.13, Volume 1, "The ACTIVE SOLVER Command"
        Section 2.4.5.2, Volume 3, "Eigenproblem Solution Specifications"

2.   Improvements are made to the consistency checking of the time point data for the execution of modal integration and direct integration time history analyses. If the number of time points to be processed by these analyses exceeds the maximum allowable 4,000,000, the following error message is now given and the indicated action taken:

```
**** ERROR_STTINT -- Maximum allowable number of integration time points
                     (4 million) exceeded for loading 1002    .
                     Scan mode entered and analysis terminated.
```

Documentation:

Section 2.4.5.4, Volume 3, "The DYNAMIC ANALYSIS Command"
Section 2.4.5.5.6, Volume 3, "Mode Superposition Analysis"
Section 2.4.5.5.7, Volume 3, "Transient Analysis"
Section 2.4.5.5.8, Volume 3, "Direction Integration (PHYSICAL) Analysis"

3.   The COMPUTE MODAL DAMPING RATIO AVERAGE BY ELEMENT command has been enhanced such that a contribution to the composite modal damping ratios are now computed for viscous damper elements.

Documentation:

Section 2.4.3.4, Volume 3, "Specification of Damping Properties"
Section 2.4.3.7, Volume 3, "The Viscous Damper Element for Linear and Nonlinear
                        Dynamic Analysis"

2.3   Finite Elements

1.   A new 8 node solid element called the IPSLIM has been implemented. This new element
      has the same node ordering as the IPSL and IPLS elements. This new element is more
      flexible than the IPSL and IPLS elements and will converge faster. A comparison of the
      results for the example of a cantilever beam presented in Section 2.3.9.3 of Volume 3 of the
      User Reference Manual is shown below:

ELEMENT	MESH	SXX STRESS (ksi) $x = 9", y = 0", z = 2"$	MAXIMUM DISPLACEMENT(in) $x = 21", y = 2", z = 2"$
IPLS	B-1	2.280	0.0552
	B-2	2.448	0.0597
IPQS	B-1	2.492	0.0613
IPSL	B-2	2.448	0.0597
IPSLIM	**B-1**	**2.499**	**0.0613**
	B-2	**2.500**	**0.0613**
IPSQ	B-2	2.500	0.0612
THEORETICAL (30)		2.500	0.0612

      The IPSLIM element not only converges faster than the IPSL and IPLS but produces almost
      the same results for the same number of elements as the twenty-noded IPSQ and IPQS
      elements.

2.   Three new four node elements have been implemented. A new plane stress called the
      Q6CDRL and a new plate bending element called the PBMITC have been implemented.
      These elements were combined to form a new four node flat plate element called the
      SBMITC which has six degrees-of-freedom per node.

      The Q6CDRL is an isoparametric four node element based on a variational formulation. This
      formulation provides a rotational degree of freedom about the planar z-axis (commonly
      referred to as the drilling degree of freedom). This drilling degree of freedom is very useful
      for interaction with plane frame elements. In addition, incompatible modes with a correction
      method were implemented in order to avoid shear locking. The Q6CDRL successfully
      addresses all other common issues of plane stress elements, such as spurious modes, failure
      to pass the patch test and geometric invariance.

The PBMITC element is based on the well known MITC4 element (mixed interpolation of tensorial components). It is an isoparametric four node plate bending element based on Mindlin's plate theory, which includes transverse shear strains (applicable for moderately thick plates). This element does not exhibit any of the common thick plate bending issues such as shear locking and spurious modes.

By combining the Q6CDRL and PBMITC (uncoupled in-plane and out-of-plane behavior), a versatile four node isoparametric flat plate element (SBMITC) is now available for general use in shell and plate problems.

2.4   General

1.   PRINT MEMBER LENGTH SORTED LIMIT v now prints the total number of members that meet the specified limit after the list of member lengths.

This only applies to LIMITS with decimal numbers, which indicates a length value and where the number of members meeting the criterium is unknown, and not to LIMITS with an integer number, which indicates a specified number of member lengths to print.

Example:

```
{ 4367} > UNITS CM
{ 4368} > PRINT MEMBER LENGTH SORTED LIMIT 20.0
     (output omitted)
**** INFO_PRTMBL - Found   10 members <    20.000
```

Documentation:

"Section 2.1.14.2.1 PRINT MEMBER LENGTH SORTED", Volume 1, GTSTRUDL Reference Manual.

2.   A new option has been added to the PRINT command, PRINT MEMBER BETA ANGLES, which allows you to print only BETA angles, without the other CONSTANTS.

Syntax:

PRINT MEMBER BETA (ANGLES) (NONZERO (ONLY) (TOLERANCE v_1)) –

$\left\{ \begin{array}{l} \rightarrow \text{ INDIVIDUALLY} \\ \text{GROUPED (TOLERANCE } v_2) \end{array} \right\} \left(\left\{ \begin{array}{l} \text{DEGREES} \\ \text{RADIANS} \end{array} \right\} \right)$ –

(COMMAND (FORMAT)) (MEMBERS list)

Examples:

```
PRINT MEMBER BETA ANGLES NONZERO ONLY TOLERANCE 0.1 DEGREES
PRINT MEMBER BETA GROUPED COMMAND FORMAT
```

Documentation:

Section 2.1.14.2.3 PRINT MEMBER BETA ANGLES, Volume 1, GTSTRUDL
Reference Manual

3.   A GLOBAL option has been added to LIST FORCES. This option will print member
forces in the global reference frame, instead of the standard local (member) reference
frame.

Syntax:

LIST FORCES (GLOBAL)

Example:

LIST FORCES GLOBAL MEMBERS 1 TO 10

Documentation:

Section 2.1.14.4, Static Analysis Computed Results - LIST Command, Volume 1,
GTSTRUDL Reference Manual.

4.   Two new options have been added to the LIST JOINT FORCES command, GLOBAL
(WITH TOTAL) and MEMBERS *m_list*.

The GLOBAL option causes the member and element forces to be rotated from the local
coordinate system to the global coordinate system, and in the case of members with
ECCENTRICITIES, transformed from the flexible end of the member to the joint -
therefore LIST JOINT FORCES GLOBAL will not match LIST FORCES even after
resolving the coordinate systems for members with ECCENTRICITIES. The WITH
TOTAL option will print the summation of all listed members and elements for each
DOF for each joint and active load. The GLOBAL option always outputs all 6 global
DOF, instead of being restricted to member DOF.

The MEMBERS *m_list* option restricts the printed forces to members or elements in
m_list and any joint that no member in *m_list* is incident to will have an informative
message printed indicating this condition ("No members or elements in the specified list

are attached to this joint."), so it is suggested that the MEMBERS *m_list* option only be used in conjunction with JOINT *list*, instead of the ALL JOINTS default, to avoid a large number of these informative messages.

<u>Syntax</u>:

$$
\underline{\text{LIST}}\ \underline{\text{JO}}\text{INT}\ \underline{\text{FORCES}}
\begin{Bmatrix}
\rightarrow \underline{\text{LOCAL}} \\
\underline{\text{GLOBAL}}\ (\underline{\text{WITH}})\ (\underline{\text{TOTAL}})
\end{Bmatrix}
-
$$

$$
\begin{Bmatrix}
\rightarrow \underline{\text{ALL}}\ (\underline{\text{JO}}\text{INTS}) \\
\underline{\text{JO}}\text{INTS list}
\end{Bmatrix}
(\underline{\text{MEMB}}\text{ERS m\_list})
$$

<u>Examples</u>:

```
{  188} > OUTPUT FIELD F
{  189} > OUTPUT DECIMAL 3
{  190} > LIST JOINT FORCES JOINT 3


-----------------------------------------------------------------------------------------------------------
-
 ---   LOADING - 1

-----------------------------------------------------------------------------------------------------------
-

       RESULTANT JOINT FORCES OUTPUT BY LOADING

          MEMBER/    REF

/-JOINT-/-ELEMENT-/-FRAME-/--------------------FORCES--------------------/--------------------MOMENTS------------------/--ECC-
/
                        FORCE X        FORCE Y        FORCE Z        MOMENT X        MOMENT Y        MOMENT Z

3       1       LOCAL        6.148       -6.119        0.000        0.000        0.000       11.185   NO
3       2       LOCAL        6.148       -6.119        0.000        0.000        0.000      -11.185   NO
3       3       LOCAL       -6.148       -6.119        0.000        0.000        0.000       11.476   NO
3       4       LOCAL       -6.119       -6.148        0.000        0.000        0.000      -11.476   NO

{  191} > LIST JOINT FORCES GLOBAL JOINT 3


-----------------------------------------------------------------------------------------------------------
-
 ---   Loading - 1

-----------------------------------------------------------------------------------------------------------
-

       GLOBAL joint forces output by loading

          MEMBER/    REF

/-JOINT-/-ELEMENT-/-FRAME-/--------------------FORCES--------------------/--------------------MOMENTS------------------/--ECC-
/
                        FORCE X        FORCE Y        FORCE Z        MOMENT X        MOMENT Y        MOMENT Z

3       1       GLOBAL       6.119        6.148        0.000        0.000        0.000       11.185   NO
3       2       GLOBAL       6.119        6.148        0.000        0.000        0.000      -11.185   NO
3       3       GLOBAL      -6.119       -6.148        0.000        0.000        0.000       11.476   NO
3       4       GLOBAL      -6.119       -6.148        0.000        0.000        0.000      -11.476   NO

{  192} > LIST JOINT FORCES GLOBAL WITH TOTAL JOINT 3


-----------------------------------------------------------------------------------------------------------
-
 ---   Loading - 1

-----------------------------------------------------------------------------------------------------------
-
```

```
      GLOBAL joint forces output by loading

          MEMBER/   REF

/-JOINT-/-ELEMENT-/-FRAME-/-------------------FORCES-------------------/-------------------MOMENTS-----------------/--ECC-
/
                          FORCE X       FORCE Y       FORCE Z      MOMENT X      MOMENT Y      MOMENT Z

3        1      GLOBAL      6.119         6.148         0.000         0.000         0.000        11.185    NO
3        2      GLOBAL      6.119         6.148         0.000         0.000         0.000       -11.185    NO
3        3      GLOBAL     -6.119        -6.148         0.000         0.000         0.000        11.476    NO
3        4      GLOBAL     -6.119        -6.148         0.000         0.000         0.000       -11.476    NO
                          --------------- --------------- --------------- --------------- --------------- ---------------
                Totals      0.000         0.000         0.000         0.000         0.000         0.000
```

```
{  193} > LIST JOINT FORCES GLOBAL WITH TOTAL JOINT 3 MEMBERS 1 2
```

```
----------------------------------------------------------------------------------------------------------------------
-
 ---   Loading - 1
----------------------------------------------------------------------------------------------------------------------
-
      GLOBAL joint forces output by loading

          MEMBER/   REF

/-JOINT-/-ELEMENT-/-FRAME-/-------------------FORCES-------------------/-------------------MOMENTS-----------------/--ECC-
/
                          FORCE X       FORCE Y       FORCE Z      MOMENT X      MOMENT Y      MOMENT Z

3        1      GLOBAL      6.119         6.148         0.000         0.000         0.000        11.185    NO
3        2      GLOBAL      6.119         6.148         0.000         0.000         0.000       -11.185    NO
                          --------------- --------------- --------------- --------------- --------------- ---------------
                Totals     12.237        12.295         0.000         0.000         0.000         0.000
```

Documentation:

Section 2.1.14.4, Static Analysis Computed Results - LIST Command, Volume 1, GTSTRUDL Reference Manual.

5.   Modest efficiency and performance improvements have been made to the GTSES stiffness analysis solver. However, the solver remains a single-processor, 32-bit solver governed by 32-bit virtual memory allocation and addressing limitations.

Documentation:

Section 2.1.13.2, The STIFFNESS ANALYSIS Command, Volume 1.
Section 2.1.13.3.2, Split Solver Static Analysis Using the GTSES/GT64M Sparse Equation Solvers, Volume 1.
Section 2.1.13.4, The ACTIVE SOLVER Command, Volume 1.

6.   The RENAME command has been added. This command allows you to change the name of an existing component - joint, member/element, load or group.

Syntax:

RENAME *type old_name new_name*

where,

type	is the type of component to rename. The allowable types are: JOINT (or NODE), MEMBER (or ELEMENT), LOAD, GROUP.
old_name	is the current component name. This name must already exist. The name may be either integer or alpha-numeric.
new_name	is the new name. There may not already be a component of this type with the new name. The new name may be either integer or alpha-numeric.

Example:

RENAME MEMBER 1 1001

After this command LIST FORCES MEMBER 1 will generate an error message, but LIST FORCES MEMBER 1001 will work properly.

Documentation:

Section 2.1.12.20, The RENAME Command, Volume 1.

7. The MEMBER PROPERTIES Command has been enhanced giving the user the ability to define member properties for Channels and Polygonal Tubes by specifying their dimensions.

The following are examples of the two new options for the MEMBER PROPERTIES Command:

MEMBER PROPERTIES
        1 TO 10 CHANNEL TOTAL DEPTH 16.97 -
                WEB THICKNESS 0.585 FLANGE WIDTH 10.425 -
                FLANGE THICKNESS 0.985

        11 TO 20 POLYGONAL TUBE -
                DIAMETER BETWEEN FLATS 14.35 -
                NUMBER OF SIDES 8 -
                THICKNESS 0.985

These new options are available for prismatic and variable members. The PRINT MEMBER PROPERTIES FOR DESIGN and the PRINT MEMBER PROPERTIES FOR DESIGN WITH DEFINITIONS commands now print data for these members. Also, members whose properties have been defined through these options can be code checked by the GTSTRUDL Codes that support such cross-sections.

See Section 2.1.9 on Volume 1 of the GTSTRUDL User Reference Manual for more information.

8.   The CALCULATE SOIL SPRINGS command has been enhanced to:

1) Allow a single KS value without an element list, which indicates that GTSTRUDL should find all elements and element faces that lie on the specified plane and create an assumed element list. This makes adding soil springs to an entire slab on one level much simpler.

2) Added a NO SUPPORT CHECK option to the COMPRESSION ONLY nonlinear springs capability. NLS elements will be added to the model without regard to the current support status of the involved joints as opposed to the current requirement that soil springs can be added only to joints with the appropriate DOF (FX, FY or FZ) restrained. This allows you greater flexibility in modeling, but leaves the responsibility of creating a stable analysis model up to you. An informational message is generated if the NO SUPPORT CHECK option is used, but the appropriate DOF is fixed - which will result in the created NLS being nonfunctional:

```
**** WARNING_SSPR03 - The NO SUPPORT CHECK option will apply nonlinear springs to
            joints without regard to support status.  This message is to
            inform you the following joints have a restrained (fixed) DOF
            in the specified spring direction and the applied NLS will
            have no effect on a subsequent analysis unless the DOF is released.
```

The NO SUPPORT CHECK option cannot be used with the INPLANE option.

3) The name of the generated NLS has been added to the output if the COMPRESSION ONLY option is used. In addition, the informative comments output below the spring value report now reflect the actual names of the generated NLS so they can be copied from the output to be added to your input or pasted into the Command window.

Documentation:

Section 2.1.12.13, The CALCULATE SOIL SPRING VALUES Command, Volume 1.

9.   A new option, LAUNCH, has been added to the RUN command. This option utilizes the Windows ShellExecuteEx utility to open programs with a specified file, but unlike the

standard RUN command, you do not need to know the program installation path. This is especially useful for running utility programs like Microsoft's Excel, where the installation path may be different on various computers.

Syntax:

RUN LAUNCH (WAIT) (PROGRAM) '*program*' (FILE) '*file*'

Elements:

> *program =*   For the LAUNCH version, *program* is the name (e.g. Excel) of the program to be launched.

> *file =*   an input file or other command line options for *program*, only available to the LAUNCH version.

Explanation:

LAUNCH

> This version of the RUN command uses the Windows "ShellExecuteEx" functionality to run a program (*program* above), even though you do not know where it is installed. The program must be registered in Windows such that the "assoc" DOS command can report the program name given its standard file extension. For example, if you issue the DOS command "assoc .gti" you get the response ".gti=GTStrudl.Input". The element *file* is sent to *program* as an argument. The HIDE option is not available for LAUNCH. PROGRAM and FILE are optional words and have no effect on the command operation. If *file* is specified as "NULL", then *program* will be launched with no arguments.

WAIT   Indicates that GTSTRUDL command processing should stop until *program* has completed. If this option is not used, GTSTRUDL will continue and process the next command if you are running a macro or an input file. Use the WAIT option if the results of *program* are required for subsequent GTSTRUDL commands.

Example:

```
RUN LAUNCH WAIT PROGRAM 'Excel' FILE 'MyData.xls'
```

This will open Microsoft Excel as if you had double-clicked the file 'MyData.xls'.

Documentation:

Section 2.1.12.17 The RUN Command, Volume 1, GTSTRUDL Reference Manual.

10.   The CALCULATE SOIL SPRINGS and CALCULATE PRESSURE commands have been enhanced to allow the use of the PBMITC, SBMITC, and IPSLIM elements.

Documentation:

Section 2.1.12.13 The CALCULATE SOIL SPRING VALUES Command, Volume 1, GTSTRUDL Reference Manual.

Section 2.1.12.19 The CALCULATE PRESSURE Command, Volume 1, GTSTRUDL Reference Manual.

11.   The LARGE PROBLEM SIZE command has been improved to take advantage of the increased resources available to computers with a 64-bit operating system. This improvement can help decrease the run time for very large jobs.

