GT STRUDL® Version 2016 R1 Release Guide

PROCESS, POWER & MARINE

Release Guide

Release Date: March 2016

PART OF HEXAGON

INTERGRAPH®

Notice

This GT STRUDL Release Guide is applicable to GT STRUDL Version 2016 R1 and later versions for use on PCs under the Microsoft Windows operating systems.

Copyright

U.S. Government Restricted Rights Legend

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

Disclaimer of Warranties

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 and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries. Other brands and product names are trademarks of their respective owners.

Table of Contents

CHAPTER 5   P<small>RERELEASE</small> F<small>EATURES</small>

This page intentionally left blank.

Chapter 1

Introduction

Version 2016 R1 covers GT STRUDL operating on PC's under the Windows 10, 8 and 7 operating systems. For users who are accustomed to our older version numbering system, the version is internally known as Version 35.1.

Chapter 2 of this release guide presents the new features and enhancements which have been added since the release of Version 2016. In particular, Chapter 2 briefly describes new features including the following significant new features:

• The addition of the Indian Standard steel design code IS800-07.

• The addition of new Indian Standard steel tables and a Smart 3D table for built-up tubes.

• Enhancements to GTMenu when placing members and when creating groups and also the addition of scroll bars for dockable dialogs.

• The addition of the Hot Spot Influence Function (HSIF) method for fatigue stress computation for offshore structures.

Chapter 3 provides you with details regarding error corrections that have been made since the Version 2016 release. Chapter 4 describes known problems with Version 2016 R1. 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 Design of Flat Plates Based on the Results of Finite Element Analysis (The DESIGN SLAB Command)

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

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

Chapter 2

New Features in Version 2016 R1

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 2016 R1. This release guide is also available online upon execution of GT STRUDL under Help - Reference Documentation - GT STRUDL Release Guide.

2.1   Steel Design

1.   A new Indian Standard IS 800 : 2007 design code (adopted on 2007) has been implemented as a released feature. The GT STRUDL code name is called IS800-07 which is primarily based on the *IS 800 : 2007, Indian Standard, General Construction in Steel – Code of Practice*, Third Revision, Reaffirmed in 2012. This new code, IS800-07, may be used to select or check any of the following shapes:

> I shapes
> Channels
> Rectangular Hollow Sections (Tubes)
> Circular Hollow Sections (Pipes)
> Single Angles
> Tees
> Double Angles
> Solid Rectangular Bars
> Solid Round Bars

The documentation for the IS800-07 code may be found by selecting Help and then Reference Documentation, Reference Manuals, Steel Design, and IS800-07 in the GT STRUDL Output Window.

2.2   Steel Tables

1.   The following new Indian Standard tables have been added:

ISHOLLOW - square and rectangular hollow (tube) sections

ISTEES - tee sections

ISTUBES - circular tube sections

These Indian Standard tables are in addition to the following tables for Indian Standard profiles which already existed in GT STRUDL:

ISBEAMS and ISCOLUMN - I sections

ISCHAN and ISCHAMCP - channel sections

ISEQANGL and ISUNANGL - single angle sections

2.   The following Intergraph Smart 3D built-up tube table has been added:

BUTube

This table contains over 900 tubular sections ranging in diameter from 508 mm to 2400 mm.

