GT STRUDL® Version 2015 Release Guide

Release Guide

Release Date: February 2015

PART OF HEXAGON

INTERGRAPH®

Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2015 and later versions for use on PCs under the Microsoft Windows 8, Windows 7 and Windows Vista operating systems.

Copyright

Copyright © 2015 Intergraph® Corporation. All Rights Reserved. Intergraph is part of **Hexagon**.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
300 Intergraph Way
Huntsville, AL 35813

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Manuals, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on eCustomer, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.

b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, http://www.bis.doc.gov/complianceandenforcement/liststocheck.htm, the U.S. Department of Treasury Specially Designated Nationals List, http://www.treas.gov/offices/enforcement/ofac/, and the U.S. Department of State Debarred List, http://www.pmddtc.state.gov/compliance/debar.html.

c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.

d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, and GT STRUDL are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Microsoft, Windows, Windows 8, Windows 7 and Windows Vista are registered trademarks of Microsoft Corporation. AutoCAD is a registered trademark of Autodesk Corporation. Other brands and product names are trademarks of their respective owners.

Table of Contents

Chapter **Page**

NOTICES iii

Table of Contents v

CHAPTER 1

Introduction 1-1

CHAPTER 2 NEW FEATURES IN VERSION 2015

2.1 CAD Modeler 2-1
2.2 Dynamics 2-6
2.3 General 2-6
2.4 GTMenu 2-7
2.5 Offshore 2-13
2.6 SPLM Licensing 2-14
2.7 Tekla Structures Interface 2-14
2.8 Base Plate Wizard 2-15

CHAPTER 3 ERROR CORRECTIONS

3.1 Dynamic Analysis 3-1
3.2 General 3-2
3.3 GTMenu 3-2
3.4 GTSTRUDL Output Window 3-4
3.5 Nonlinear Analysis 3-4
3.6 Offshore 3-5
3.7 Steel Design 3-5

CHAPTER 4 KNOWN DEFICIENCIES

4.1 Finite Elements 4-1

4.2 General Input/Output 4-1

4.3 GTMenu 4-2

CHAPTER 5 PRERELEASE FEATURES

5.1 Introduction 5.1-1

5.2 Design Prerelease Features 5.2-1

5.2.1 LRFD3 Steel Design Code 5.2-1

5.2.2 ACI Code 318-99 . . . . . 5.2-2

5.2.3 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command) 5.2-5

5.2.4 ASCE4805 Code for the Design of Steel Transmission Pole Structures 5.2-15

5.3 Analysis Prerelease Features 5.3-1

5.3.1 Calculate Error Estimate Command 5.3-1

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command 5.3-5

5.4 General Prerelease Features 5.4-1

5.4.1 Rotate Load Command 5.4-1

5.4.2 Reference Coordinate System Command 5.4-5

5.4.2-1 Printing Reference Coordinate System Command . . 5.4-8

5.4.3 GTMenu Point Coordinates and Line Incidences Commands . . 5.4-9

This page intentionally left blank.

Chapter 1

Introduction

Version 2015 covers GT STRUDL operating on PC's under the Windows 8, Windows 7 and Windows Vista operating systems. Version 2015 changes the version naming convention to reflect the year in which the version was available to users. Internally, this version is known as Version 34.0.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 33. In particular, Chapter 2 briefly describes the new GT STRUDL CAD Modeler which is an AutoCAD based graphical modeler. Chapter 3 provides you with details regarding error corrections that have been made since the Version 33 release. Chapter 4 describes known problems with Version 2015. 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 GT STRUDL 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 section 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 or AISC14 codes which are released features.
- 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 a released feature.
- 5.2.3 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)
- 5.2.4 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
5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES 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 GT STRUDL capabilities.

Chapter 2

New Features in Version 2015

This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GT STRUDL in Version 2015. This release guide is also available online upon execution of GT STRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1 CAD Modeler

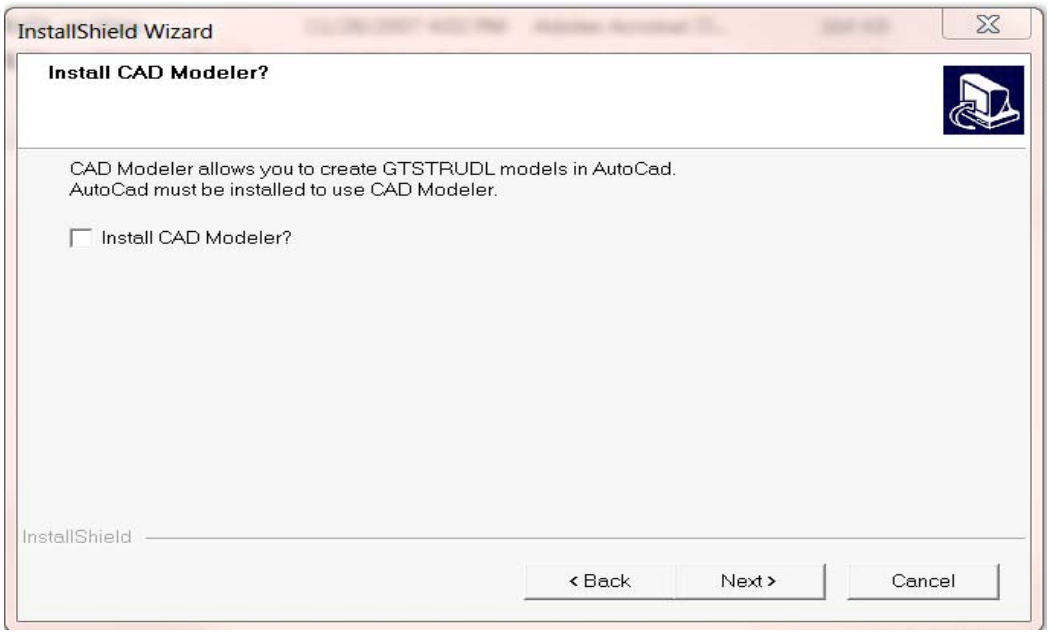
A new modeling feature has been added to GT STRUDL 2015 and is available to all supported customers. This new modeling feature is called GT STRUDL CAD Modeler and uses the well known AutoCAD® program. CAD Modeler is an add-on to AutoCAD which allows you to create GT STRUDL Input Files (.gti files) graphically using AutoCAD's powerful CAD tools and graphical display capabilities. AutoCAD must be installed in your computer before installing and running CAD Modeler. The CAD Modeler extensions to AutoCAD will be added to AutoCAD during installation of GT STRUDL. To use CAD Modeler, you must have one AutoCAD version 2012 - 2015 installed on your computer and be proficient in the use of AutoCAD.

A summary of some of the important features of CAD Modeler are shown below:

- Create GT STRUDL models using AutoCAD functionality
- View and modify existing GT STRUDL models in an AutoCAD environment
- Switch between different floors or levels and also use a grid to facilitate model creation and modification
- Generate GT STRUDL joints and control their properties such as constraints and spring values
- Choose any of the steel shapes available in GT STRUDL standard tables
- Generate GT STRUDL members and be able to define their cross-section, material, releases, elastic connections and member eccentricities
- Generate GT STRUDL finite elements
- Load your model using self-weight loadings, joints loads, members loads, finite element loads and area loads
- Create Load Combinations
- Define Groups and control the display of members and finite elements with Groups and include the Group definitions in the GT STRUDL input file created by CAD Modeler
- Generate member and finite element meshes along a curve, between lines and extruded polyline surfaces.

