

GT STRUDL[®]

GT STRUDL Release Guide Version 28.1

Volume 1 of 1

January 2006
Computer-Aided Structural Engineering Center
School of Civil & Environmental Engineering
Georgia Institute of Technology
Atlanta, Georgia 30332-0355
U.S.A.

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

NOTICES

This GTSTRUDL® Release Guide is applicable to Version 28.1, with a release date in the GTSTRUDL title block of January 2006.

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

GTMMenu and its documentation were developed as an enhancement to GTSTRUDL authored by the Computer-Aided Structural Engineering Center, Georgia Institute of Technology.

DISCLAIMER

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

Commercial Software Rights Legend

Any use, duplication, or disclosure of this software by or for the U.S. Government shall be restricted to the terms of a license agreement in accordance with the clause at DFARS 227.7202-3 (October 1998).

This material may be reproduced by or for the U.S. Government pursuant to the copyright license under the clause at DFARS 252-227-7013, September 1989.

© Copyright 1998, 1999, 2000, 2002, 2005, 2006
Georgia Tech Research Corporation
Atlanta, Georgia 30332-0355
U.S.A.

ALL RIGHTS RESERVED

S))))))))))))))Q

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

© Windows XP, Windows 2000, Windows NT, Windows ME, and Windows 98 are registered trademarks of Microsoft Corporation, Redmond Washington.

© Excel is a registered trademark of Microsoft Corporation, Redmond Washington.

Table of Contents

| | |
|---|------|
| NOTICES | ii |
| DISCLAIMER | ii |
| Commercial Software Rights Legend | ii |
| CHAPTER 1 | |
| Introduction | 1-1 |
| CHAPTER 2 | |
| New Features in Version 28.1 | 2-1 |
| 2.1 GTSTRUDL Startup | 2-1 |
| 2.2 Dynamics | 2-7 |
| 2.3 General | 2-8 |
| 2.4 Static Analysis | 2-8 |
| 2.5 Offshore | 2-8 |
| 2.6 Steel Design | 2-11 |
| CHAPTER 3 ERROR CORRECTIONS | 3-1 |
| 3.1 Dynamic Analysis | 3-1 |
| 3.2 General | 3-1 |
| 3.3 Model Wizard | 3-1 |
| 3.4 Offshore | 3-2 |
| 3.5 Steel Design | 3-2 |
| CHAPTER 4 KNOWN DEFICIENCIES | 4-1 |
| 4.1 Finite Elements | 4-1 |
| 4.2 General Input/Output | 4-2 |
| 4.3 GTMenu | 4-3 |
| 4.4 Rigid Bodies | 4-4 |
| 4.5 Scope Environment | 4-4 |

| | | |
|-----------|---|--------|
| CHAPTER 5 | PRERELEASE FEATURES | 5.1-1 |
| 5.2 | Design Prerelease Features | 5.2-1 |
| 5.2.1 | LRFD3 Steel Design Code and Parameters | 5.2-1 |
| 5.2.2 | GTSTRUDL LRFD3 Profile Tables | 5.2-27 |
| 5.2.3 | GTSTRUDL BS5950 Steel Design Code and Parameters | 5.2-31 |
| 5.2.4 | GTSTRUDL Indian Standard Design Code IS800 | 5.2-53 |
| 5.2.5 | GTSTRUDL Profile Tables for the Design based on the IS800 Code | 5.2-61 |
| 5.2.6 | Steel Deflection Check and Design | 5.2-69 |
| 5.2.7 | Brazilian Table | 5.2-75 |
| 5.2.8 | ACI Code 318-99 | 5.2-79 |
| 5.2.9 | Rectangular and Circular Concrete Cross-Section Tables | 5.1-83 |
| 5.2.10 | ASD9-E Code | 5.2-85 |
| 5.3 | Analysis Prerelease Features | 5.3-1 |
| 5.3.1 | The CALCULATE ERROR ESTIMATE Command | 5.3-1 |
| 5.3.2 | Output of Response Spectrum Results | 5.3-5 |
| 5.3.3 | FORM STATIC LOAD Command -- Automatic Generation of Static Equivalent Earthquake Loads | 5.3-9 |
| 5.3.4 | FORM UBC97 LOAD Command -- Automatic Generation of Seismic Loads According to 1997 UBC | 5.3-19 |
| 5.3.5 | FORM IS1893 LOAD Command - Automatic Generation of Static Seismic Loads According to IS 1893 | 5.3-29 |
| 5.3.6 | Element Properties Command for Nonlinear Hysteretic Spring Element | 5.3-35 |
| 5.3.7 | Nonlinear Dynamic Analysis | 5.3-41 |
| 5.3.8 | The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis | 5.3-53 |
| 5.4 | General Prerelease Features | 5.4-1 |
| 5.4.1 | Align Command | 5.4-1 |
| 5.4.2 | ROTATE LOAD Command | 5.4-5 |
| 5.4.3 | RUN Command | 5.4-9 |
| 5.4.4 | COUTPUT Command | 5.4-11 |

| | | |
|---------|--|--------|
| 5.4.5 | Reference Coordinate System Command | 5.4-13 |
| 5.4.5-1 | Printing Reference Coordinate System Command | 5.4-16 |
| 5.4.6 | Hashing Algorithm to Accelerate Input Processing | 5.4-17 |
| 5.4.7 | Add/Delete Without Commands | 5.4-19 |
| 5.4.8 | The List Sum Forces Command | 5.4-23 |
| 5.4.9 | GTMMenu Point and Line Incidences Commands | 5.4-33 |

This page intentionally left blank.

Chapter 1

Introduction

Version 28.1 covers GTSTRUDL operating on PC's under the Windows XP and Windows 2000 operating systems. Chapter 2 presents the new features and enhancements which have been added since the Version 28 release. Chapter 3 provides you with details regarding error corrections that have been made since the Version 28 release. Chapter 4 describes known problems with Version 28.1. Chapter 5 describes prerelease features -- new features which have been developed and subjected to limited testing, or features for which the user documentation have not been added to the GTSTRUDL User Reference Manual. The command formats and functionality of the prerelease features may change before they become supported features based on additional testing and feedback from users.

The Prerelease features in Version 28.1 are the same as those in Version 28 and are included in this Release Guide for completeness. The Prerelease features are subdivided into Design, Analysis, and General categories. The features in these categories and their sections numbers in Chapter 5 are shown below:

- 5.2 Design Prerelease Features
 - 5.2.1 LRFD3 Steel Design Code and Parameters
 - 5.2.2 LRFD3 Tables
 - 5.2.3 BS5950 Steel Design Code and Parameters
 - 5.2.4 Steel Design by Indian Standard Code IS800
 - 5.2.5 IS800 Tables
 - 5.2.6 Steel Deflection Check and Design
 - 5.2.7 Brazilian Table
 - 5.2.8 ACI Code 318-99
 - 5.2.9 Rectangular and Circular Concrete Cross Section Tables
 - 5.2.10 ASD9-E Code
- 5.3 Analysis Prerelease Features
 - 5.3.1 Calculate Error Estimate Command
 - 5.3.2 Output of Response Spectrum Results
 - 5.3.3 Form Static Load Command
 - 5.3.4 Form UBC97 Load Command
 - 5.3.5 Form IS1893 Load Command
 - 5.3.6 Element Properties Command for Nonlinear Hysteretic Spring Element
 - 5.3.7 Nonlinear Dynamic Analysis
 - 5.3.8 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

- 5.4 General Prerelease Features
 - 5.4.1 Align Command
 - 5.4.2 Rotate Load Command
 - 5.4.3 Run Command
 - 5.4.4 Coutput Command
 - 5.4.5 Reference Coordinate System Command
 - 5.4.6 Hashing Algorithm to Accelerate Input Processing
 - 5.4.7 Add/Delete Without Commands
 - 5.4.8 List Sum Forces Command

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

Note that GTMenu is described in Volume 2 of the Version 28 Release Guide.

Chapter 2

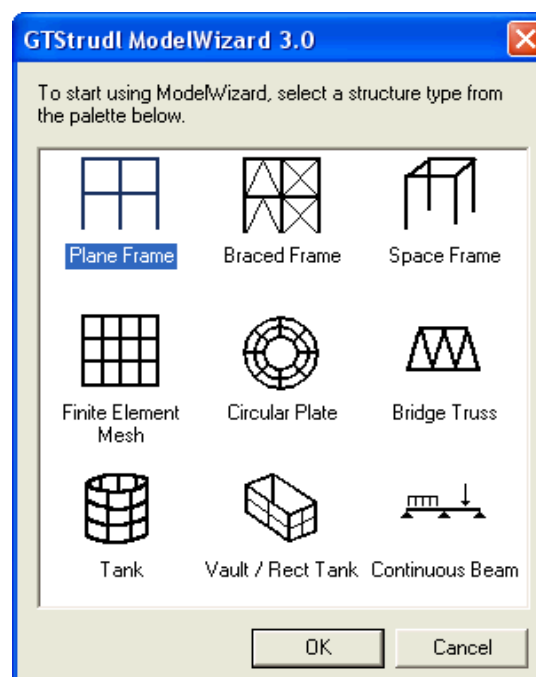
New Features in Version 28.1

This chapter provides you with details regarding new features and enhancements that have been added to many of the functional areas of GTSTRUDL in Version 28.1. This release guide is also available online upon execution of GTSTRUDL under Help/Reference Documentation/GTSTRUDL 28.1 Release Guide.

2.1 GTSTRUDL Startup

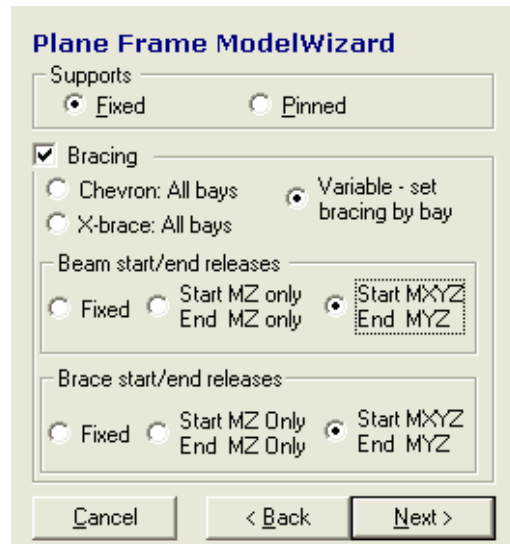
Model Wizard

1. New Braced Frame, Circular Plate and Vault /Rectangular Tank options have been added to the Model Wizard as shown below:



The new Braced Frame Wizard can be used to generate Chevron and X Braced Frames. You may specify which bays contain the bracing as well as specify member releases for the beams and braces. Examples of several of the Braced Frame Wizard dialogs are shown below:

Bracing type and member release dialog:

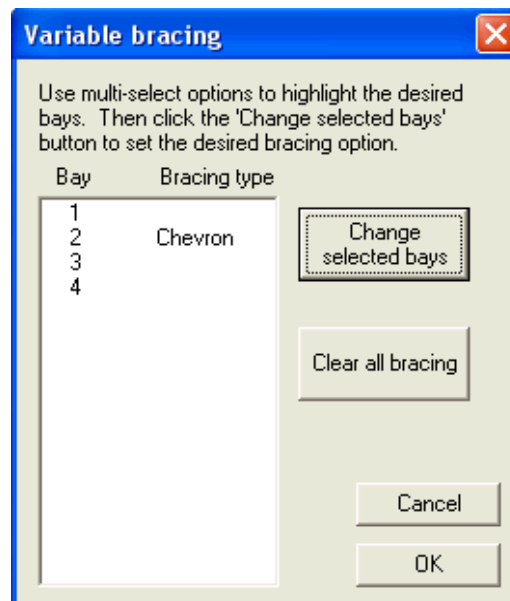


The **Plane Frame ModelWizard** dialog box contains the following settings:

- Supports:** ☒ Fixed, ☐ Pinned
- Bracing:** ☒ Bracing
 - ☐ Chevron: All bays, ☒ Variable - set bracing by bay
 - ☐ X-brace: All bays
- Beam start/end releases:**
 - ☐ Fixed, ☐ Start MZ only, ☒ Start MXYZ
 - ☐ End MZ only, ☐ End MYZ
- Brace start/end releases:**
 - ☐ Fixed, ☐ Start MZ Only, ☒ Start MXYZ
 - ☐ End MZ Only, ☐ End MYZ

Buttons at the bottom: **Cancel**, **< Back**, **Next >**

Variable bracing dialog - specify which bays contain bracing ;



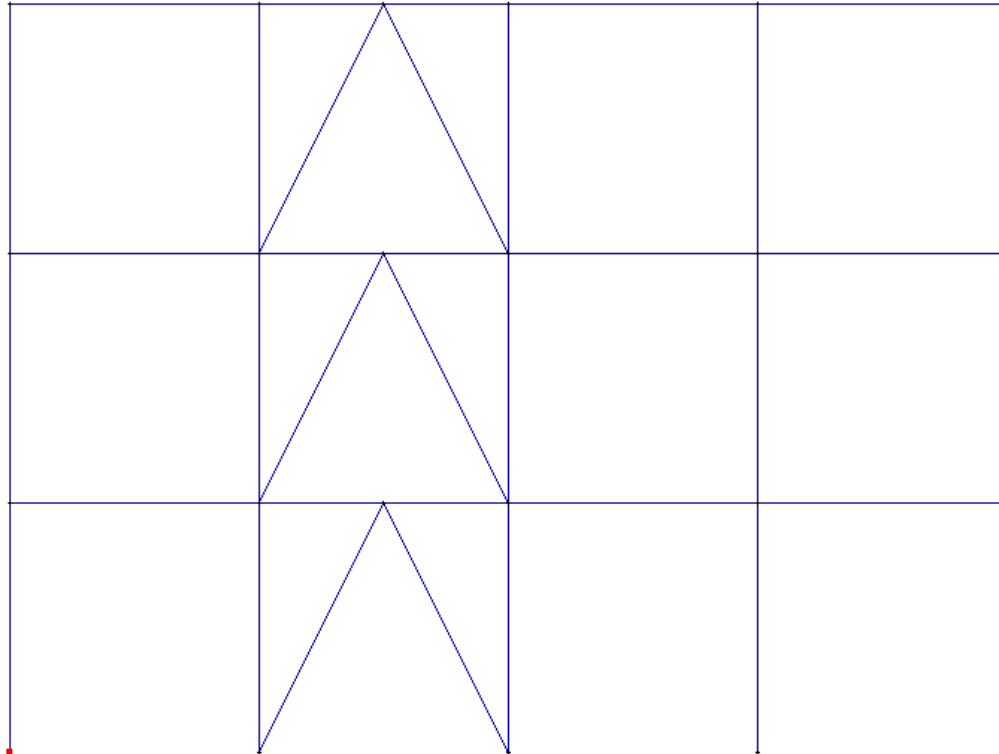
The **Variable bracing** dialog box includes the following elements:

Use multi-select options to highlight the desired bays. Then click the 'Change selected bays' button to set the desired bracing option.

| Bay | Bracing type |
|-----|--------------|
| 1 | Chevron |
| 2 | |
| 3 | |
| 4 | |

Buttons: **Change selected bays**, **Clear all bracing**, **Cancel**, **OK**

Pinned and fixed supports may be specified at the base of the columns as well as dead, live, and lateral wind loads. An example of the frame created using the new Braced Frame Wizard is shown below:



The new Circular Plate Model Wizard allows you to create a finite element mesh of a circular plate including a circular plate with a hole at the center using SBHQ6 and SBHT6 elements. An example of the geometry dialog including the specification of a hole at the center (inside radius) of the plate is shown:

Circular Plate Model Wizard

Now create the tank's basic geometry.