2.3   GTMenu

1.   Horizontal and vertical Scroll Bars have been added to Dockable dialogs as shown
below. The scroll bars make it easier to manipulate dialogs when GTMenu is not in
full screen mode or when using a low resolution display.
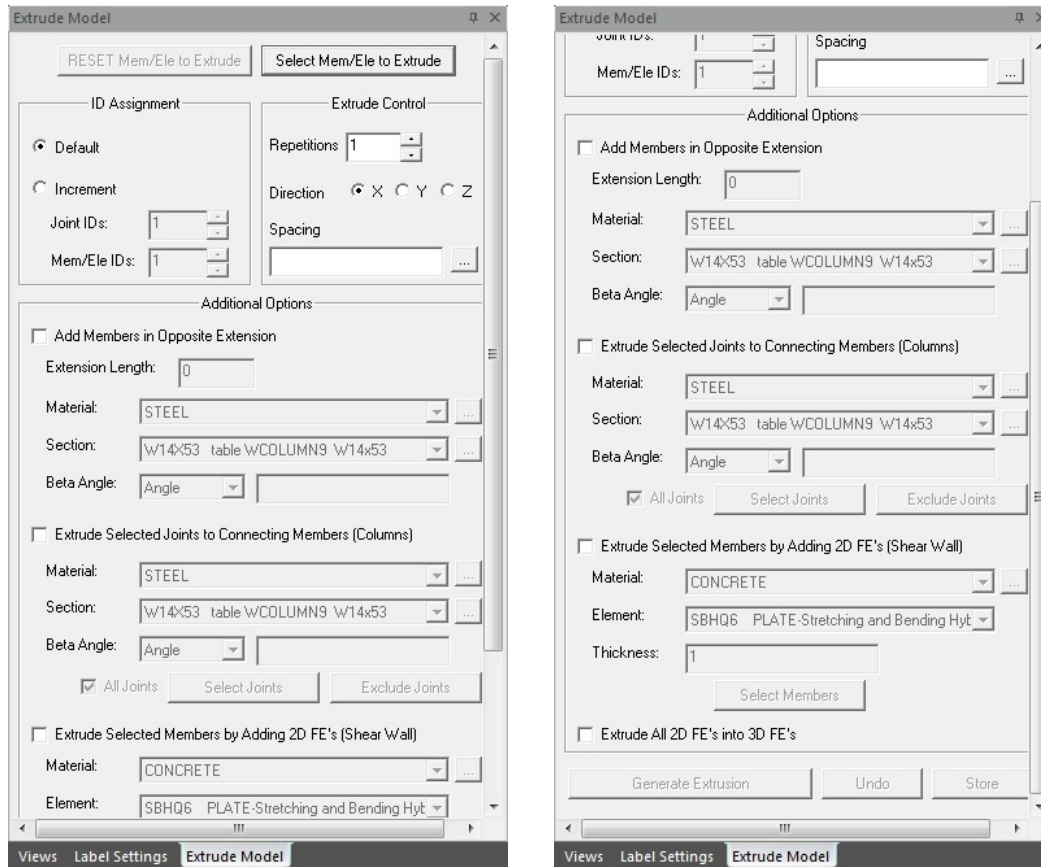
2.   Member End Releases can now be defined in the Place Members Dialog as shown
          below:
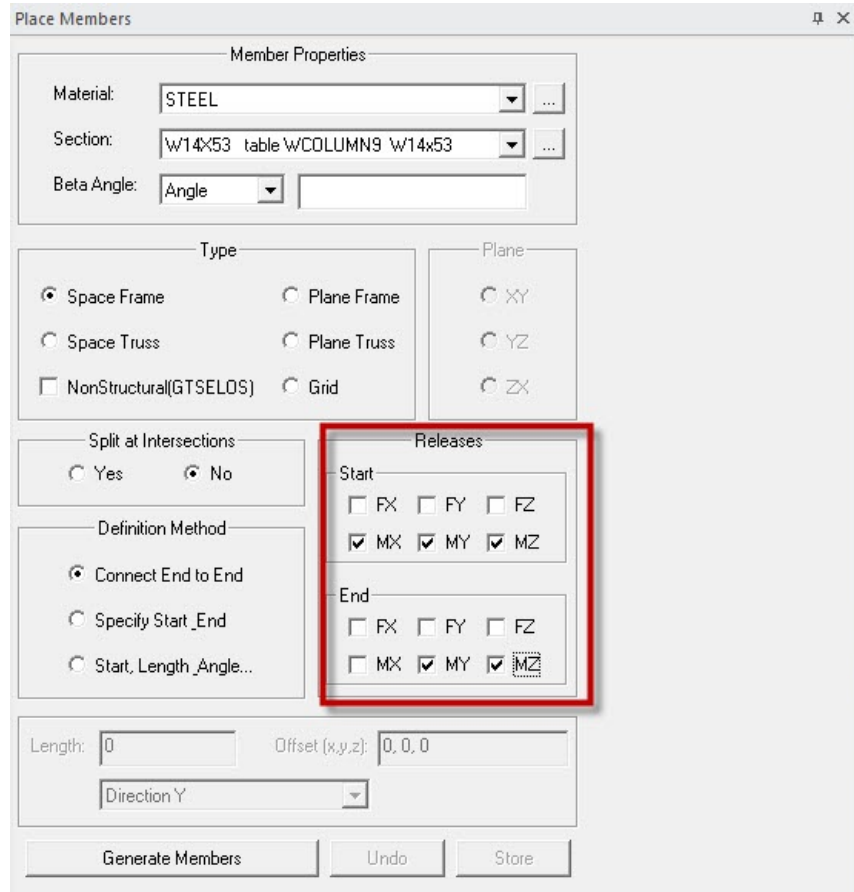
If the Split at Intersection option is selected, the releases prescribed will be applied
to the splitting of the new members only.