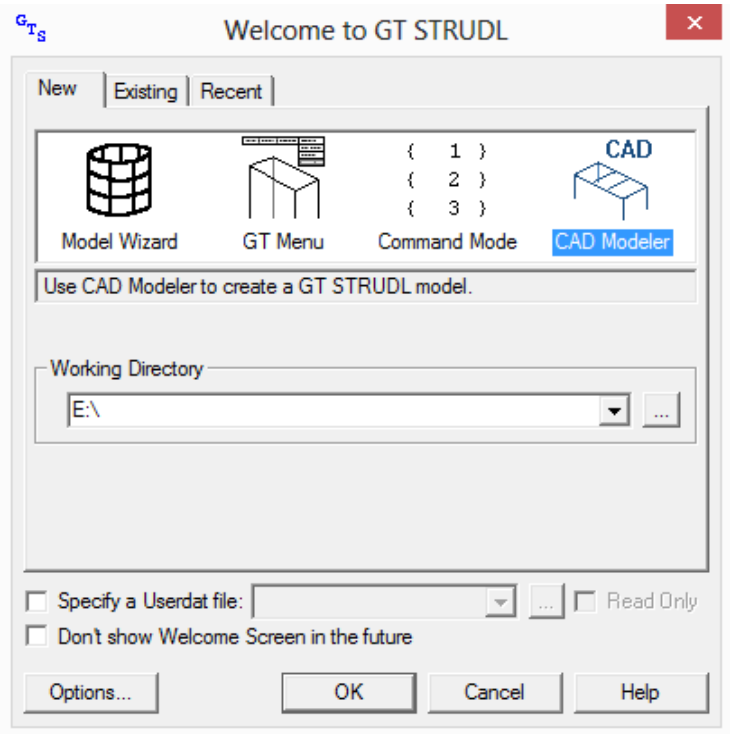
- Create 2D area meshes of triangular finite element meshes for regions defined using closed polylines with internal boundary and point constraints
- Use AutoCAD functions such as move, copy, delete, rotate and mirror in the creation and modification of your models
- Switch between 3D Solid and wireframe views of your model
- Locate potential errors in your model by checking and eliminating duplicate and floating joints
- Create GT STRUDL input files (.gti files) identical to the ones created by GTMenu
- View results such as the deformed shape, member force/moment diagrams, finite element contours and code check pass-fail results
- Use any of the available GT STRUDL units at any time

In order to install CAD Modeler, check the box "Install CAD Modeler?" in the dialog shown below during the GT STRUDL installation procedure.

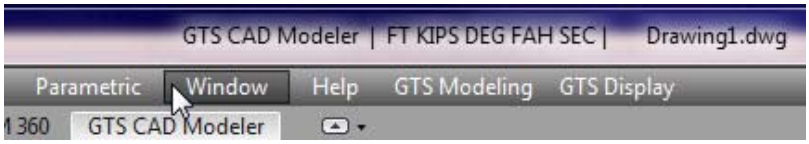


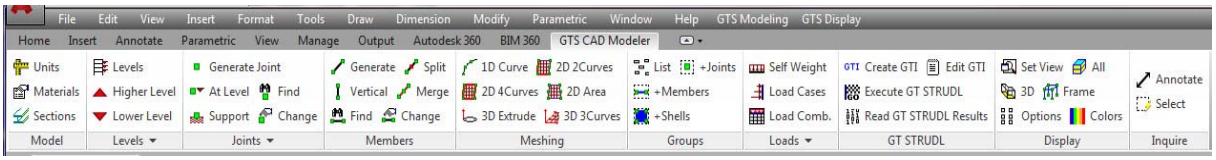
CAD Modeler Installation can also be launched independently, after GTSTRUDL installation, by executing the file "CADM_setup.exe" which is located in the CADModeler folder in the GTStrudl 2015 installation directory.

CAD Modeler is launched from the Welcome to GT STRUDL dialog (previously called the Startup Wizard) by selecting the “CAD Modeler” icon as shown below:



A new instance of AutoCAD, having CAD Modeler automatically loaded, is then created. AutoCAD will have two additional menus called (“GTS Modeling” and “GTS Display” next to the AutoCAD main menus and the “GTS CAD Modeler” tab in the ribbon area. as shown in the figures below:

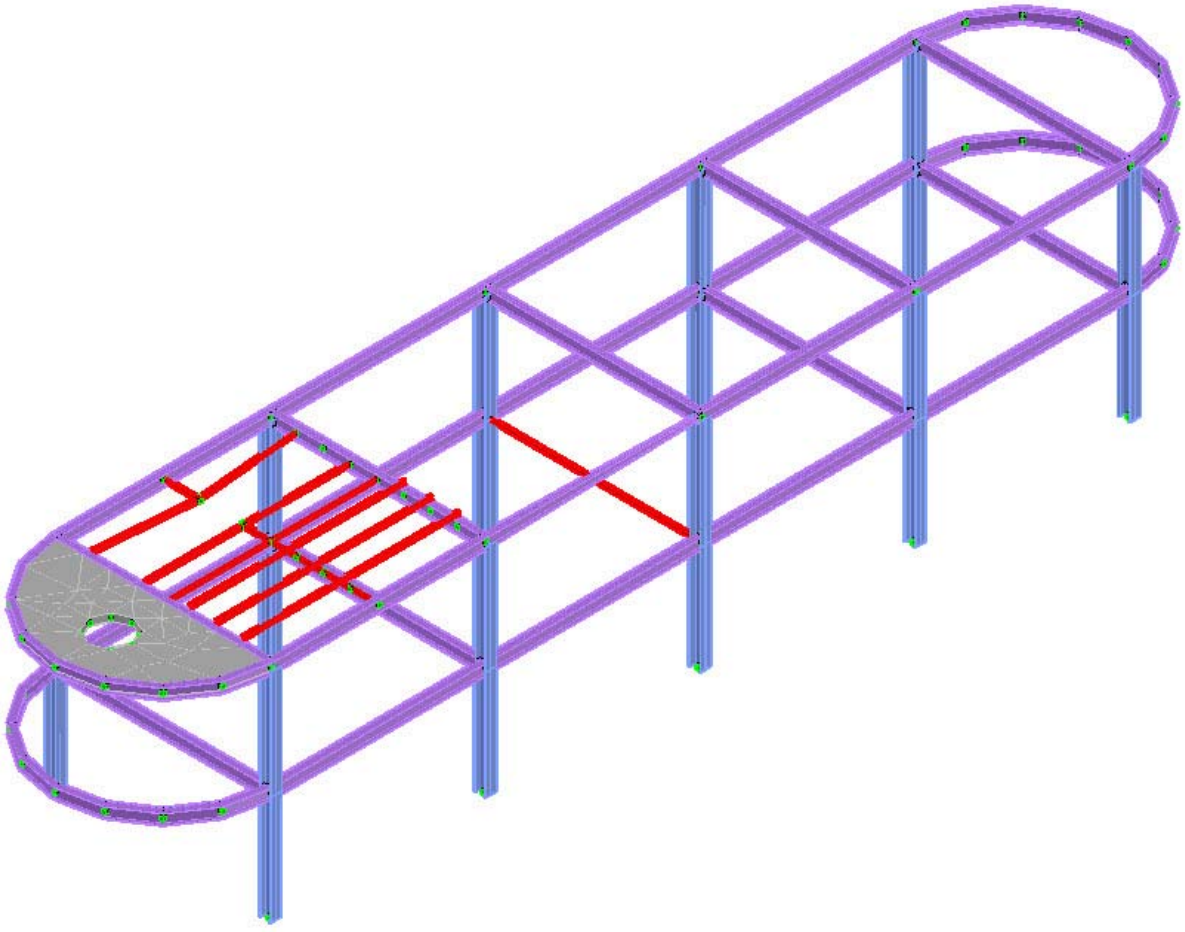




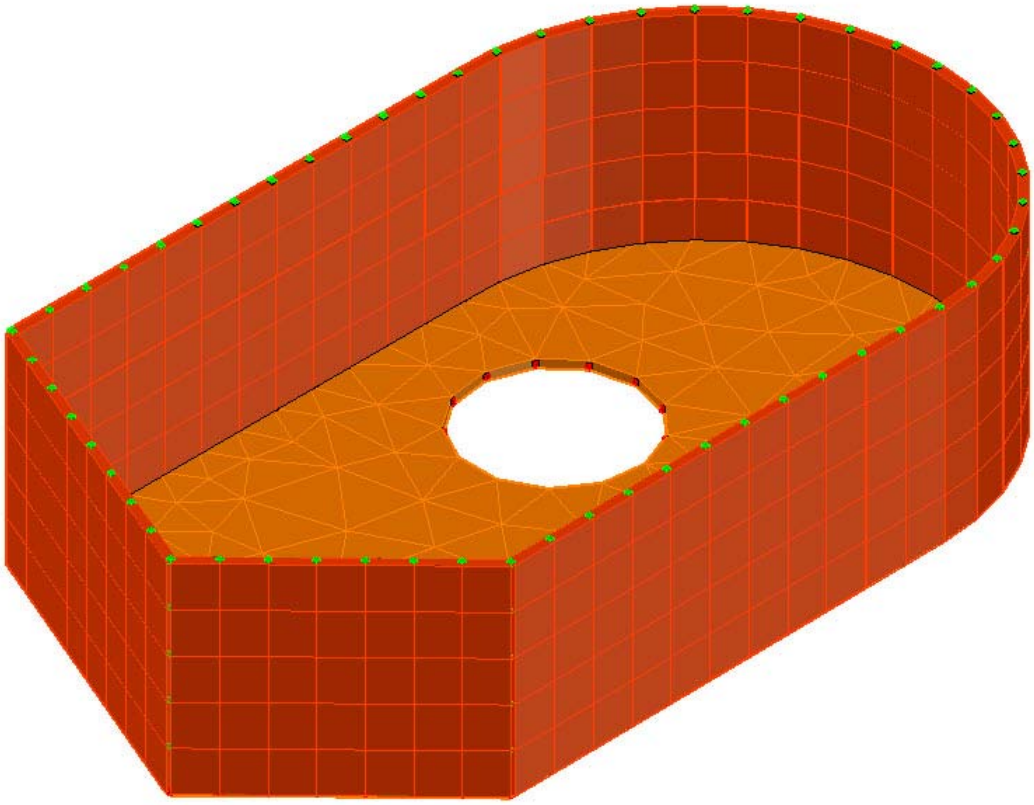
A new CAD Modeler Getting Started manual is available under Help - Reference Documentation.

Two examples of models created in CAD Modeler are shown in the following figures. The first model is a building and the second model is a tank modeled with finite elements:

Building with Frame Members and Finite Elements



Finite Element Model of Tank



2.2 Dynamics

1. The dynamic analysis LIST command (LIST RESPONSE SPECTRUM, LIST TRANSIENT, etc.) no longer leaves holes in the resulting report for inactive/deleted joints when the JOINTS ALL list option is used.

   Documentation

   Section 2.4.6, Volume 3

2.3 General

1. The number of compression-only nonlinear spring elements that can be created by the CALCULATE SOIL SPRING VALUES command has been increased from 9,999 to 999,999.

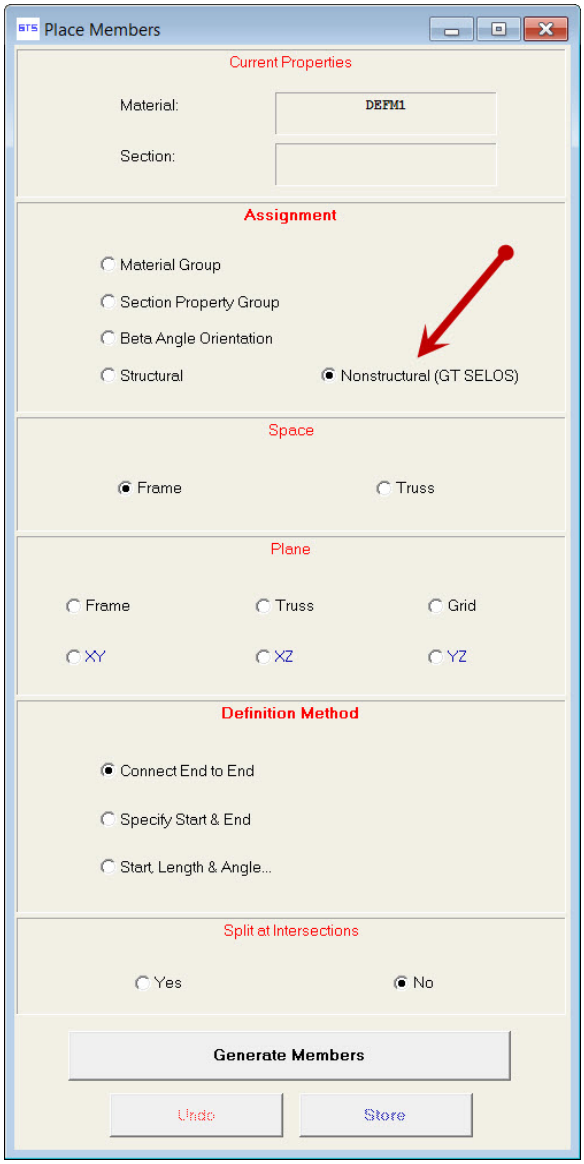
   Documentation

   Section 2.1.12.13, Volume 1.

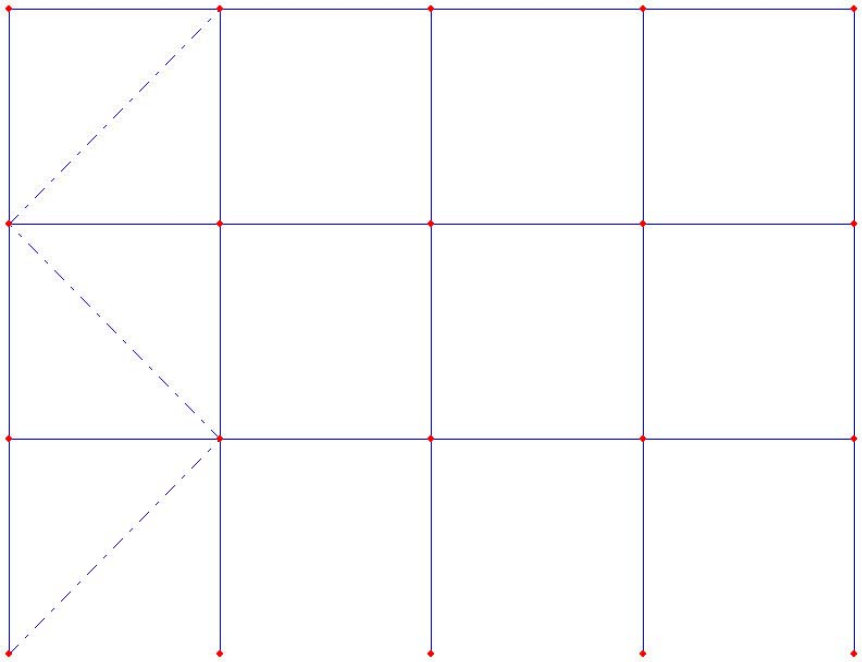
2. The allowable number of characters for an individual input line has been increased to 1000 from the previous limit of 278. If a single input line exceeds 1000 characters, an error message is generated and the SCAN flag is turned on. The total number of characters allowed for a single command, which is the summation of all input lines with continuations plus the final input line for the command, is now 524,288 characters. Previously the limit was 50,000 characters for a single command.

2.4 GTMenu

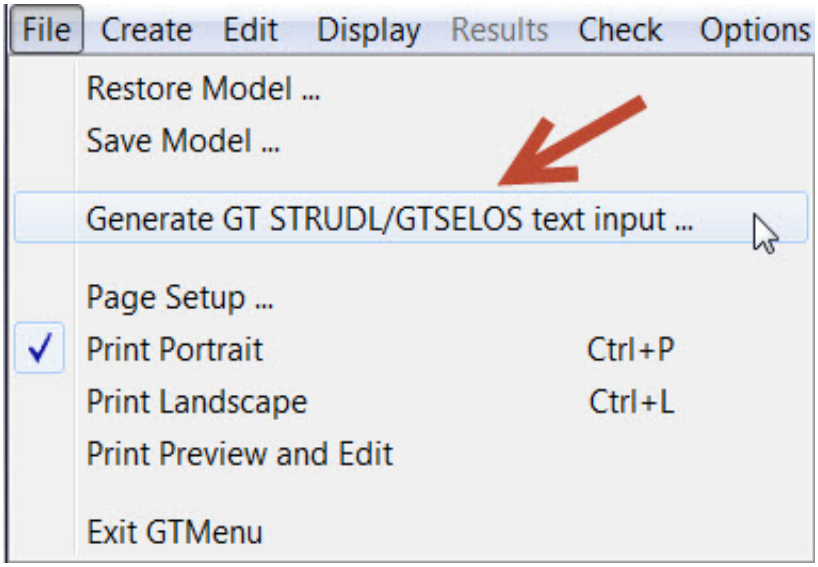
1. You may now create nonstructural members in GTMenu. Non-structural members are members which can be loaded for wave and wind load application in GT SELOS but their stiffness is neglected. The Place Members, Generate Members Using Construction Points and Lines and Edit Member Data dialogs now have an option which allows you to specify nonstructural members for GTSELOS. The modified Place Members dialog is shown in the following figure:



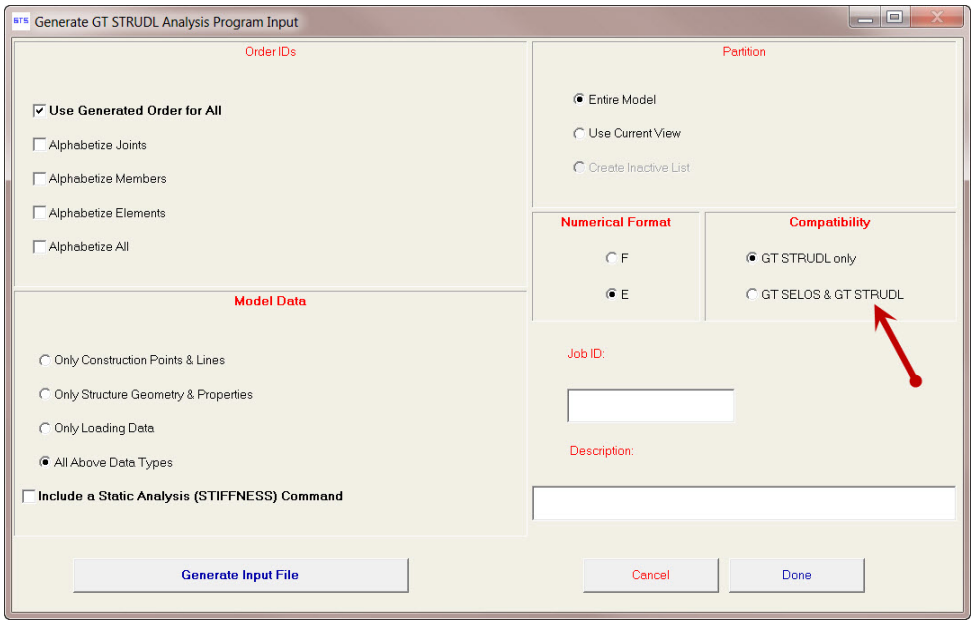
When nonstructural members are created in GTMenu, they will be displayed with a dashed line in order to be able to easily distinguish them from structural members. An example of a structure with nonstructural members used for the diagonals is shown below:



2. GTMenu will now also create GTSELOS(offshore) text input files from the File pulldown as shown in the following figure:



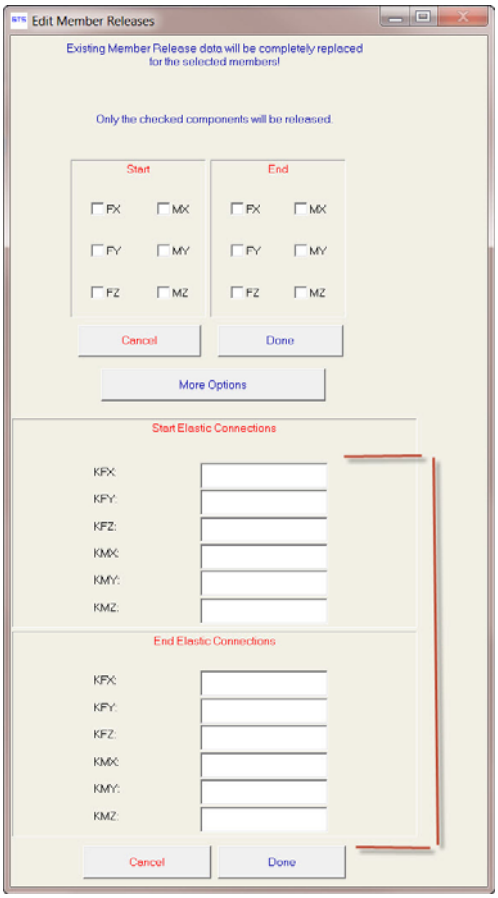
The Generate GT STRUDL Analysis Program Input dialog has been modified to allow the user to create an input file which contains both GT STRUDL and GTSELOS commands as shown below:



An example of an input file created using the GTSELOS & GT STRUDL option in the above dialog is shown below with the commands that will be recognized only by GTSELOS having a single dollar sign followed by a blank at the start of a line. A portion of an input file which contains GT STRUDL and GTSELOS commands is shown below with the lines that will be recognized only by GTSELOS with a single $ sign at the start of the line and which are also in red:

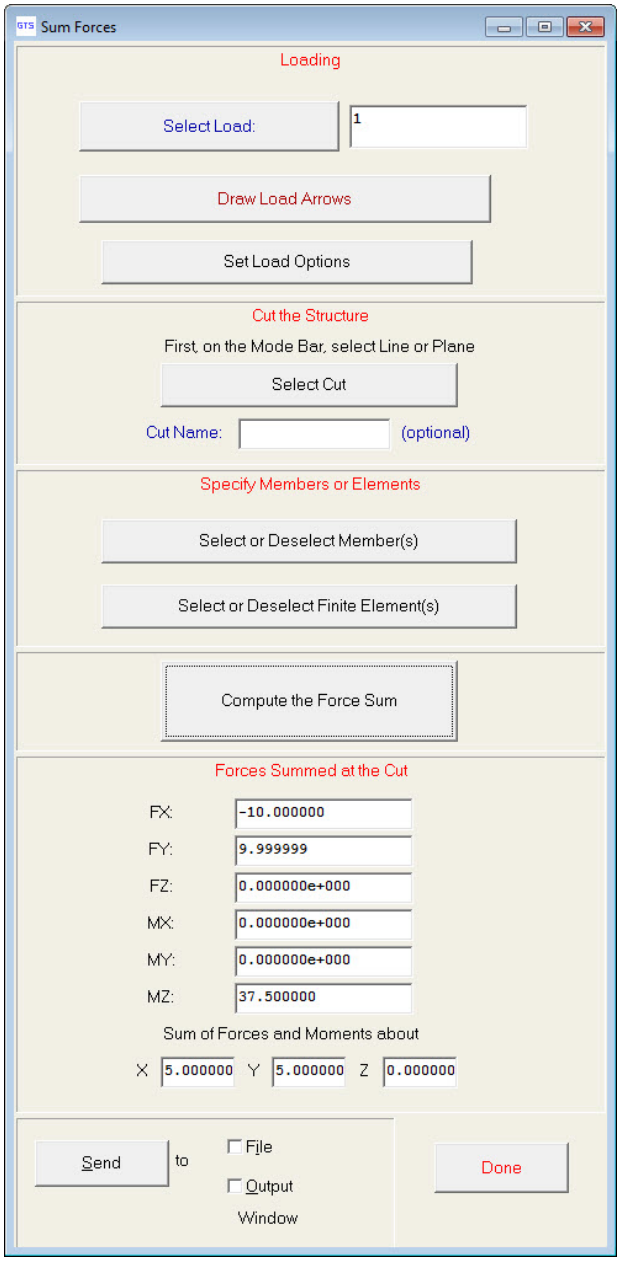
```
$ SELOS     '         '
STRUDL      '         '
$$
$$      This    GTSTRUDL file created from GTMenu on 10/22/2014
$$

UNITS   FEET   KIPS   DEG    FAH    $   LBM
JOINT COORDINATES GLOBAL
     1               0.0000000E+00   0.0000000E+00   0.0000000E+00
     2               1.0000000E+01   0.0000000E+00   0.0000000E+00
     3               2.0000000E+01   0.0000000E+00   0.0000000E+00
     4               3.0000000E+01   0.0000000E+00   0.0000000E+00
     5               4.0000000E+01   0.0000000E+00   0.0000000E+00
     6               0.0000000E+00   1.0000000E+01   0.0000000E+00
     7               1.0000000E+01   1.0000000E+01   0.0000000E+00
     8               2.0000000E+01   1.0000000E+01   0.0000000E+00
     9               3.0000000E+01   1.0000000E+01   0.0000000E+00
     10              4.0000000E+01   1.0000000E+01   0.0000000E+00
     11              0.0000000E+00   2.0000000E+01   0.0000000E+00
     12              1.0000000E+01   2.0000000E+01   0.0000000E+00
     13              2.0000000E+01   2.0000000E+01   0.0000000E+00
     14              3.0000000E+01   2.0000000E+01   0.0000000E+00
     15              4.0000000E+01   2.0000000E+01   0.0000000E+00
     16              0.0000000E+00   3.0000000E+01   0.0000000E+00
     17              1.0000000E+01   3.0000000E+01   0.0000000E+00
     18              2.0000000E+01   3.0000000E+01   0.0000000E+00
     19              3.0000000E+01   3.0000000E+01   0.0000000E+00
     20              4.0000000E+01   3.0000000E+01   0.0000000E+00
$ END JOINT COORDINATES
$$
UNITS   FEET   KIPS   DEG    FAH    $   LBM
$$
$$
TYPE   SPACE FRAME
$$
MEMBER INCIDENCES
$   '28      '      '1      '      '7      '     NONSTRUCTURAL
$   '29      '      '7      '      '11     '     NONSTRUCTURAL
$   '30      '      '11     '      '17     '     NONSTRUCTURAL
```