12.   More comprehensive checking is now performed when processing the MEMBER LOAD command to detect when the user has not specified the load component or type of load.

13.   The SAVE/RESTORE process has been enhanced to allow data bases larger than 2 GB. Because GTSTRUDL relies on standard 32 bit file utilities, individual files are limited to 2 GB. When internal data bases exceed 2 GB, a second file is created during the SAVE process. This file uses the same user specified file name as the standard SAVE file, but with the extension ".gts2". This extension is fixed and is not based on the extension of the standard SAVE file, which has a default extension of ".gts" but may be different if the user specifies an extension. A successful SAVE of a data base greater than 2 GB will issue a message similar to this:

```
{   79} > save 'test'

****INFO_XXSAVE - Data base > 2GB and saved in files
C:\32\LargeProblem\2Files\test.gts
C:\32\LargeProblem\2Files\test.gts2
```

Only the standard SAVE file is referenced in the RESTORE command, but **both files** (the standard SAVE file and the second .gts2 file) are required for a successful RESTORE of a data base larger than 2 GB. If the SAVE file is copied or moved to a different folder, be sure to include the .gts2 file if it exists. If the .gts2 file is missing, RESTORE will fail with an error similar to this:
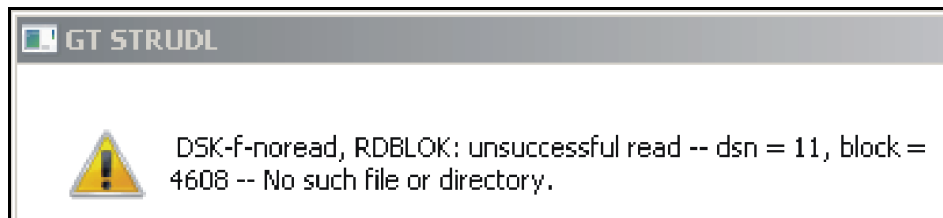
A successful RESTORE will issue a message similar to this:

```
{   80} > RESTORE 'C:\32\LargeProblem\2Files\test.gts'

****INFO_XXREST - RESTORE complete from 2 files:
C:\32\LargeProblem\2Files\test.gts
C:\32\LargeProblem\2Files\test.gts2
```

The SAVE/RESTORE process for data bases less than 2GB is unchanged.

14.   The NJP parameter has been added to the GTSES/GT64M ASSEMBLE command. The
new command syntax is shown below:

$$\begin{Bmatrix} \underline{\text{GTSES}} \\ \underline{\text{GT64M}} \end{Bmatrix} \underline{\text{ASS}}\text{EMBLE} \ \underline{\text{FOR}} \ \underline{\text{STAT}}\text{ICS} \ (\underline{\text{NJP}} \ \begin{Bmatrix} \rightarrow 6 \\ v_{\text{NJP}} \end{Bmatrix}) \ (\underline{\text{OOC}})$$

The optional NJP parameter, v_{NJP}, is used to control the number of joints that are
assembled into a typical global stiffness sub-matrix partition, where NJP = 6 is the
default. NJP = 6 is adequate for the majority of cases; however, in the rare case when the
assembly process terminates due to a failure to allocate sufficient virtual memory , re-
executing the GTSES/GT64M ASSEMBLE FOR STATICS command with a larger
value of NJP – e.g. NJP = 12 – may permit the assembly to complete successfully.

Documentation

Section 2.1.13.3.2, Volume 1, Split Solver Static Analysis Using the GTSES/GT64M
Sparse Equation Solvers.

2.5   GTMenu

1.   Dynamic mass specified using the INERTIA OF JOINTS command can now be
      displayed. The revised Results pulldown and the Display Joint Mass dialog are shown
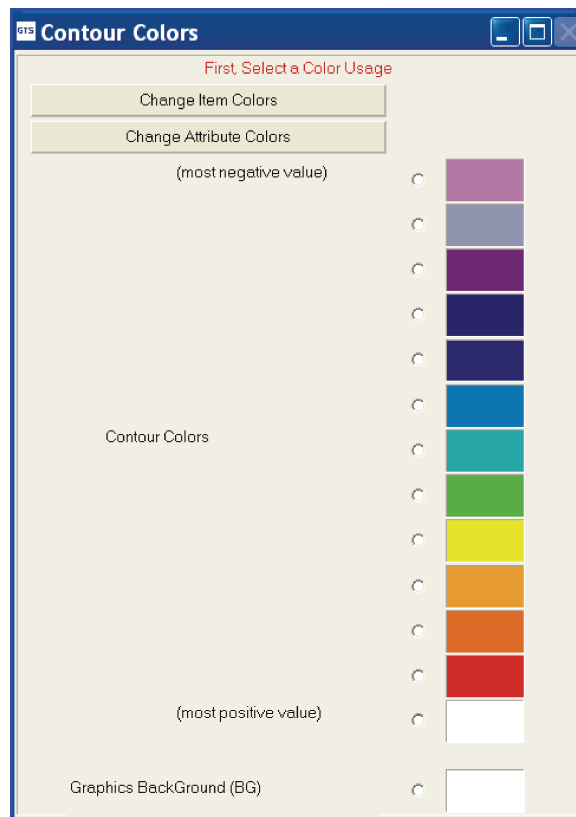   below:
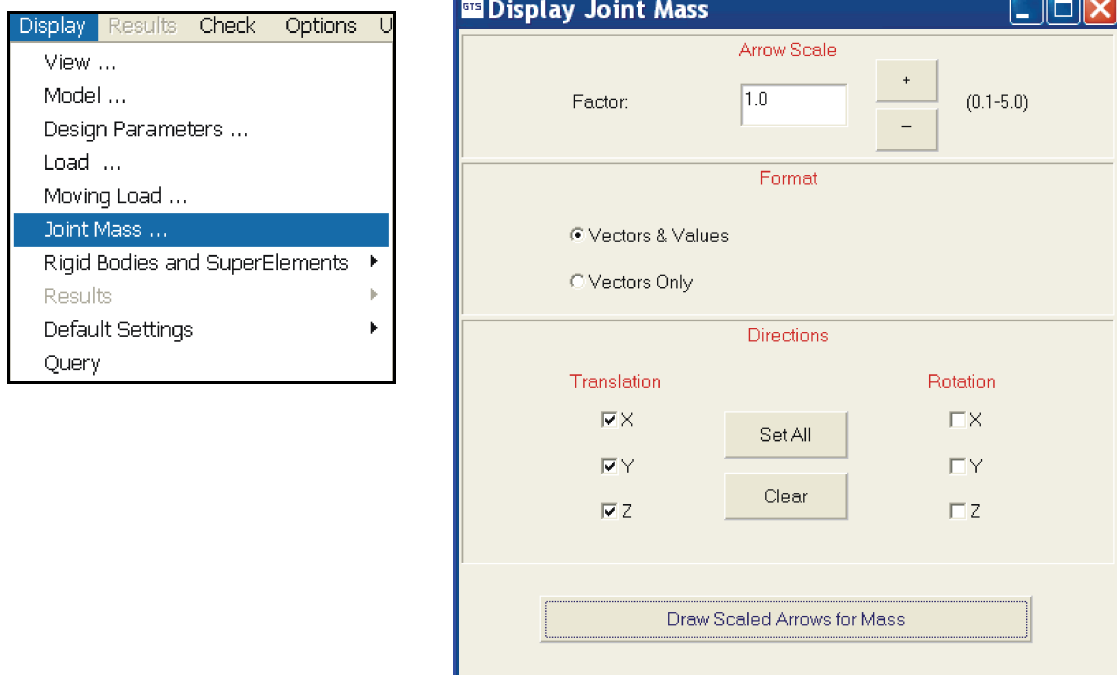
An example of the display of joint masses using the above dialog is shown below:
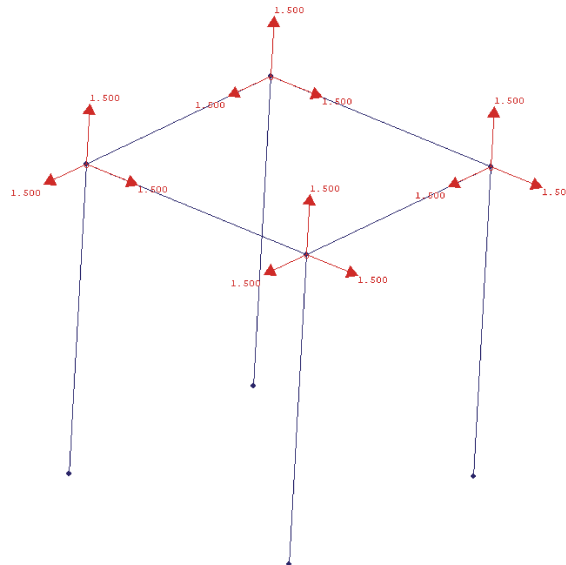
2.   The colors used for contour plots can now be changed. The revised Options pulldown is
         shown below:
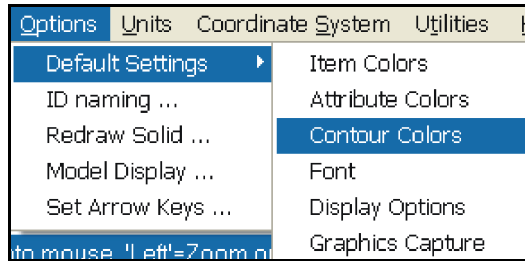
The new Contour Colors dialog is shown below:

3.   Different colors can now be used for members and finite elements for the deformed
      structure, mode shapes and transient animation as shown on the revised Attribute Colors
      dialog below:
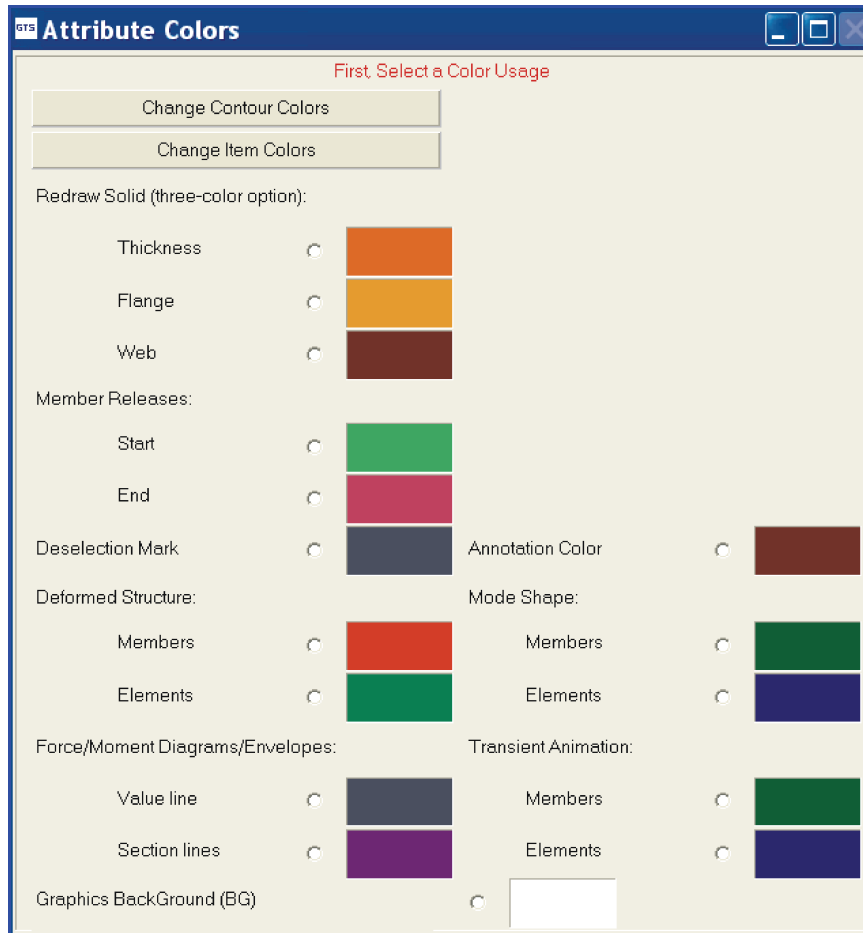
4.   A right mouse click can now be set using the Inquire Button so that only joint, members or elements are selected. This setting is retained until you change it using the Inquire Button. By default, the selection is set to All Types.

        The right mouse click setting is now displayed at the top of the Graphics Display. The Inquire Button pulldown and the display of the right mouse click setting at the top of the Graphics Display are shown below:
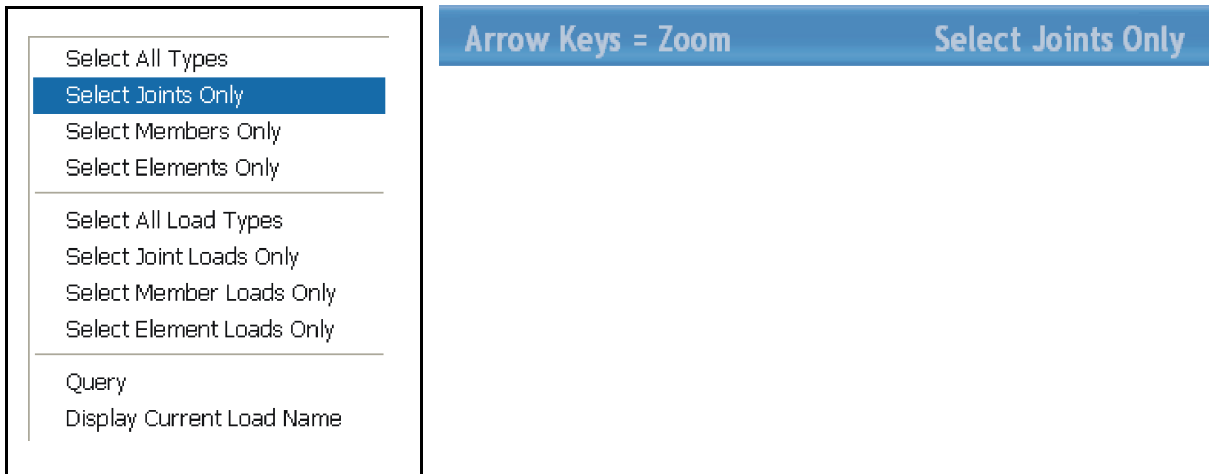
5.   GTMenu can now produce selected result graphs for a member in a separate graphical
        display window. The results available for selection are local y displacement, local z
        displacement and local force and moment diagrams and envelopes.