3.   The Create/Edit Groups Dialog has been improved to allow Physical Members to be created using the dialog. If members are added to a Physical Member Group, they will be checked since Physical Members require all the Members to be collinear, contiguous, and to have the same cross section and beta angle. Also, there should not be internal member releases in the physical member. The revised dialog is shown below:
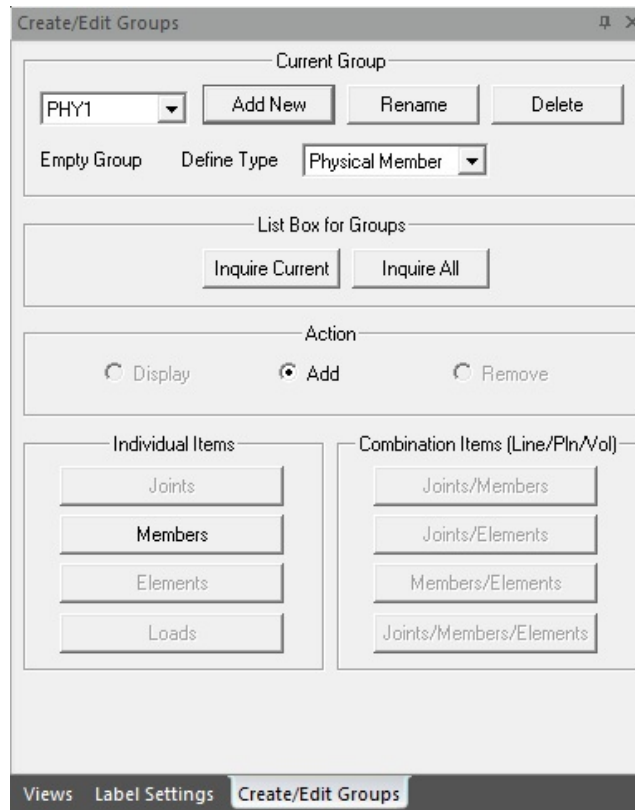
More information regarding Physical Members may be found in Section 2.1.2.10.1 of Volume 1 of the Reference Manuals.

4.   The Color by Section option of the Display Model dialog has been improved to allow
        the user to select which sections are to be shown as solids. This feature allows the
        user to easily check if the correct properties have been assigned to the members.