Specify the dimensions of the circular plate,

Outside Radius: 10

☒ Inside Radius: 2

Radial Divisions: 18

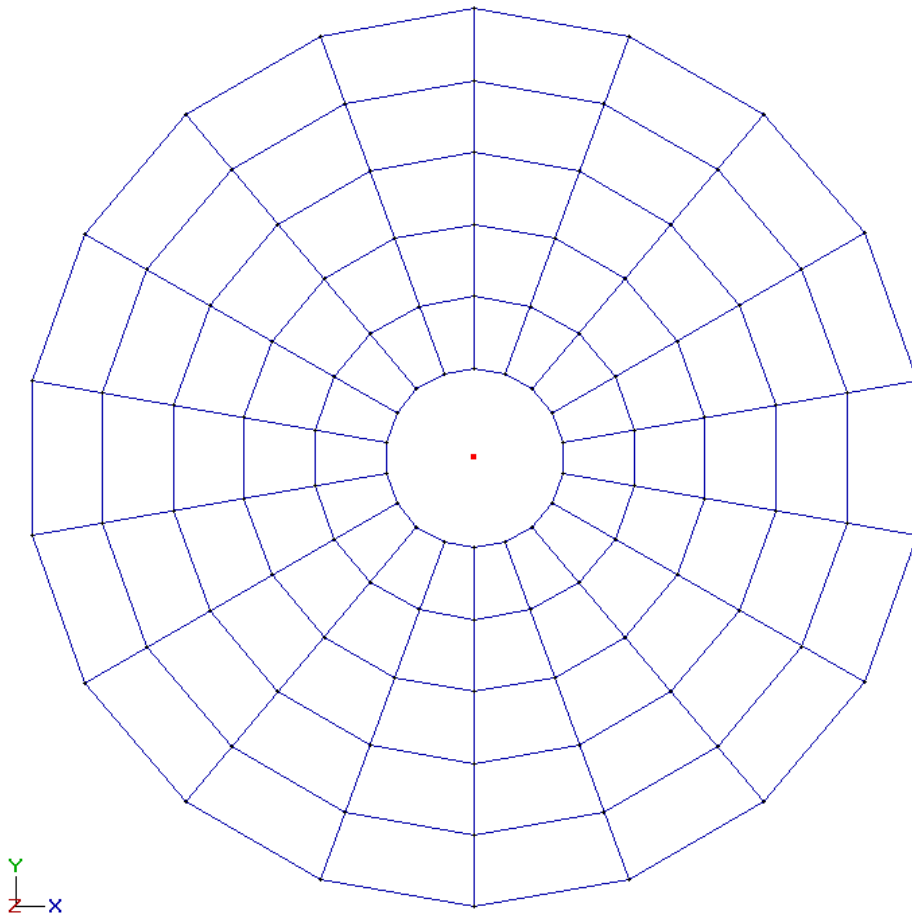
Circumferential Divisions: 5

and the normal axis direction.

☐ X ☐ Y ☒ Z

Cancel < Back Next >

An example of the finite element mesh created using the Circular Plate Wizard is shown below:



You may also specify a variety of support conditions including pinning or fixing the inner and/or outer edges of the circular plate, pinning or fixing all joints on the plate, elastic soil springs, and compression only nonlinear springs normal to the plate with in-plane springs specified as a ratio of the normal springs' stiffness. An example of the circular plate support dialog is shown:

Circular Plate ModelWizard

☐ No supports. I will add supports later.

☐ Supports at plate edges

☒ Outside ☐ Fixed ☒ Inside ☐ Fixed

☐ Pinned ☐ Pinned

☐ Supports at all plate joints

☐ Fixed ☒ Pinned

☒ Elastic Soil Springs for plate joints

Ks of: Kips

per cubic

☒ Pinned supports (release MXYZ)

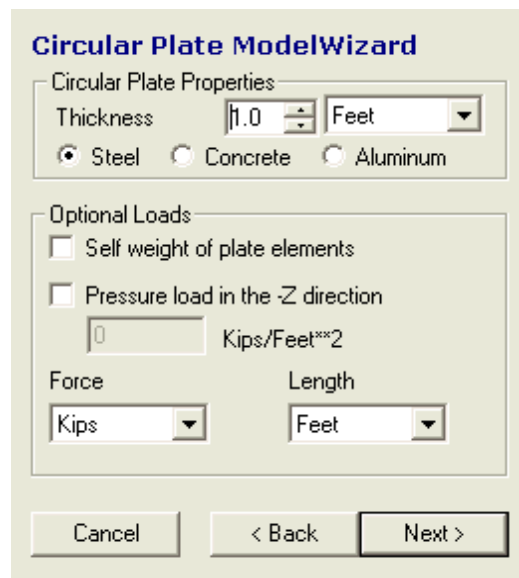
☐ Compression only (unrestrained liftoff)

☐ In-plane springs

In-plane spring ratio

Cancel < Back Next >

Self weight and normal pressure loads may also be specified as shown in the dialog below:



Circular Plate ModelWizard

Circular Plate Properties

Thickness

☒ Steel ☐ Concrete ☐ Aluminum

Optional Loads

☐ Self weight of plate elements

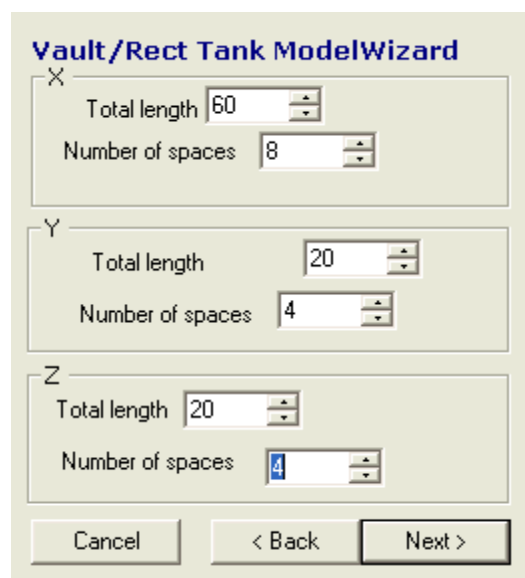
☐ Pressure load in the -Z direction

Kips/Feet**2

Force Length

Cancel < Back Next >

The new Vault / Rectangular Tank Wizard allows you to create a finite element mesh of a rectangular tank including the tank bottom using SBHQ6 elements. The geometry of the tank is created by specifying the dimensions of the tank in the global directions and the number of divisions of finite elements to use in each direction. The global Y direction is assumed to be the vertical direction of the tank. The geometry dialog is shown below:



Vault/Rect Tank ModelWizard

X

Total length

Number of spaces

Y

Total length

Number of spaces

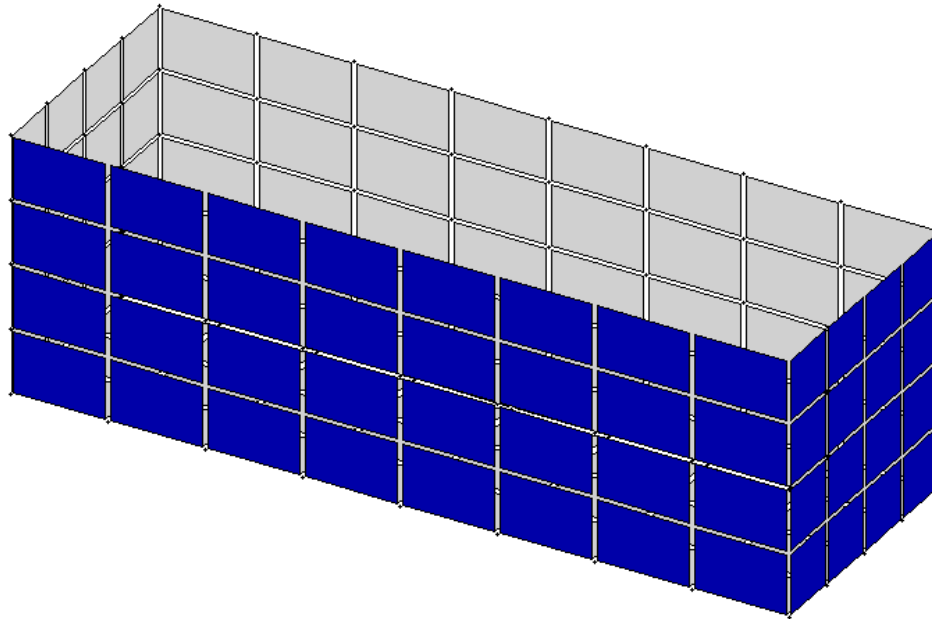
Z

Total length

Number of spaces

Cancel < Back Next >

An example of the finite element mesh created using the Rectangular Tank Wizard is shown below:



The dialogs for specifying the thicknesses for the walls and bottom of the tank and for specifying the supports at the bottom of the tank are shown below:

Vault / Rect Tank ModelWizard

Specify a thickness and material:

Bottom
Bottom thickness: 12 Inches
☐ Steel ☒ Concrete ☐ Aluminum

Walls
Wall thickness: 12 Inches
☐ Steel ☒ Concrete ☐ Aluminum

Cancel < Back Next >

Vault / Rect Tank ModelWizard

Select a support status for the tank bottom

☐ No supports. I will add supports later.

☐ Supports at all bottom joints
☐ Fixed ☒ Pinned

☒ Elastic Soil Springs for bottom joints
Ks of: 10 Kips per cubic Feet

Cancel < Back Next >

Self weight, internal pressure, and hydrostatic loadings may be specified as shown in the dialog below:

Vault / Rect Tank ModelWizard
Internal loading information:

Loading units:
Force: Pounds
Length: Feet

☒ Self weight of tank elements

☐ Internal pressure of:
0 Pounds/Feet²

☒ Hydrostatic load
Gamma = 62.4 Pounds/Feet³
Height = 15 Feet
above the bottom of the tank.

Cancel < Back Next >

2.2 Dynamics

1. A new eigenvalue analysis procedure, designated as GTHCLANCZOS, has been implemented. The GTHCLANCZOS method is a modified form of the GTLANCZOS method, in which the Lanczos tridiagonalization of the stiffness and dynamic matrices is performed on matrix hypercolumn blocks consisting of a command-specified number of matrix elements. By default, the number of hypercolumn matrix elements is taken as 10,000,000. The GTHCLANCZOS method is most useful when an eigenvalue analysis is to be performed on a model having greater than 60,000 degrees of freedom (10,000 six-degree-of-freedom joints), to be followed later by a transient analysis and/or a response spectrum analysis. The GTHCLANCZOS method is specified in the EIGENPROBLEM PARAMETERS commands as shown in the example below:

```
EIGENPROBLEM PARAMETERS
NUMBER OF MODES 15
SOLVE USING GTHCLANCZOS
...
...
END
```

2.3 General

1. AREA LOAD error checking for illegal member configurations has been improved. An illegal member configuration is one where the areas to be loaded are not simply bounded. For example, X bracing should not be included in an AREA LOAD request, but declared INACTIVE before the AREA LOAD command. In addition, more modeling errors, including overlapping members (where some of all of two members' centroidal axes are co-incident), are detected and reported.

2.4 Static Analysis

1. The statistical output from the new GTHCS equation solver has been improved to now output information regarding the number of degrees of freedom, the number of terms in the skyline and the number of hyper-columns.

2.5 Offshore

1. The Efthymiou SCF equations for the computation of stress concentrations have been added to the FATIGUE MEMBER command for offshore fatigue analysis. The FATIGUE MEMBER command now provides for the selection of the Kuang, Smedley, and Efthymiou SCF equations as indicated by the new command syntax shown in its entirety below:

5.3.1 FATIGUE MEMBER Command

General form:

FATIGUE MEMBER (DATA) (position data) (SCF data)

list_m (position data) (SCF data)

C

C

C

list_m (position data) (SCF data)

where

$$\text{position data} = \left\{ \begin{array}{l} \underline{\text{PIPE}} \left\{ \begin{array}{l} \rightarrow \underline{\text{EQUAL}} (V_E) \\ \underline{\text{UNEQUAL}} V_1, \dots, V_i, \dots, V_n \end{array} \right\} \\ (y_1 \ z_1, y_2 \ z_2, \dots, y_n \ z_n) (\underline{\text{YFACTOR}} \ v_y \ \underline{\text{ZFACTOR}} \ v_z) \end{array} \right\}$$

SCF data =

$$\text{SCF} \left\{ \begin{array}{l} \rightarrow (\underline{\text{CONSTANT}}) \text{ (constant specs)} \\ \underline{\text{VARIABLE}} \text{ variable specs} \\ \underline{\text{SIMPLIFIED}} (\underline{\text{FATIGUE}}) (\underline{\text{MINIMUM}} (\underline{\text{SCF}}) \left\{ \begin{array}{l} \rightarrow 1.5 \\ v_{\min} \end{array} \right\}) \end{array} \right\}$$

constant specs =

$$\left\{ \begin{array}{l} (\underline{\text{COMBINED}}) [\underline{\text{START}}] V_{\text{ST}} [\underline{\text{END}}] V_{\text{EN}} \\ \underline{\text{SEPARATE}} * \left\{ \begin{array}{l} [\underline{\text{START}}] [\underline{\text{AXIAL}}] V_{\text{AS}} [\underline{\text{BENDING}}] V_{\text{BS}} \\ [\underline{\text{END}}] [\underline{\text{AXIAL}}] V_{\text{AE}} [\underline{\text{BENDING}}] V_{\text{BE}} \end{array} \right\} \end{array} \right\}$$

variable specs =

$$\left\{ \begin{array}{l} \underline{\text{EQUATIONS}} \left\{ \begin{array}{l} * \left[\begin{array}{l} \underline{\text{KUANG}} \left\{ \begin{array}{l} \rightarrow \underline{\text{K}} \\ \rightarrow \underline{\text{Y}} \\ \underline{\text{T}} \end{array} \right\} \\ * \left[\begin{array}{l} \rightarrow \underline{\text{T}} \\ \rightarrow \underline{\text{X}} \\ \underline{\text{K}} \\ \underline{\text{Y}} \end{array} \right] \end{array} \right\} \\ \underline{\text{SMEDLEY}} \left\{ \begin{array}{l} \rightarrow \underline{\text{T}} \\ \rightarrow \underline{\text{X}} \\ \underline{\text{K}} \\ \underline{\text{Y}} \end{array} \right\} \end{array} \right\} (\underline{\text{MINIMUM}} (\underline{\text{SCF}}) \left\{ \begin{array}{l} \rightarrow 1.5 \\ v_{\min} \end{array} \right\}) \\ \underline{\text{EFTHYMIU}} \\ \underline{\text{USE}} \left\{ \begin{array}{l} \rightarrow \underline{\text{BRACE}} (\underline{\text{SIDE}}) \\ \underline{\text{CHORD}} (\underline{\text{SIDE}}) \end{array} \right\} * \left\{ \begin{array}{l} \text{Start specs} \\ \text{End specs} \end{array} \right\} \end{array} \right\}$$

and where,

$$\text{Start specs} = \left\{ \begin{array}{l} \text{START} \left\{ \begin{array}{l} \text{SADDLE [AXIAL]} \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{ASS} \end{array} \right\} \quad [\text{IPB}] \left\{ \begin{array}{l} \rightarrow 0.0 \\ v_{IPSS} \end{array} \right\} \quad [\text{OPB}] \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{OPSS} \end{array} \right\} \\ \text{CROWN [AXIAL]} \left\{ \begin{array}{l} \rightarrow v_{ASS} \\ v_{ASC} \end{array} \right\} \quad [\text{IPB}] \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{IPSC} \end{array} \right\} \quad [\text{OPB}] \left\{ \begin{array}{l} \rightarrow 0.0 \\ v_{OPSC} \end{array} \right\} \end{array} \right\} \end{array} \right\}$$

$$\text{End specs} = \left\{ \begin{array}{l} \text{END} \left\{ \begin{array}{l} \text{SADDLE [AXIAL]} \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{AES} \end{array} \right\} \quad [\text{IPB}] \left\{ \begin{array}{l} \rightarrow 0.0 \\ v_{IPES} \end{array} \right\} \quad [\text{OPB}] \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{OPES} \end{array} \right\} \\ \text{CROWN [AXIAL]} \left\{ \begin{array}{l} \rightarrow v_{AES} \\ v_{AEC} \end{array} \right\} \quad [\text{IPB}] \left\{ \begin{array}{l} \rightarrow 1.0 \\ v_{IPEC} \end{array} \right\} \quad [\text{OPB}] \left\{ \begin{array}{l} \rightarrow 0.0 \\ v_{OPEC} \end{array} \right\} \end{array} \right\} \end{array} \right\}$$

The EQUATIONS option now specifies that the KUANG and/or SMEDLEY or EFTHYMIU equations can be used to compute SCF's based on specified or computed joint geometry classifications for the brace member. The KUANG equations are used to compute SCF's for brace members of joints classified as K, Y, and T, while the SMEDLEY equations are used to compute SCF's for brace members of joints classified as T, X, K, and Y. The EFTHYMIU equations are used to compute SCF's for brace members of joints having K, T, Y, and X classifications, which cannot be individually selected as with the KUANG and SMEDLEY equations. Note that either KUANG alone, SMEDLEY alone, or both KUANG and SMEDLEY equations may be selected with any combination of applicable joint classifications. If the KUANG and/or SMEDLEY equations are selected, then the EFTHYMIU equations cannot be selected. If the EFTHYMIU equations are selected, then the KUANG and SMEDLEY equations cannot be selected. Figure 5.3-6 shows examples of T, X, K, and Y joint classifications.

2. A new and more efficient command has been implemented for fatigue analysis. The new PERFORM FATIGUE ANALYSIS command can now be used instead of the existing COMPUTE FATIGUE LIFE command. The complete syntax of the PERFORM FATIGUE ANALYSIS command is shown below:

$$\text{PERFORM FATIGUE (ANALYSIS)} \left\{ \begin{array}{l} \text{PSD} \\ \text{DISCRETE} \end{array} \right\} (\text{BASE PERIOD } v_b) -$$

(stress information) (deletions) (REPORT (SCF DIAGNOSTICS))

where

$$\text{stress information} = \left(* \left\{ \begin{array}{l} \underline{\text{MINIMUM (STRESS)}} S_{\text{MIN}} \\ \underline{\text{MAXIMUM (STRESS)}} S_{\text{MAX}} \\ \underline{\text{NUMBER (OF DIVISIONS)}} i_D \end{array} \right\} \right)$$

$$\text{deletions} = \left(\underline{\text{DELETE FATIGUE}} * \left\{ \begin{array}{l} \underline{\text{STRESSES}} \\ \underline{\text{TRANSFER (FUNCTIONS)}} \\ \underline{\text{EXCEEDANCES}} \end{array} \right\} \right)$$

The PERFORM FATIGUE ANALYSIS command executes the fatigue life computations on a joint-by-joint basis, which dramatically improves the efficiency of the fatigue analysis computations and increases the size of the fatigue analysis job that can be solved (number of fatigue wave loads and number of fatigue members). The PERFORM FATIGUE ANALYSIS command performs all fatigue analysis computations including automatic joint classification if requested, computation of fatigue stresses, computation of transfer functions, and computation of fatigue damage and life. The PERFORM FATIGUE ANALYSIS command should not be used in conjunction with the split fatigue analysis commands described in Section 5.6 of Volume 8.