        The modified Results pulldown and the new Graph Member Results dialog are shown
        below:
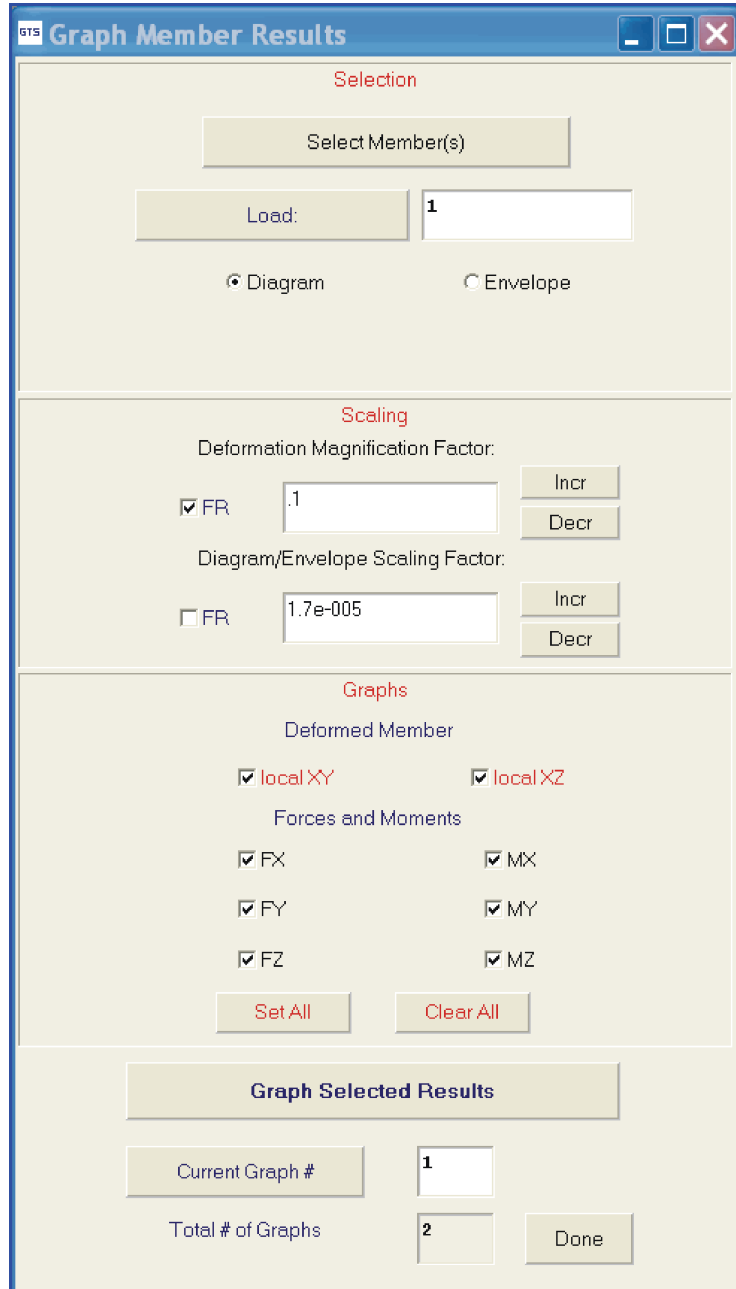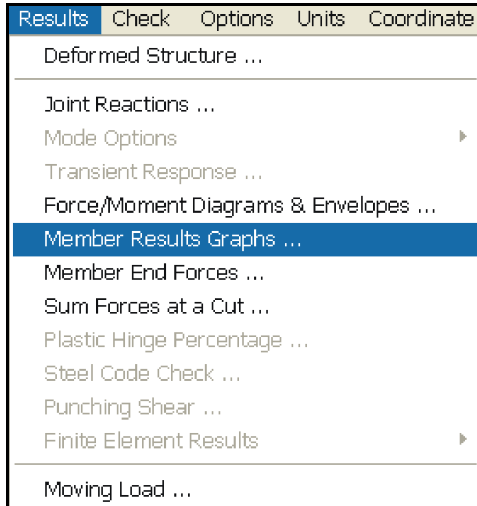
An example of the selected result graphs produced in a separate window is shown below:
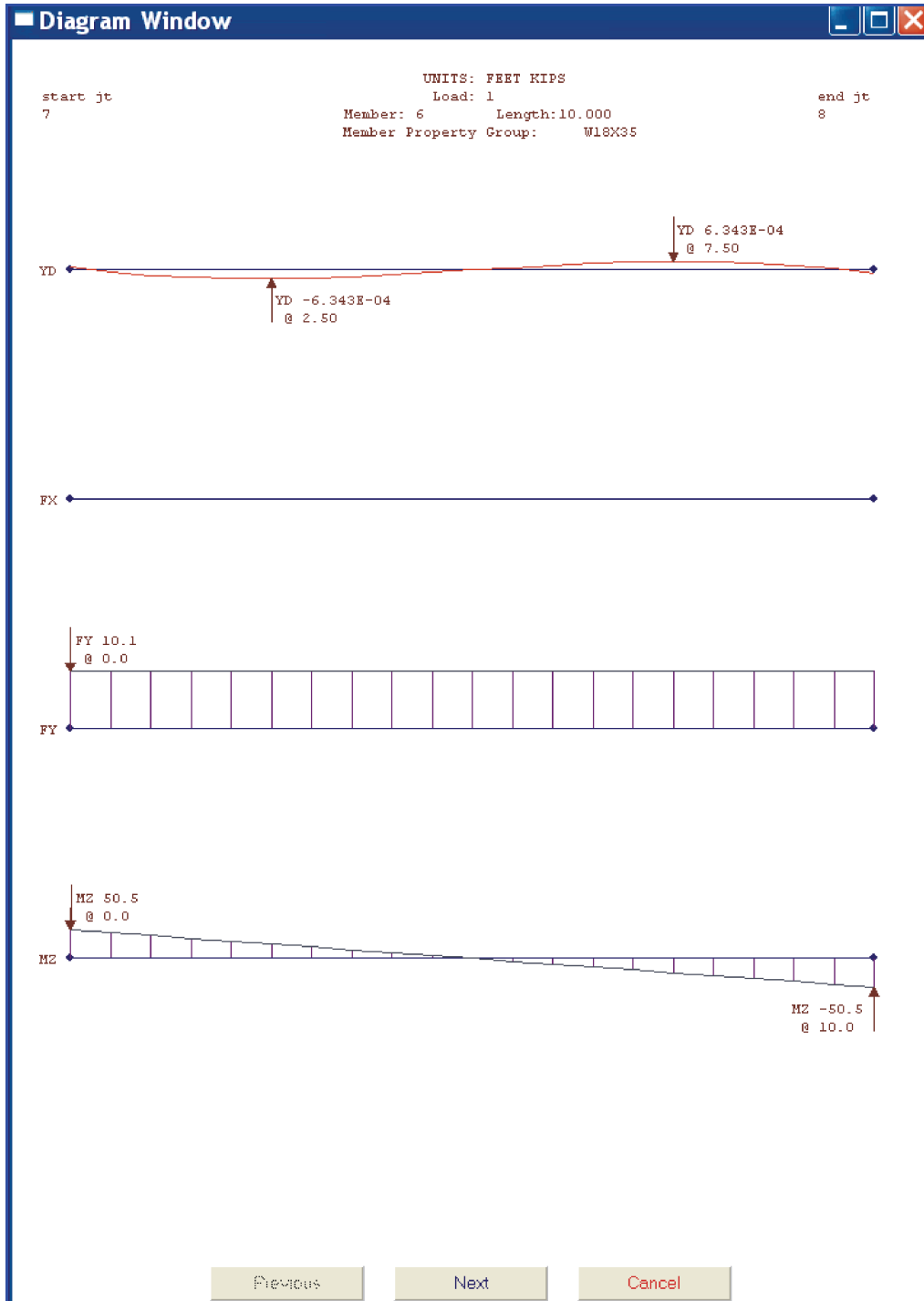
Graphs for multiple members are available by clicking on the Next button shown above. In addition, the graphs may be printed or edited in the Scope Editor.

6.   Finite element edges and faces may now be labeled using the revised Display Model
      dialog which is shown below:

7.   An automatic mesh feature for 2D elements has been implemented as a pre-release
      feature on an experimental basis to mesh structures using only the structure's outline
      defined by Construction Lines including internal boundaries and line constraints. Users
      should save their models before using this feature due to its experimental nature. This
      feature is called GTMesh and is available using the hotkey <Ctrl>8 (press Ctrl and 8
      simultaneously).

      The GTMesh dialog is shown below which allows the user to specify the external and
      internal boundaries and constraints as well as setting the mesh spacing.

An example of the boundary outline including the boundary around an internal opening is shown below:
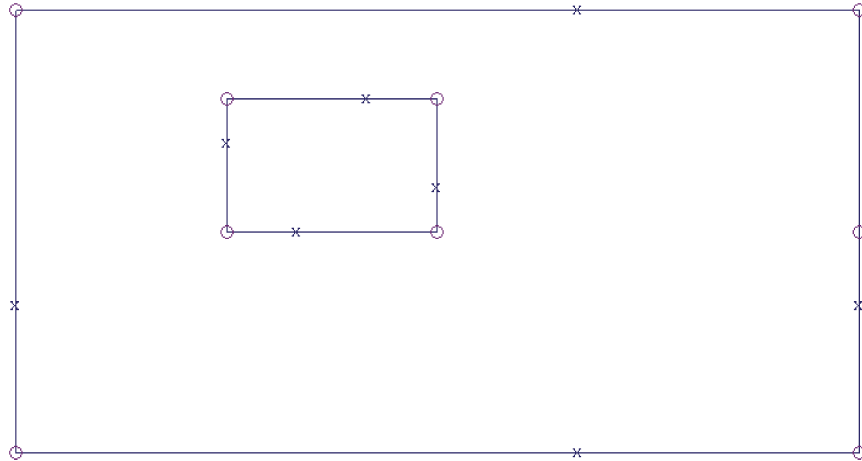
The resulting mesh using triangular elements (SBHT6) is shown below:
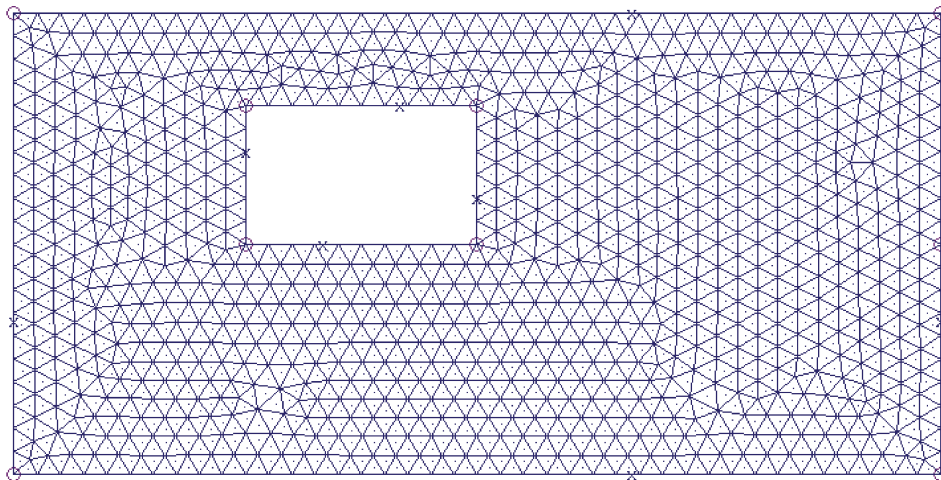
A mesh generated for the same region using quadrilateral elements (SBHQ6) is shown below:
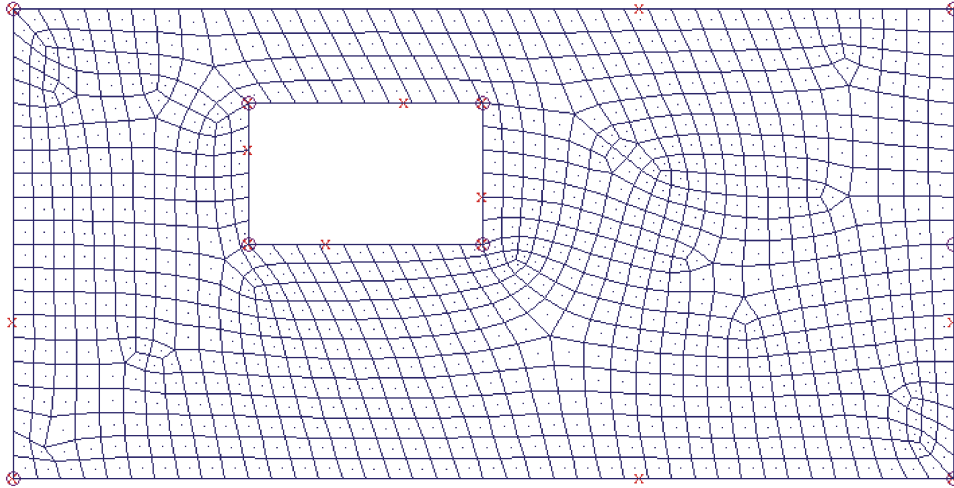
8.   When refining a finite element mesh, there is now an option to split frame members which are located in the mesh in the same way automatically. The revised Refine a Finite Element Mesh Dialog is shown below:
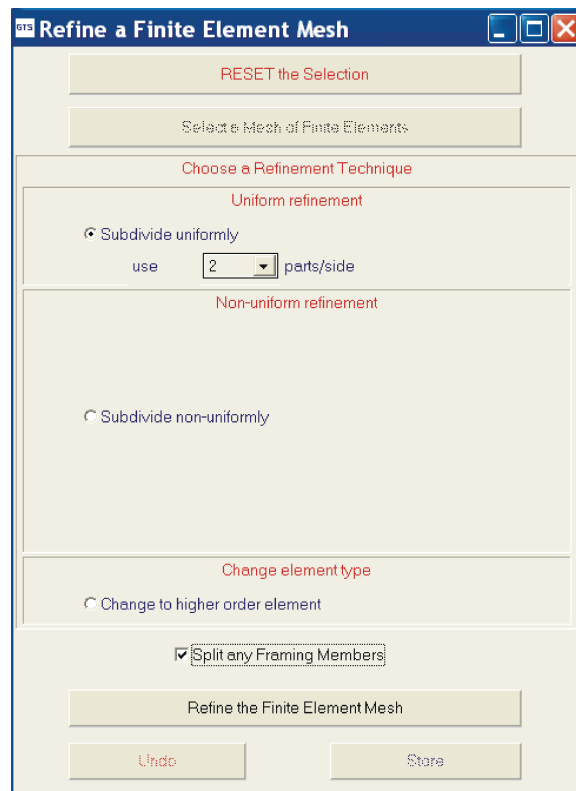
An example illustrating the automatic simultaneous refinement of members attached to a finite element mesh is shown in the following figures with the members labeled before and after the refinement to indicate the new members which were automatically created:
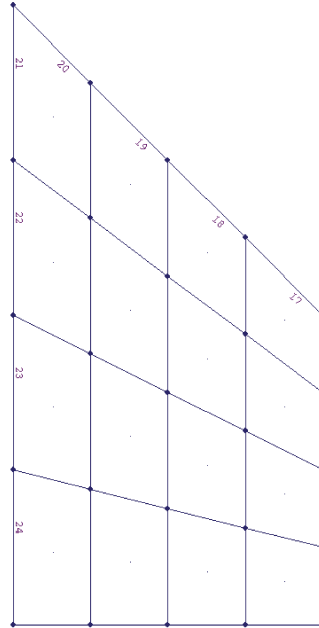
Before Refinement with original members labeled
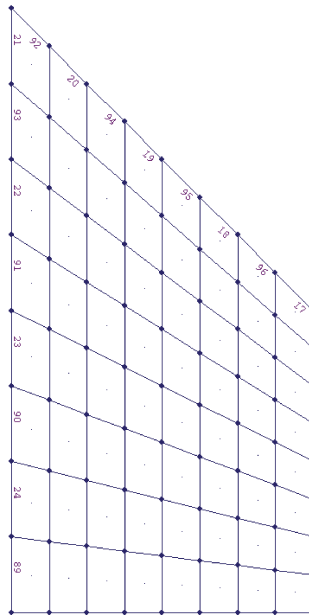
After Refinement with new split members labeled

9.   Redraw Solid will now draw members specified using the CHANNEL and POLYGONAL TUBE option of the MEMBER PROPERTIES Command. Section Names for Model Display, and Descriptors for the Property Legend Dialog are created for these shapes as well as for I-SHAPES. The naming convention is as follows:

Section Names for Model Display.

Shape	Prismatic Member	Variable Member
I-SHAPE	I(YD)x(ZD)	varISHP
CHANNEL	C(YD)x(ZD)	varCHAN
POLYGONAL TUBE	P(SD)-(OD)	varPOLY

Descriptor for Property Legend Dialog

Shape	Prismatic Member	Variable Member
I-SHAPE	I (YD)x(ZD)x(WBTK)	I (YD)x(ZD)
CHANNEL	C (YD)x(ZD)x(WBTK)	C (YD)x(ZD)
POLYGONAL TUBE	Polygon (SD) (OD)x(TH)	P(SD) (OD)x(TH)

YD	=	Depth of the member.
ZD	=	Width of the flange.
WBTK	=	Web thickness.
SD	=	Sides in the polygon.
OD	=	Outer diameter through the corners.
TH	=	Thickness of the Polygonal Tube.

10.   GTMenu now allows Landscape printing of graphics. Select Portrait (the default orientation) or Landscape from the File menu. The current orientation is shown by a check on the appropriate line in the File menu, and is remembered during the current GTSTRUDL session, but the next time you run GTSTRUDL, the default orientation will return to Portrait. In addition, when Print Portrait or Print Landscape is chosen from the File menu, the standard printer selection dialog will be displayed.

Print Preview and Edit will open the Scope Editor with the image in the current orientation. The Print button on the Button Bar will use the current orientation without the option to select a printer and send the current display to the default printer.

No automatic determination of the "proper" orientation is done, so you can use the Print Preview and Edit option to determine the proper orientation for the current display. Note that selecting an orientation (Portrait or Landscape) and then canceling the Printer Selection dialog will change the orientation without actually printing.

The new File menu:

GTMenu Structural Modeling Tool Version 32	
File Create Edit Display Results Check Optio	Print Portrait Ctrl+P
Restore Model ...	✔ Print Landscape Ctrl+L
Save Model ...	Print Preview and Edit
Generate GT STRUDL text input ...	
Page Setup ...	
✔ Print Portrait Ctrl+P	
Print Landscape Ctrl+L	
Print Preview and Edit	
Exit GTMenu	
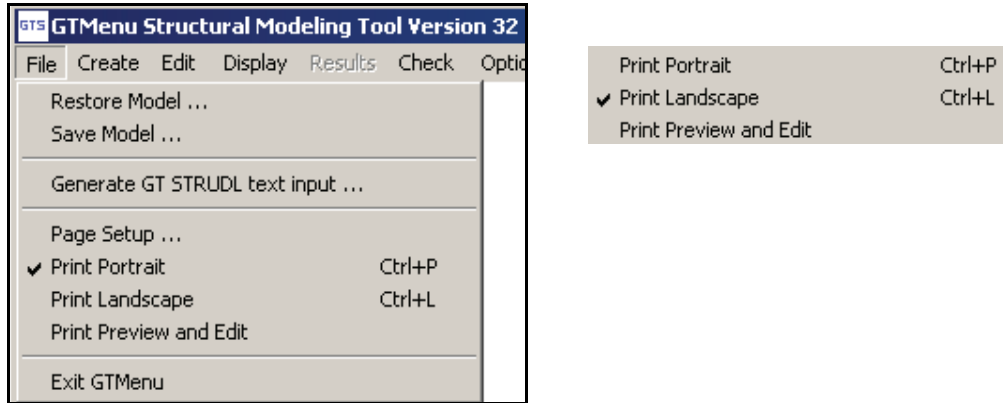
Note: A new sample template (border box) for Landscape printing is supplied in the 32 installation folder: ...GTStrudl\32\LandscapeTemplate.ssc

11.   You may now produce a contour plot of the mode shapes for a model which contains two or three dimensional finite elements. The revised Results pulldown dialog with the new Mode Contours option available under Mode Options is shown below:

Results Check Options Units Coordinate System Utilities Help		
Deformed Structure ...		Display Model
Joint Reactions ...		
Mode Options ▶	Mode Shape ...	
Transient Response ...	Mode Contours ...	
Force/Moment Diagrams & Envelopes ...	m mouse.	
Member Results Graphs ...		
Member End Forces ...	**Arrow Keys** =	
Sum Forces at a Cut ...		
Plastic Hinge Percentage ...		
Steel Code Check ...		
Punching Shear ...		
Finite Element Results ▶		
Moving Load ...		