        This option can be accessed directly through a button in the Display Model dialog,
        or through a button in the Design Parameters Dialog. Shown below in the left figure
        is the revised Display Model dialog. When the Color by Section option is selected,
        the Section Properties legend is popped up as shown in the figure on the right which
        allows you to select which sections are to be drawn as solids with the colors shown
        in the legend. You can select one or more of the sections in the legend using
        Windows multi-select functionality and the members are immediately drawn as
        solids. If you select the Refresh List button, all sections are drawn as solids. If you
        select the Label on Screen button, you can place a legend on the screen.
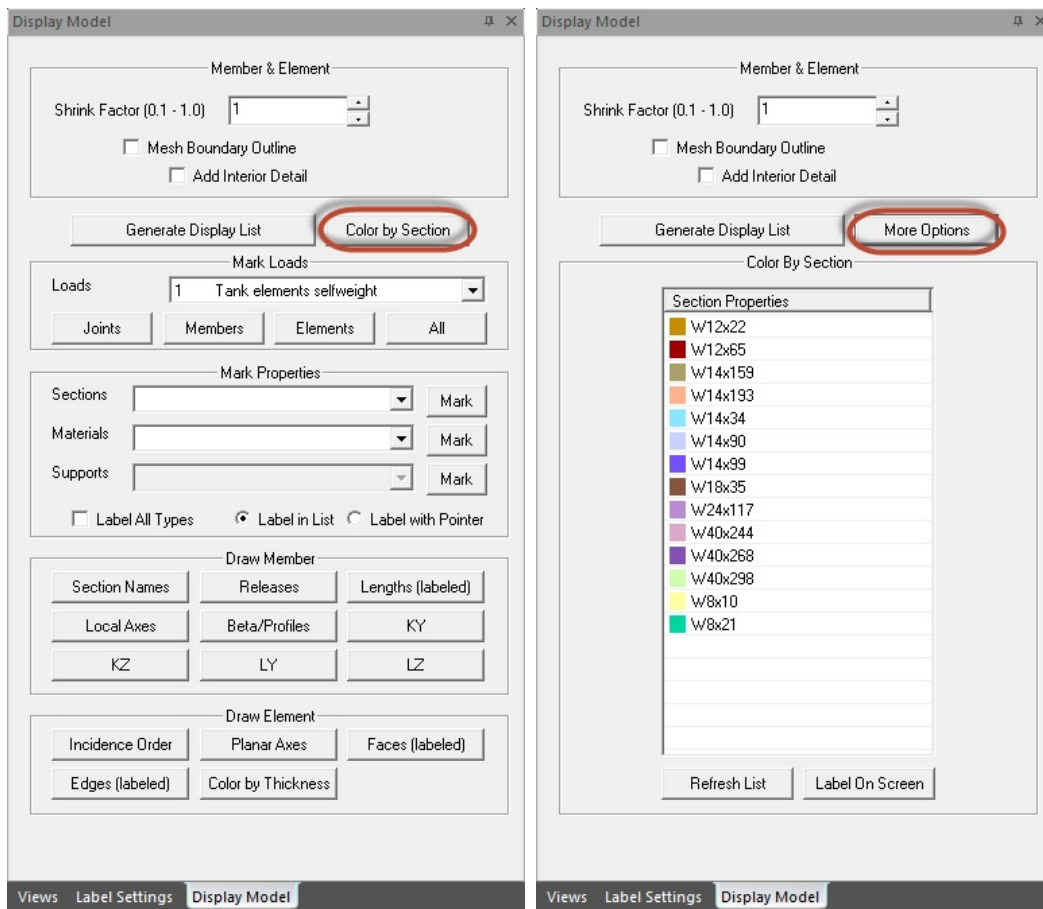
To return to the Display Model dialog, select the More Options button.