The REPORT SCF DIAGNOSTICS option causes SCF equation diagnostic information and joint classification information to be reported during the fatigue analysis computations. If not given, this report, which can be quite lengthy, is suppressed. All other command options are identical to those of the COMPUTE FATIGUE LIFE command described in Section 5.5.3, Volume 8.

3. For the APILRFD1 code, the reduction for FYLD has been removed as it is not needed for LRFD.

2.6 Steel Design

1. A new parameter called 'Class3' has been added to EC3 code. This parameter allows the user to request that the code check or design to be performed based on the class 3 classification. A user specified value of 'YES' for this parameter indicates that when a code check or design is performed for EC3 code, equations based on the Class 3 classification should be used. When a value of 'YES' has been specified for parameter 'Class3', EC3 code check will assume that the member is a class 3 cross-section. The default value for this

- parameter is 'NO'. This indicates that the program computes the classification of the member based on the cross-section properties.
2. The Summarize command for the critical section prints the summary results for the section that has the highest actual/allowable ratio. When the KL/r actual/allowable ratio is the highest ratio during a code check or design, the Summarize command for the critical section outputs that section's summary results. In general, prismatic sections have the same KL/r ratio for each loading and section. Since the KL/r value is the same for all sections, when the Summarize command is issued and the KL/r is the highest actual/allowable ratio, the summary results for the last loading and last section are printed. In this version of GTSTRUDL, the summarize output for the critical section has been modified to also print the section with the highest stress value. The section which has the highest stress value also has the highest KL/r ratio.

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. The FORM MISSING MASS command now functions as documented and assumes a damping ratio if the word RATIO or PERCENT is omitted after DAMPING. Previously, an error message would be output and a damping ratio of 0.0 would be assumed. (GPRF 2005.03)
2. The INERTIA OF JOINTS FROM LOADS command will no longer abort if member/finite element loads are present in any of the loading conditions specified in the command and any of the members/finite elements have undefined properties. (GPRF 2005.05)

3.2 General

1. The FORM LOAD command will now copy member loads on the IPCABLE element to the new loading condition. Previously, an error message would be output and member loads on the IPCABLE element would not be copied to the new loading condition. (GPRF 2005.06)
2. An abort will no longer occur if a model containing a self weight loading was saved under a version prior to Version 28 and then is subsequently restored in Version 28 and the STIFFNESS ANALYSIS command is specified. (GPRF 2005.07)
3. Users have reported cases where they have encountered the Scan flag being On during an analysis and after specifying SCAN OFF, a subsequent analysis still reported that Scan was On. This problem has been corrected. (No GPRF issued)

3.3 Model Wizard

1. The new Model Wizard discussed in the Version 28 Release Guide was inadvertently omitted from the Version 28 installation. The Model Wizard in Version 28.1 includes the new features in Version 28 plus the additional features discussed in Section 2.1 of this Release Guide. (No GPRF issued)

3.4 Offshore

1. In Version 28, the simplified fatigue analysis for standard fatigue members aborted. The simplified fatigue analysis should actually ignore such members and now does so in Version 28.1. (No GPRF issued)

3.5 Steel Design

1. Automatic K-factor computations now compute the K-factors correctly when the parameter FRLY or FRLZ has been specified. (GPRF 2005.04)

CHAPTER 4

KNOWN DEFICIENCIES

This chapter describes known problems or deficiencies in Version 28.1. These deficiencies have been evaluated and based on our experience, they are seldom encountered or there are workarounds. The following sections describe the known problems or deficiencies by functional area.

4.1 Finite Elements

1. The ELEMENT LOAD command documentation indicates that header information such as type and load specs are allowed. If information is given in the header and an attempt is made to override the header information, a message is output indicating an invalid command or incorrect information is stored. (GPRF 90.06)
2. Incorrect results (displacements, stresses, reactions, frequencies, ... etc.) will result if a RIGIDITY MATRIX is used to specify the material properties for the IPSL, IPSQ, and TRANS3D elements. (GPRF 93.09)
3. The CALCULATE RESULTANT command may either abort or print out an erroneous error message for cuts that appear to be parallel to the Planar Y axis. (GPRF 94.21)
4. If a superelement is given the same name as a member or finite element, an abort will occur in the DEVELOP STATIC PROPERTIES command. (GPRF 95.08)
5. The curved elements, TYPE 'SCURV' and 'PCURV' will produce incorrect results for tangential member loads (FORCE X). An example of the loading command which will produce this problem is shown below:

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

where member (element) 1 is a 'SCURV' or 'PCURV' element.
(GPRF 99.13)

4.2 General Input/Output

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

```
GENERATE 5 MEM ID 1 INC 1 FROM 1 INC 1 TO 2 INC 1  
REPEAT 2 TIMES ID 5 FROM 7 INC 1 TO 8 INC 1
```

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

2. Rigid body elements can not be deleted or inactivated as conventional finite elements. The specification of rigid body elements as conventional finite elements in the INACTIVE command or in DELETIONS mode will cause an abort in a subsequent stiffness, nonlinear, or dynamic analysis. (GPRF 97.21)
3. The path plus file name on a SAVE or RESTORE is limited to 256 characters. If the limitation is exceeded, the path plus file name will be truncated to 256 characters. This is a Windows limitation on the file name including the path. (No GPRF issued)
4. Object groups, created by the DEFINE OBJECT command, may not be used in a GROUP LIST as part of a list. If the OBJECT group is the last group in the list, processing will be correct. However, if individual components follow the OBJECT group, they will fail. Also, you can not copy members or joints from the OBJECT group into a new group.

(GPRF 99.26)

5. Numerical precision problems will occur if joint coordinate values are specified in the JOINT COORDINATES command with more than a total of seven digits. Similar precision problems will occur for joint coordinate data specified in automatic generation commands. (GPRF 2000.16)
6. Internal member results will be incorrect under the following conditions:
 1. Dynamic analysis is performed (response spectra or time history)
 2. Pseudo Static Loadings are created
 3. Buckling Analysis is Performed
 4. Internal member results are output or used in a subsequent steel design after the Buckling Analysis.

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

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

Workaround:

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

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

(GPRF 2004.14)

4.3 GTMenu

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

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

(GPRF 95.18)

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

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

(GPRF 95.21)

4. 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.

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

(No GPRF issued)

5. Projected element loads will be displayed incorrectly when they are created or when they are displayed using Display Model 6 Loads.

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

(No GPRF issued)

4.4 Rigid Bodies

1. Response spectrum analysis may abort if rigid bodies and/or joint ties with slave releases are present in the model. (GPRF 99.18)
2. Static and dynamic analyses will abort if member releases are specified for rigid bodies. (GPRF 2005.02)

4.5 Scope Environment

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

CHAPTER 5

PRERELEASE FEATURES

5.1 Introduction

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

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

The Prerelease features in Version 28.1 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 and Parameters
- 5.2.2 LRFD3 Tables
- 5.2.3 BS5950 Steel Design Code and Parameters
- 5.2.4 Steel Design by Indian Standard Code IS800
- 5.2.5 IS800 Tables
- 5.2.6 Steel Deflection Check and Design
- 5.2.7 Brazilian Table
- 5.2.8 ACI Code 318-99
- 5.2.9 Rectangular and Circular Concrete Cross Section Tables
- 5.2.10 ASD9-E Code

5.3 Analysis Prerelease Features

- 5.3.1 Calculate Error Estimate Command
- 5.3.2 Output of Response Spectrum Results
- 5.3.3 Form Static Load Command
- 5.3.4 Form UBC97 Load Command
- 5.3.5 Form IS1893 Load Command

- 5.3.6 Element Properties Command for Nonlinear Hysteretic Spring Element
- 5.3.7 Nonlinear Dynamic Analysis
- 5.3.8 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis
- 5.4 General Prerelease Features
 - 5.4.1 Align Command
 - 5.4.2 Rotate Load Command
 - 5.4.3 Run Command
 - 5.4.4 Coutput Command
 - 5.4.5 Reference Coordinate System Command
 - 5.4.6 Hashing Algorithm to Accelerate Input Processing
 - 5.4.7 Add/Delete Without Commands
 - 5.4.8 List Sum Forces Command
 - 5.4.9 GTMenu Point Coordinates and Line Incidences Commands

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

5.2 Design Prerelease Features

5.2.1 LRFD3 Steel Design Code and Parameters

LRFD3 Code
American Institute of Steel Construction
Load and Resistance Factor Design
AISC LRFD Third Edition

LRFD3.1 LRFD3 Code

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

| | |
|--|--|
| I shapes | Subjected to bi-axial bending and axial forces |
| Channels | Subjected to bi-axial bending and axial forces |
| Single Angles | Subjected to bi-axial bending and axial forces |
| Double Angles | Subjected to bi-axial bending and axial forces |
| Tees | Subjected to bi-axial bending and axial forces |
| Round HSS (Pipes) | Subjected to bi-axial bending, axial, and torsional forces |
| Rectangular and Square HS (Structural Tubes) | Subjected to bi-axial bending, axial, and torsional forces |
| Plate Girders | Subjected to bi-axial bending and axial forces |

The term I shapes is used to mean rolled or welded I and H beams and columns, universal beams and columns, joists, universal bearing piles, W, S, M, and HP profiles with doubly symmetric cross-sections.

The 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.

Second order elastic analysis using factored loads is required by the GTSTRUDL LRFD3 code. Second order effects may be considered by using GTSTRUDL Nonlinear Analysis (Section 2.5 or Volume 3 of the User Reference Manual). GTSTRUDL LRFD3 code check does not consider the technique discussed in Section C1.2 of AISC, *Manual of*

Steel Construction, Load & Resistance Factor Design, Third Edition, for determination of M_u (B_1 and B_2 factors) in lieu of a second order analysis.

The following assumptions are made throughout the LRFD3 code.

1. Open cross-sections (I shapes, channels, single angles, double angles, tees, and plate girders) are normally not used in situations wherein significant torsional moments must be resisted by the member. Torsional stresses are usually small for open cross-sections when compared to axial and bending stresses, and may be neglected. No checks are made for torsion in open cross-sections (I shapes, channels, single angles, double angles, tees, and plate girders). The designer is reminded to check the torsional stresses for open cross-sections (I shape, channels, single angles, double angles, tees, and plate girders) whenever they become significant.
2. Torsional stresses are checked for round HSS (pipes), rectangular and square HSS (structural tubes) based on the Section 6.1 on Page 16.2-8 of the AISC LRFD Third Edition. Combined torsion, shear, flexure, and/or axial forces are also checked for round HSS (pipes), rectangular and square HSS (structural tubes) based on the Section 7.2 on Page 16.2-10 of the AISC LRFD Third Edition. Closed cross-sections (HSS) are frequently used in situations wherein significant torsional moments must be resisted by the members. Generally the normal and shear stresses due to warping in closed cross-sections (HSS) are insignificant and the total torsional moment can be assumed to be resisted by pure torsional shear stresses (Saint-Venant's torsion).
3. Web stiffeners are considered for web shear stress, but they are not designed.
4. Modified column slenderness for double angle member is considered (Section E4 of the AISC LRFD Third Edition). Modified column slenderness of the double angle member is computed based on the user specified or designed number of the intermediate connectors.
5. Double angles contain an adequate number of intermediate connectors (stitch plates) which make the two angles act as one, Tee-like section.

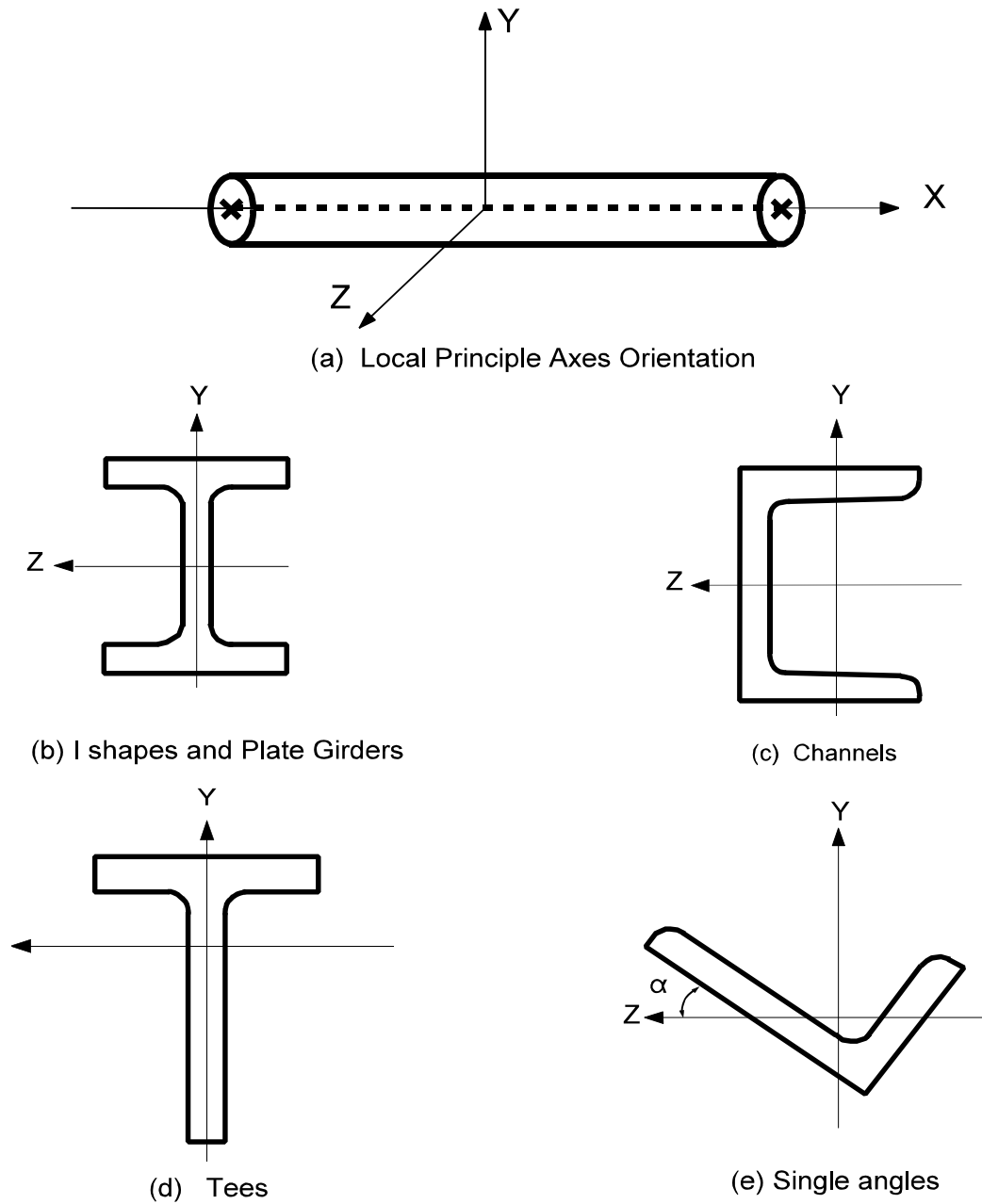
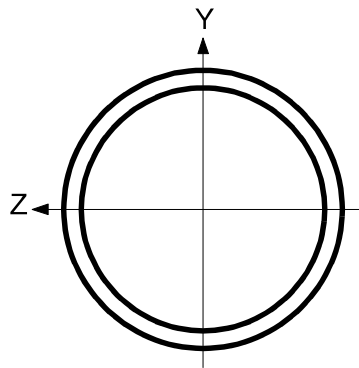
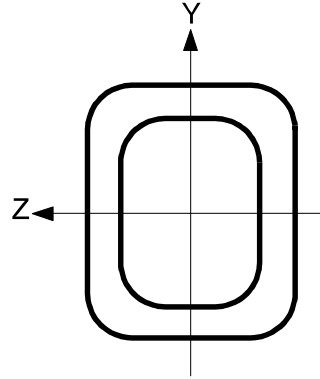