The Element Mode Contours dialog and an example of the a mode shape contour plot are shown below:
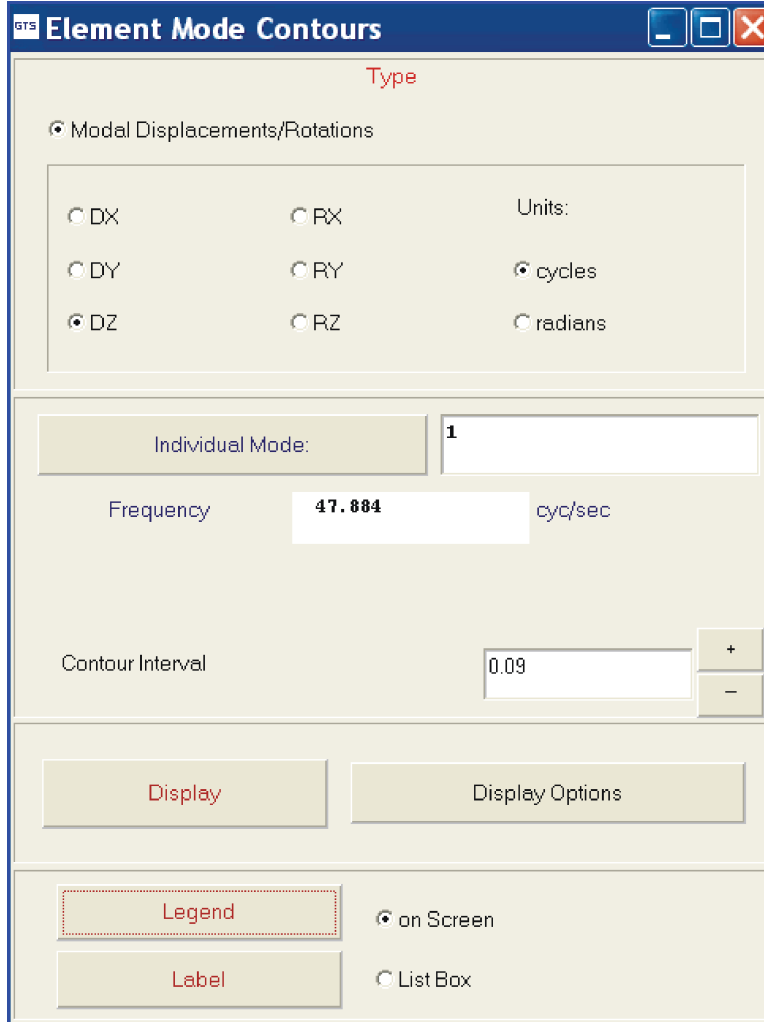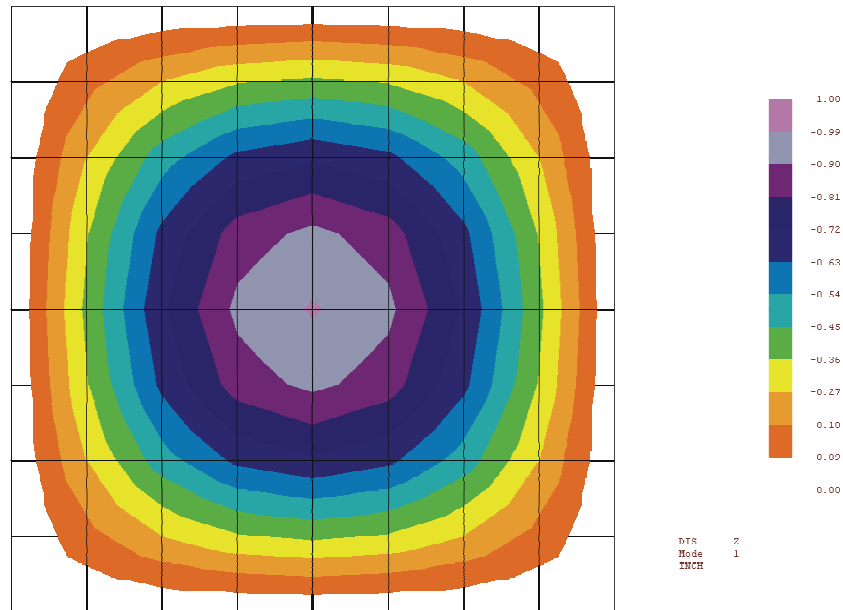
12.   A new option has been added to the Results pulldown which will allow the user to
       graphically select the joints and elements to obtain the resultant forces along a cut which
       is parallel to a global plane. The modified Results pulldown is shown below:
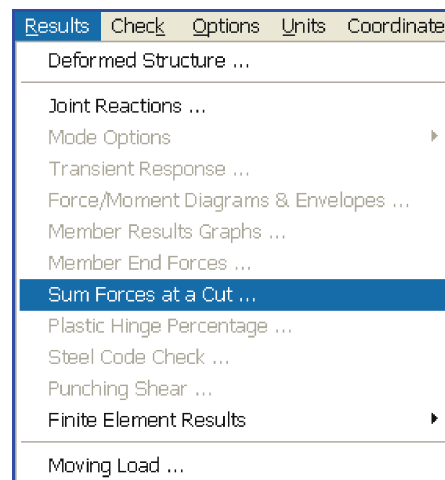
The new Sum Forces dialog is shown below along with the resulting output from selecting joints and elements using this dialog:
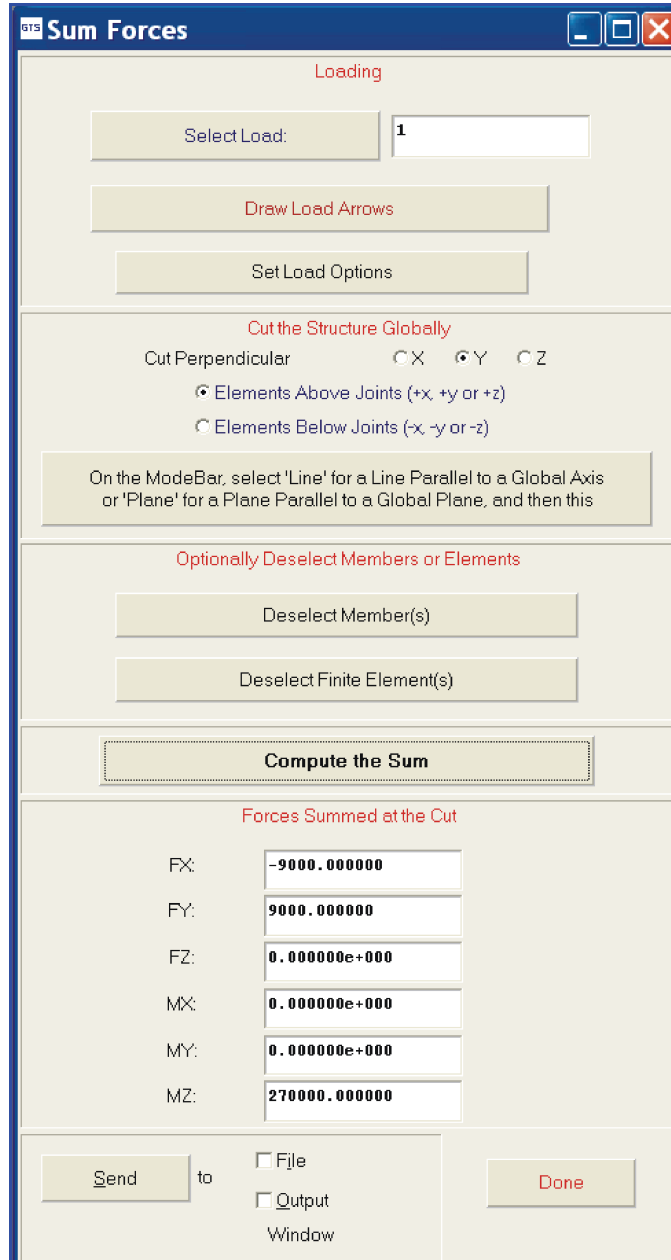
13.   Redraw Solid and Display Model Profile functions will now draw the nine Unistrut
        Profiles which are available in GTSTRUDL as shown below:
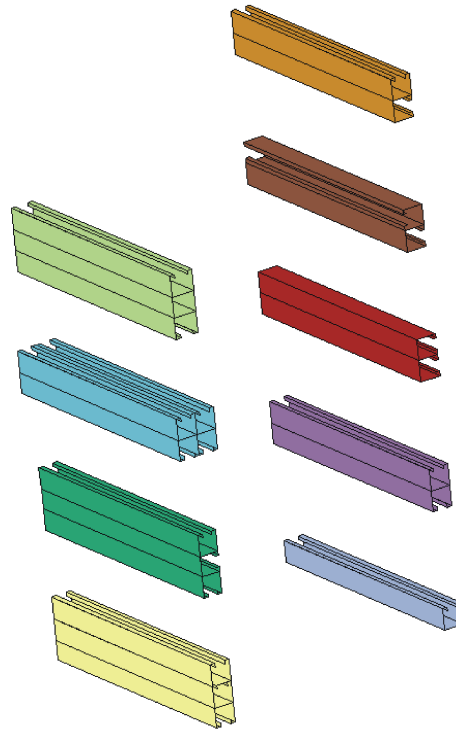
14.   Multipoint and rectangular fencing are now available in the revised Mode Bar as shown
        below:

○ Rectangular	○ Include Only Completely Enclosed Items
○ Multi-point	○ Include Items Clipped and Enclosed within the Closed Fence Boundary
○ EXIT Fencing	○ Include Only Items Clipped by the Specified Lines

| ⊙ Hit | ○ List | ○ All | ⊙ Fencing | ○ Line | ○ Plane | ○ Volume | ○ Bound |
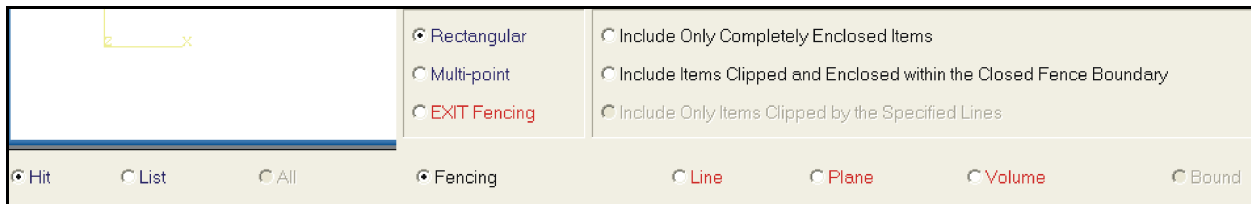
        A multi-point fence is specified as a sequence of vertices, while a rectangular fence is
        specified as a selection window (diagonally opposite points). In either case, the options
        to select only what is totally enclosed or to include items that cross a fence boundary are
        available for joint, member and element selection.

15.   The Display Parameters dialog has been enhanced so users may now change the Units in the dialog and also display the following additional parameters:

LX, FRLX, FRLY, FRLZ, FRUNCLF, TBLNAM, Yield Strength and Tensile Strength.

The enhanced Display Parameters dialog is shown below:
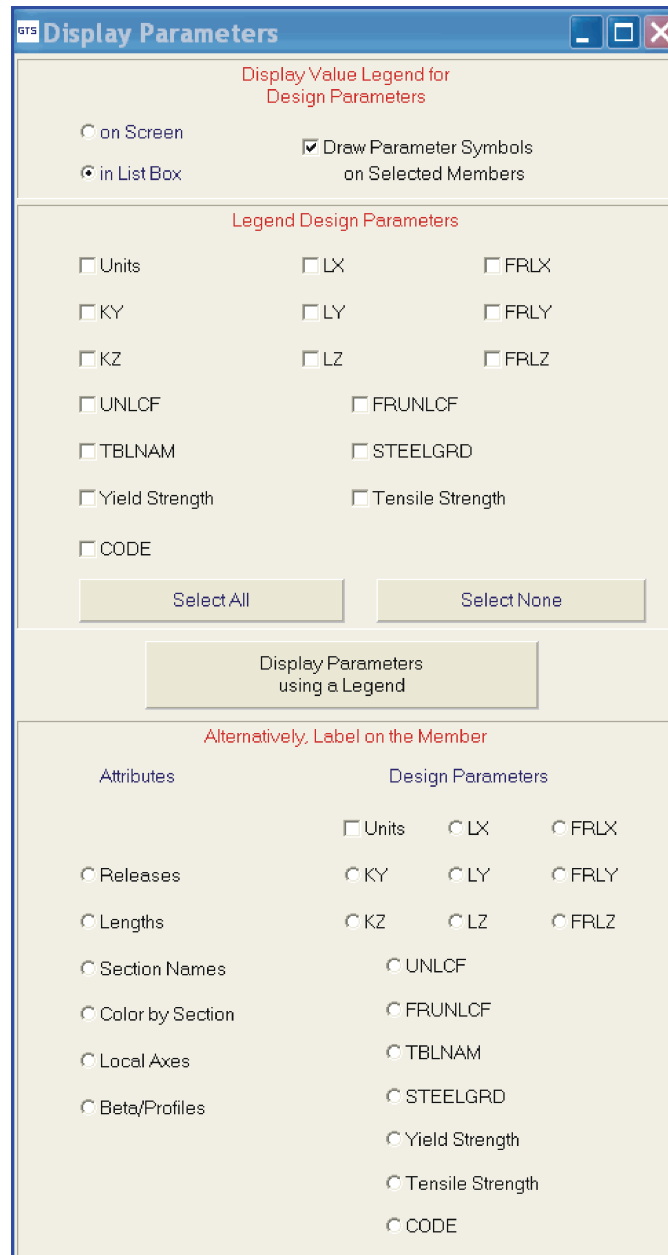
16.   The hotkey "L S" have been added to Label Supports.

17.   When Refining a Finite Element Mesh and changing to a higher order element, the mapping shown below is now followed which shows the lower order element and the new higher order element:

        Plane stress triangles map to the LST element

        Plane stress quads map to the IPQQ element

        Plate bending triangles produce an error saying unavailable

        Plate bending quads map to the IPBQQ element

        Tridimensional 6 node TRIP maps to the WEDGE15 element

        Tridimensional 8 node bricks map to the IPSQ element

        Plate triangles produce an error saying unavailable

        Plate quads produce an error saying unavailable

18.   The Generate Input File feature now generates an input file which includes the following:

        Nonlinear Effects

        Nonlinear Spring Properties

        Nonlinear Spring Elements

        Object commands and group titles

        Elastic connections at the ends of members

        TYPE RIGID PINs

19.   When checking for duplicate items such as joints, members or elements, the duplicates are now labeled in the Graphical Display window if display is set for joints, members or elements in the Label Settings dialog.

20.   When checking for floating joints, the floating joints are now labeled in the Graphical Display window if display is set for joints in the Label Settings dialog.

21.   Finite elements are now labeled closer to their centroid.

22.   Force and Moment Diagrams and value labels can now be rotated according to the Beta angles on the members. A FY Diagram is plotted in the local xy plane by default; a FZ Diagram is plotted in the local xz plane by default; a MY Diagram is plotted in the local xz plane by default; and a MZ diagram is plotted in the local xy plane by default.

23.   The Inquire dialog no longer disappears when the Graphics Window is activated.

24.   Member loads on cable elements can now be displayed.

25.   Split Members is now able to split ALL members. The ALL option is now available on the Mode Bar.

26.   You may now create new Member Property Groups when creating members. A new Add Member Property Group option has been added to the Section Property Group dialog during member creation.

27.   Member Elastic Connections are now listed in the Inquire Output pop-up.

28.   The efficiency of splitting members has been improved.

29.   The color used for animation of mode shapes is now the same as the Attribute Color set for mode shape display.

30.   Refining a finite element mesh is somewhat faster now in that it does not display the refinement until all elements have been split. Previously, the elements would be drawn as they were split.

31.   Input files created from the File - Generate GT STRUDL text input ... menu pick or from the GTMENU GENERATE INPUT FILE command now include loadings created with the CREATE LOADING COMBINATION TYPE RMS and CREATE LOADING COMBINATION TYPE ABSOLUTE commands (Section 2.1.11.3.4, Volume 1). Because these loadings require results to exist, they are generated as comments, preceded by an informational comment, which will be one line in the generated input file:

```
$    This load must come after analysis commands.
$    Remove the $ to use this loading command.
```

For example, if in the generated input file, load 2002 appears like this:

```
$    This load must come after analysis commands...
$    CREATE LOADING COMBINATION  2002 TYPE RMS   -
$      SPEC    2001           1.0000    999              1.0000
```

Add the appropriate analysis commands and remove the $ comment characters for load 2002:

```
COMPUTE RESPONSE SPECTRUM DISPLACEMENTS FORCES REACTIONS -
      MODAL COMBINATIONS RMS CQC

UNITS CYCLES SECOND
FORM MISSING MASS LOAD 999 'MISSING MASS LOAD' FROM RESPONSE SPECTRUM -
      LOAD 1001 CUTOFF FREQUENCY 8.529112 DAMPING RATIO 0.10
LOAD LIST 999
STIFFNESS ANALYSIS

CREATE LOADING COMBINATION  2002 TYPE RMS  -
      SPEC     2001              1.0000    999              1.0000
```

32.   The Button Bar now has hotkeys, as follows:

       \<ctrl>p for print

       \<ctrl>q for query

       \<ctrl>a for annotate

       \<ctrl>t for label settings

       \<ctrl>z for zoom

       \<ctrl>r for rotate

       \<ctrl>d for redraw wireframe

       \<ctrl>s for redraw solid

33.   Graphics hotkeys have been adapted to the Diagram window, produced from the Member Results dialog (Results -> Member Results Graphs):

       specifically,   "m"   = inquire for the current member;

                       "j"   = inquire for the current joints.