An example showing the use of this feature is shown below where only the members which have the property of W12x45 are drawn as solids with a legend placed in the graphical display area.
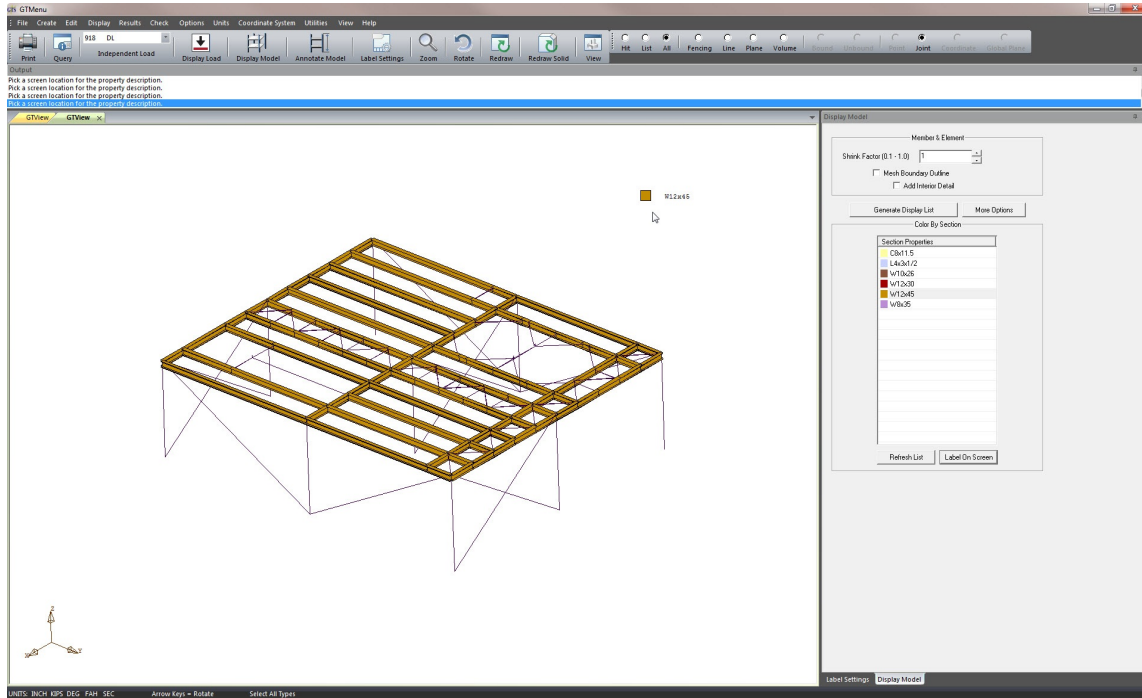
You may also access this feature using the Color by Section option in the Display Parameters dialog as shown below. You can then return to the Display Parameters dialog by selecting the Back to Parameters button as shown in the figure on the right below.
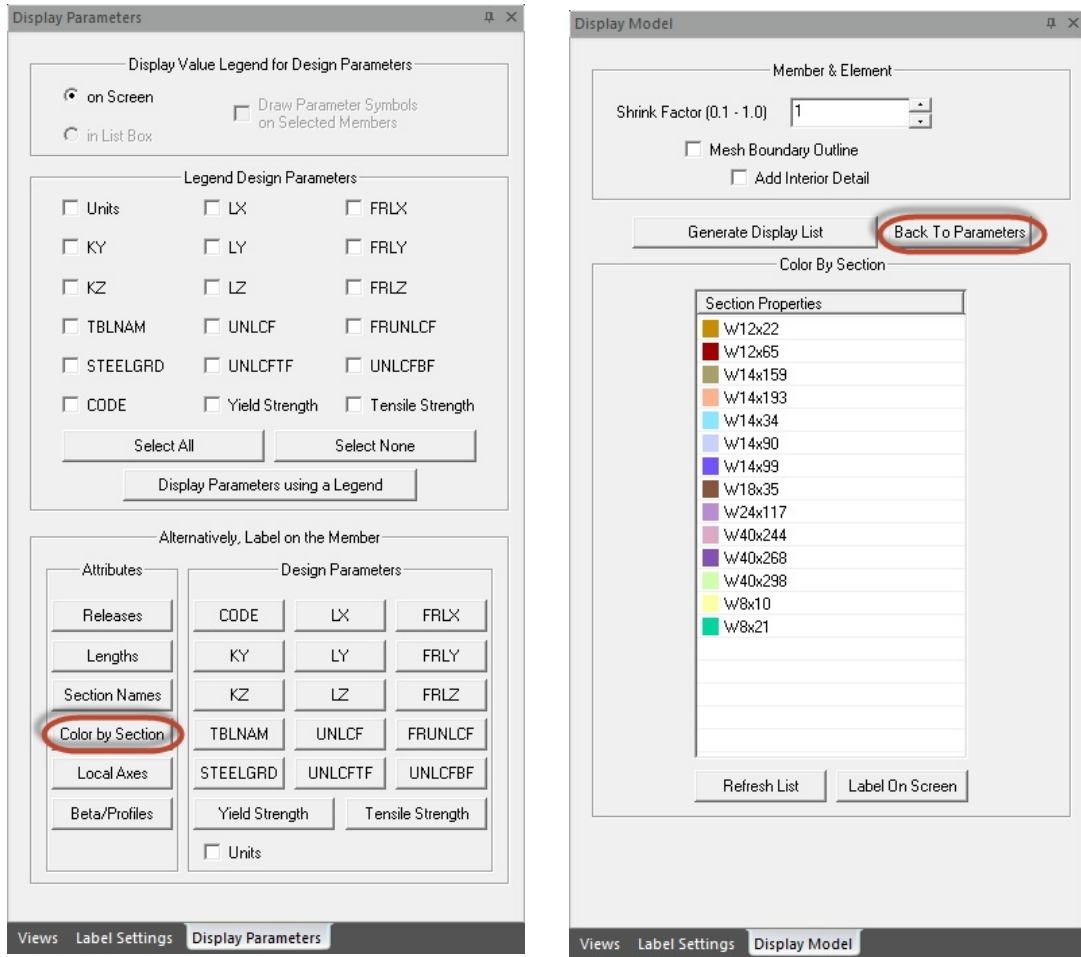
2.4   CAD Modeler

1.   You may now import user define cross section tables that were created in GT
        STRUDL (cross section tables in a userdat.ds file). User defined cross sections, that
        have been created in GT STRUDL, can be imported in CAD Modeler from the Menu
        GTS Modeling > Cross Sections > Import from User Dataset" or by typing
        GTSOpenDS at the command prompt. Using the "Select User Dataset DS File"
        dialog, you can select the dataset file (*.ds) that includes the user defined cross
        sections to be imported. By pressing "Open" the cross sections in the selected .ds file
        are added to the available built-in tables in CAD Modeler..