3. GTMenu now has the ability to create and edit Member Elastic Connections. The Edit Member Data dialog has been modified where you may now specify elastic spring constants (KFX,KFY,KFZ,KMX,KMY,KMZ) for the connections at the start and end of members. The revised Edit Member Releases dialog is shown on the following page:



4. GTMenu now allows the use of multiple monitors for graphical mouse inputs, and it will also restrict the input area to the graphics window so that it is clearer that input is required. Mouse input was previously restricted to the first monitor rendering GTMenu useless for graphical mouse input if moved to a second monitor.

5. The dialog for Sum Forces at a Cut has been modified as shown on the next page. In the Sum Forces dialog, you select a cut using a Line or Plane and now you select the members and elements that you want considered by clicking on them with the mouse in Hit mode or listing them if you are in List Mode. After selecting the members and elements, you then press the Compute the Force Sum button and the results will be shown at the bottom of the dialog.
   A new feature has been added when you send the output to the output window or a file. Now, not only the results will be sent but also the LIST SUM FORCES command that will produce the same results as the dialog. This allows you to incorporate the same LIST SUM FORCES command into subsequent revisions of your model.



Revised Sum Forces dialog

```
------------------------------------------------------------------------
   Output from GTMenu follows:


Units:  FEET  KIPS  LBM   DEG   FAH
Loading: 1

Forces Summed at the Cut
FX:     -10.0000000
FY:       9.9999990
FZ:       0.0000000
MX:       0.0000000
MY:       0.0000000
MZ:      37.5000000
Sum of Forces and Moments about
X:   5.0000  Y:   5.0000  Z:   0.0000

LIST SUM FORCES -
JOINTS -
    11          12          13          14          15        -
MEMBERS -
    9           10          11          12
```

Sum Forces Output sent to the GT STRUDL Output Window

2.5 Offshore

1. The *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 release feature. The GT STRUDL 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 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 GT STRUDL Output Window.

2. Numerical precision has been increased from 32 bit to 64 bit for the computation of GTSELOS linear (Airy) wave kinematics. This improvement also affects the random linear wave model and the multi-generation wave model as both are based on the linear wave. The impact of this improvement is most noticeable for shallow waves in deep water. For cases of wave heights less than six meters in still water depths greater than 300 meters, the 32-bit precision model was unable to compute the expected 0.0 linear

wave kinematics as depth elevations approached the sea bed. On the other hand, the 64-bit precision improvements allow for the correct and consistent computation of linear wave kinematics under such conditions.

2.6 SPLM Licensing

1. SPLM (Smart Plant Licensing Manager) licensing has been added to GT STRUDL. SPLM licensing requires that the Smart Plant Licensing Manager service is running on your computer and that it is configured to contact a server with the appropriate GT STRUDL license tags. Contact your Intergraph software supplier for more information about installing SPLM on your computer. GT STRUDL 2015 does not support the SPLM License Checkout feature allowing off-network licensing. SPLM licensing must be selected during installation to be used. To change from a Sentinel hardware lock to SPLM, you must uninstall GT STRUDL 2015 and then re-install and select SPLM when choosing the license type.

2.7 Tekla Structures Interface

1. A new interface GTSTRUDL_TS has been added to facilitate translations from Tekla Structures models into GT STRUDL input files. It is typically included in the folder \Utilities\Tekla folder in the GTStrudl installation directory. See the file ReadMe_GT_TS.pdf for how to install GTSTRUDL_TS. Since this interface is called from within Tekla Structures, Tekla Structures must be installed before GTSTRUDL_TS is installed.

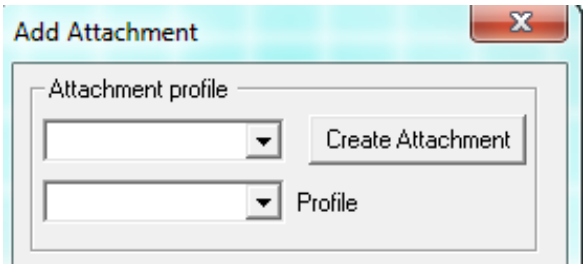
2.8 Base Plate Wizard

1. Attachments may now be rotated about their X and Y axes to easily model attachments that are at an angle to the base plate, such as the lower knee brace of a pipe rack. X and Y rotations are treated as Euler angles to create a projected footprint in the base plate. Z rotations may also be specified and if so are applied after any X or Y rotations.

   X and Y rotations may only be applied to attachments modeled as RIGID SOLID "footprints". Attachments modeled as finite element extensions do not support X and Y rotations.

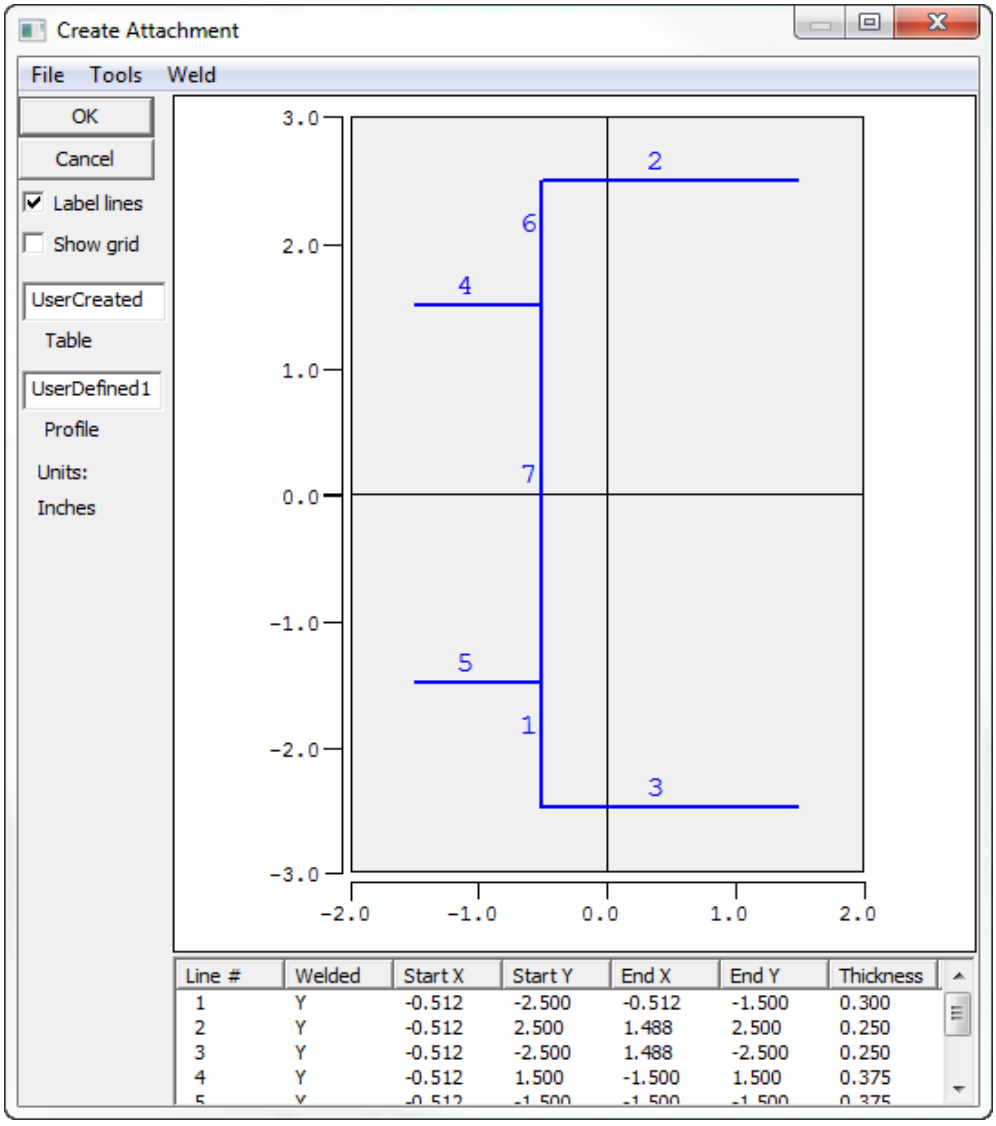
   See Section 3 of the Base Plate Wizard Users Guide for more details.