2.6   **GTSTRUDL Output Window**

1.   The Macros dialog has been improved to allow the use of a Personal Macros folder not in
        the Common Macros folder, which previously was the only option. Separate Personal
        Macros are only available if you enable them in the Macros page in the GTSTRUDL
        Options dialog. If the "Use a separate Personal Macros folder" box is not checked,
        Personal Macros are assumed to be located in the Common Macros folder, which is the
        folder referenced in the "Macro folder location" box. The Personal Macros folder option
        was added in version 31, but was not noted in the Release Guide.
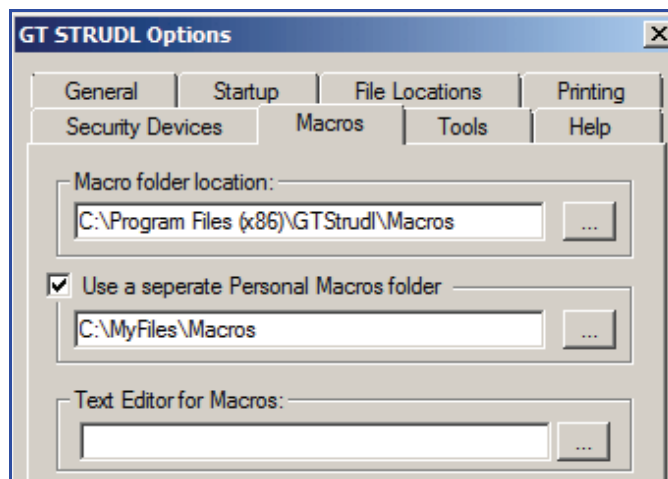
2.   The "Display Response Spectrum File Data" dialog has been enhanced by the addition of
        a check box option that allows you to indicate that the response spectrum acceleration
        values from the selected curve are to be displayed in terms of g's. The check box,
        entitled "Acceleration Unit = g," is shown in check mode below:
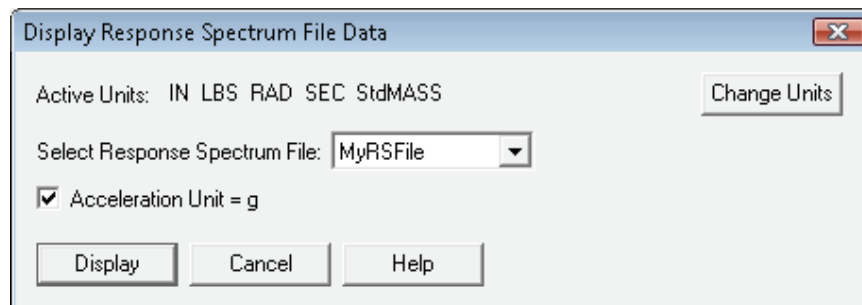
3.   The Edit - Paste verification message size has been increased to 15 lines from 10, and for
        paste buffers larger than 15 lines, the first 10 lines of the buffer are displayed plus the last
        5 lines of the buffer.

2.7   Model Wizard

1.   The Finite Element Mesh Wizard (2D Mesh) has three new elements added: PBMITC, SBMITC and Q6CDRL. See the documentation for these new elements included in this Release Guide and Volume 3 of the GTSTRUDL Reference Manual.

2.8   Nonlinear

1.   A new optional parameter, NGP i_{NGP}, has been added to the NONLINEAR EFFECTS, PLASTIC SEGMENT option that provides for the specification of either two or three Gauss quadrature points for the numerical integration of the plastic segment equilibrium equations. The specification of NGP 3 provides for improved plastic segment accuracy over a greater range of segment lengths when compared to that associated with NGP 2, the default value that was originally implemented as a fixed constant.

        Example

                NONLINEAR EFFECTS
                        PLASTIC SEGMENT NGP 3 END -
                        FIBER GEOMETRY NTF 2 NTW 1 NBF 14 ND 12 LH 10.0 -
                        STEEL FY 50.0 FSU 50.00001 ESU 1.0 MEMBER 'COL4' 'COL8'

        Documentation

                The NONLINEAR EFFECTS Command; Section 2.5.2.1, Volume 3 of the GTStrudl Reference Manual.

2.   In previous versions, member loads could not be applied to members for which the PLASTIC SEGMENT nonlinear effect was specified. This restriction has been removed and all member load types are now supported for the PLASTIC SEGMENT nonlinear effect.

        Documentation

                The NONLINEAR EFFECTS Command; Section 2.5.2.1, Volume 3 of the GTStrudl Reference Manual.

3.   As of Version 32, the presence in the structural model of nonlinear elements of type(s) IPCABLE, NLS, NLS4PH, and the friction bearing isolation element (NLSFBI), will prevent the execution of any linear static or dynamic analysis operation. An example of a typical error message that is produced if this situation occurs is shown as follows:

```
**** ERROR_CKNLEL --      Nonlinear members or elements detected during the
                          execution of a linear stiffness and/or dynamic
                          analysis.  Such members/elements are not permitted
                          for linear analyses.  Scan mode entered and analysis
                          terminated.
```

If the nonlinearity present in a model is due only to specifications by the NONLINEAR EFFECTS command (geometry, tension/compression-only, plastic hinge, etc.) then all affected members will be assumed linear for any subsequent linear analysis operation, static or dynamic.

2.9   Offshore

1.   The offshore APIWSD21 punching shear check now can perform a code check for *joints with thickened cans*. Joints with thickened cans are discussed in the Section 4.3.5 of the API WSD 21st Edition. Also see GTSTRUDL's detailed discussion of joints with thickened cans in Section 4.4, Provisions 4.3-4, (Pa)c, (Qu)c, and (Qf)c of the APIWSD21 users reference manual.

There are four new parameters applicable to the *joints with thickened cans* option of the APIWSD21which are described as follows:

Parameter Name	Default Value	Meaning
Thickened Can Parameters		
Tc	0.0	Thickness of the chord thickened can (Figure 2.8-1). A user specified value initiates allowable capacity computation according to the Section 4.3.5, *joints with thickened cans* of the API WSD 21st Edition. Chord with thickened can capacity computation requires the user to specify values for parameters 'Tc', 'a-Can', 'b-Can', and 'c-Can'.
a-Can	0.0	Thickened can distance 'a' to the brace 1 (Figure 2.8-1). Thickened can capacity computation requires the user to specify a value for parameter 'Tc'.
b-Can	0.0	Thickened can distance 'b' to the brace 2 (Figure 2.8-1). Thickened can capacity computation requires the user to specify a value for parameter 'Tc'.
c-Can	0.0	Thickened can distance 'c' to the brace 3 (Figure 2.8-1). Thickened can capacity computation requires the user to specify a value for parameter 'Tc'.
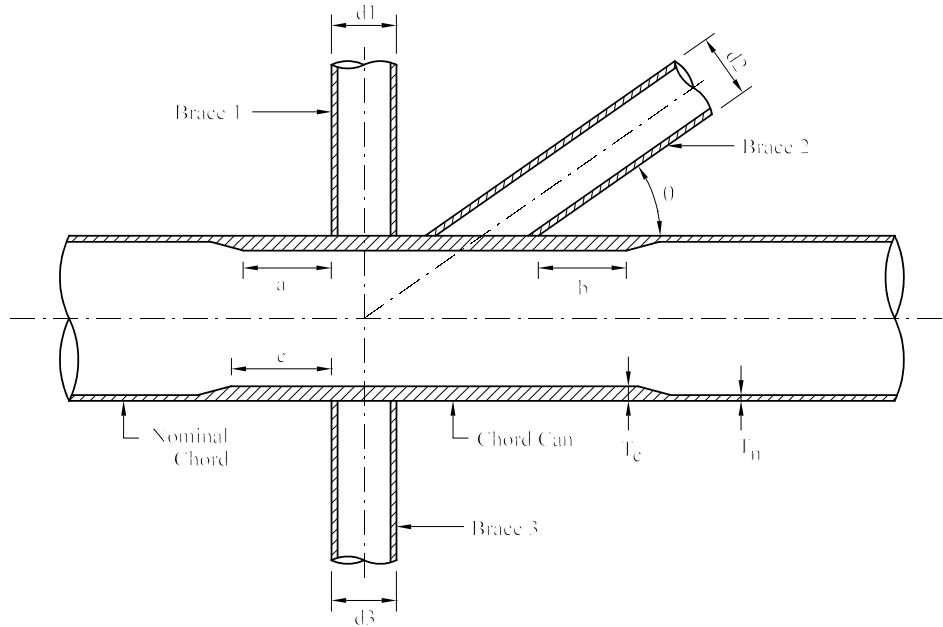
Brace	Length, L_c
1	$2a + d_1$
2	$2b + d_2/\sin\theta$
3	$2c + d_3$

Figure 2.8-1   Chord thickened can geometry

2.   The offshore steel design code APIWSD21, which is based on the *API Recommended
        Practice 2A-WSD (RP 2A-WSD) Twenty-First Edition steel design code for Basic
        Stresses, Hydrostatic Pressure, and Punching Shear*, has been moved to the release
        status. The 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 then *APIWSD21: API RP 2A-WSD, 21st Edition* in the
        GTSTRUDL Output Window.

3.   A new *International Standard ISO 19902:2007(E), First Edition, 2007-12-01, Petroleum
        and natural gas industries – Fixed steel offshore structures* has been implemented as a
        prerelease feature. The GTSTRUDL code name for this new offshore code is "19902-
        07". The 19902-07 code performs *Basic, Hydrostatic Pressure, and Punching Shear
        stresses* check according to *International Standard ISO 19902:2007(E)*. This new code,
        19902-07, may be used to select or check Circular Hollow Sections (Pipes). The
        prerelease documentation for the 19902-07 code may be found by selecting the help
        menu and then Reference Documentation, Reference Manuals, Offshore Loading,
        Analysis and Design, and 19902-07: ISO 19902:2007(E), First Edition in the
        GTSTRUDL Output Window.

4.   The GTSELOS WAVE command has been enhanced to include the selection of an
        updated version of the STREAM FUNCTION wave model (Section 7.10, Volume 7,
        User Reference Manual). The original implementation of the stream function wave
        model was due to Dean (1965) and demonstrated significant deficiencies over a wide
        range of water depths. The enhanced WAVE command provides for the additional
        selection of the Fenton (1988) stream function model, which is a higher order Fourier
        Series solution that produces significantly improved solutions, particularly in shallow
        water depths.

2.10   Steel Design

1.   Efficiency improvements have been added to the EC3-2005 code. Time to code check
        (CHECK) or design (SELECT) for jobs with large number of loadings have been
        reduced. Also the GTSES option has been improved to allow models with a large
        number of load cases to be processed more efficiently during a code check or design.

2.   Under previous versions of GTSTRUDL, the message below indicates that nonlinear
        analysis is required for code checking based on LRFD codes:

```
 **** WARNING_STTECC -- Code check based on the LRFD code requires nonlinear analysis.
                        Nonlinear analysis has not been performed for the following loadings:
                        1
```

        According to the AISC 13th Edition Code (GTSTRUDL AISC13 code), nonlinear
        analysis is required regardless of the method (LRFD or ASD) used. For GTSTRUDL
        Version 32, the message above has been modified to reflect this AISC 13th Edition
        requirement:

```
 **** WARNING_STTECC -- Code check based on the AISC13 code requires nonlinear analysis.
                        Nonlinear analysis has not been performed for the following loadings:
                        1
```

3.   The steel design code EC3-2005, which is based on the *Eurocode 3: Design of Steel
        Structures, Part 1-1 General rules and rules for buildings, DS/EN 1993-1-1:2005(E)*, has
        been moved to release status. The documentation for the EC3-2005 code may be found
        by selecting the *Help* menu and then *Reference Documentation*, *Reference Manuals*,
        Steel Design and then *EC3-2005: Eurocode 3, 2005* in the GTSTRUDL Output Window.

4.   When a value of "YES" has been specified for the parameter "TowerCK," the
        slenderness ratios are checked according to the transmission tower code provisions. This
        option is applicable to the AISC13 and ASD9 codes for single and double angle cross-
        sections and was a new option that was added to GTSTRUDL Version 31.
        In GTSTRUDL Version 32, this option has been modified so that the AISC13 or ASD9
        slenderness ratios L/r, KL/r, B7 TEN, and B7 COMP are checked as informative
        provisions rather than pass/fail conditions. This means that as of Version 32, only the
        transmission tower slenderness ratios SLENT and SLENC are checked as pass/fail
        provisions. Previously, all slenderness ratios L/r, KL/r, B7 TEN, B7 COMP, SLENT,
        and SLENC (transmission tower, AISC13, and ASD9) were checked as a pass/fail
        provisions.

5.   Additional error checking for parameters CODE, TBLNAM, and STEELGRD have been
        added into the steel design parameter command. The user specified parameter values for
        the parameters CODE, TBLNAM, and STEELGRD are now checked against accepted

values and if the specified value is incorrect, an error message is given and the scan mode is set.

6.   Efficiency improvements have been made to both the member selection and code checking functions when the external file solver is active and analysis results are stored in files on external storage devices. In particular, virtual memory demands have been reduced, enabling the CHECK and SELECT commands to handle the required numbers of members and loading conditions more efficiently. The external file solver is activated by the ACTIVE SOLVER GTSES or GT64M command (Section 2.1.13.4, Volume 1) or when the GTSES or GT64M options are specified in the STIFFNESS ANALYSIS command (Section 2.1.13.2, Volume 1) or the SOLVE option of the EIGENPROBLEM PARAMETERS command (Section 2.4.5.2, Volume 3).

7.   Geometric checks in Codes N690-94, NF-2004, and NF-2007 for local stability are now stored as informative checks. This will prevent GTSTRUDL from reporting geometric checks instead of stress checks on the Code Check or Design Summary.

8.   The parameter ALSTRINC which can be used to modify the allowable stresses can now accept negative values in order to allow the designer to reduce the allowable stresses for a list of loads. This parameter is available for the ASD9, ASD9-E, UNISTRUT, APIWSD20 and APIWSD21 Codes and will be available for the N690-94 code in Version 32.

9.   The steel design code, ASCE4805, which is based on the 2005 edition of the ASCE/SEI, *Design of Steel Transmission Pole Structures* Specification has been implemented as a pre-release feature. The ASCE/SEI 48-05 Specification is based on ultimate strength methods using factored loads.

        The ASCE4805 Code may be used to select or check any of the following shapes:

        Design for axial force, bi-axial bending, and torsion:

                Pipes
                Regular Polygonal Tube
                Structural Tubing

        The documentation for the ASCE4805 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and "ASCE4805" in the GTSTRUDL Output Window.

2.11 Steel Tables and GTTABLE

1.   The output of the Table print commands will no longer overflow the output field and be printed as "**************" when the value is larger than the reserved field. The values that are too large for the reserved fields are automatically converted to an exponential format before being printed. This will prevent any print overflow problem.

2.12 Base Plate Wizard

1.   The command "SET ELEMENTS HASHED" is now added to the GTSTRUDL input file for models greater than 200 joints to increase processing speed.

2.   The Base Plate Wizard Release Guide can be viewed from the Help menu.
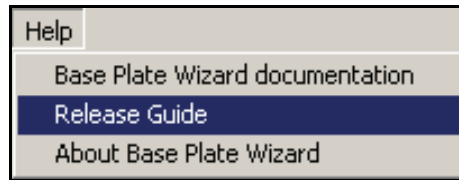
3.   The Nonlinear Analysis option for Generate Input now includes the ability to request NONLINEAR ANALYSIS GTSES, which uses the external file solver. This can run much faster for very large models, but note that the external results files will now be included in your working directory.
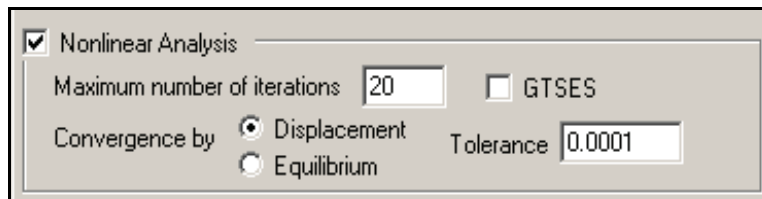
4.   The Open New Attachment File dialog (available from the Options menu) has been changed to give more control over handling attachment data not in the default Attachment File. The changed dialog looks like this:

If you check the "Change my default attachment file to this one" box, these new features appear:
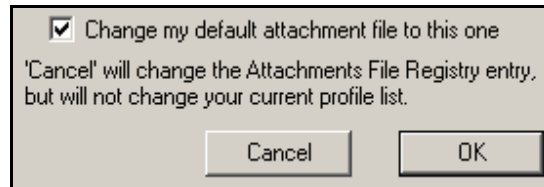
See Appendix A2 in the Base Plate Wizard User Manual for an explanation of the various options.