2.5   Offshore

1.   The Hot Spot Influence Function (HSIF) method for fatigue stress computation has
        been added to the offshore fatigue analysis capability. The purpose of the HSIF
        method is to provide for the computation of effective hot spot stresses in arbitrary
        weld configurations that may be best modeled as 2-D/3-D finite element
        substructures that are too complex and cumbersome to be included in the one-
        dimensional frame member models normally employed for fatigue analysis. This is
        accomplished by using a hot spot influence matrix to transform member section
        forces to a set of weld stress components that are further used as input to the S-N
        curve for the computation of fatigue cycles to failure. The influence matrices are
        specified by the new FATIGUE MEMBER HSIF DATA command, which is
        described in Section 5.3.1.2, Volume 8, GT STRUDL Reference Manual.

        Documentation:

                Joint and Member Data for Fatigue Analysis, Section 5.3, Volume 8, GT
                STRUDL Reference Manual.

                The FATIGUE MEMBER Command for the HSIF method, Section 5.4,
                Volume 8, GT STRUDL Reference Manual.

2.6   DBX

1.   The WRITE command now opens DBX files with the attribute SHARE='DENYRW',
        which prevents another program from opening the DBX file until GT STRUDL has
        closed it. This will prevent programs that use DBX from reading DBX files before
        they are complete.

Chapter 3

Error Corrections

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

3.1   Finite Elements

1.   The LIST STRESSES/STRAINS commands no longer abort for elements Q6CDRL, PBMITC and SBMITC when you have specified the command USE EXTERNAL FILE SOLVER under the DYNAMIC PARAMETERS command before or after a STIFFNESS ANALYSIS which used the standard equation solver.

In addition, the LIST STRESSES/STRAINS commands no longer report incorrect stress or strain results when you use the following sequence of commands where you have specified using the external file solver but the EIGENPROBLEM PARAMETER command specified using the standard GTLANCZOS eigensolver instead of the GTSELANCZOS or GT64MLANCZOS eigensolvers:

DYNAMIC PARAMETERS
        USE EXTERNAL FILE SOLVER
END

EIGENPROBLEM PARAMETERS
        SOLVE USING GTLANCZOS
        NUMBER OF MODES 2
END

DYNAMIC ANALYSIS EIGENVALUE
RESPONSE SPECTRUM LOAD 1
        SUPPORT ACCELERATION
        TRANSLATION Z 1.0 FILE 'S.7GD51'
END

DAMPING PERCENT 5.0 2

PERFORM RESPONSE SPECTRUM ANALYSIS

COMPUTE RESPONSE SPECTRUM FORCE STRESSES -
 MODE COMBINATION RMS
CREATE PSEUDO STATIC LOAD 2 FROM RMS OF LOAD 1

LIST STRESSES STRAINS

(GPRF 2016.02)

3.2 GTMenu

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

1.   GTMenu will no longer abort if more than one instance of GTMenu is opened in the
        same Working Directory.

2.   The default size of the Inquiry Output Dialog has been modified, so that a proper size
        is shown when in floating mode.

3.   An abort will no longer occur when using the Sum Forces at Cut dialog when the
        selection of a cut was cancelled or there were no joints selected.

4.   The Edit Loads Dialog will now show the independent loads in the drop-down.

5.   The Undo option in the Place Members Dialog will no longer delete some existing
        joints when the Split at Intersection option has been used.

6.   New elements, members, and joints will now be drawn correctly if a Query has been
        used right before generating these objects.

7.   Hit mode no longer behaves erratically when turning fencing on/off.

8.   The Fencing option has been fixed/implemented for all of the dialogs which could
        use this function.

9.   Dockable Dialog are no longer unresponsive after pressing the Escape key.

10.   The Element Stresses Dialog now correctly shows which stress results are available for various element types.
11.   The Principal Vector option in the Results Menu is now disabled when there are solid (Tridimensional) elements in the current view.