2. A new Create Attachment feature has been added to the Add Attachment dialog.



   This new feature helps you create a custom attachment to model beam seats, built-up shapes and other non-standard profile attachments. Capabilities are supplied to create attachments based on standard shapes (W, Channel, etc.) with your dimensions, extract from an existing .gtbp file or by entering data (X, Y and thickness) for attachment lines. In addition, automatic splitting of intersecting lines is available so you don't have to calculate the intersection point. Below is a sample of the Create Attachment dialog with a custom channel with two stiffener plates added.

   See Section "3.2 The Create Attachment Dialog" of the Base Plate Wizard Users Guide for more details.


Create Attachment
File Tools Weld
OK
Cancel
Label lines
Show grid
UserCreated
Table
UserDefined1
Profile
Units:
Inches
3.0
2.0
1.0
0.0
-1.0
-2.0
-3.0
-2.0
-1.0
0.0
1.0
2.0
1
2
3
4
5
6
7
Line # Welded Start X Start Y End X End Y Thickness
1 Y -0.512 -2.500 -0.512 -1.500 0.300
2 Y -0.512 2.500 1.488 2.500 0.250
3 Y -0.512 -2.500 1.488 -2.500 0.250
4 Y -0.512 1.500 -1.500 1.500 0.375

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 Dynamic Analysis

1. Nonlinear dynamic analysis of models containing tension/compression-only members now converges to the correct solution when the tension/compression-only members cycle through changes in axial force between tension and compression.

   (GPRF 2014.05)

2. The CREATE PSEUDO STATIC LOAD command no longer produces zero stress and strain results under the following condiitons:

   a. The active solver has been specified as either GTSES or GT64M.

   b. The command specifies a response spectrum mode combination from two or more source loads.

   c. The command is not preceded by another CREATE PSEUDO STATIC command that specifies a response spectrum mode combination from a single source load only.

   (GPRF 2014.10)

3. The CREATE PSEUDO STATIC LOAD command produces the following error message:

   **** ERROR_EFDCPS -- Model inconsistencies detected during creation of pseudo static load RMS01 . Analysis terminated and scan mode entered.

**** WARN_STDSY1 -- SCAN mode is ON. Analysis will be halted.

when the following three conditions are present:

a. The active solver has been specified as either GTSES or GT64M.

b. Elements of different types having different numbers of nodal stress and strain components are present in the model, where the last element defined (element name and incidences given/created) has the largest number of such components. For example, a plate model may contain both SBHQ6 and SBHT6 element types. Because the SBHQ6 element has more nodal stress and strain components than the SBHT6 element, this condition is satisfied if a SBHQ6 element is the last element defined.

c. The command specifies a response spectrum mode combination from two or more source loads.

d. The command follows another CREATE PSEUDO STATIC command that specifies a response spectrum mode combination from a single source load only.

(GPRF 2014.11)

3.2 General

1. The algorithm to find maximum shear or moment locations along a member has been corrected. This capability is used in options of the SELECT and CHECK commands and for shear and moment diagrams in GTMenu.

   (GPRF 2014.04)

3.3 GTMenu

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

1. The syntax error in the CONVERGENCE TOLERANCE EQUILIBRIUM command that is produced by the GTMENU File -> Generate GTSTRUDL Text Input ... function has been corrected.
2. The Check - Duplicates & Floating Joints... - Duplicate Elements function now works properly for planar elements with 3 nodes and solid elements with 6 nodes.

3. An abort will no longer occur when finite element results have been contoured and a dialog which normally pops in the dialog area such as the Deformed Structure dialog is moved over the graphical display area.

4. An abort will no longer occur when a large number of Construction Points and Lines are read from an input file followed by entering GTMenu and then changing the views and then repeatedly zooming in on part of the model.

5. The Results Sum Forces dialog will no longer abort when making a cut on a skewed geometry. This dialog has been revised as previously described in Chapter 2.

6. An abort will no longer occur when deleting elements using Plane Mode.

7. Element loads may now be displayed in Hit mode in the Display Load dialog when Element loads are selected. Previously, an abort would occur or GTMenu would no longer respond.

8. GTMenu will now display local member loads in the member's local coordinate system when Input System is selected in the Display Load dialog. Previously, the local loads would always be transferred to the global system and the global components displayed.

9. You can now change the primary direction of the joint, member and element numbering when using Construction Points and Lines to create 2D and 3D member and finite element models. The V primary direction of numbering which is specified at the bottom of the 2D and 3D member and element Spacing dialogs will now work so the joints and members or elements will now be numbered first in the V direction.

10. The Generate Display List option in the Display Model dialog now works using Plane or Volume mode for selection of the Display List.

11. An abort will no longer occur when entering GTMenu if your model contains the TRANS3D solid element.

12. Animation of deformed shapes will no longer abort if your model contains the TRANS3D, IPSQ or IPQS solid elements.

13. An abort will no longer occur when editing member id's in line, plane or volume modes and you select the No button in either the "Ok to continue" pop-up or the "Continue with current selection" pop-up after selecting the line, plane or volume.

14. The List mode selection now works for more than one item when editing Construction Points, Lines and Surfaces.

15. You may now delete Construction Lines and Curves.

16. Dialogs requiring a response are no longer popped behind other dialogs. This problem has prevented the user from finding and responding to the dialogs. This more frequently occurred when creating Views and either the Enter Data dialog for the new view or the Process View dialog asking you to accept the new view would pop behind the other dialogs and be inaccessible.

3.4 GT STRUDL Output Window

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

1. The active units are now displayed correctly at the top of the List Dynamic Analysis Results dialog (Results - Dynamic Analysis Results - List).

3.5 Nonlinear Analysis

1. A new parameter, EYOUNG, has been added to the existing set of parameters, FY, ESH, FSU, ESU, and ALPHA, that are used to define the constitutive behavior of the steel in plastic hinges and segments. EYOUNG is used to define Young's modulus for the plastic hinge/segment steel and is taken as 29,000 ksi by default.

   Documentation:

   > The NONLINEAR EFFECTS Command, Section 2.5.2.1, Volume 3, GT STRUDL Reference Manual.

   (GPRF 2014.07)

2. The CURVE option for the specification of material constitutive characteristics

for steel, concrete, and general custom plastic hinges/segments has been enhanced with the addition of an associated "yield" strain parameter that is used for the computation of the plastic hinge/segment "yield" state as reported by the LIST PLASTIC HINGE STATUS command.

Documentation:

The NONLINEAR EFFECTS Command, Section 2.5.2.1, Volume 3, GT STRUDL Reference Manual.

The DEFINE PLASTIC HINGE CROSS SECTION Command, Section 2.5.2.2.1, Volume 3, GT STRUDL Reference Manual.

The LIST PLASTIC HINGE STATUS Command, Section 2.5.6.5, Volume 3, GT STRUDL Reference Manual.

(GPRF 2014.07)

3.6 Offshore

1. When a GT STRUDL DEFINE command (DEFINE GROUP, DEFINE OBJECT, etc.) is read by GTSELOS, an error condition is flagged that prevents GTSELOS from performing any further analysis operations. This has been corrected for Version 2015, whereby such commands are now identified by GTSELOS as unrecognized" and processing is permitted to continue normally. This problem occurs only in Version 33 of GTSELOS.