5.   A new option has been added to Attachments to allow specification of attachment lines as "Contact Only", corresponding to lines of the attachment that are in contact with the plate, but are not welded therefore allowing the attachment line and the plate to separate under tension. This option is available with the "Edit Weld Status" button when editing an attachment. Standard profiles may have the weld status edited, but they will become 'arbitrary' (user defined) profiles upon saving. See "Section 3.2.1 Changing Weld Status for Attachments" in the Base Plate Wizard User Manual for an explanation of how to edit the weld status and how Contact Only is modeled.
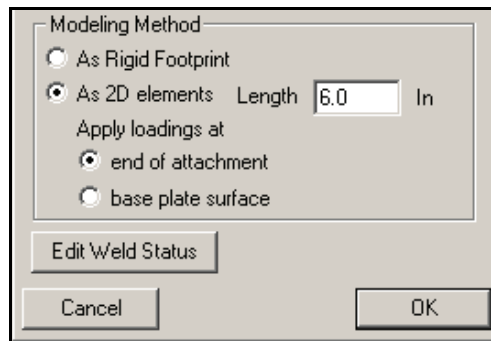
New 'Edit Weld Status' button.

6.   The Plate Stress results dialog has a new option to include the location (X and Y coordinates) of each joint. Check the 'Location' box to add new columns to the right of the display.

Joint	Load	Surf	Von Mises	S1	S2	Tmax	Theta	X	Y
61	2	Top	15520.3	17629.8	6027.0	5801.4	0.000	6.00	5.00
61	2	Bot	15520.3	-6027.0	-17629.8	5801.4	0.000	6.00	5.00
72	2	Top	13402.4	15233.1	5252.2	4990.4	0.000	6.00	6.00
72	2	Bot	13402.4	-5252.2	-15233.1	4990.4	0.000	6.00	6.00

7.   When allowable forces are specified for anchors, you can now elect to use an interaction ratio limit of 1.2. Limits for tension and shear remain fixed at 1.0.
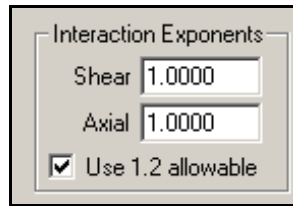
8.   New options have been added to Results Summary (Section 11.1) to add more information to the Anchor results output.
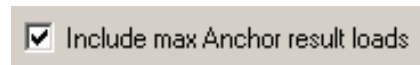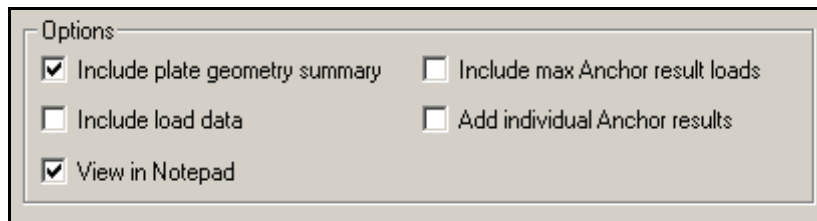
This option will add the names of the loads that caused the maximum interaction, tension and shear ratios. Load names are printed underneath the standard anchor results output.

```
Anchor P/F Intact Tension   T allow Ratio   Shear  V allow Ratio
------ --- ------ -------- -------- ----- -------- -------- -----
 B1      F  1.0625   2125.0   2000.0 1.063   706.9   1500.0 0.471
                T-1      T-1                      5
```
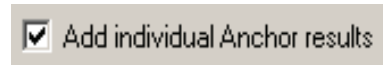

This option will output the individual anchor results ordered by loading following the standard anchor results output.

```
Individual anchor results, by load
Anchor     Load I_ratio P/F    Axial A_Allow A_Ratio    Shear V_Allow V_Ratio
------ -------- ------- ---  ------- ------- -------  ------- ------- -------
   B1      A  0.0903  P    -106.1  2000.0  0.0530     55.9  1500.0  0.0373
   B2         0.0938  P     -23.6  2000.0  0.0118    123.0  1500.0  0.0820

   B1      2  0.2021  P    -262.9  2000.0  0.1315    106.0  1500.0  0.0707
   B2         0.1252  P    -109.0  2000.0  0.0545    106.1  1500.0  0.0707
```

9.   The Anchor Results dialog has a new option to create commands for the ANCHOR
        CONFIGURATION command (Section 2.9.2 "The ANCHOR CONFIGURATION
        Command", Volume 3, GTSTRUDL Reference Manual). You can create commands to
        transfer the calculated loads for each anchor, and optionally the anchor coordinates, from
        the Base Plate Wizard to a GTSTRUDL input file used to check the anchor capacity with
        ACI Appendix D.

        Click the 'Write ANCHOR LOADS' button to bring up the Write Anchor command
        dialog.

        See Section 11. Results in the Base Plate Wizard User Guide.

10.   Two new Anchor Patterns have been added: "4 corners + interior fill" to create a grid of
        anchors, and "Arc/Circle" to place anchors in a circular arrangement. See Section 4.1
        Adding Anchors for documentation of these two new patterns. The new Anchor Patterns
        dialog looks like this:
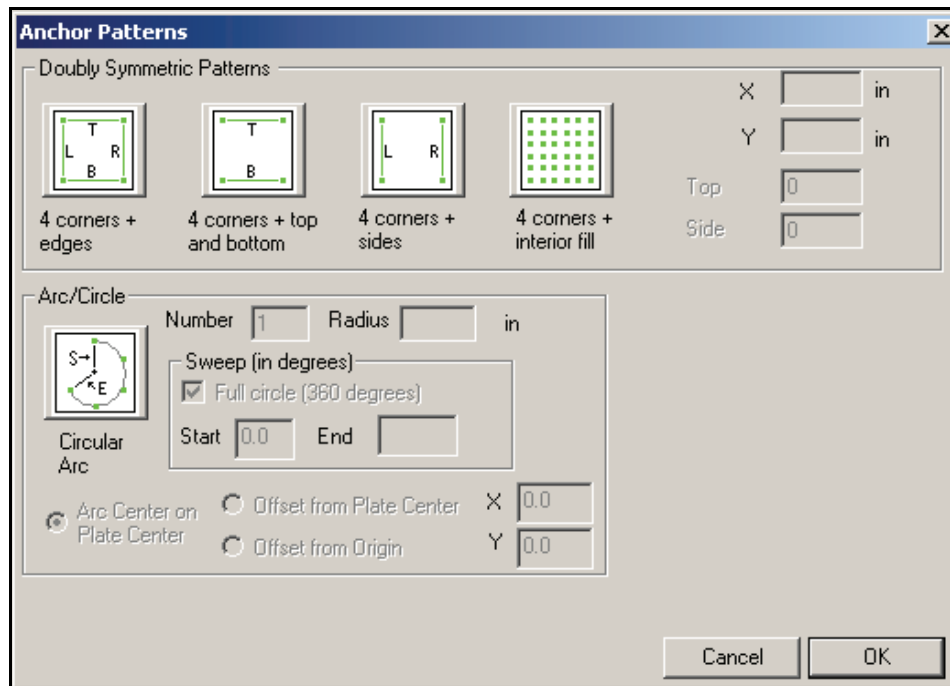
2.13   Base Plate Anchors

1.   A new capability to calculate the capacity of structural anchors in concrete by the
        methods of ACI 318-05 Appendix D has been added. Commands are provided to enter
        the anchor properties, base plate configuration and request a capacity check. The new
        commands are documented in Volume 3, Section 2.9.

        • The ANCHOR TYPE command is used to describe anchor properties such as effective
        embedment length and diameter.

$$\text{ANCHOR TYPE 'a\_name'} \begin{Bmatrix} cast\text{-}in \\ post\text{-}installed \end{Bmatrix} -$$

$$\underline{\text{HEF}} \ h_1 \ \underline{\text{DIAMETER}} \ d_1 \ \underline{\text{FUTA}} \ f_1 \ (\underline{\text{ASE}} \ a_1) -$$

$$(\begin{Bmatrix} \rightarrow \underline{\text{BRITTLE}} \\ \underline{\text{DUCTILE}} \end{Bmatrix}) \ (\underline{\text{WASHER}} \ t_1)$$

• The ANCHOR CONFIGURATION command is used to describe the geometry and properties of the base plate to be checked, along with relevant properties of the bearing surface, including edge location.

> ANCHOR CONFIGURATION 'ac_name' ('ac_title')
> > *(plate)*
> > *bearing_surface*
> > *anchor_location*
> > *anchor_loads*
> > *(rows)*
> > *(edge)*
> > *(edge_distances)*
> > *(phi_category)*
> > *(sup_reinforce)*
> END (ANCHOR CONFIGURATION)

• The PRINT ANCHOR command will output the specified anchor type or configuration data.

> PRINT ANCHOR TYPE
> PRINT ANCHOR CONFIGURATION

• The CHECK ANCHOR CONFIGURATION command checks a previously created anchor configuration with the requirements of ACI 318 "Appendix D - Anchoring to Concrete".

$$\underline{\text{CHECK}} \ \underline{\text{ANCHOR}} \ (\underline{\text{CONFIGURATIONS}}) \ (\{\rightarrow \underline{\text{ACI-05}}\}) -$$

$$(\underline{\text{SEISMIC}}) \begin{Bmatrix} \rightarrow \underline{\text{ALL}} \\ list \end{Bmatrix}$$

2.14   Scope Editor

1.   A **Circle** has been added to the Tools menu. Circles are created by holding down the left
        mouse button and then dragging the circle to determine the orientation and diameter of
        the diameter of the circle.
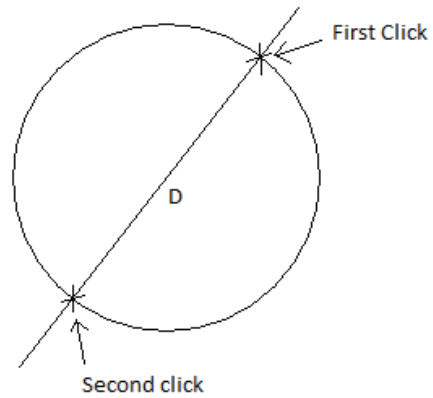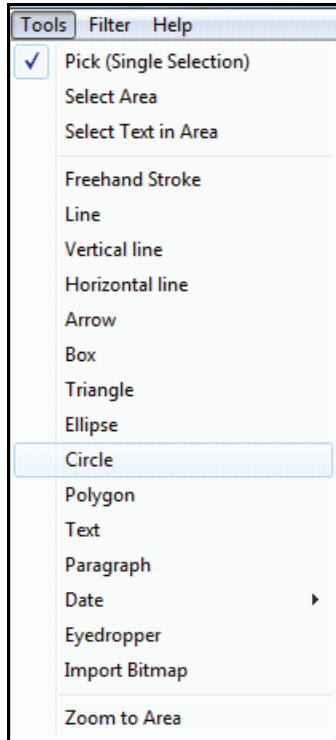
2.   **Triangle** has been added to the Tools menu. Triangles are created by left-clicking three points to indicate the vertices of the triangle.
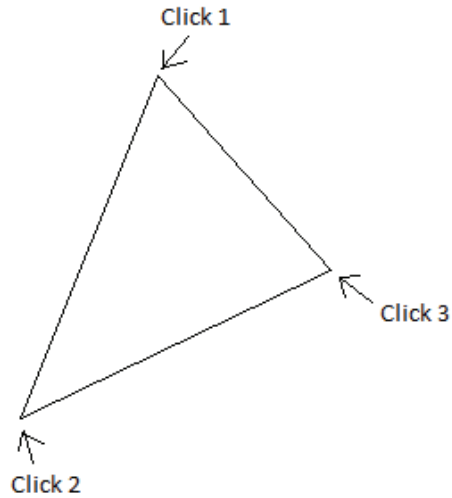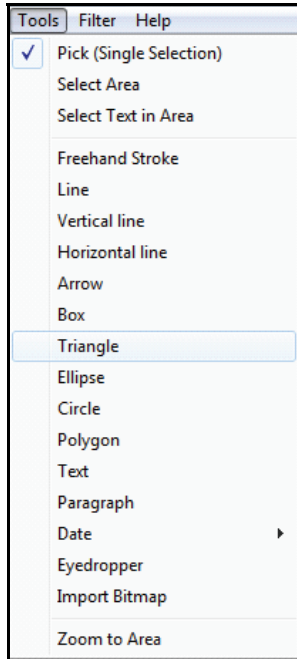
3.   **Polygon** has been added to the Tools menu. Left-click to initiate construction. A new vertex is added for each new left-click until the first vertex is clicked, closing the polygon.
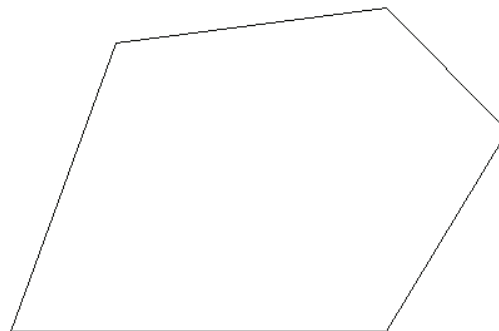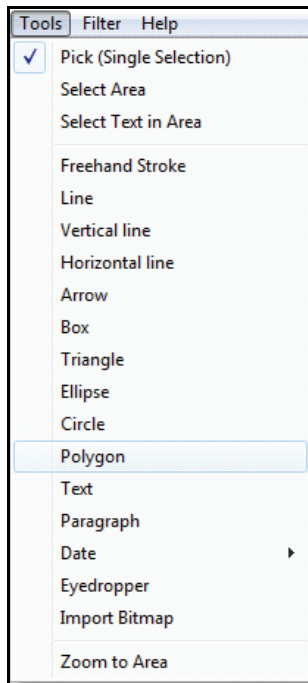
4.   The **Fill shape** and **Outline Only** options have been added to the Tools menu available from the right-mouse click pop-up menu. **Fill shape** causes the selected (highlighted) objects to become filled with the current global brush color (set with Format - Color...). **Outline only** causes the selected (highlighted) objects to become outlined shapes, without interior color.
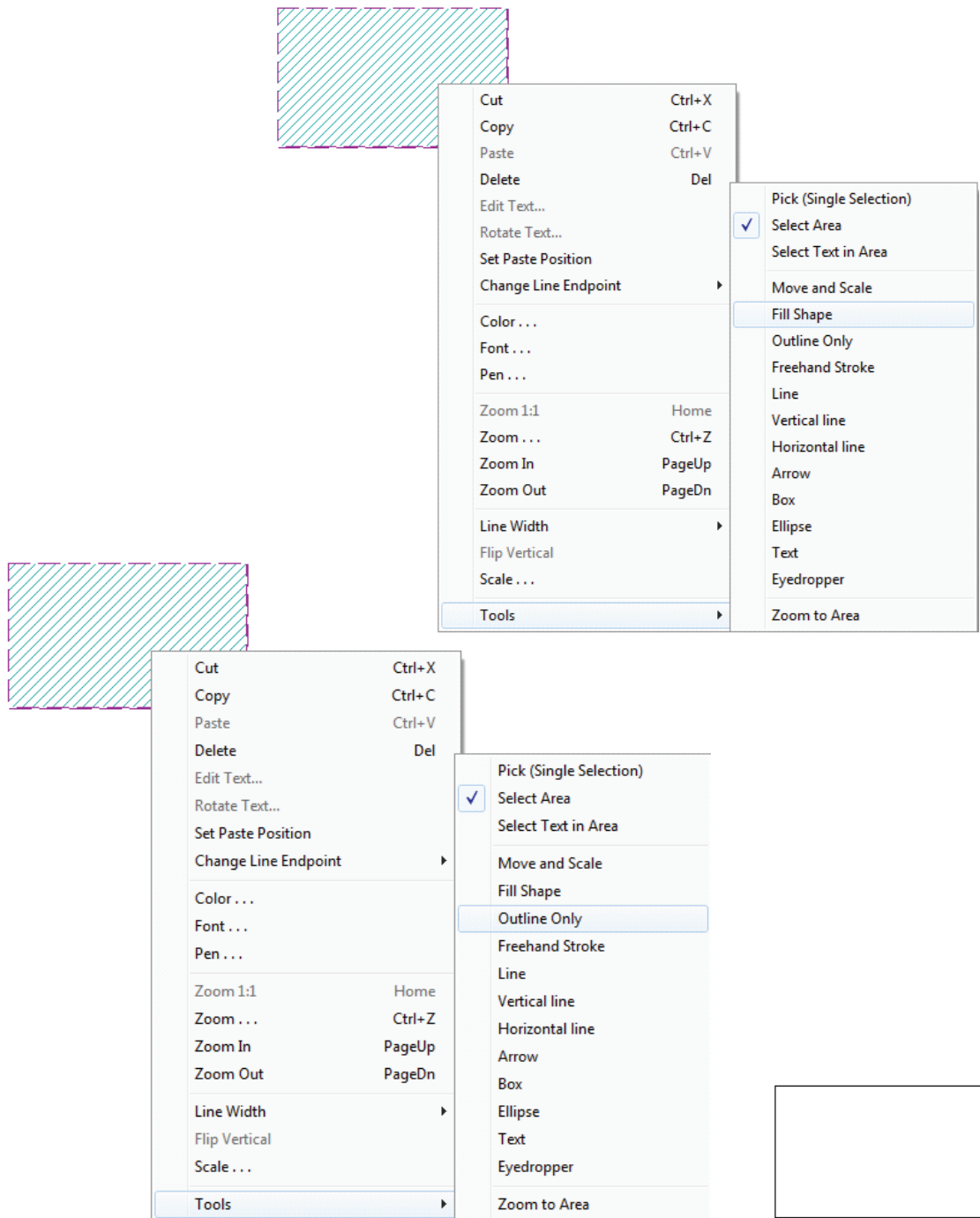
5.   **Export Rich Text** has been added to the File menu. This selection creates a file with ".rtf" extension that can be read in various Microsoft programs. This RTF file contains all elements that are currently visible on the Scope Editor.
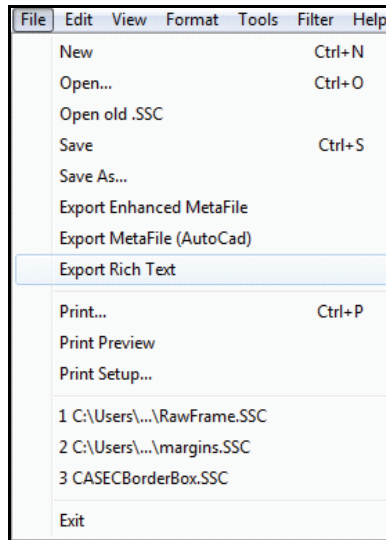
6.   The **Move and Scale** option has been added to the Tools menu available from the right-mouse click pop-up menu. This option allows many items to be moved and scaled easily.