3.3   Nonlinear

1.   Nonlinear dynamic analysis now correctly computes analysis results for the case when the model contains plane and space truss members having tension/ compression-only nonlinearity in combination with friction damper nonlinearity.

(GPRF 2016.01)

Documentation:

The NONLINEAR EFFECTS Command, Section 2.5.2.1, Volume 3.

Nonlinear Dynamic Analysis, Section 2.4.10, Volume 3

Chapter 4

Known Deficiencies

This chapter describes known problems or deficiencies in Version 2016 R1. 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   CAD Modeler
*(GPRF's are **not** issued for CAD Modeler unless specifically noted below)*

1.   Loads are not copied or mirrored when using the Copy or Mirror commands.

2.   The Beta angles and Loads are not rotated or mirrored when using the Rotate or Mirror commands.

4.2   Finite Elements

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

4.3   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 when all of the following conditions are present:

   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 GT STRUDL.

(GPRF 2004.14)

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

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 GT STRUDL 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 GT STRUDL Reference Manual.

   (GPRF 95.21)

3.   The Check Load option in CHECK MODEL dialog 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 GT STRUDL 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.
        (No GPRF issued)

6.   The Deformed Structure display with the Deform between Joints option may produce inconsistent results for nonlinear geometric frame members. The deformed structure may show a discontinuity at the joints.
        (No GPRF issued)

7.   GTMenu is limited to 10,000 Member Property Groups. If more than 10,000 property groups are created, incorrect results may occur.
        (No GPRF issued)

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

    5.2.2   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   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:

DESIGN SLAB (REINFORCEMENT) (USING) –

$$
\left\{ \begin{array}{l} \text{WOOD (AND) (ARMER)} \left\{ \begin{array}{l} \rightarrow \text{AVERAGE} \\ \text{MAXIMUM} \end{array} \right\} \\ \text{CALCULATE (RESULTANT) (ELEMENT) (FORCES)} \end{array} \right\} \text{ (ALONG)} -
$$

$$
(\text{CUT} \left\{ \begin{array}{l} \text{'a'} \\ i_1 \end{array} \right\}) \left\{ \begin{array}{l} \text{JOINTS} \\ \text{NODES} \end{array} \right\} \text{ list}_1 \text{ ELEMENT list}_2 \text{ (TABLE} \left\{ \begin{array}{l} \rightarrow \text{ASTM} \\ \text{UNESCO} \end{array} \right\}) -
$$

$$
{}^{*} \left[\begin{array}{l} \text{TOP (FACE) (BARS } i_2) \text{ (SPACING } v_1) \\ \text{BOTTOM (FACE) (BARS } i_3) \text{ (SPACING } v_2) \\ \text{BOTH (FACES) (BARS } i_4) \text{ (SPACING } v_3) \end{array} \right] -
$$

$$
\left\{ \begin{array}{l} \rightarrow \text{INNER (LAYER)} \\ \text{OUTER (LAYER)} \end{array} \right\} \text{ (COVER } v_4) \text{ (LINEAR (TOLERANCE) } v_5) -
$$

(TORSIONAL (MOMENT) (WARNING) 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.
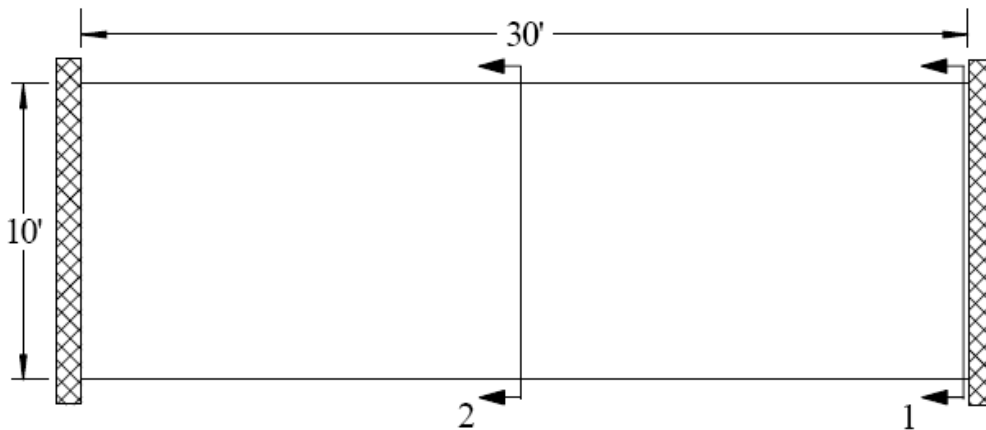
Figure 5.2.3-1   Example Flat Plate Structure (PLAN)