   (No GPRF issued)

3.7 Steel Design

1. The SELECT or CHECK command with the ALL MEMBERS option no longer aborts when the GTSES or GT64M is specified for the ANALYSIS and the first member created in the model has been deleted.

   (GPRF 2014.06)

2. The documentation error for the ELASTIC CONNECTION option of the MEMBER RELEASE command (Section 11.1 of the Volume UNISTRUT) has been corrected. This command has been modified to accept "WELD OFFSET DISTANCE" or "JOINT OFFSET X".

elastic connection specs = ELASTIC (CONNECTION) { type specs / spring specs }

type specs =

TYPE i (DESIGN (FACTOR) v_7) ({ { (WELD) OFFSET (DISTANCE) v_8 / (JOINT) OFFSET (X) V_8 } / EXPONENT v_9 })

spring specs = ELASTIC (CONNECTION) [KFX] v_1 [KFY] v_2 – [KFZ] v_3 [KMX] v_4 [KMY] v_5 [KMZ] v_6

(GPRF 2014.09)

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 33. 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)

4.2 General Input/Output

1. 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)
2. 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)

5. GTMenu is limited to 1,000 views. If more than 1,000 views are created, incorrect displays may occur.

6. GTMenu is limited to 10,000 Member Property Groups. If more than 10,000 property groups are created, incorrect results may occur.

Chapter 5

Prerelease Features

5.1 Introduction

This chapter describes new features that have been added to GT STRUDL 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 GT STRUDL 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 or AISC14 codes which are released features. Consequently, this will be the last release with the LRFD3 code available.
- 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. Consequently, this will be the last release with the 318-99 code available.
- 5.2.3 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)
- 5.2.4 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

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES 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 GT STRUDL 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 or AISC14 codes which are released features.

The LRFD3 code of GT STRUDL 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	

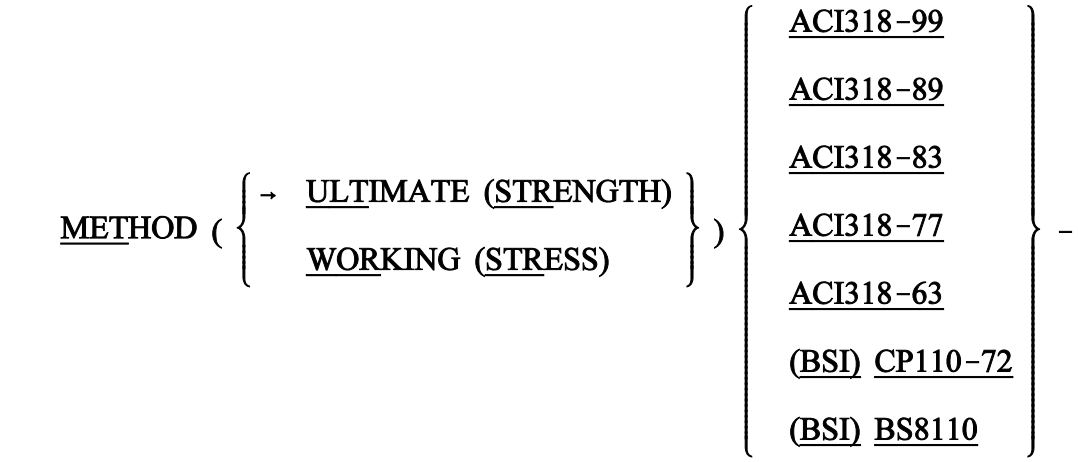
Design for bi-axial bending, axial, and torsional forces:

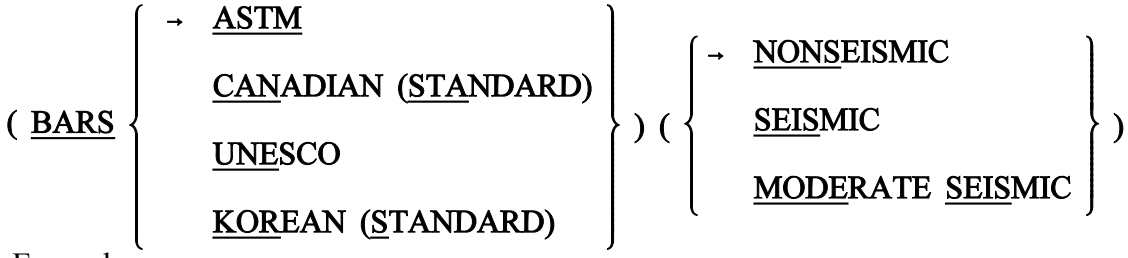
Round HSS (Pipes)
Rectangular and Square HSS (Structural Tubes)

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





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	$29x10^6$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 GT STRUDL 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 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:

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

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

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

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

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

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

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 5.2.3-1. 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.

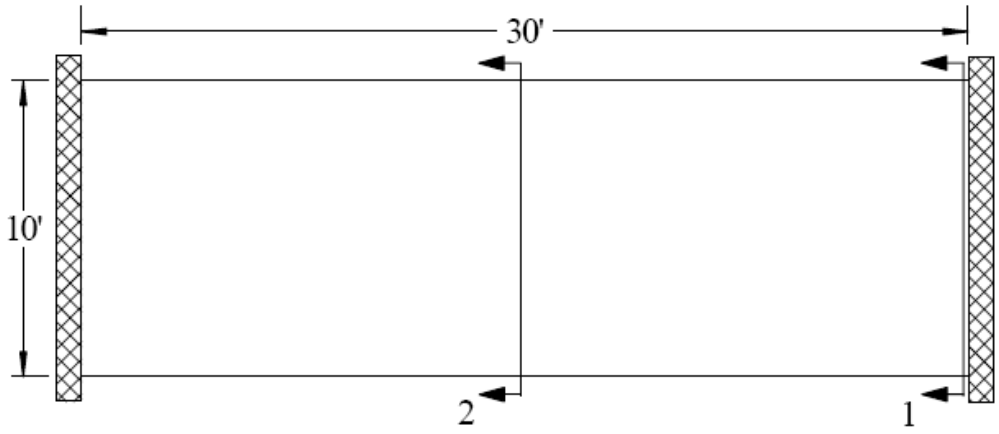


Figure 5.2.3-1 Example Flat Plate Structure (PLAN)

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.

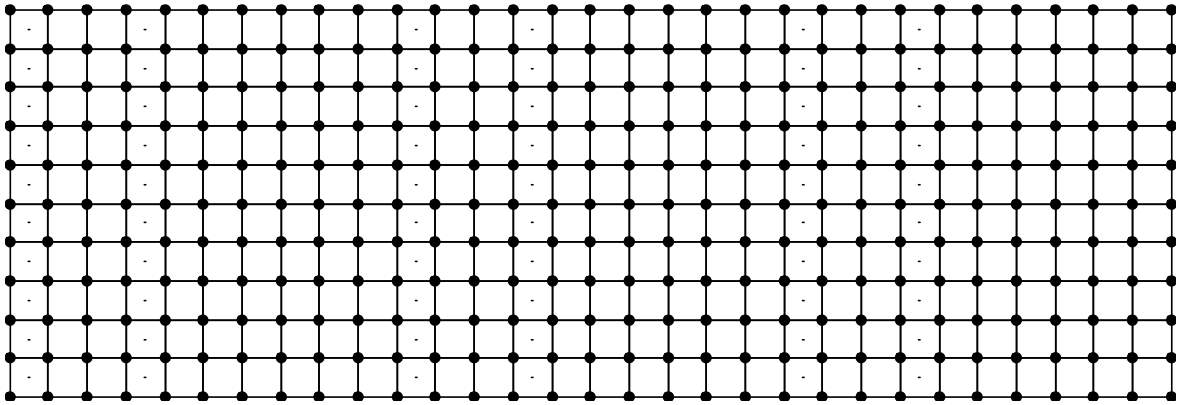


Figure 5.2.3-2 Example Finite Element Model

Definition of Cut Cross-Sections

Two “cuts” are considered for the verification example, as shown in Figure 5.2.3-1.