Figure LRFD3.1-1 Local Axes for Design with LRFD3

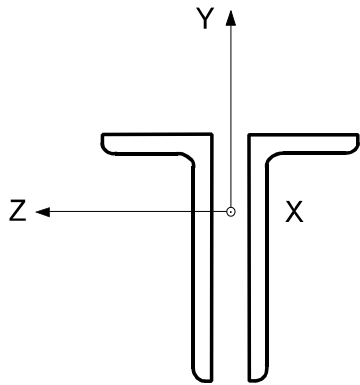


(f) Pipes

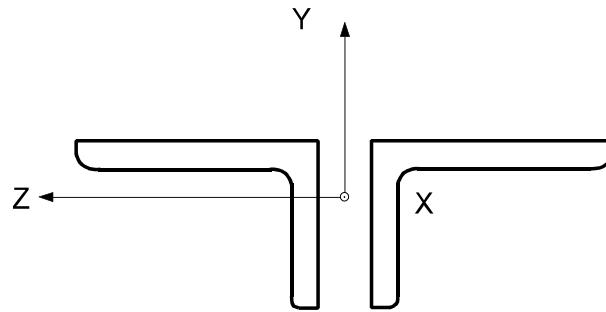


(g) Structural tubing

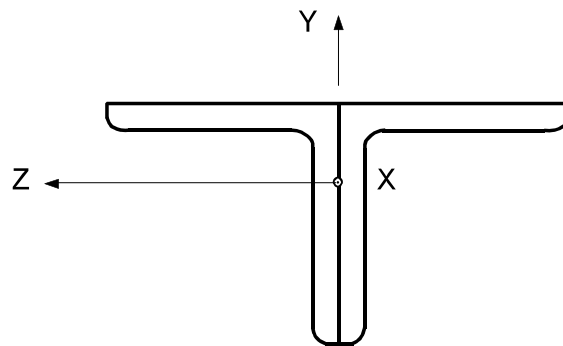
Figure LRFD3.1-1 Local Axes for Design with LRFD3 (continued)



(h) Long legs back-to-back
double angle with spacing



(i) Short legs back-to-back
double angle with spacing



(j) Equal legs back-to-back
double angle in contact

Figure LRFD3.1-1 Local Axes for Design with LRFD3 (Continued)

The sections of the AISC LRFD Third Edition specifications (96) which are considered by the GTSTRUDL LRFD3 code are summarized below:

| <u>Section</u> | <u>Title</u> |
|-----------------------|--|
| Chapter D | Tension members |
| Section B7 | Limiting slenderness ratios |
| Section D1 | Design tensile strength |
| Chapter E | Columns and other compression members |
| Section B7 | Limiting slenderness ratios |
| Table B5.1 | Limiting width to thickness ratio for unstiffened compression elements |
| Table B5.1 | Limiting width to thickness ratio for stiffened compression elements |
| Section E2 | Design compressive strength for flexural buckling |
| Section E3 | Design compressive strength for flexural-torsional buckling |
| Section E4 | Built-up member. |
| Section E4.1 | Design strength. Modified column slenderness |
| Section E4.2 | Detailing requirements |
| Appendix E | Columns and other compression members |
| Appendix E3 | Design compressive strength for flexural-torsional buckling |
| Appendix B | Design requirements |
| Appendix B5.3a | Unstiffened compression elements |
| Appendix B5.3b | Stiffened compression elements |
| Appendix B5.3c | Design properties |
| Section B5.3d | Design strength |
| Chapter F | Beam and other flexural members |
| Section F1.1 | Yielding |
| Section F1.2 | Lateral-Torsional Buckling |
| Section F1.2a | Doubly symmetric shapes and channels with $L_b \neq L_r$ |
| Section F1.2b | Doubly symmetric shapes and channels with $L_b > L_r$ |
| Section F1.2c | Tees and Double angles |
| Appendix F | Beams and other flexural members |
| Appendix F1 | Design for flexure |
| Table A-F1.1 | Nominal strength parameters |

| <u>Section</u> | <u>Title</u> |
|-----------------------|---|
| Appendix F2 | Design for shear |
| Appendix F2.2 | Design shear strength |
| Appendix F2.3 | Transverse stiffeners |
| Appendix G | Plate Girders |
| Appendix G1 | Limitations |
| Appendix G2 | Design flexural strength |
| Appendix G3 | Design shear strength |
| Appendix G4 | Transverse stiffeners |
| Appendix G5 | Flexure-shear |
| Chapter H | Member under combined forces |
| Section H1 | Symmetric members subject to bending and axial force |
| Section H1.1 | Doubly and singly symmetric member in flexure and tension |
| Section H1.2 | Doubly and singly symmetric member in flexure and compression |

Load and Resistance Factor Design Specification for Single-Angle Members

| <u>Section</u> | <u>Title</u> |
|-----------------------|--|
| Section 2 | Tension |
| Section 3 | Shear |
| Section 4 | Compression |
| Section 5 | Flexure |
| Section 5.1 | Flexure Design Strength |
| Section 5.3 | Bending About Principal Axes |
| Section 6 | Combined Forces |
| Section 6.1 | Members in Flexure and Axial Compression |
| Section 6.2 | Members in Flexure and Axial Tension |

Load and Resistance Factor Design Specification for Steel Hollow Structural Sections

| <u>Section</u> | <u>Title</u> |
|----------------|---|
| Table 2.2-1 | Limiting Wall Slenderness for Compression Elements |
| Section 3 | Tension Members |
| Section 3.1 | Design Tensile Strength |
| Section 4 | Column and Other Compression Members |
| Section 4.2 | Design Compressive Strength |
| Section 5 | Beams and Other Flexural Members |
| Section 5.1 | Design Flexural Strength |
| Section 5.2 | Design Shear Strength |
| Section 6 | Torsion Members |
| Section 6.1 | Design Torsional Strength |
| Section 7 | Members Under Combined Forces |
| Section 7.1 | Design for Combined Flexure and Axial Force |
| Section 7.2 | Design for Combined Torsion, Shear, Flexure, and/or Axial Force |

Tensile or compressive axial strengths, bi-axial bending, shear strengths, and combined strengths are considered for all cross-sections except single and double angle (tension or compression axial strengths only). Parameters allowing for the changes which occur in structural steel at high temperatures have been included and may be invoked at the user's discretion.

The detailed explanation of the code parameters, cross-section properties, general nomenclature, and code equations are as follows.

1. Table LRFD3.1-1 Shows the parameters used by LRFD3 code. Table LRFD3.1-1 contains the applicable parameter names, their default values, and a brief description of the parameters.
2. Section LRFD3.2 Describes the cross-section properties used for each shape.
3. Section LRFD3.3 Contains detailed discussion of the parameters used by the LRFD3 code and they are presented in the alphabetic order in this section.
4. Sections LRFD3.4 Describes the subsections in the Section LRFD3.4.

- | | | |
|-----|-------------------|--|
| 5. | Section LRFD3.4.1 | Defines the symbols used in the LRFD3 code provisions. |
| 6. | Section LRFD3.4.2 | Contains detailed discussion of the code provisions and the equations applicable to the I shape cross-sections subjected to axial and bending forces. |
| 7. | Section LRFD3.4.3 | Contains detailed discussion of the code provisions and the equations applicable to the single angle cross-sections subjected to axial force only. |
| 8. | Section LRFD3.4.4 | Contains detailed discussion of the code provisions and the equations applicable to the double angle cross-sections subjected to axial force only. |
| 9. | Section LRFD3.4.5 | Contains detailed discussion of the code provisions and the equations applicable to the round HSS (pipe) cross-sections subjected to axial, bending, and torsional forces. |
| 10. | Section LRFD3.4.6 | Contains detailed discussion of the code provisions and the equations applicable to the rectangular and square HSS (structural tube) cross-sections subjected to axial, bending, and torsional forces. |
| 11. | Section LRFD3.4.7 | Contains detailed discussion of the code provision and the equations applicable to the plate girder cross-sections subjected to axial and bending forces. |
| 12. | Section LRFD3.4.8 | Contains detailed discussion of the code provision and the equations applicable to the channel cross-sections subjected to the axial and bending forces. |
| 13. | Section LRFD3.4.9 | Contains detailed discussion of the code provision and the equations applicable to the tee cross-sections subjected to the axial and bending forces. |

Table LRFD3.1-1
LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|---|
| CODE | Required | <p>Identifies the code to be used for member checking or member selection. Specify LRFD3 for code name. Second order elastic analysis using factored loads is required by the GTSTRUDL LRFD3 code. Second order effect may be considered by using GTSTRUDL Nonlinear Analysis (Section 2.5 of Volume 3 of the User Reference Manual). LRFD3 code is applicable to the following cross-sections:</p> <ol style="list-style-type: none"> 1. I shape, Channel, Tee, Double Angle, and Plate Girder profiles subjected to axial and bending. 2. Round HSS (Pipe), Rectangular and Square HSS (Structural Tube) profiles subjected to axial, bending, and torsional forces. 3. Single Angle profile subjected to axial force only. <p>See Sections LRFD3.2, LRFD3.3, and LRFD3.4 for a more detailed description.</p> |
| TBLNAM | WSHAPES9 | Identifies the table of profiles to be used during selection (SELECT command). See Table LRFD3.1-3 for a list of available table names. |
| CODETOL | 0.0 | Percent variance from 1.0 for compliance with the provisions of a code. The ratio of Actual/Allowable must be less than or equal to $[1.0 + \text{CODETOL}/100]$. |
| PF | 1.0 | Area reduction factor for holesout in members subject to axial tension. |
| a | 10000.0 (inches) | The clear distance between transverse stiffeners. This parameter is used to compute a/h ratio which is used in the computation of the limiting shear strength. The default value indicates that the shear check does not consider transverse stiffeners. |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|--|
| SECTYPE | Computed | Indicates that the cross-section is Rolled or Welded shape. This parameter is used to compute the value of F_r . F_r is the compressive residual stress in flange. ROLLED = rolled shape. Compressive residual stress is equal to 10 ksi. WELDED = welded shape. Compressive residual stress is equal to 16.5 ksi. |

Material Properties

| | | |
|----------|----------|--|
| STEELGRD | A36 | Identifies the grade of steel from which a member is made. See Tables LRFD3.1-4 and LRFD3.1-5 for steel grades and their properties. |
| Fy | Computed | Yield stress of member. Computed from parameter 'STEELGRD' if not given. |
| Fu | Computed | Minimum tensile strength of member. Computed from parameter 'STEELGRD' if not given. |
| Fyf | Fy | Minimum yield stress of the flange. If not specified, it is assumed equal to the parameter 'Fy'. This parameter is used to define a hybrid cross-section. See parameter 'Fyw' also. |
| Fyw | Fy | Minimum yield stress of the web. If not specified, it is assumed equal to the parameter 'Fy'. This parameter is used to define a hybrid plate cross-section. See parameter 'Fyf' also. |
| RedFy | 1.0 | Reduction factor for parameter 'Fy'. This factor times parameter 'Fy' gives the F_y value used by the code. Used to account for property changes at high temperatures. |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|--|--------------------------|--|
| <u>Material Properties</u> (continued) | | |
| RedFu | 1.0 | Reduction factor for parameter 'Fu'. Similar to parameter 'RedFy'. |
| REDE | 1.0 | Reduction factor for E, the modulus of elasticity. Similar to parameter RedFy. |
| <u>Slenderness Ratio</u> | | |
| SLENCOMP | 200 | Maximum permissible slenderness ratio (KL/r) for a member subjected to axial compression. When no value is specified for this parameter, the value of 200 is used for the maximum slenderness ratio. |
| SLENTEN | 300 | Maximum permissible slenderness ratio (L/r) for a member subjected to axial tension. When no value is specified for this parameter, the value of 300 is used for the maximum slenderness ratio. |
| <u>K-Factors</u> | | |
| COMPK* | NO | <p>Parameter to request the computation of the effective length factors KY and KZ (Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual).</p> <p>YES = compute KY and KZ factors.</p> <p>KY = compute KY only.</p> <p>KZ = compute KZ only.</p> <p>NO = use default or specified values for KY and KZ.</p> |

*K-factor Leaning Columns Concept has not been implemented for the automatic K-factor Computation.

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|------------------------------|--------------------------|--|
| <u>K-Factors</u> (continued) | | |
| KY | 1.0 | Effective length factor for buckling about the local Y axis of the profile. See Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual for GTSTRUDL computation of effective length factor, KY. |
| KZ | 1.0 | Effective length factor for buckling about the local Z axis of the profile. See Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual for GTSTRUDL computation of effective length factor, KZ. |
| Print-K | YES | Parameter to print the computed K-factor values after the default code check or select command output (TRACE 4 output). The default value of 'YES' for this parameter indicates that the computed K-factor values should be printed after the code check or select command output. The column names attached to the start and end of the code checked member is also printed. This printed information allows the user to inspect the automatic detection of the columns attached to the start and end of the designed member. A value of 'NO' indicates that K-factor values and the names of the attached columns to the start and end of the designed member should not be printed. |
| SDSWAYY | YES | Indicates the presence or absence of sidesway about the local Y axis. YES = sidesway permitted. NO = sidesway prevented. |

K-factor Leaning Columns Concept has not been implemented for the automatic K-factor Computation.

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|------------------------------|--------------------------|---|
| <u>K-Factors</u> (continued) | | |
| SDSWAYZ | YES | Indicates the presence or absence of sidesway about the local Z axis. YES=sidesway permitted. NO=sidesway prevented. |
| GAY | Computed | G-factor at the start joint of the member. GAY is used in the calculation of effective length factor KY (see parameters COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual). |
| GAZ | Computed | G-factor at the start joint of the member. GAZ is used in the calculation of effective length factor KZ (see parameters COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual). |
| GBY | Computed | G-factor at the end joint of the member. GBY is used in the calculation of effective length factor KY (see parameters COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual). |
| GBZ | Computed | G-factor at the end joint of the member. GBZ is used in the calculation of effective length factor KZ (see parameters COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A of the User Reference Manual). |

Buckling Length

| | | |
|----|----------|--|
| LY | Computed | Unbraced length for buckling about the local Y axis of the profile. The default is computed as the length of the member. |
|----|----------|--|

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|------------------------------------|--------------------------|--|
| <u>Buckling Length</u> (continued) | | |
| LZ | Computed | Unbraced length for buckling about the local Z axis of the profile. The default is computed as the length of the member. |
| FRLY | 1.0 | Fractional form of the parameter LY, allows unbraced length to be specified as fractions of the total length. Used only when LY is computed. |
| FRLZ | 1.0 | Fractional form of the parameter LZ, similar to FRLY. Used only when LZ is computed. |
| <u>Flexural-Torsional Buckling</u> | | |
| KX | 1.0 | Effective length factor for torsional buckling about the local X axis of the profile. This parameter is used in flexural-torsional buckling stress, F_e computations. |
| LX | Computed | Unbraced length for torsional buckling about the local X axis of the profile. The default is computed as the length of the member. This parameter is used in flexural-torsional buckling stress, F_e computations. |
| FRLX | 1.0 | Fractional form of the parameter LX, allows unbraced length to be specified as fractions of the total length. Used only when LX is computed. |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|--|
| <u>Bending Strength</u> | | |
| CB | Computed | Coefficient used in computing allowable compressive bending strength (AISC LRFD Third Edition, Section F1.2a, Equation F1-3). |
| UNLCF | Computed | Unbraced length of the compression flange. The default is computed as the length of the member. In this parameter no distinction is made between the unbraced length for the top or bottom flange. See UNLCFTF or UNLCFBF. |
| FRUNLCF | 1.0 | Fractional form of the parameter UNLCF, allows unbraced length to be specified as fractions of the total length. Used only when UNLCF is computed. |
| UNLCFTF | Computed | Unbraced length of the compression flange for the top flange. When no value is specified, UNLCF and FRUNLCF are used for this parameter. |
| UNLCFBF | Computed | Unbraced length of the compression flange for the bottom flange. When no value is specified, UNLCF and FRUNLCF are used for this parameter. |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter | Default | |
|--------------------------------|----------------|--|
| <u>Name</u> | <u>Value</u> | <u>Meaning</u> |
| <u>Channel Parameter</u> | | |
| Tipping | YES | This is the parameter indicating that the tipping effect should be considered. When the load is applied to the top flange of the channel and the flange is not braced, there is a tipping effect that reduces the critical moment. A value of YES for this parameter indicates that the flange is unbraced and the flange is loaded as such that causes tipping effect. In this case the reduced critical moment may be conservatively approximated by setting the warping buckling factor X_2 equal to zero. A value of NO indicates that the tipping effect does not happen and the warping buckling factor is computed based on the Equation F1-9 of the AISC LRFD Third Edition. |
| <u>Single Angle Parameter</u> | | |
| Cby | Computed | Coefficient used in computing elastic lateral-torsional buckling moment, M_{ob} , (AISC LRFD Third Edition, Section 5.3 on the page 16.3-6) for major axis bending (bending about the Y axis). |
| <u>Tee Parameter</u> | | |
| SFYBend | 1.0 | Parameter to specify safety factor for the computation of the limit state of Y axis (minor axis) bending of the tee section. |
| <u>Double Angle Parameters</u> | | |
| nConnect | 0 | Number of connectors between individual angles. The user specified value is used during code check. When the SELECT MEMBER (design) is requested, the user specified value is used unless more connectors are required. If the |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter | Default | |
|--|----------------|---|
| <u>Name</u> | <u>Value</u> | <u>Meaning</u> |
| <u>Double Angle Parameters</u> (continued) | | |
| | | designed number of connectors are larger than the user specified value, the computed number of connectors are used and printed after the SELECT MEMBER result. The default value of zero indicates that the angles are connected at the ends only. Following are additional options that you can specify for this parameter: |
| | 0 | = angles are connected at the ends of the member. |
| | -1 | = requesting the number of connectors to be computed during code check. |
| | -2 | = bypass modified column slenderness equations. This will bypass the check for the Section E4.1 of the AISC LRFD Third Edition. |
| ConnType | WELDED | Type of the intermediate connectors that are used for double angle. Choices are: SNUG and WELDED. |
| | SNUG | = intermediate connectors that are snug-tight bolted. |
| | WELDED | = intermediate connectors that are welded or fully tensioned bolted. This is the default. |
| L | Computed | Actual physical member length is used to design a number of connectors and to check connector spacing (Section E4.2 of the AISC LRFD), and also used in the computation of the modified column slenderness, $(KL/r)_m$ (Section E4.1 of the AISC LRFD). This parameter is used to compute the distance between connectors $a = L(n+1)$, where 'a' is the distance between connectors, 'L' is the physical member length, and 'n' is the number of connectors. The default is computed as the length of the member. |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter | Default | |
|--|----------------|---|
| <u>Name</u> | <u>Value</u> | <u>Meaning</u> |
| <u>Double Angle Parameters</u> (continued) | | |
| K | 1.0 | Effective length factor for an individual component (single angle). This parameter is used to design a number of connectors and to check the connector spacing (Section E4.2 of the AISC LRFD). |
| SFYBend | 1.0 | Parameter to specify safety factor for the computation of the limit state of Y axis (minor axis) bending of the double angle section. |