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.
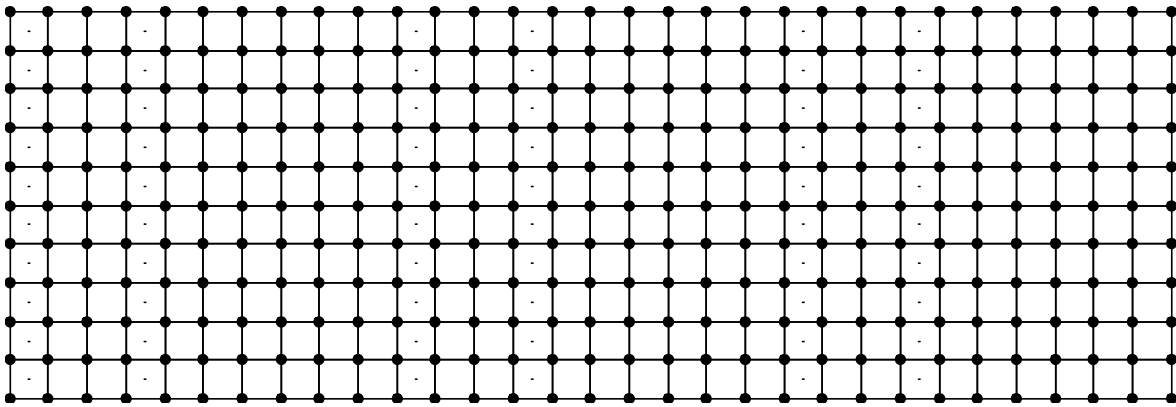
Figure 5.2.3-2   Example Finite Element Model

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
```

5.2.2 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) -

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

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

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

The L_2 norm is given by:

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

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,

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

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

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

Normalized Percent Maximum Difference

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

Normalized Percent Difference from Average Method

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

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

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

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

1.   Enter GTMenu.

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

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

4.   Select the Loading.

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

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

5.3.2   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 1º.

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:

$$\underline{\text{RO}}\text{TATE} \quad \underline{\text{LOA}}\text{DING} \quad \left\{ \begin{array}{c} i_R \\ \\ {}'a_R{}' \end{array} \right\} \; (\underline{\text{ANGLES}} \;) \left[\underline{\text{T1}} \right] r_1 \left[\underline{\text{T2}} \right] r_2 \left[\underline{\text{T3}} \right] r_3$$

Elements:

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

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

Explanation:

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

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

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

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

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:

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

Explanation:

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

5.4.3   GTMenu Point and Line Incidences Commands

GTMenu can now write construction geometry commands to an input file, which can be read later into 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

$$\left\{ \ 'a_1' \ \right\} \quad \text{coordinate-specs}_1$$

.
.
.

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

Elements:

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

$$\left\{ \ 'a_1' \ \right\} \quad type_1 \quad incidence\text{-}specs_1$$

.
.
.

$$\left\{ \ 'a_n' \ \right\} \quad type_n \quad incidence\text{-}specs_n$$

Elements:

$$type \quad = \quad \left\{ \begin{array}{l} \rightarrow \underline{LINE} \\[2mm] \underline{POLY}NOMINAL \ (\underline{CUR}VE) \\[2mm] \underline{ARC} \ (\underline{TEM}PLATE) \\[2mm] \underline{CENTERED} \ (\underline{ARC}) \ \underline{PER}CENT \ v_1 \\[2mm] \underline{BEZIER} \ (\underline{CUR}VE) \\[2mm] \underline{SPLINE} \ (\underline{CUR}VE) \ (\underline{ORDER} \ k_2) \end{array} \right\}$$

$$incidence\text{-}specs \quad = \quad \left\{ \ point_1' \ \right\} \ \left\{ \ 'point_2' \ \right\} . \ . \ . \ \left\{ \ 'point_p' \right.$$

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