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 GT STRUDL 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
```

5.2.4 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 GT STRUDL 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) -

* { ENERGY (NORM)
MAX DIFFERENCE
DIFFERENCE FROM AVERAGE
PERCENT MAX DIFFERENCE
PERCENT DIFFERENCE FROM AVERAGE
NORMALIZED PERCENT MAX DIFFERENCE
NORMALIZED PERCENT DIFFERENCE FROM AVERAGE } -

(AT) * { TOP
MIDDLE
BOTTOM } (SURFACES) (FOR) { → ALL
ELEMENT list }

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:

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

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,

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

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

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

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

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

5.3.2 The CALCULATE ECCENTRIC MEMBER BETA ANGLES Command

General form:

CALCULATE ECCENTRIC (MEMBER) (BETA) (ANGLES) (WITHOUT - COMMAND (LISTING))

Explanation:

Section 1.10.4 states that the member beta angle (the orientation of the member cross section principal axes) is defined with respect to the joint-to-joint position of the member before member eccentricities are applied. However, in certain structural modeling situations it may be more desirable to be able to specify a beta angle value that is defined with respect to the eccentric position of the member, after member eccentricities are applied. To this end, the CALCULATE ECCENTRIC MEMBER BETA ANGLES command has been implemented in order to provide beta angle information that can be used to construct CONSTANTS commands that specify beta angle values that reflect such a need. When issued, the CALCULATE ECCENTRIC MEMBER BETA ANGLES command produces a report that includes the member name, the member's originally-specified or -computed joint-to-joint beta angle value, and an adjusted joint-to-joint beta angle value that if specified, produces a member orientation and associated analysis behavior as if the original beta angle were defined with respect to the eccentric position of the member. The report also includes a listing of CONSTANTS/BETA commands for all affected members that can be easily copied and pasted into a GTSTRUDL command text file. If this command listing is not desired, it can be eliminated by giving the WITHOUT COMMAND LISTING option. An example of the report is reproduced below:

```
{  657} > CALCULATE ECCENTRIC MEMBER BETA ANGLES

**** WARNING_CHKECCBTA -- The CALCULATE ECCENTRIC MEMBER BETA ANGLES command is
                         a prerelease feature.  User feedback and suggestions
                         are welcome.


   ****************************
   *RESULTS OF LATEST ANALYSIS*
   ****************************

   PROBLEM - None

   ACTIVE UNITS  FEET KIP  RAD  DEGF SEC
```

```
The following report lists adjusted beta angle values that if specified, produce member
orientations, including corresponding analysis behavior, as if the ORIGINALLY-SPECIFIED
beta angles were defined with respect to the eccentric position of the member.  This
report is for information purposes only.  No computational action is taken.
```

Eccentric Member Beta Angle Check Results

Member	Original Beta Angle	Adjusted Beta Angle
11002	0.06655	0.09484
12002	-0.02815	0.00884
11003	-3.04469	-3.06850
13002	1.26565	2.52545
14002	1.16144	2.31630
15002	1.05723	2.10572
16002	0.95302	1.89668
13003	1.26565	-0.61557
14003	1.16144	-0.79819
15003	1.05723	-1.03473
16003	0.95302	-1.24443
17002	-0.06191	0.01547
18002	-0.44292	-0.58340
18003	3.13987	3.35983

CONSTANTS/BETA Commands for Adjusted Beta Angles

```
UNITS RAD
CONSTANTS
  BETA       0.09484  MEMBER  '11002   '
  BETA       0.00884  MEMBER  '12002   '
  BETA      -3.06850  MEMBER  '11003   '
  BETA       2.52545  MEMBER  '13002   '
  BETA       2.31630  MEMBER  '14002   '
  BETA       2.10572  MEMBER  '15002   '
  BETA       1.89668  MEMBER  '16002   '
  BETA      -0.61557  MEMBER  '13003   '
  BETA      -0.79819  MEMBER  '14003   '
  BETA      -1.03473  MEMBER  '15003   '
  BETA      -1.24443  MEMBER  '16003   '
  BETA       0.01547  MEMBER  '17002   '
  BETA      -0.58340  MEMBER  '18002   '
  BETA       3.35983  MEMBER  '18003   '
```

Note that members are listed only if they are active, they have global eccentricities, and the originally-specified beta angle and the adjusted beta angle differ by more than 1^{o}.

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 GT STRUDL User Reference Manual.

2.1.11.4.6 The ROTATE LOAD Command

General form:

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

Elements:

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

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

Explanation:

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

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

REFERENCE (COORDINATE) (SYSTEM) $\left\{ \begin{matrix} i_1 \\ 'a_1' \end{matrix} \right\}$ –

$$\left\{ \begin{matrix} (\text{ORIGIN } [\text{X}]\ v_x\ [\text{Y}]\ v_y\ [\text{Z}]\ v_z)\ (\text{ROTATION } [\text{R1}]\ v_1\ [\text{R2}]\ v_2\ [\text{R3}]\ v_3) \\ \left\{ \begin{matrix} \text{JOINT} \left\{ \begin{matrix} i_2 \\ 'a_2' \end{matrix} \right\} \\ \text{X } v_4\ \text{Y } v_5\ \text{Z } v_6 \end{matrix} \right\} \left\{ \begin{matrix} \text{JOINT} \left\{ \begin{matrix} i_2 \\ 'a_2' \end{matrix} \right\} \\ \text{X } v_4\ \text{Y } v_5\ \text{Z } v_6 \end{matrix} \right\} \left\{ \begin{matrix} \text{JOINT} \left\{ \begin{matrix} i_2 \\ 'a_2' \end{matrix} \right\} \\ \text{X } v_4\ \text{Y } v_5\ \text{Z } v_6 \end{matrix} \right\} \end{matrix} \right\}$$

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 GT STRUDL 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 GT STRUDL 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 GT STRUDL. When this option is used, a reference system is created and its definitions of the system origin, rotation angles, as well as the transformation matrix between the global coordinate system and the reference system are generated and stored as would be for the first option. Therefore, if any of the coordinates for the joints used to specify a reference system is changed after the REFERENCE COORDINATE SYSTEM command has been given, the definition of the reference system remains unchanged. For this reason, care must be taken in using the three joints option in conjunction with the changes of joint coordinates. The reference system should be deleted first if any of the coordinates of the joints used to define the reference system are to be changed. Under the DELETIONS mode, the complete definition of the reference coordinate system is destroyed.

Examples:

a)
```
UNITS DEGREES
REFERENCE COORDINATE SYSTEM 'FLOOR2' -
      ORIGIN 0.0 15.0 0.0 R1 30.
```

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

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

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

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

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

d)
```
CHANGES
REFERENCE COORDINATE SYSTEM 'FLOOR2' -
      ORIGIN  10  20  30
ADDITIONS
```

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

e)
```
DELETIONS
REFERENCE  SYSTEM 2
ADDITIONS
```

The above command deletes Reference System 2.

5.4.2-1 Printing Reference Coordinate System Command

General form:

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

Explanation:

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

5.4.3 GTMenu Point and Line Incidences Commands

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

(1) GTMenu POINT COORDINATES

General Form:

GTMenu POINT COORDINATES

$$\left\{ \text{'a}_1\text{'} \right\} \quad \text{coordinate-specs}_1$$

.
.
.

$$\left\{ \text{'a}_n\text{'} \right\} \quad \text{coordinate-specs}_n$$

Elements:

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

where,

'a_1', 'a_2', ..., 'a_n' = 1 to 8 character alphanumeric Point identifiers beginning with P (i.e. P1 P2 ...)

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

(2) **GTMenu LINE INCIDENCES**

General Form:

GTMenu LINE INCIDENCES

$\left\{ \text{'a}_1\text{'} \right\}$ type_1 incidence-specs_1

.
.
.

$\left\{ \text{'a}_n\text{'} \right\}$ type_n incidence-specs_n

Elements:

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

$\text{ıcidence-specs} = \left\{ \text{point}_1\text{'} \right\} \left\{ \text{'point}_2\text{'} \right\} \ldots \left\{ \text{'point}_p\text{'} \right.$

where,

‘a_1’, ‘a_2’, ..., ‘a_n’	=	1 to 8 character alphanumeric Line/Curve identifiers beginning with C (.i.e. C1, C2).
‘$point_1$’, ..., ‘$point_p$’	=	1 to 8 character alphanumeric Point identifiers beginning with P (i.e. P1, P2).
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