Round HSS (Pipes) Shear Check Parameters

| | | |
|-----|----------|--|
| avy | Computed | The length of essentially constant shear in the Y axis direction of a member. This parameter is used to check the Y direction shear of a round HSS (pipe) cross-section (96). This parameter is similar to the variable 'a' in the Equation 5.2-2 of the AISC LRFD HSS specification in the Section 16.2 of the LRFD Third Edition. The default is computed as the length of the member. |
| avz | Computed | The length of essentially constant shear in the Z axis direction of a member. This parameter is used to check the Z direction shear of a round HSS (pipe) cross-section (96). This parameter is similar to the variable 'a' in the Equation 5.2-2 of the AISC LRFD HSS specification in the Section 16.2 of the LRFD Third Edition. The default is computed as the length of the member. |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter Name | Default Value | Meaning |
|--|--------------------------|---|
| <u>Round HSS (Pipes) Torsion Check Parameter</u> | | |
| LX | Computed | This parameter is to specify the distance between torsional restraints. LX is used in the equation 6.1-2 on Page 16.2-8 of AISC LRFD Third Edition (96). This parameter is similar to the variable 'a' in the Equation 5.2-2 of the AISC LRFD HSS specification in the Section 16.2 of the LRFD Third Edition. The default is computed as the length of the member. |

Rectangular Hollow Structural Section (HSS) Parameters

| | | |
|---------|----------|--|
| Cby | Computed | Coefficient used in computing limiting compressive bending strength (AISC LRFD Third Edition, Section F1.2a, Equation F1-3) for minor axis bending (bending about the Y-axis). |
| UNLCW | Computed | Unbraced length of the compression web about the local Y axis of the profile. The default is taken as length of member. |
| FRUNLCW | Computed | Fractional form of the parameter UNLCW allows unbraced length to be specified as a fraction of the total length. Used only when UNLCW is computed. |

Plate Girder Parameters

| | | |
|------|-----|--|
| Fyst | Fy | Minimum yield stress of the transverse stiffeners material. If not specified, it is assumed equal to the parameter Fy. |
| Ast | 0.0 | Parameter to specify the transverse stiffeners area. This parameter is used to check Appendix G4 of AISC LRFD 3 rd Edition. The specified transverse stiffeners area is checked to see if it is smaller than the computed value from Equation A-G4-1 of Appendix G4 of AISC LRFD 3 rd Edition. The |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter | Default |
|------------------|----------------|
| <u>Name</u> | <u>Value</u> |

| <u>Meaning</u> |
|----------------|
|----------------|

Plate Girder Parameters (continued)

default value of 0.0 indicates that the transverse stiffeners area of Appendix G4 is not checked.

| | | |
|---------|-----|--|
| Ist | 0.0 | Parameter to specify the transverse stiffeners moment of inertia. This parameter is used to check Appendix F2.3 of AISC LRFD 3 rd Edition for the required transverse stiffeners moment of inertia. The default value of 0.0 indicates that the transverse stiffeners moment of inertia according to Appendix F2.3 is not checked. |
| Dstiff | 2.4 | Parameter to specify the factor D that is used in the Equation A-G4-1 of Appendix G4 of AISC LRFD 3 rd Edition. A default value of 2.4 for single plate stiffeners is assumed. The value of factor D (parameter 'Dstiff') in the Equation A-G4-1 is dependent on the type of transverse stiffeners used in a plate girder. Alternate values are as follows: 1.0 = for stiffeners in pairs. This is the default value when the specified value for the parameter 'NumBars' is greater than 1. 1.8 = for single angle stiffeners. 2.4 = for single plate stiffeners. This is the default value when the specified value for the parameter 'NumBars' is equal to 1. |
| NumBars | 1.0 | Parameter to specify a number of single plate stiffeners. The default value for this parameter indicates 1 single plate stiffener. |
| Stiff-H | 0.0 | Parameter to specify the single plate stiffeners cross-section's height. Parameters 'Stiff-H', 'Stiff-W', and 'NumBars' are used for the automatic computation of the |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter | Default | |
|------------------|----------------|----------------|
| <u>Name</u> | <u>Value</u> | <u>Meaning</u> |

Plate Girder Parameters (continued)

parameters ‘Ast’ and ‘Ist’. The automatic computation of the parameters ‘Ast’ and ‘Ist’ is based on the rectangular bar stiffeners geometry. If transverse stiffeners are not rectangular bar, parameters ‘Ast’ and ‘Ist’ should be specified.

| | | |
|---------|-----|---|
| Stiff-W | 0.0 | Parameter to specify the single plate stiffeners cross-section’s width. See parameter ‘Stiff-H’ for more information. |
|---------|-----|---|

Force Limitation

| | | |
|-------|--------------|---|
| FXMIN | 0.5 (lb) | Minimum axial force to be considered by the code; anything less in magnitude is taken as zero. |
| FYMIN | 0.5 (lb) | Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero. |
| FZMIN | 0.5 (lb) | Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero. |
| MXMIN | 20.0 (in-lb) | Minimum torsional moment to be considered by the code; anything less in magnitude is taken as zero. |
| MYMIN | 20.0 (in-lb) | Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero. |
| MZMIN | 20.0 (in-lb) | Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero. |

Table LRFD3 (continued)

LRFD3 Code Parameters

| Parameter | Default | |
|--|----------------|--|
| <u>Name</u> | <u>Value</u> | <u>Meaning</u> |
| <u>Output Processing and System Parameters</u> | | |
| SUMMARY | NO | Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; see Sections 2.9 and 7.2 of Volume 2A of the User Reference Manual for an explanation. |
| PrintLim | NO | Parameter to request to print the section limiting values for limit state and load and resistance factor codes. The default output from CHECK or SELECT command prints the section force values. A value of "YES" for this parameter indicates that the section limiting values should be printed instead of default section forces. |
| TRACE | 4.0 | Flag indication when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A of the User Reference Manual for an explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces. |
| VALUES | 1.0 | Flag indicating if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A of the User Reference Manual for the explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties. |

Table LRFD3.1-2
GTSTRUDL AISC Codes*

| Code Name | Parameter Table | Application |
|----------------------|----------------------------|---|
| LRFD3 | LRFD3.1-1 Volume 2C | Checks compliance of I shapes, channels, tees, and double angles (subject to bi-axial bending and axial force), round HSS (Pipes), rectangular and square HSS (structural tubes) (subjected to bi-axial bending, axial, and torsional forces), single and double angles (subjected to axial forces only) shape profiles to the 1999 AISC LRFD, Third Edition, Specification (96). |
| LRFD2 | LRFD2 Volume 2A | Checks compliance of I shapes, pipes, structural tubing, plate girders (subjected to bi-axial bending and axial force), Single and Double Angles (subjected to axial forces only) shape profiles to the 1993 AISC LRFD, Second Edition, Specification (81). |
| ASD9 | ASD9 Volume 2A | Checks compliance of I shapes, Single angles, Channels, Tees, Double angles, Solid round bars, Pipes, Solid Squares and Rectangular bars, and Structural tubing shape profiles to the 1989 AISC ASD, Ninth Edition, Specification (72). |
| 78AISC | 2.2.3.1 Volume 2B | Checks compliance of I shapes, Single angles, Channels, Tees, Solid round bars, Pipes, Solid Squares and Rectangular bars, and Structural tubing (use code name DBLANG for Double angle) shape profiles to the 1978 AISC Specification (33), Eighth Edition, including 1980 updates. |
| 69AISC | 2.2.3.1 Volume 2B | Checks compliance of I shapes, Single angles, Channels, Tees, Solid round bars, Pipes, Solid Squares and Rectangular bars, and Structural tubing (use code name DBLANG for Double angle) shape profiles to the 1969 AISC Specification (16), Seventh Edition, including supplements 1, 2, and 3. |

* For latest (up to date) version of this table, see Table 2.1-1a of Volume 2A.

Table LRFD3.1-2 (continued)
GTSTRUDL AISC Codes*

| <u>Code Name</u> | <u>Parameter Table</u> | <u>Application</u> |
|-------------------------|-------------------------------|--|
| W78AISC | 2.2.3.1 Volume 2B | Similar to 78AISC code, except limited to checking I shape profiles. This code is identical to the 78AISC code which was available in older versions of GTSTRUDL (i.e., version V1M7 and older). |
| DBLANG | 2.2.3.1 Volume 2B | Checks compliance of Double angle profiles to the 1969 AISC Specification (16), Seventh Edition, including supplements 1, 2, and 3. |
| W69AISC | 2.2.3.1 Volume 2B | Similar to 69AISC code, except limited to checking I shape profiles. This code is identical to the 69AISC code which was available in older versions of GTSTRUDL (i.e., version V1M7 and older). |

* For latest (up to date) version of this table, see Table 2.1-1a of Volume 2A.

This page intentionally left blank.

5.2.2 GTSTRUDL LRFD3 Profile Tables

Table LRFD3.1-3

**GTSTRUDL Profile Tables for the
Design based on the LRFD3 Code**

| <u>Profile Shapes</u> | <u>Reference</u> |
|------------------------------|---|
| I shapes | See Table D-1 in Appendix D for list of applicable table names for I shapes, W, S, M, HP shapes, wide flange shapes, universal beam shapes, universal column shapes, etc. |
| Single Angles | See Table D-2 in Appendix D for list of single angle table names applicable to LRFD3 code. |
| Double Angles | See Table D-3 in Appendix D for list of double angle table names applicable to LRFD3 code. |
| Round HSS | See Table D-4 in Appendix D for list of round HSS (pipe, circular hollow section) table names applicable to LRFD3 code. |
| Rectangular HSS | See Table D-5 in Appendix D for list of rectangular and square HSS (structural tube, rectangular and square hollow section) table names applicable to LRFD3 code. |
| Channels | See Table D-6 in Appendix D for a list of channel table names applicable to LRFD3 codes. |
| Tees | See Table D-7 in Appendix D for a list of tee table names applicable to LRFD3 codes. |
| Plate Girders | See Table D-9 in Appendix D for a list of plate girder table names applicable to LRFD3 codes. |

Table LRFD3.1-4

**ASTM Steel Grades and Associated Values of F_y and F_u Based on the
1999 AISC LRFD Third Edition Specifications
Applicable Shapes: W, M, S, HP, L, 2L, C, MC, WT, MT, and ST
shapes from AISC Tables**

| Steel Grade ASTM Designation | Group Number Per ASTM A6 F_y , Minimum Yield Stress (ksi) F_u , Minimum Tensile Strength (ksi) | | | | |
|------------------------------------|--|----------|----------|----------|----------|
| | Group 1 | Group 2 | Group 3 | Group 4 | Group 5 |
| A36 | 36 58 | 36 58 | 36 58 | 36 58 | 36 58 |
| A529-G50 | 50 65 | 50 65 | NA | NA | NA |
| A529-G55 | 55 70 | 55 70 | NA | NA | NA |
| A572-G42 | 42 60 | 42 60 | 42 60 | 42 60 | 42 60 |
| A572-G50 | 50 65 | 50 65 | 50 65 | 50 65 | 50 65 |
| A572-G55 | 55 70 | 55 70 | 55 70 | 55 70 | 55 70 |
| A572-G60 | 60 75 | 60 75 | 60 75 | NA | NA |
| A572-G65 | 65 80 | 65 80 | 65 80 | NA | NA |
| A913-G50 | 50 60 | 50 60 | 50 60 | 50 60 | 50 60 |
| A913-G60 | 60 75 | 60 75 | 60 75 | 60 75 | 60 75 |
| A913-G65 | 65 80 | 65 80 | 65 80 | 65 80 | 65 80 |
| A913-G70 | 70 90 | 70 90 | 70 90 | 70 90 | 70 90 |

Table LRFD3.1-4 (continued)

ASTM Steel Grades and Associated Values of F_y and F_u Based on the
1999 AISC LRFD Third Edition Specifications

Applicable Shapes: W, M, S, HP, L, 2L, C, MC, WT, MT, and ST shapes from AISC Tables

| Steel Grade ASTM Designation | Group Number Per ASTM A6 F_y , Minimum Yield Stress (ksi) F_u , Minimum Tensile Strength (ksi) | | | | |
|------------------------------------|--|---------|-----------------|-----------------|-----------------|
| | Group 1 | Group 2 | Group 3 | Group 4 | Group 5 |
| A992 ^a | 50 | 50 | 50 | 50 | 50 |
| | 65 | 65 | 65 | 65 | 65 |
| A242 | 50 | 50 | 46 ^b | 42 ^a | 42 ^a |
| | 70 | 70 | 67 ^b | 63 ^a | 63 ^a |
| A588 | 50 | 50 | 50 | 50 | 50 |
| | 70 | 70 | 70 | 70 | 70 |

a Applicable to W shapes only.

b Applicable to W and HP shapes only.

NA Indicates that shapes in the corresponding group are not produced for that grade of steel. GTSTRUDL assumes F_y and F_u to be zero in such cases and will not select profiles for these combinations of group number and steel grade. Minimum yield stresses (F_y) and minimum tensile strengths (F_u) were obtained from the summary of ASTM specifications included in the 1999 AISC LRFD Third Edition specification.

Table LRFD3.1-5

**ASTM Steel Grades and Associated Values of F_y and F_u Based on the
1999 AISC LRFD Third Edition Specifications
Applicable Shapes: Round HSS, Steel Pipe, and Rectangular HSS**

| Steel Grade ASTM Designation | Applicable Shape Series F_y , Minimum Yield Stress (ksi) F_u , Minimum Tensile Strength (ksi) | | |
|--|---|------------|-----------------|
| | Round HSS | Steel Pipe | Rectangular HSS |
| A53-GB | NA | 35 60 | NA |
| A500-GB | 42 58 | NA | 46 58 |
| A500-GC | 46 62 | NA | 50 62 |
| A501 | 36 58 | NA | 36 58 |
| A618-GI A618-GII Thickness # 3/4 | 50 70 | NA | 50 70 |
| A618-GI A618-GII Thickness > 3/4 | 46 67 | NA | 46 67 |
| A618GIII | 50 65 | NA | 50 65 |
| A242-G46 | NA | NA | 46 67 |
| A242-G50 | NA | NA | 50 70 |
| A588 | NA | NA | 50 70 |
| A847 | 50 70 | NA | 50 70 |

NA Not applicable. See Table LRFD3.1-4 for more explanation.

5.2.3 GTSTRUDL BS5950 Steel Design Code and Parameters

00BS5950 Code British Standard BS 5950-1:2000

00BS5950.1 00BS5950 Code

The 00BS5950 code of GTSTRUDL may be used to select or check any of the following shapes:

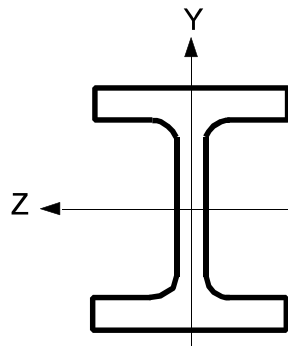
| | |
|----------------------------------|--------------------------------------|
| I shapes | Subjected to bending and axial force |
| Single Angles | Subjected to axial force only |
| Circular Hollow Sections (Pipes) | Subject to bending and axial force |

The term I shapes is used to mean rolled or welded I and H beams and columns, universal beams and columns, joists, universal bearing piles, W, S, M, and HP profiles with doubly symmetric cross-sections.