• Select elements to be moved and/or scaled.
• Right click a selected element and choose the **Move and Scale** option from the Tools menu. This will create a bounding rectangle encompassing all the selected elements, with one control square at each corner. This bounding box will be your guide to where and what size the completed operation will look like.
• To move, place the cursor in one of the control squares. Left-click and drag the bounding rectangle to one corner of its new position. Release the mouse when the rectangle is properly placed. To leave the bounding rectangle in place (scale only), don't move the mouse before releasing.
• To scale, place the cursor in one of the control squares - the corner opposite the selected control square will be considered the anchor for scaling. Left-click and drag the corner of the bounding rectangle. The bounding rectangle will resize. When the desired size is reached, release the left mouse button. To leave the current scaling (move only), don't move the mouse before releasing.
• The selected elements will then be redrawn with the correct translation and scaling to fill the bounding rectangle.

Cut	Ctrl+X
Copy	Ctrl+C
Paste	Ctrl+V
Delete	Del
Edit Text...	
Rotate Text...	
Set Paste Position	
Change Line Endpoint	▶
Color . . .	
Font . . .	
Pen . . .	
Zoom 1:1	Home
Zoom . . .	Ctrl+Z
Zoom In	PageUp
Zoom Out	PageDn
Line Width	▶
Flip Vertical	
Scale . . .	
Tools	▶

- Pick (Single Selection)
- ✓ Select Area
- Select Text in Area
- Move and Scale
- Fill Shape
- Outline Only
- Freehand Stroke
- Line
- Vertical line
- Horizontal line
- Arrow
- Box
- Ellipse
- Text
- Eyedropper
- Zoom to Area

7.   The **Options** dialog, available from the View menu, has a new option to specify a
       Landscape template to be used when GTMenu sends a file in Landscape orientation.
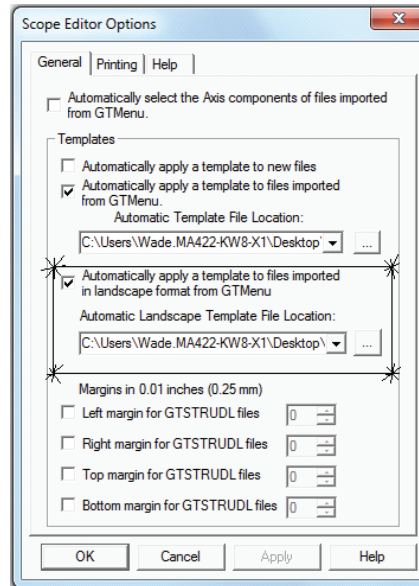
8.   A new feature has been added to Template processing, allowing you to set the margins
       for each Template within the Template itself by adding a text entry beginning with
       "<<Margins" followed by the margin sizes Left, Top, Right, Bottom in 0.01 inch (0.25
       mm) increments. Margin values are separated by spaces and/or commas. All four
       margin values are required. If this option is used, the margin values specified in the
       Options dialog are ignored. "Template processing" is what happens when you check one
       of the "Automatically apply a template..." check boxes in the Options dialog, as shown
       above.

       This new feature allows you to easily switch between Templates without having to
       respecify margins in the Options dialog.

       Example:
       See "PortraitTemplate.ssc" in C:\Program Files (x86)\GTStrudl\32. The text entry above
       the title box is:
              <<Margins 75, 75, 75, 125
       which sets the Left margin as 0.75" (19 mm), theTop margin as 0.75" (19 mm), the Right
       margin as 0.75" (19 mm) and the Bottom margin as 1.25" (32 mm).

This page intentionally left blank.

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   DBX

1.   In previous versions the results files created by the WRITE DBX MEMBER RESULTS command include results data for finite elements found in the command member list. Finite elements are now skipped.

      (No GPRF Issued)

2.   The WRITE JOINT ATTRIBUTES command will no longer abort if an undefined joint is included in the list of joints.

      (No GPRF Issued)

3.   The WRITE MEMBER RESULTS command will no longer write results for the first two nodes of finite elements in the specified member list. Finite elements are now ignored in this command.

      (No GPRF Issued)

4.   Previous versions of WRITE DYNAMIC STIFFNESS could possibly be missing the final matrix rows due to incorrectly closing the generated file. Usually, when GTSTRUDL is closed, the file buffer (unwritten data) is automatically flushed (written to the file), but for very large output files the automatic feature can fail, resulting in an incomplete output file. This has been corrected in Version 32.

      (No GPRF Issued)

3.2   Dynamic Analysis

1.   The COMPUTE MODAL DAMPING command no longer aborts when rigid body modes,. i.e. modes that have zero-value eigenvalues, have been computed by the preceding eigenvalue analysis. The computed modal damping ratio is set equal to 0.0 for such modes.

      (GPRF 2011.09)

2.   The TIMES ALL option of the COMPUTE TRANSIENT and COMPUTE STEADY STATE commands no longer limits the number of time points for which transient and steady state responses are computed to 65536. All specified time points are now processed under the TIMES ALL option of these commands. Note that the maximum permitted number of time points is 4 million (see Section 2.2, Item 2).

      (GPRF 2011.13)

3.   GTSTRUDL no longer aborts when using the LIST DYNAMIC PARTICIPATION FACTORS ORDER BY Command with the absence of mass participating in one of the principal directions.

      (GPRF 2011.04)

3.3   External File Solver

1.   When the External File Solver is active, the deletion of existing entities (i.e. joints, members, and finite elements) and the addition of new entities between successive analysis steps will no longer create inconsistencies between the number and order of the entities reflected in the analysis results data stored in the external files before the deletions/additions operations and the number and order of the entities actually present in the model following the deletions/additions operations."

      (GPRF 2010.05)

3.4   Finite Elements

1.   Incorrect results will no longer result if a RIGIDITY MATRIX is specified for elements IPSL, IPSQ and TRANS3D. The Rigidity Matrix input is not supported for

these elements as shown in the Element Dictionary in Section 2.3.4 of Volume 3 of the GTSTRUDL Reference Manuals. Now, the following error message is output and the scanning mode is entered:

**** ERROR_STERR5-4 -- RIGIDITY MATRIX IS NOT SUPPORTED FOR ELEMENT xxxx SCANNING MODE ENTERED.

(GPRF 93.09)

2.   The following warning message for temperature gradient loadings on the BPHT, BPHQ, SBHT, SBHQ, SBHT6 and SBHQ6 elements will now output the correct loading name. An example of the warning message is shown below:

****WARNING_THCE30 -- NOT ALL JOINTS FOR ELEMENT 1   IN LOADING 3   HAVE A SPECIFIED TEMPERATURE GRADIENT.   A VALUE OF 0.0 WILL USED FOR THE TEMPERATURE GRADIENT.

Previously, the warning message could output an incorrect loading name.

 (No GPRF issued)

3.5   General

1.   Changes to stiffness and flexibility matrix member properties specified by giving the new matrix property values in the MEMBER PROPERTIES STIFFNESS/FLEXIBILITY command in CHANGES mode are not reflected in the results of any subsequent analysis (stiffness, dynamic, nonlinear, etc.) in all versions of GTStrudl prior to Version 32. This problem has been corrected and such changes are now properly reflected in subsequent analysis results.

(GPRF 2011.01)

2.   The UNSYMMETRIC SECTION option of the   LIST SECTION STRESSES command has been corrected to utilize all the significant decimal values of the moment of inertia with respect to the Z axis (IZ).

(GPRF 2010.09)

3.   EXISTING MEMBERS ONLY and EXISTING ELEMENTS ONLY now work correctly when used with the CONSTANTS command.

(No GPRF issued)

4.   When multiple master joints are specified for a given slave joint, the documented action is taken (i.e. all but the first master-slave specification are ignored; Sections 2.6.5.1 and 2.6.5.2.2, Volume 3 of the reference manual); however, no message so indicating is issued. This situation has been improved. Now, a warning message is issued indicating that the second and later specifications of additional master joints for the same slave joint is invalid with scan mode turned on.

(GPRF 2011.05)

5.   The PRINT MEMBER PROPERTIES command for PIPE tables which prints the ID, OD, and TH-PIPE values are now printing these values.

(GPRF 2011.08)

An example of PRINT MEMBER PROPERTIES output is as follows:

```
{   52} >
{   53} > MEMBER PROPERTIES
{   54} >  1 TO 3   T 'WSPIPE'  'P12X33'
{   55} >  4 TO 6   T 'M/S/HP13'  'S6X12.5'
{   56} >
{   57} > PRINT MEMBER PROPERTIES
1
*****************************************
*  PROBLEM DATA FROM INTERNAL STORAGE  *
*****************************************

 JOB ID - FR418      JOB TITLE - Check PRINT MEMBER PROPERTIES for PIPE

 ACTIVE UNITS -  LENGTH          WEIGHT          ANGLE       TEMPERATURE     TIME
                 INCH            LB              DEG         DEGF            SEC


 MEMBER PROPERTIES----------------------------------------------------------------------------------------/
 MEMBER/SEG TYPE                      AX       AY       AZ       IX       IY       IZ       SY       SZ
                                      YD       ZD       YC       ZC       EY       EZ
                                      ID       OD       TH-PIPE  SC

 1       TABLE     WSPIPE         9.817    4.909    4.909  383.700  191.824  191.824   30.090   30.090
                   P12x33                 12.750   12.750    6.375    6.375    0.000    0.000
                                         12.250   12.750    0.250    0.000

 2       TABLE     WSPIPE         9.817    4.909    4.909  383.700  191.824  191.824   30.090   30.090
                   P12x33                 12.750   12.750    6.375    6.375    0.000    0.000
                                         12.250   12.750    0.250    0.000

 3       TABLE     WSPIPE         9.817    4.909    4.909  383.700  191.824  191.824   30.090   30.090
                   P12x33                 12.750   12.750    6.375    6.375    0.000    0.000
                                         12.250   12.750    0.250    0.000

 4       TABLE     M/S/HP13       3.660    1.392    1.594    0.167    1.800   22.000    1.080    7.340
                   S6x12.5                 6.000    3.330    3.000    1.665    0.000    0.000

 5       TABLE     M/S/HP13       3.660    1.392    1.594    0.167    1.800   22.000    1.080    7.340
                   S6x12.5                 6.000    3.330    3.000    1.665    0.000    0.000

 6       TABLE     M/S/HP13       3.660    1.392    1.594    0.167    1.800   22.000    1.080    7.340
                   S6x12.5                 6.000    3.330    3.000    1.665    0.000    0.000


 *****************************************
 *  END OF DATA FROM  INTERNAL STORAGE  *
 *****************************************
```

6.   The LOCATE DUPLICATE JOINTS command no longer possibly aborts when properties have not been specified for the existing members or elements. The absence of properties did not always cause an abort, but now the absence of properties is handled properly in all situations.

(GPRF 2011.10)

7.   The GTSTRUDL abort, which may occur when the external file solver is active, has been eliminated. The abort was reported one time under the very extreme conditions of a very large model having numerous and assorted stiffness and dynamic analyses and is therefore deemed very unlikely to occur. The external file solver writes and stores analysis results in files on the hard drive and is activated in conjunction with the activation of the high performance sparse equation solvers when the ACTIVE SOLVER GTSES/GT64M command is given prior to any subsequent stiffness and dynamic analysis executions. The external file solver also is activated by the DYNAMIC PARAMETERS/USE EXTERNAL FILE SOLVER command.

(GPRF 2011.14)

8.   The abort of the FORM UBC97 LOAD command caused by the presence of undefined joints has been corrected.

(GPRF 2011.15)

9.   The DEAD LOAD and SELF WEIGHT LOAD commands with the INCLUDE FINITE ELEMENTS option now state that the dead load for IPCABLE elements is not computed.

(No GPRF issued)

10.   The CALCULATE PRESSURE command no longer aborts if the GTSES/GT64M solvers are used.

(No GPRF issued)

11.   The error that limited the maximum number of CREATE PSEUDO STATIC LOAD commands to 100 when the external file solver is active has been corrected.

(GPRF 2010.06)

12.   GTSTRUDL static and dynamic analyses no longer abort when a slave joint makes reference to an inactive or deleted master joint.

(GPRF 2011.16)

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

1.   Member Elastic Connections are no longer lost if changes are made in GTMenu.

2.   In some instances, loadings which had exactly 5 characters in their name would not draw the loads. This has been corrected.

3.   Debug output will not longer be printed when performing a Redraw Solid for barbell concrete wall sections.

4.   If the first section of a variable section is prismatic, Redraw Solid will now draw it as a straight line.

5.   Split members now works for more than 20 different split operations while the Split Member dialog is active. You may have up to 1000 different split operations while the Split Member dialog is active.

6.   Generate Input File now checks member data more completely in order to eliminate an abort.

7.   An abort has been corrected when performing a Diagram Envelope.

8.   GTMenu no longer marks all joints when they are being selected to display supports.

9.   The aspect ratio calculation has been removed for 1-D elements such as the IPCABLE element.

10.   Split members with variable spacing and absolute specs now works correctly if the members are long and the length unit is mm.

11.   An abort no longer occurs if an element has a thickness but the type of element has not been specified.

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

1.   The Braced Plane Frame no longer generates incorrect models if only AB or CD knee bracing is selected for all bays.

3.8   Nonlinear Analysis

1.   Members having stiffness/flexibility matrix properties no longer have the potential to cause GTSES nonlinear analysis convergence problems when such members also are designated as having nonlinear effects of any kind. Nonlinear effects for such members are now ignored and the following message is issued when this condition is detected:

```
 **** WARNING_NLCNCK -- Nonlinear effects have been specified for
                        members having stiffness/flexibility
                        matrix properties. Nonlinear effects for
                        such members are ignored.
```

(GPRF 2011.03)

2.   Plane and space frame members that have been designated as NLS CONNECTION members by the NONLINEAR EFFECTS command may cause problems that prevent the successful completion of a nonlinear static analysis under the following conditions:

1)   There are two or more members that are designated as NLS CONNECTION.

2)   Different members specify different NLS CONNECTION degrees of freedom with different nonlinear spring curves at different member ends (start or end), as illustrated by the following command sequence:

```
NONLINEAR EFFECTS
  GEOMETRY ALL MEMBERS
  NLS CONNECTION MEMBERS 5 6 START MOMENT Y 'My' Z 'Mz'
  NLS CONNECTION MEMBERS   7 START MOMENT Y 'My' END MOMENT Y 'My'
```

In the commands above, the fact that the NLS CONNECTION specs specify different nonlinear spring curves for different degrees of freedom among members 5, 6, and 7, may precipitate this problem.

3)   During the nonlinear analysis, the displacement or rotation of a NLS CONNECTION spring exceeds the displacement/rotation bounds specified for the spring in the NONLINEAR SPRING PROPERTIES curve data.

When these conditions occur, the nonlinear analysis may abort or terminate prematurely, accompanied by an unintelligible error message.

This problem has been corrected.

(GPRF 2011.07)

3.   Nonlinear static analyses for multiple active loading conditions in which the friction bearing isolation element, i.e. element type NLSFBI, is used no longer encounter equilibrium convergence failures in the analyses for the second and subsequent loading conditions.

(GPRF 2011.11)

3.9   Offshore

1.   The random abort of the PERFORM FATIGUE ANALYSIS command has been corrected. This condition was caused by the case when the first wave of the wave data set has fewer time stepping points than a subsequent wave in the data set.

(No GPRF issued)

2.   A sequence of PERFORM FATIGUE ANALYSIS DETERMINISTIC command executions following a COMPUTE FATIGUE LJF INCLUDE command may produce slightly different fatigue damage and fatigue life results depending on the order in which multiple wave load files were previously processed by the READ WAVE LOADS NEW command. This discrepancy has been corrected.

(No GPRF issued)

3.   The DIRECTION option of the LIST FATIGUE STRESSES and LIST FATIGUE TRANSFER FUNCTIONS commands now reports the requested fatigue analysis results only for the specified wave DIRECTIONs. In previous versions, the DIRECTION option was ignored and the requested fatigue analysis results were always reported for all available wave directions.

(No GPRF issued)

3.10   Static Analysis

1.   The command STIFFNESS ANALYSIS GTSES WITHOUT REDUCE BAND will now always turn off bandwidth reduction.

(No GPRF issued)

2.   When the DUMP OFF command is given and a self-weight load is present, the GTSES and GT64M solvers now complete the solution. Previously, the solvers would exit the assembly process without performing the solution.