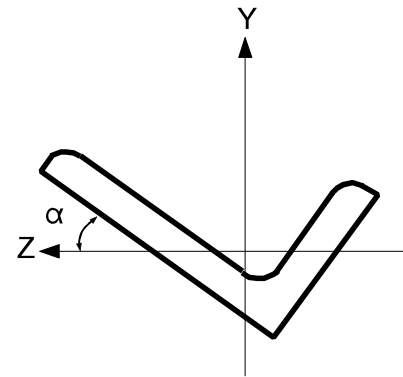
The code is primarily based on the BS 5950-1:2000 "British Standard Structural use of steelwork in building, Part 1: Code of practice for design rolled and welded sections" amendment number 13199, issued May 2001. The 00BS5950 code utilizes the limit state design techniques of the BSI (British Standard Institution) specification.

The following assumptions are made throughout the 00BS5950 code.

1. Torsional stresses are usually small when compared to axial and bending stresses, and may be neglected. No checks are made for torsion. The designer is reminded to check the torsional stresses whenever they become significant.
2. Web stiffeners are considered for web shear stress, but they are not designed.



(a) I shapes



(b) Single angles

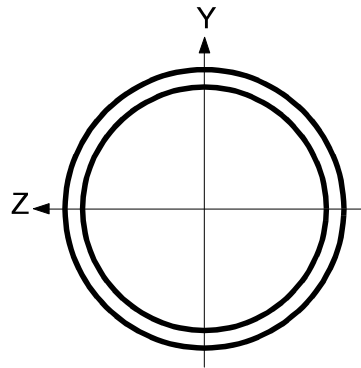
(c) Circular Hollow Sections
(Pipes)

Figure 00BS5950.1-1 Local Axes for Design with 00BS5950

The sections of the BS 5950-1:2000 specifications (95) which are considered by the GTSTRUDL 00BS5950 code are summarized below:

| <u>Section</u> | <u>Title</u> |
|----------------|--|
| 3. | Properties of materials and section properties |
| 3.5 | Classification of cross-sections |
| 3.5.1 | General |
| 3.5.2 | Classification |
| 3.5.3 | Flanges of compound I- or H-sections |
| | Table 11. Limiting width-to-thickness ratios for sections other than CHS and RHS |
| 3.5.5 | Stress ratios for classification |
| 3.5.6.2 | I- or H-sections with equal flanges |
| 3.5.6.4 | Circular hollow sections |
| 3.6.2.2 | Effective area |
| 3.6.2.3 | Effective modulus when web is fully effective |
| 3.6.4 | Equal-leg angle sections |
| 3.6.5 | Alternative method |
| 3.6.6 | Circular hollow sections |
| 4. | Design of structural members |
| 4.2.3 | Shear capacity |
| 4.2.5 | Moment capacity |
| 4.2.5.2 | Low shear |
| 4.2.5.3 | High shear |
| 4.3 | Lateral-torsional buckling |
| 4.3.4 | Destabilizing load. |
| 4.3.5 | Effective length for lateral-torsional buckling |
| | Table 13. Effective length L_E for beams without intermediate restraint |
| 4.3.6.2 | I-, H-, channel and box sections with equal flanges |
| 4.3.6.4 | Buckling resistance moment |
| 4.3.6.5 | Bending strength p_b |

| <u>Section</u> | <u>Title</u> |
|----------------|---|
| 4.3.6.6 | Equivalent uniform moment factor m_{LT} Table 18. Equivalent uniform moment factor m_{LT} for lateral-torsional buckling |
| 4.3.6.9 | Ratio ϕ_w |
| 4.4.5 | Shear buckling resistance |
| 4.4.5.2 | Simplified method |
| 4.4.5.3 | More exact method |
| 4.6 | Tension members |
| 4.6.1 | Tension capacity |
| 4.7.2 | Slenderness |
| 4.7 | Compression members |
| 4.7.2 | Slenderness |
| 4.7.4 | Compression resistance |
| 4.7.5 | Compressive strength Table 23. Allocation of strut curve |
| 4.8 | Members with combined moment and axial force |
| 4.8.2 | Tension members with moments |
| 4.8.2.2 | Simplified method |
| 4.8.2.3 | More exact method |
| 4.8.3 | Compression members with moments |
| 4.8.3.2 | Cross-section capacity |
| 4.8.3.3 | Member buckling resistance |
| 4.8.3.3.1 | Simplified method Table 26. Equivalent uniform moment factor m for flexural buckling |
| 4.8.3.3.2 | More exact method for I- or H-sections with equal flanges Table 26. Equivalent uniform moment factor m for flexural buckling |

| <u>Section</u> | <u>Title</u> |
|----------------|---|
| 4.8.3.3.3 | More exact method for CHS, RHS or box sections with equal flanges Table 26. Equivalent uniform moment factor m for flexural buckling |
| 4.9 | Members with biaxial moments |
| Annex B | (normative) Lateral-torsional buckling of members subject to bending |
| B.2 | Buckling resistance |
| B.2.1 | Bending strength. |
| B.2.2 | Perry factor and Robertson constant |
| B.2.3 | Uniform I, H and channel sections with equal flanges |
| Annex C | (normative) Compressive strength |
| C.1 | Strut formula |
| C.2 | Perry factor and Robertson constant |
| Annex H | (normative) Web buckling resistance |
| H.1 | Shear buckling strength |
| Annex I | (normative) Combined axial compression and bending |
| I.2 | Reduced plastic moment capacity |
| I.2.1 | I- or H-section with equal flanges |

Tensile or compressive axial stresses, bi-axial bending, shear stresses, and combined stresses are considered for all cross-sections except single angles (tension or compression axial stresses only). Provisions for columns in simple construction are included. Parameters allowing for the changes which occur in structural steel at high temperatures have been included and may be invoked at the user's discretion.

The detailed explanation of the code parameters, cross-section properties, general nomenclature, and code equations are as follows.

1. Table 00BS5950.1-1 Shows the parameters used by 00BS5950 code. Table 00BS5950.1-1 contains the applicable parameter names, their default values, and a brief description of the parameters.
2. Section 00BS5950.2 Describes the cross-section properties used for each shape.
3. Section 00BS5950.3 Contains detail discussion of the parameters used by the 00BS5950 code and they are presented in the alphabetic order in this section.
4. Sections 00BS5950.4 Describes the subsections in the Section 00BS5950.4.
5. Section 00BS5950.4.1 Defines the symbols used in the 00BS5950 code provisions.
6. Section 00BS5950.4.2 Contains detailed discussion of the code provisions and the equations applicable to the I shape cross-sections subjected to bending and axial forces.
7. Section 00BS5950.4.3 Contains detailed discussion of the code provisions and the equations applicable to the single angle cross-sections subjected to axial force only.
8. Section 00BS5950.4.4 Contains detailed discussion of the code provisions and the equations applicable to the circular hollow sections (CHS, pipes) subjected to bending and axial forces.

Table 00BS5950.1-1

00BS5950 Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|---|
| CODE | Required | Identifies the code to be used for member checking or member selection. Specify 00BS5950 for code name. See Table 00BS5950.1-2 and Sections 00BS5950.2, 00BS5950.3, and 00BS5950.4 for a more detailed description. |
| TBLNAM | UNIBEAMS | Identifies the table of profiles to be used during selection (SELECT command). See Table 00BS5950.1-3 for choices. |
| METHOD | EXACT | <p>Identifies the design method. This parameter indicates the type of method that should be used for the shear or combined capacity checks.</p> <p>BOTH = Use simplified and the more exact methods. See Sections 4.4.5, 4.8.2 and 4.8.3 of BS 5950-1:2000 (95).</p> <p>EXACT = Use the more exact method. See Sections 4.4.5.3, 4.8.2.3, 4.8.3.3.2, and 4.8.3.3.3 of BS 5950-1:2000 (95).</p> <p>SIMPLIFY = Use simplified method. See Sections 4.4.5.2, 4.8.2.2 and 4.8.3.2 of BS 5950-1:2000 (95).</p> |
| SECTYPE | ROLLED | <p>Indicates that the cross-section is rolled or welded shape. This parameter is used to determine the equations that are applicable to the rolled or welded shape.</p> <p>ROLLED = Member is hot rolled.</p> <p>WELDED = Member is welded/coldformed.</p> |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|---|
| SHRAREAF | Computed | <p>SHeaR AREA Factor is used for the computation of the shear area. When an alternate value other than COMPUTE or TABLE is specified, shear area is computed as the SHRAREAF times the cross sectional area ($A_v = A_Y = \text{SHRAREAF} \times A_X$).</p> <p>COMPUTE = Compute the shear area based on the Section 4.2.3 of BS 5950-1:2000 (95) except for single and double angles. Shear area for single and double angles are extracted from GTSTRUDL or USER table.</p> <p>TABLE = Shear area from GTSTRUDL or USER table is used.</p> |
| a | 254000.0(mm) | <p>Distance between web stiffeners. This parameter is used to compute the a/d ratio where a/d is the ratio of the distance between stiffeners to web depth. An arbitrary high value of 254000.0 (mm) has been assumed as a default to indicate that the web stiffeners are absent. A value is necessary to account for web stiffeners in the shear capacity calculation (Provisions '4.4.5.2' and '4.4.5.3').</p> |
| SimpSupp | NO | <p>Indicates that a member is simply supported or not. This parameter is used to determine the equations that are applicable to the simply supported members (Provisions '4.2.5.2Z', '4.2.5.3Z', '4.2.5.2Y', and '4.2.5.3Y').</p> <p>NO = Member is not simply supported.</p> <p>YES = Member is simply supported.</p> |
| CODETOL | 0.0 | <p>Percent variance from 1.0 for compliance with the provisions of a code. The ratio of actual/limiting must be less than or equal to $[1.0 + \text{CODETOL}/100]$.</p> |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| Parameter Name | Default Value | Meaning |
|----------------------------|--------------------------|--|
| PF | 1.0 | Area reduction factor for holesout in members subject to axial tension. |
| <u>Material Properties</u> | | |
| STEELGRD | S235JRG2 | Identifies the grade of steel from which a member is made. See Table 00BS5950.1-4 for STEEL GRaDes and their properties. |
| Py | Computed | Design strength p_y (yield stress) of member. Computed from parameter STEELGRD if not given. |
| REDPy | 1.0 | Reduction factor for parameter Py. This factor times parameter Py gives the design strength (p_y) value used by the code. Used to account for property changes at high temperatures. |
| Pyf | Py | Design strength of the flange. If not specified, it is assumed equal to the parameter Py. This parameter is used to define a hybrid cross-section, see parameter Pyw also. |
| Pyw | Py | Design strength of the web. If not specified, it is assumed equal to the parameter Py. This parameter is used to define a hybrid cross-section, see parameter Pyf also. |
| REDE | 1.0 | Reduction factor for E, the modulus of elasticity. Similar to REDPy. |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| Parameter Name | Default Value | Meaning |
|--|--------------------------|--|
| <u>Slenderness Ratio</u> | | |
| SLENCOMP | Computed | Maximum permissible slenderness ratio (L_E/r , KL/r) for a member subjected to axial compression. The default value for maximum compression slenderness ratio is equal to 180. |
| SLENTEN | Computed | Maximum permissible slenderness ratio (L/r) for a member subjected to axial tension. Only a user specified value will initiate the slenderness ratio check for a tension member. |
| <u>Effective Length for a Compression Member</u> | | |
| EFLEY | 1.0 | Effective factor value used for the computation of nominal effective length, $L_{Ey} = EFLEY \times LY$ for a compression member. Nominal effective length, L_{Ey} , is used in the computation of maximum slenderness ratio about the local Y axis of the profile. See Table 00BS5950.1-5 or Sections 4.7.2, 4.7.3, and Table 22 of BS 5950-1:2000 (95) for the EFLEY values. |
| LY | Computed | Unbraced length for buckling about the local Y axis of the cross-section. This parameter is used to compute nominal effective length L_{Ey} for a compression member ($L_{Ey} = EFLEY \times LY$). The default value is computed as a length of the member. |
| FRLY | 1.0 | Fractional form of the parameter LY, allows unbraced length to be specified as fractions of the total length. Used only when default value of 'Computed' is used for parameter LY ($LY = FRLY \times \text{Member Length}$). |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| Parameter Name | Default Value | Meaning |
|--|--------------------------|--|
| <u>Effective Length for a Compression Member</u> (continued) | | |
| EFLEZ | 1.0 | Effective factor value used for the computation of nominal effective length, $L_{Ez} = EFLEZ \times LZ$ for a compression member. Nominal effective length, L_{Ez} , is used in the computation of maximum slenderness ratio about the local Z axis of the profile. See Table 00BS5950.1-5 or Sections 4.7.2, 4.7.3, and Table 22 of BS 5950-1:2000 (95) for the EFLEZ values. |
| LZ | Computed | Unbraced length for buckling about the local Z axis of the cross-section. This parameter is used to compute nominal effective length L_{Ez} for a compression member ($L_{Ez} = EFLEZ \times LZ$). The default value is computed as a length of the member. |
| FRLZ | 1.0 | Fractional form of the parameter LZ, allows unbraced length to be specified as fractions of the total length. Used only when default value of 'Computed' is used for parameter LZ ($LZ = FRLZ \times \text{Member Length}$). |

Effective Length for Lateral-Torsional Buckling

| | | |
|----|-----|---|
| LE | LLT | Effective length of a member for lateral torsional buckling of a beam with restraints at the ends. Default value is the effective length between restraints against lateral-torsional buckling of a member under bending, see parameter LLT ($LE = EFLE \times LLT$). See Table 00BS5950.1-6 for alternative values and also see Table 13 and 14 of the BS5950-1:2000 (95). |
|----|-----|---|

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| Parameter | Default | |
|--|----------------|---|
| <u>Name</u> | <u>Value</u> | <u>Meaning</u> |
| <u>Effective Length for Lateral-Torsional Buckling</u> (continued) | | |
| EFLE | 1.0 | Effective factor value used for the computation of the effective length, LE of a member under bending. Used only when default value of LLT is used for parameter LE ($LE = EFLE \times LLT$, see Table 00BS5950.1-6 and parameter LE). |
| LLT | Computed | Segment length between restraints against lateral-torsional buckling (unbraced length). This parameter generally used to specify the segment length of the compression flange restraint against lateral-torsional buckling (unbraced length of the compression flange). Computed as length of member. |
| FRLLT | 1.0 | Fractional value used for the computation of the unbraced lateral-torsional buckling length of a member, LLT. Used only when default value of 'Computed' is used for parameter LLT ($LLT = FRLLT \times \text{Member Length}$). |

Equivalent Uniform Moment Factors

| | | |
|-----|----------|---|
| mLT | Computed | Equivalent uniform moment factor for lateral-torsional buckling (m_{LT}) which is used in the member buckling resistance equations. This parameter modifies Z axis bending buckling capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation. |
| my | Computed | Equivalent uniform moment factor for flexural buckling (m_y) which is used in the member buckling resistance equations. This parameter modifies Y axis bending capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation. |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| Parameter Name | Default Value | Meaning |
|--|--------------------------|--|
| <u>Equivalent Uniform Moment Factors</u> (continued) | | |
| mz | Computed | Equivalent uniform moment factor for flexural buckling (m_z) which is used in the member buckling resistance equations. This parameter modifies Z axis bending capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation. |
| myz | Computed | Equivalent uniform moment factor for lateral flexural buckling (m_{yz}) which is used in the member out-of-plane buckling resistance equations. This parameter modifies Y axis bending capacity in combined axial and bending capacity equations. See Section 00BS5950.3 for more explanation. |
| SDSWAYY | YES | Indicates the presence or absence of SiDeSWAY about the local Y axis. YES = Sidesway permitted. NO = Sidesway prevented. |
| SDSWAYZ | YES | Indicates the presence or absence of SiDeSWAY about the local Z axis. YES = Sidesway permitted. NO = Sidesway prevented. |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| Parameter | Default | |
|--|----------------|--|
| <u>Name</u> | <u>Value</u> | <u>Meaning</u> |
| <u>Equivalent Uniform Moment Factors</u> (continued) | | |
| DESTLDY [*] | YES | <p>Indicates the presence or absence of a DESTabilizing LoaD which causes movement in the member local Y axis direction (and possibly rotation about the member local Y axis). Destabilizing load conditions exist when a load is applied in the local Z axis direction of a member and both the load and the member are free to deflect laterally (and possibly rotationally also) relative to the centroid of the member. This parameter is only applicable to LOADS list or ALL LOADS of the PARAMETERS command.</p> <p>YES = Destabilizing load. NO = Normal load.</p> |
| DESTLDZ [*] | YES | <p>Indicates the presence or absence of a DESTabilizing LoaD which causes movement in the member local Z axis direction (and possibly rotation about the member local Z axis). Destabilizing load conditions exist when a load is applied to the top flange (local Y axis load) of a member and both the load and the flange are free to deflect laterally (and possibly rotationally also) relative to the centroid of the member. This parameter is only applicable to LOADS list or ALL LOADS of the PARAMETERS command.</p> <p>YES = Destabilizing load. NO = Normal load.</p> |
| <u>Force Limitation</u> | | |
| FXMIN | 2.224 (N) | Minimum axial force to be considered by the code; anything less in magnitude is taken as zero. Units are in newtons (N). |
| FYMIN | 2.224 (N) | Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero. |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| <u>Parameter Name</u> | <u>Default Value</u> | <u>Meaning</u> |
|-------------------------------------|---------------------------------|--|
| <u>Force Limitation</u> (continued) | | |
| FZMIN | 2.224 (N) | Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero. |
| MYMIN | 2260.0 (mm-N) | Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero. |
| MZMIN | 2260.0 (mm-N) | Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero. |
| <u>Output Processing</u> | | |
| MXTRIALS | 500.0 | Maximum number of profiles to be tried when designing a member. Default is larger than the number of profiles in most tables. |
| SUMMARY | NO | Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; see Sections 2.9 and 7.2 of Volume 2A for an explanation. |
| PrintLim | NO | Parameter to request to print the section limiting values for limit state and load and resistance factor codes. This parameter is applicable to the steel design CHECK and SELECT commands. The default output from CHECK or SELECT command prints the section force values. A value of "YES" for this parameter indicates that the section limiting values should be printed instead of default section forces. |

Table 00BS5950.1-1 (continued)

00BS5950 Code Parameters

| <u>Parameter Name</u> | <u>Default Value</u> | <u>Meaning</u> |
|--------------------------------------|---------------------------------|---|
| <u>Output Processing</u> (continued) | | |
| TRACE | 4.0 | Flag indication when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A for the explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces. |
| VALUES | 1.0 | Flag indication if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A for the explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties. |

Table 00BS5950.1-2

GTSTRUDL British Standard Codes

| Code Name | Parameter Table | <u>Application</u> |
|----------------------|----------------------------|---|
| 00BS5950 | 00BS5950.1-1 Volume 2C | Checks compliance of I shapes and Circular Hollow Sections (Pipes) (subjected to bi-axial bending and axial force), and also Single Angles (subjected to axial forces only) profiles to the BS 5950-1:2000, Part 1 specification (95), adopted May, 2001. |
| BS5950 | BS5950 Volume 2A | Checks compliance of I, Single angle, Channel, Tee, Double angle, Solid round bar, Pipe, Solid square and Rectangular bar, and Structural tubing shape profiles to the BS 5950: Part 1: 1990 specification (79), adopted July 31, 1990. |
| BS449 | BS449 Volume 2A | Checks compliance of I, Single angle, Channel, Tee, Pipe, and Structural tubing shape profiles to the British Standard 449, part 2 Metric Units, Specifications for the Use of Structural Steel in Building, British Standard Institution, October 1969, with amendments through July 1975. |

Table 00BS5950.1-3

**GTSTRUDL Profile Tables for the
Design based on the 00BS5950 Code**

| <u>Profile Shapes</u> | <u>Reference</u> |
|------------------------------|--|
| I shapes | See Table D-1 in Appendix D for list of Applicable Table names for Universal Beams, Universal Columns, Joists, Universal Bearing Piles, I shapes, W, S, M, HP shapes, Wide Flange shapes, etc. |
| Single Angles | See Table D-2 in Appendix D for list of single angle Table names applicable to 00BS5950 code. |
| Circular Hollow Sections | See Table D-4 in Appendix D for list of Circular Hollow Section (Pipe, Round HSS) Table names applicable to 00BS5950 code. |

Table 00BS5950.1-4

Steel Grades Based on the BS 5950-1:2000 (00BS5950) and 1993 Eurocode (EC3) Specification

| Steel Grade | Nominal Yield Strength, f_y (N/mm ²) | | | | | | | | Ultimate Tensile Strength, f_u (N/mm ²) | | |
|-------------|--|-------------|-------------|-------------|--------------|---------------|---------------|---------------|---|---------------|---------------|
| | t # 16 | 16 < t # 40 | 40 < t # 63 | 63 < t # 80 | 80 < t # 100 | 100 < t # 150 | 150 < t # 200 | 200 < t # 250 | t # 100 | 100 < t # 150 | 150 < t # 250 |
| S185 | 185 | 175 | ! | ! | ! | ! | ! | ! | 290 | ! | ! |
| S235JR | 235 | 225 | ! | ! | ! | ! | ! | ! | 340 | ! | ! |
| S235JRG1 | 235 | 225 | ! | ! | ! | ! | ! | ! | 340 | ! | ! |
| S235JRG2 | 235 | 225 | 215 | 215 | 215 | 195 | 185 | 175 | 340 | 340 | 320 |
| S235J0 | 235 | 225 | 215 | 215 | 215 | 195 | 185 | 175 | 340 | 340 | 320 |
| S235J2G3 | 235 | 225 | 215 | 215 | 215 | 195 | 185 | 175 | 340 | 340 | 320 |
| S235J2G4 | 235 | 225 | 215 | 215 | 215 | 195 | 185 | 175 | 340 | 340 | 320 |
| S275JR | 275 | 265 | 255 | 245 | 235 | 225 | 215 | 205 | 410 | 400 | 380 |
| S275J0 | 275 | 265 | 255 | 245 | 235 | 225 | 215 | 205 | 410 | 400 | 380 |
| S275J2G3 | 275 | 265 | 255 | 245 | 235 | 225 | 215 | 205 | 410 | 400 | 380 |
| S275J2G4 | 275 | 265 | 255 | 245 | 235 | 225 | 215 | 205 | 410 | 400 | 380 |
| S275N | 275 | 265 | 255 | 245 | 235 | 225 | ! | ! | 370 | 350 | ! |
| S275NL | 275 | 265 | 255 | 245 | 235 | 225 | ! | ! | 370 | 350 | ! |

Table 00BS5950.1-4 (continued)

Steel Grades Based on the BS 5950-1:2000 (00BS5950) and 1993 Eurocode (EC3) Specification

| Steel Grade | Nominal Yield Strength, f_y (N/mm ²) | | | | | | | | Ultimate Tensile Strength, f_u (N/mm ²) | | |
|-------------|--|-------------|-------------|-------------|--------------|---------------|---------------|---------------|---|---------------|---------------|
| | t # 16 | 16 < t # 40 | 40 < t # 63 | 63 < t # 80 | 80 < t # 100 | 100 < t # 150 | 150 < t # 200 | 200 < t # 250 | t # 100 | 100 < t # 150 | 150 < t # 250 |
| S355JR | 355 | 345 | 335 | 325 | 315 | 295 | 285 | 275 | 490 | 470 | 450 |
| S355J0 | 355 | 345 | 335 | 325 | 315 | 295 | 285 | 275 | 490 | 470 | 450 |
| S355J2G3 | 355 | 345 | 335 | 325 | 315 | 295 | 285 | 275 | 490 | 470 | 450 |
| S355J2G4 | 355 | 345 | 335 | 325 | 315 | 295 | 285 | 275 | 490 | 470 | 450 |
| S355K2G3 | 355 | 345 | 335 | 325 | 315 | 295 | 285 | 275 | 490 | 470 | 450 |
| S355K2G4 | 355 | 345 | 335 | 325 | 315 | 295 | 285 | 275 | 490 | 470 | 450 |
| S355N | 355 | 345 | 335 | 325 | 315 | 295 | ! | ! | 470 | 450 | ! |
| S355NL | 355 | 345 | 335 | 325 | 315 | 295 | ! | ! | 470 | 450 | ! |
| S420N | 420 | 400 | 390 | 370 | 360 | 340 | ! | ! | 520 | 500 | ! |
| S420NL | 420 | 400 | 390 | 370 | 360 | 340 | ! | ! | 520 | 500 | ! |
| S460N | 460 | 440 | 430 | 410 | 400 | ! | ! | ! | 550 | ! | ! |
| S460NL | 460 | 440 | 430 | 410 | 400 | ! | ! | ! | 550 | ! | ! |

Table 00BS5950.1-5

**Effective Factor Values EFLEY and EFLEZ for
Nominal Effective Length L_{Ey} and L_{Ez} computation
British Standard BS 5950-1:2000 Specification**

| a) non-sway mode* | | | |
|--|--|-------------------------------------|-----------------------|
| Restraint (in the plane under consideration) by other parts of structure | | | EFLEY and EFLEZ |
| Effectively held in position at both ends | Effectively restrained in direction at both ends | | 0.7 |
| | Partially restrained in direction at both ends | | 0.85 |
| | Restrained in direction at one end | | 0.85 |
| | Not restrained in direction at either end | | 1.0 |
| b) sway mode* | | | |
| One end | Other end | | EFLEY and EFLEZ |
| Effectively held in position and restrained in direction | Not held in position | Effectively restrained in direction | 1.2 |
| | | Partially restrained in direction | 1.5 |
| | | Not restrained in direction | 2.0 |
| * Excluding angle, channel or T-section struts designed in accordance with Section 4.7.10 of the BS 5950-1:2000 (95) | | | |

Example:

PARAMETERS

| | | | | |
|---------|-----|-----------|----|--------------------------------|
| 'EFLEY' | 1.5 | MEMBER 1 | \$ | $L_{Ey} = 1.5LY$ for member 1 |
| 'EFLEZ' | 1.2 | MEMBER 25 | \$ | $L_{Ez} = 1.2LZ$ for member 25 |

LY and LZ are the unbraced length for buckling about the local Y and Z axis of the cross-section (see parameter LY and LZ).

Table 00BS5950.1-6

Effective Length L_E **British Standard BS 5950-1:2000 Specification**

| Conditions of restraint at supports | Alternate values for Parameter LE | Loading conditions | |
|---|--------------------------------------|--------------------|---------------|
| | | Normal | Destabilizing |
| | | DESTLDZ = NO | DESTLDZ = YES |
| Default value for parameter LE | LLT | EFLLT×LLT | EFLLT×LLT |
| Compression flange laterally restrained. Nominal torsional restraint against rotation about longitudinal axis. | | | |
| Both flanges fully restrained against rotation on plan | A1 | 0.7LLT | 0.85LLT |
| Compression flange fully restrained against rotation on plan | A2 | 0.75LLT | 0.9LLT |
| Both flanges partially restrained against rotation on plan | A3 | 0.8LLT | 0.95LLT |
| Compression flange partially restrained against rotation on plan | A4 | 0.85LLT | 1.0LLT |
| Both flanges free to rotate on plan | A5 | 1.0LLT | 1.2LLT |
| Compression flange laterally unrestrained. Both flanges free to rotate on plan | | | |
| Partial torsional restraint against rotation about longitudinal axis provided by connection of bottom flange to supports | A6 | 1.0LLT + 2D | 1.2LLT + 2D |
| Partial torsional restraint against rotation about longitudinal axis provided only by pressure of bottom flange onto supports | A7 | 1.2LLT + 2D | 1.4LLT + 2D |

Example:

PARAMETERS

| | | | |
|-----------|-------|----------|----------------------------------|
| 'DESTLDZ' | 'NO' | LOAD 2 | |
| 'DESTLDZ' | 'YES' | LOAD 5 | |
| 'LE' | 'A3' | MEMBER 1 | \$ LE = 0.8LLT for load 2 and |
| | | | \$ LE = 0.95LLT for load 5 |
| 'LE' | 'A7' | MEMBER 8 | \$ LE = 1.2LLT+2D for load 2 and |
| | | | \$ LE = 1.4LLT+2D for load 5 |

1. D is the depth of cross-section (table property YD).
2. Default value for parameter EFLLT is equal to 1.0.
3. For cantilevers and other types of beams not in Table 00BS5950.1-6, use parameter EFLLT to specify the effective length factor ($LE = EFLLT \times LLT$).

5.2.4 GTSTRUDL Indian Standard Design Code IS800

A new steel design code named IS800 has been added. This code is based on the Indian Standard, IS:800-1984, Code of Practice for General Construction in Steel, Second Revision. Applicable cross-sections for the IS800 code as follows:

| | |
|---------------|-------------------|
| I-shapes | Solid Round Bars |
| Channels | Pipes |
| Single Angles | Solid Square Bars |
| Tees | Solid Rectangular |
| Double Angles | Structural Tubes |

Table IS800

IS800 Code Parameters

| <u>Parameter Name</u> | <u>Default Value</u> | <u>Meaning</u> |
|-----------------------|----------------------|--|
| CODE | Required | Identifies the code to be used for member checking or member selection. Specify IS800 for code name. See Section IS800 for a more detailed description. |
| TBLNAM | ISBEAMS | Identifies the table of profiles to be used during selection. See Table 2.1-2a for choices. |
| CODETOL | 0.0 | Percent variance from 1.0 for compliance with the provisions of a code. The ratio of Actual/Allowable must be less than or equal to $[1.0 + \text{CODETOL}/100]$. |
| PF | 1.0 | Area reduction factor for holesout in members subject to axial tension. |
| a | 254000.0(mm) | Distance between web stiffeners. This parameter is used to compute a/h ratio. The a/h ratio is the ratio of the distance between stiffeners to the web depth. An arbitrary high value of 254000.0 (mm) has been assumed as a default to indicate that web stiffeners are absent. A value is necessary to account for web stiffeners in the allowable shear stress calculation (Provision '6.4.2 Y' and '6.4.2 Z'). |

Table IS800 (continued)

IS800 Code Parameters

| Parameter Name | Default Value | Meaning |
|----------------------------|--------------------------|--|
| <u>Material Properties</u> | | |
| STEELGRD | A36 | Identifies the grade of steel from which a member is made. See Table 2.1-3 in Volume 2A for steel grades and their properties. |
| FY | Computed | Yield stress of member. Computed from STEELGRD if not given. |
| REDFY | 1.0 | Reduction factor for FY. This factor times FY gives the f_y value used by the code. Used to account for property changes at high temperatures. |
| REDE | 1.0 | Reduction factor for E, the modulus of elasticity. Similar to REDFY. |

Slenderness Ratio

| | | |
|----------|----------|--|
| SLENCOMP | Computed | Maximum permissible slenderness ratio (KL/r) for member subjected to axial compression. When no value is specified for this parameter, the value of 180 is used for the maximum slenderness ratio. |
| SLENTEN | Computed | Maximum permissible slenderness ratio (L/r) for member subjected to axial tension. When no value is specified for this parameter, the value of 400 is used for the maximum slenderness ratio. |

Table IS800 (continued)

IS800 Code Parameters

| <u>Parameter Name</u> | <u>Default Value</u> | <u>Meaning</u> |
|---------------------------|--------------------------|--|
| <u>K-Factors</u> | | |
| COMPK | NO | <p>Parameter to request the computation of the effective length factors KY and KZ (Sections 2.2 and 2.3 of Volume 2A).</p> <p>YES = Compute KY and KZ factors. See the COLUMN/BEAM LINE command (Section 2.3 of Volume 2A).</p> <p>KY = Compute KY only.</p> <p>KZ = Compute KZ only.</p> <p>NO = Use default or specified values for KY and KZ.</p> |
| KY | 1.0 | Effective length factor for buckling about the local Y axis of the profile. See Sections 2.2 and 2.3 of Volume 2A for GTSTRUDL computation of effective length factor, KY. |
| KZ | 1.0 | Effective length factor for buckling about the local Z axis of the profile. See Sections 2.2 and 2.3 of Volume 2A for GTSTRUDL computation of effective length factor, KZ. |
| SDSWAYY | YES | <p>Indicates the presence or absence of sidesway about the local Y axis.</p> <p>YES = sidesway permitted.</p> <p>NO = sidesway prevented.</p> |
| SDSWAYZ | YES | <p>Indicates the presence or absence of sidesway about the local Z axis.</p> <p>YES = sidesway permitted.</p> <p>NO = sidesway prevented.</p> |

Table IS800 (continued)

IS800 Code Parameters

| Parameter Name | Default Value | Meaning |
|------------------------------|--------------------------|---|
| <u>K-Factors</u> (continued) | | |
| GAY | Computed | G-factor at the start joint of the member. GAY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A). |
| GAZ | Computed | G-factor at the start joint of the member. GAZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A). |
| GBY | Computed | G-factor at the end joint of the member. GBY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A). |
| GBZ | Computed | G-factor at the end joint of the member. GBZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A). |
| <u>Buckling Length</u> | | |
| LY | Computed | Unbraced length for buckling about the local Y axis of the profile. Computed as length of member. |
| LZ | Computed | Unbraced length for buckling about the local Z axis of the profile. Computed as length of member. |

IS800 Code Parameters

| Parameter Name | Default Value | Meaning |
|------------------------------------|--------------------------|---|
| <u>Buckling Length</u> (Continued) | | |
| FRLY | 1.0 | Fractional form of the parameter LY. Allows the unbraced length to be specified as fractions of the total length. Used only when LY is computed. |
| FRLZ | 1.0 | Fractional form of the parameter LZ, similar to FRLY. Used only when LZ is computed. |
| <u>Bending Stress</u> | | |
| UNLCF | Computed | Unbraced length of the compression flange. Computed as length of member. In this parameter no distinction is made between the unbraced length for the top or bottom flange. See UNLCFTF or UNLCFBF. |
| FRUNLCF | 1.0 | Fractional form of the parameter UNLCF. Allows the unbraced length to be specified as a fraction of the total length. Used only when UNLCF is computed. |
| UNLCFTF | Computed | Unbraced length of the compression flange for the top flange. When no value is specified, UNLCF and FRUNLCF are used for this parameter. |
| UNLCFBF | Computed | Unbraced length of the compression flange for the bottom flange. When no value is specified, UNLCF and FRUNLCF are used for this parameter. |