(No GPRF issued)

3.   Static analysis using the GTSES/GT64M sparse equation solvers could prematurely complete, with no results computed, when one of the active independent loads was computed by the SELF-WEIGHT or DEAD LOAD command for at least one active member/finite element having a DENSITY value equal to exactly 0.0. This condition has been corrected.

(GPRF 2011.02)

3.11 Steel Design

1.   The uninitialized variable in the AISC13 code which resulted into an incorrect lateral-torsional buckling (LTB) computation has been fixed. Also the user specified values for the parameters UNLCFTF and UNLCFBF are correctly considered in the lateral-torsional buckling equations.

(GPRF 2012.01)

This page intentionally left blank.

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 32. 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   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.   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)

3.   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)

4.   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)

4.2   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 7 INC 1 TO 8 INC 1

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)

4.3   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)

4.4   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)

4.5   Offshore Analysis and Design – GTSELOS

1. The GTSELOS linear wave implementation produces incorrect wave kinematics and wave forces at water depths greater than approximately 525 feet (160 meters).

    (No GPRF issued)

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

5.2   Design Prerelease Features

5.2.1   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.

5.2.2   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.

5.2.3   Rectangular and Circular Concrete Cross Section Tables

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

5.2.5   ISO 19902:2007(E) code for fixed steel offshore structure.

5.2.6   ASCE4805 Steel Design Code. This code is for the ultimate strength design of steel transmission pole structures.

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 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.

5.2.2 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.

$$\underline{\text{METHOD}} \; (\left\{ \begin{array}{l} \rightarrow \; \underline{\text{ULTIMATE}} \; (\underline{\text{STRENGTH}}) \\ \underline{\text{WORKING}} \; (\underline{\text{STRESS}}) \end{array} \right\}) \left\{ \begin{array}{l} \underline{\text{ACI318-99}} \\ \underline{\text{ACI318-89}} \\ \underline{\text{ACI318-83}} \\ \underline{\text{ACI318-77}} \\ \underline{\text{ACI318-63}} \\ (\underline{\text{BSI}}) \; \underline{\text{CP110-72}} \\ (\underline{\text{BSI}}) \; \underline{\text{BS8110}} \end{array} \right\} -$$

$$(\; \underline{\text{BARS}} \left\{ \begin{array}{l} \rightarrow \; \underline{\text{ASTM}} \\ \underline{\text{CANADIAN}} \; (\underline{\text{STANDARD}}) \\ \underline{\text{UNESCO}} \\ \underline{\text{KOREAN}} \; (\underline{\text{STANDARD}}) \end{array} \right\}) \; (\left\{ \begin{array}{l} \rightarrow \; \underline{\text{NONSEISMIC}} \\ \underline{\text{SEISMIC}} \\ \underline{\text{MODERATE}} \; \underline{\text{SEISMIC}} \end{array} \right\})$$
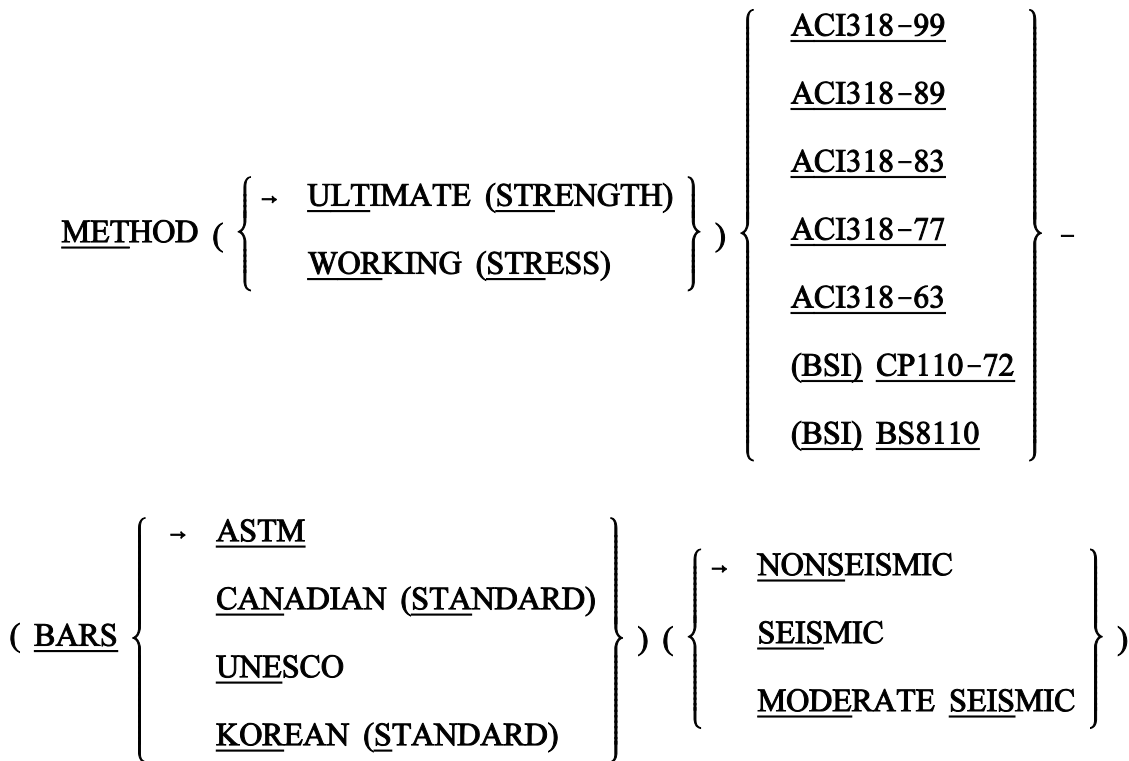
Example:

```
        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/6.70) $\sqrt{f_c'}$ 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.

5.2.3   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

5.2.4 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 Reference Manuals). 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:

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

$$\left\{ \begin{array}{l} \underline{WOOD} \ (\underline{AND}) \ (\underline{ARMER}) \left\{ \begin{array}{l} \to \underline{AVER}AGE \\ \underline{MAXI}MUM \end{array} \right\} \\ \underline{CALC}ULATE \ (\underline{RES}ULTANT) \ (\underline{ELEMENT}) \ (\underline{FOR}CES) \end{array} \right\} \ (\underline{AL}ONG) -$$

$$(\underline{CUT} \left\{ \begin{array}{l} 'a' \\ i_1 \end{array} \right\}) \ \left\{ \begin{array}{l} \underline{JOINTS} \\ \underline{NODES} \end{array} \right\} \ list_1 \ \underline{ELEMENT} \ list_2 \ (\underline{TABLE} \left\{ \begin{array}{l} \to \underline{ASTM} \\ \underline{UNESCO} \end{array} \right\}) -$$

$$ ^{*} \left\{ \begin{array}{l} \underline{TOP} \ (\underline{FACE}) \ (\underline{BARS} \ i_2) \ (\underline{SP}ACING \ v_1) \\ \underline{BOTTOM} \ (\underline{FACE}) \ (\underline{BARS} \ i_3) \ (\underline{SP}ACING \ v_2) \\ \underline{BOTH} \ (\underline{FACES}) \ (\underline{BARS} \ i_4) \ (\underline{SP}ACING \ v_3) \end{array} \right\} -$$

$$\left\{ \begin{array}{l} \to \underline{INNER} \ (\underline{LAYER}) \\ \underline{OUTER} \ (\underline{LAYER}) \end{array} \right\} \ (\underline{COVER} \ v_4) \ (\underline{LINEAR} \ (\underline{TOL}ERANCE) \ v_5) -$$

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

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 5.2.4-1. 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.
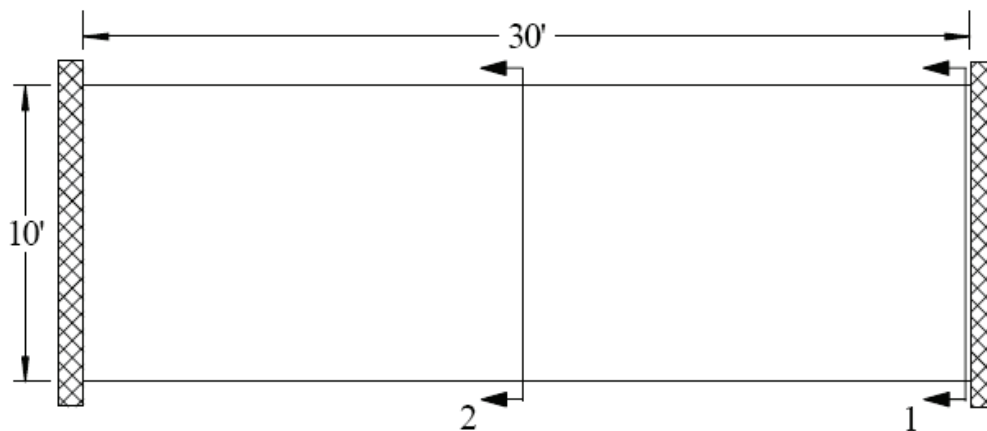
Figure 5.2.4-1   Example Flat Plate Structure (PLAN)

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.
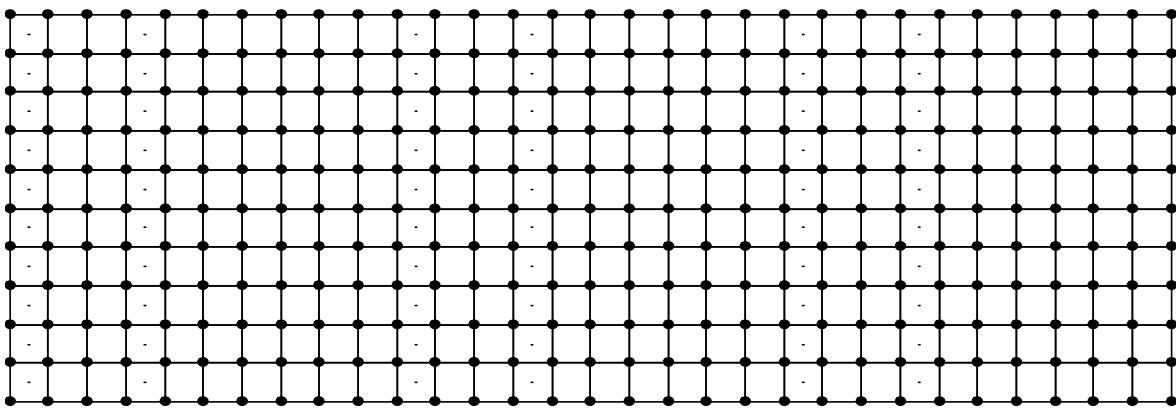
Figure 5.2.4-2   Example Finite Element Model

Definition of Cut Cross-Sections

Two "cuts" are considered for the verification example, as shown in Figure 5.2.4-1.

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
```

5.2.5   ISO 19902:2007 Code for the Design of Fixed Steel Offshore Structures

A new *International Standard ISO 19902:2007(E), First Edition, 2007-12-01, Petroleum and natural gas industries – Fixed steel offshore structures* has been implemented as a prerelease feature. The GTSTRUDL code name for this new offshore code is "19902-07". The 19902-07 code performs *Basic, Hydrostatic Pressure, and Punching Shear stresses* check according to *International Standard ISO 19902:2007(E)*. This new code, 19902-07, may be used to select or check Circular Hollow Sections (Pipes).   The prerelease documentation for the 19902-07 code may be found by selecting the help menu and then Reference Documentation, Reference Manuals, Offshore Loading, Analysis and Design, and 19902-07: ISO 19902:2007(E), First Edition in the GTSTRUDL Output Window.

5.2.6   ASCE4805 Code for the Design of Steel Transmission Pole Structures

The steel design code, ASCE4805, which is based on the 2005 edition of the ASCE/SEI, *Design of Steel Transmission Pole Structures* Specification has been implemented as a pre-release feature. The ASCE/SEI 48-05 Specification is based on ultimate strength methods using factored loads.

The ASCE4805 Code may be used to select or check any of the following shapes:

Design for axial force, bi-axial bending, and torsion:

Pipes

Regular Polygonal Tubes

Structural Tubing

The documentation for the ASCE4805 code may be found by selecting the Help menu and then Reference Documentation, Reference Manuals, Steel Design, and "ASCE4805" in the GTSTRUDL Output Window.

5.3   Analysis Prerelease Features

5.3.1   The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

CALCULATE ERROR (ESTIMATE) (BASED ON)   -

$$
* \left\{ \begin{array}{l} \underline{\text{ENERGY}} \ (\underline{\text{NORM}}) \\ \underline{\text{MAX}} \ \underline{\text{DIFFERENCE}} \\ \underline{\text{DIFFERENCE}} \ \underline{\text{FROM}} \ \underline{\text{AVERAGE}} \\ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFFERENCE}} \\ \underline{\text{PERCENT}} \ \underline{\text{DIFFERENCE}} \ \underline{\text{FROM}} \ \underline{\text{AVERAGE}} \\ \underline{\text{NORMALIZED}} \ \underline{\text{PERCENT}} \ \underline{\text{MAX}} \ \underline{\text{DIFFERENCE}} \\ \underline{\text{NORMALIZED}} \ \underline{\text{PERCENT}} \ \underline{\text{DIFFERENCE}} \ \underline{\text{FROM}} \ \underline{\text{AVERAGE}} \end{array} \right\} \quad -
$$

$$
(\underline{\text{AT}}) \ {}^{*} \left\{ \begin{array}{l} \underline{\text{TOP}} \\ \underline{\text{MIDDLE}} \\ \underline{\text{BOTTOM}} \end{array} \right\} (\underline{\text{SURFACES}}) \ (\underline{\text{FOR}}) \left\{ \begin{array}{l} \to \underline{\text{ALL}} \\ \underline{\text{ELEMENT}} \ \text{list} \end{array} \right\}
$$

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:

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

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,

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

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:
The nodal error estimates estimate the accuracy of the data in a selected nodal output vector.

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

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

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

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
Normalized Percent Maximum Difference

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

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

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:

$$\underline{\text{ROTA}}\text{TE}\ \ \underline{\text{LOA}}\text{DING}\ \left\{ \begin{matrix} i_R \\ \\ 'a_R\ ' \end{matrix} \right\}\ (\ \underline{\text{ANGLES}}\ )\left[\underline{\text{T1}}\right]r_1\left[\ \underline{\text{T2}}\ \right]r_2\left[\ \underline{\text{T3}}\ \right]r_3$$

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:

$$\underline{\text{REFERENCE}} \ (\underline{\text{COORD}}\text{INATE}) \ (\underline{\text{SYS}}\text{TEM}) \ \begin{Bmatrix} i_1 \\ 'a_1' \end{Bmatrix} \ -$$

$$\begin{Bmatrix} (\underline{\text{ORIGIN}} \ [\ \underline{X}\ ] \ v_x \ [\ \underline{Y}\ ] \ v_y \ [\ \underline{Z}\ ] \ v_z) \ (\underline{\text{ROT}}\text{ATION} \ [\ \underline{R1}\ ] \ v_1 \ [\ \underline{R2}\ ] \ v_2 \ [\ \underline{R3}\ ] \ v_3) \\[2em] \begin{Bmatrix} \begin{bmatrix} \underline{\text{JOINT}} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X} \ v_4 \ \underline{Y} \ v_5 \ \underline{Z} \ v_6 \end{bmatrix} \end{Bmatrix} \begin{Bmatrix} \begin{bmatrix} \underline{\text{JOINT}} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X} \ v_4 \ \underline{Y} \ v_5 \ \underline{Z} \ v_6 \end{bmatrix} \end{Bmatrix} \begin{Bmatrix} \begin{bmatrix} \underline{\text{JOINT}} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \\ \underline{X} \ v_4 \ \underline{Y} \ v_5 \ \underline{Z} \ v_6 \end{bmatrix} \end{Bmatrix} \end{Bmatrix}$$

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 Manuals 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:

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

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

$$
\begin{matrix}
\cdot \\
\cdot \\
\cdot
\end{matrix}
\quad
\begin{Bmatrix}
i_1 \\
'a_1'
\end{Bmatrix}
\quad \text{coordinate-specs}_1
$$

$$
\begin{Bmatrix}
i_n \\
'a_n'
\end{Bmatrix}
\quad \text{coordinate-specs}_n
$$

Elements:

coordinate-specs   =   [$\underline{X}$] v_1 [$\underline{Y}$] v_2 [$\underline{Z}$] 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:

GTMenu LINE INCIDENCES

$$
\begin{Bmatrix} i_1 \\ 'a_1' \end{Bmatrix} \quad type_1 \quad incidence\text{-}specs_1
$$

.
.
.

$$
\begin{Bmatrix} i_n \\ 'a_n' \end{Bmatrix} \quad type_n \quad incidence\text{-}specs_n
$$

Elements:

$$
type \quad = \quad \begin{Bmatrix} \rightarrow \underline{LINE} \\ \underline{POLY}NOMINAL\ (\underline{CUR}VE) \\ \underline{ARC}\ (\underline{TEM}PLATE) \\ \underline{CEN}TERED\ (\underline{ARC})\ \underline{PER}CENT\ v_1 \\ \underline{BEZ}IER\ (\underline{CUR}VE) \\ \underline{SPL}INE\ (\underline{CUR}VE)\ (\underline{OR}DER\ k_2) \end{Bmatrix}
$$

$$
incidence\text{-}specs \quad = \quad \begin{Bmatrix} i_1 \\ 'a_1' \end{Bmatrix} \begin{Bmatrix} i_2 \\ 'a_2' \end{Bmatrix} \ldots \begin{Bmatrix} i_p \\ 'a_p' \end{Bmatrix}
$$

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