Table IS800 (continued)

IS800 Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|---|
| <u>Combined Stresses</u> | | |
| AXEFF | 0.0 | Axial stress reduction factor indicating the amount of the axial stress which is to be deducted from a corresponding bending stress acting in the opposite direction. |
| CMY | Computed | Coefficient which modifies Y axis bending stress in interaction equation (IS:800-1984 Second Ed., Section 7). |
| CMZ | Computed | Coefficient which modifies Z axis bending stress in interaction equation (IS:800-1984 Second Ed., Section 7). |
| <u>Force Limitation</u> | | |
| FXMIN | 2.2 (N) | Minimum axial force to be considered by the code; anything less in magnitude is taken as zero. |
| FYMIN | 2.2 (N) | Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero. |
| FZMIN | 2.2 (N) | Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero. |
| MYMIN | 89.0(N-mm) | Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero. |
| MZMIN | 89.0(N-mm) | Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero. |

IS800 Code Parameters

| Parameter Name | Default Value | Meaning |
|--|--------------------------|---|
| <u>Output Processing and System Parameters</u> | | |
| MXTRIALS | 500.0 | Maximum number of profiles to be tried when designing a member. Default is larger than the number of profiles in most tables. |
| PRIDTA | 1.0 | Flag for requesting output from selection procedure. 1 = no output 2 = output parameters |
| SUMMARY | NO | Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; See Sections 2.9 and 7.2 of Volume 2A for explanation. |
| TRACE | 4.0 | Flag indication when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A for explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces. |

Table IS800 (continued)

IS800 Code Parameters

| Parameter | Default |
|------------------|----------------|
| <u>Name</u> | <u>Value</u> |

| <u>Meaning</u> |
|----------------|
|----------------|

Output Processing and System Parameters (continued)

| | | |
|--------|-----|--|
| VALUES | 1.0 | Flag indication if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A for explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties. |
|--------|-----|--|

5.2.5 GTSTRUDL Profile Tables for the Design based on the IS800 Code

The profile tables for design based on the IS800 Code are shown below:

Table 2.1-2l

**GTSTRUDL Profile Tables
for the Design based on the IS800 Code***

| <u>Table Name</u> | <u>Reference</u> |
|---------------------------|---|
| Angle shape Tables | |
| ISEQANGL | Equal single leg angle cross-sections from Tables 5.1 and 5.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision. |
| ISUNANGL | Unequal single leg angle cross-sections from Tables 6.1 and 6.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision. |
| ANGLES | Equal and unequal single leg angles from 1969 AISC ASD Seventh Edition (16). |
| ANGLESM | Equal and unequal single leg angles from AISC Metric “ANGLES, Equal legs and unequal legs” table (83). |
| EQANGLE | Equal single leg angles from 1969 AISC ASD Seventh Edition (16). |
| ULANGLE | Unequal single leg angles from 1969 AISC ASD Seventh Edition (16). |
| USANGLE | Unequal single leg angles from 1969 AISC ASD Seventh Edition (16). |
| EQDBLANG | Double angles with equal legs back-to-back from 1969 AISC ASD Seventh Edition (16). |
| LLDBLANG | Double angles with long legs back-to-back from 1969 AISC ASD Seventh Edition (16). |

*See Appendix C of Volume 2A for Table description and profile names.

Table 2.1-2l (continued)
GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

| <u>Table Name</u> | <u>Reference</u> |
|---------------------------------------|--|
| Angle shape Tables (continued) | |
| SLDBLANG | Double angles with short legs back-to-back from 1969 AISC ASD Seventh Edition (16). |
| EQDBANGM | Equal leg double angle profiles from AISC Metric “DOUBLE ANGLES, Two equal leg angles, Properties of sections” table (83). |
| LLDBANGM | Unequal leg, long leg back-to-back, double angle profiles from AISC Metric “DOUBLE ANGLES, Two unequal leg angles, Properties of sections, Long legs back-to-back” table (83). |
| SLDBANGM | Unequal leg, short leg back-to-back, double angle profiles from AISC Metric “DOUBLE ANGLES, Two equal leg angles, Properties of sections, Short legs back-to-back” table (83). |
| BSEQANGL | Equal leg single angle profiles from British “EQUAL ANGLES” table (82). |
| BSEQDBAN | Equal leg double angle profiles from British “COMPOUND EQUAL ANGLES LEGS BACK TO BACK” table (82). |
| BSLLDBAN | Unequal long legs back-to-back double angle profiles from British “COMPOUND UNEQUAL ANGLES LONG LEGS BACK TO BACK” table (82). |
| BSSLDBAN | Unequal short legs back-to-back double angle profiles from British “COMPOUND UNEQUAL ANGLES SHORT LEGS BACK TO BACK” table (82). |

BAR shape Tables

| | |
|------|------------------------------------|
| BARS | Round and square solid bar shapes. |
| RBAR | Rectangular solid bar shapes. |

*See Appendix C of Volume 2A for Table description and profile names.

Table 2.1-2I (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

| <u>Table Name</u> | <u>Reference</u> |
|-----------------------------|--|
| Channel shape Tables | |
| ISCHAN | Channel sections (sloping flange channels) from Table 4.1 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision. |
| ISCHAMP | Channel sections (parallel flange channels) from Table 4.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision. |
| CHANNEL9 | Channel shapes from 1989 AISC ASD Ninth Edition (72). |
| CHANNELM | Channel shape profiles from AISC Metric “CHANNELS AMERICAN STANDARD” table (83). |
| CHANNELS | Channel shapes from 1978 AISC ASD Eighth Edition (33). |
| BSCHAN | Channel shape profiles from British “CHANNELS” table (82). |
| I shape Tables | |
| ISBEAMS | I shape beam sections (medium flange beams, junior and light weight beams) from Tables 2.1 and 2.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision. |
| ISCOLUMN | I shape column/heavy weight beam sections (column and heavy weight beams) from Table 3.1 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision. |

*See Appendix C of Volume 2A for Table description and profile names.

Table 2.1-2I (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

| Table Name | Reference |
|-----------------------------------|--|
| I shape Tables (continued) | |
| WSHAPES9 | W shapes from 1989 AISC ASD Ninth Edition (72). |
| M/S/HP9 | M, S, and HP shapes from 1989 AISC ASD Ninth Edition (72). |
| WBEAM9 | W shapes commonly used as beams from 1989 AISC ASD Ninth Edition (72). |
| WCOLUMN9 | W shapes commonly used as columns from 1989 AISC ASD Ninth Edition (72). |
| WSHAPESM | W shape from AISC Metric “WSHAPES” table (83). |
| M/S/HPM | M, S, and HP shape profiles from AISC Metric “M SHAPES, S SHAPES, and HP SHAPES” table (83). |
| WBEAMM | W shape profiles commonly used as beams from AISC Metric “WSHAPES” table (83). |
| WCOLUMNM | W shape profiles commonly used as columns from AISC Metric “WSHAPES” table (83). |
| STEELW78 | W shapes from 1978 AISC ASD Eighth Edition (33). |
| HP/S/M | HP, S, and M shapes from 1978 AISC ASD Eighth Edition (33). |
| W78BEAM | W shapes commonly used as beams from 1978 AISC ASD Eighth Edition (33). |
| W78COLUMN | W shapes commonly used as columns from 1978 AISC ASD Eighth Edition (33). |
| STEELW | W shapes from 1969 AISC ASD Seventh Edition (16). |
| WCOLUMN | W shapes commonly used as columns from 1969 AISC ASD Seventh Edition (16). |

*See Appendix C of Volume 2A for Table description and profile names.

Table 2.1-21 (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

| Table Name | Reference |
|-----------------------------------|--|
| I shape Tables (continued) | |
| UNIBEAMS | British Universal Beam profiles from 1996 BS 5950 Section Properties, 4th Edition (82). |
| UNICOL | British Universal Column profiles from 1996 BS 5950 Section Properties, 4th Edition (82). |
| JOISTS | British Joist profiles from 1996 BS 5950 Section Properties, 4th Edition (82). |
| UBPILES | I shape profiles from British “UNIVERSAL BEARING PILES” table (82). |
| HEA | H shaped (HE-A) profiles from Breite I-Träger, Reihe HE-A. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997". |
| HEB | H shaped (HE-B) profiles from Breite I-Träger, Reihe HE-B. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997". |
| HEM | H shaped (HE-M) profiles from Breite I-Träger, Reihe HE-M. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997". |
| IPE | I shaped (IPE) profiles from Mittelbreite I-Träger, IPE-Reihe. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997". |
| EUROPEAN | This table contains profiles from IPE, HEA, HEB, and HEM tables. |

*See Appendix C of Volume 2A for Table description and profile names.

Table 2.1-2I (continued)

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

| <u>Table Name</u> | <u>Reference</u> |
|--------------------------|--|
| Pipe shape Tables | |
| PIPES9 | Pipe shapes from 1989 AISC ASD Ninth Edition (72). |
| PIPESM | Pipe shapes from AISC Metric “PIPE” table (83). |
| AISCPIPE | Pipe shapes from 1978 AISC ASD Eighth Edition (33). |
| SSPIPE | Pipe shapes from ANSI B36.19-1976. |
| WSPipe | Pipe shapes from ANSI B36.10-1979 |
| CIHOLLOW | British Circular Hollow profiles from 1996 BS 5950 Section Properties, 4th Edition (82). |
| Tee shape Tables | |
| TEES9 | Tee shapes from 1989 AISC ASD Ninth Edition (72). |
| TEESM | Tee shape profiles from AISC Metric “STRUCTURAL TEES, Cut from W shapes” table (83). |
| TEES | Tee shapes from 1978 AISC ASD Eighth Edition (33). |
| TEEUBeam | Tee shape profiles from British “STRUCTURAL TEES CUT FROM UNIVERSAL BEAMS” table (82). |
| TEEUCOLU | Tee shape profiles from British “STRUCTURAL TEES CUT FROM UNIVERSAL COLUMNS” table (82). |

*See Appendix C of Volume 2A for Table description and profile names.

GTSTRUDL Profile Tables
for the Design based on the IS800 Code*

| <u>Table Name</u> | <u>Reference</u> |
|--------------------------------------|--|
| Tube shape Tables (continued) | |
| TUBES9 | Structural Tubing shapes from 1989 AISC ASD Ninth Edition (72). |
| TUBESM | Structural Tubing shapes from AISC Metric “STRUCTURAL TUBING Square and Rectangular” table (83). |
| TUBE80 | Structural Tubing shapes from 1978 AISC ASD Eighth Edition (33). |
| AISCTUBE | Structural Tubing shapes from 1969 AISC ASD Seventh Edition (16). |
| REHOLLOW | British Rectangular Hollow profiles from 1996 BS 5950 Section Properties, 4th Edition (82). |
| SQHOLLOW | British Square Hollow profiles from 1996 BS 5950 Section Properties, 4th Edition (82). |

*See Appendix C of Volume 2A for Table description and profile names.

This page intentionally left blank.

5.2.6 Steel Deflection Check and Design

Deflection check and design is now available for steel design. Deflection check and design for steel members can be performed with or without a stress code check or select. Deflection check or design is available for all steel design codes (i.e. ASD9, LRFD2, BS5950, CAN97, EC3, IS800, etc.). For physical members, only a deflection check is valid at the present time. Deflection design has not been implemented at this time for physical members or when the start and end joints are specified by the parameters 'DefStaJT' and 'DefEndJT'.

Deflection check or design is based on the member or physical member chord deflection (see LIST SECTION DISPLACEMENTS command, Section 2.1.14.6 of the Volume 1). Chord deflection is the displacement of the member or physical member relative to a line between member or physical member deflected end points in the member or physical member reference frame (i.e. the member's start and end deflection are always equal to 0.0).

Only the default value for the TRACE parameter is valid for deflection check. If a value other than 4 (default value) has been specified for the parameter TRACE, output from CHECK or SELECT will not have deflection result information. Note that the WITH AXIAL, BEAM, or GENERAL options of the SELECT command are not valid for deflection design. These options are not supported for the SELECT command.

Nine new parameters are available for the deflection check or design. The new parameters are described below:

New Deflection Check Parameters

| | |
|-----------|---------------|
| Parameter | Default Value |
|-----------|---------------|

| | |
|-------------|----------------|
| <u>Name</u> | <u>Meaning</u> |
|-------------|----------------|

| | | |
|----------|----|---|
| DefCheck | NO | Parameter to request the deflection check or design. Deflection check or design is based on the member or physical member chord deflection (see LIST SECTION DISPLACEMENTS command, Section 2.1.14.6 of Volume 1). Chord deflection is the displacement of the member or physical member relative to a line between member or physical member deflected end points in the member or physical member reference frame (i.e. the member's start and end deflection are always equal to 0.0). Values of 'NO', |
|----------|----|---|

‘YES’, ‘YES-Y’, ‘YES-Z’, ‘ONLY’, ‘ONLY-Y’, and ‘ONLY-Z’ are valid for this parameter and explanations of these parameters are as follows:

- NO = No deflection check or design.
- YES = Perform deflection code check (CHECK MEMBER command) or design a member to satisfy deflection requirement (SELECT MEMBER command). This option performs a deflection check or design for local Y and Z axis and also performs code check or design based on the user specified code. When this option is used, the design code name must be specified also.
- YES-Y = Perform deflection check or design for local Y axis deflection. Also, perform a code check or design based on the user specified code. When this option is used, the design code name must be specified also.
- YES-Z = Perform deflection check or design for local Z axis deflection. Also, perform a code check or design based on the user specified code. When this option is used, the design code name must be specified also.
- ONLY = Perform only the deflection code check (CHECK MEMBER command) or design of a member to satisfy deflection requirement (SELECT MEMBER command). This option performs a deflection check for local Y and Z axis. This option does not perform a code check or code design but does perform a deflection check or deflection design.
- ONLY-Y = Perform only the deflection check or design for local Y axis deflection. This option does not perform a code check or code design but does perform a deflection check or deflection design.
- ONLY-Z = Perform only the deflection check or design for local Z axis deflection. This option does not perform a code check or code design but does perform a deflection check or deflection design.

| GT STRUDL | | Steel Deflection Check and Design |
|-----------|----------|---|
| DefLoads | NO | Parameter to specify the loadings that you want to be used for the deflection check. A value of YES for a list of loads defines the loads to be used for the deflection check. If this parameter has not been specified, the active loads are used for the deflection check. |
| DefLimit | 360 | Parameter to specify the deflection limitation. Member length (chord length) is divided by the value of this parameter which defines the deflection limitation ($L/360$). This value is used for both local Y and Z direction deflection check. |
| DefLim-Y | Computed | Parameter to specify the local Y direction deflection limitation. This parameter is used for the local Y direction deflection check. Member length (chord length) is divided by the value of this parameter which defines the Y direction deflection limitation ($L/360$). If not specified, the value for parameter 'DefLimit' is used. |
| DefLim-Z | Computed | Parameter to specify the local Z direction deflection limitation. This parameter is used for the local Z direction deflection check. Member length (chord length) is divided by the value of this parameter which defines the Z direction deflection limitation ($L/360$). If not specified, the value for parameter 'DefLimit' is used. |
| DefPhys | NO | Parameter to define the members that you want to be deflection checked based on the member's physical length. The specified members must be part of a physical member. A value of YES indicates that the deflection check should be based on the physical member chord. Deflection design is not available for physical members at this time. |
| DefStaJT | Computed | This parameter can be used to specify the start joint and parameter 'DefEndJT' can be used to specify the end joint of the chord that the member is lying on. The deflection is checked based on the chord through the joints specified by the parameters 'DefStaJT' and 'DefEndJT'. Start joint name followed by a list of members should be specified for this parameter (ex: DefStaJT JNT1 Members 'M1' 'M2' 'M3'). Deflection design is not available for this option at this time. |

| | | |
|----------|----------|---|
| DefEndJT | Computed | This parameter can be used to specify the end joint and parameter 'DefStaJT' can be used to specify the start joint of the chord that the member is lying on. The deflection is checked based on the chord through the joints specified by the parameters 'DefStaJT' and 'DefEndJT'. End joint name followed by a list of members should be specified for this parameter (ex: DefEndJT JNT4 Members 'M1' 'M2' 'M3'). Deflection design is not available for this option at this time. |
| DefNuSec | 9 | Parameter to specify the number of sections to be used for the computation of the member deflection. The default of 9 sections is used to compute the section deflections. The deflection check is performed for the location that has the largest deflection. |

Example:

The example illustrates the usage of several of the new parameters needed in order to perform a deflection check. The user should note that both factored and unfactored loading combinations are required since deflection checks are usually performed on unfactored (service) loads.

\$ Factored load combinations for code check

```
LOADING COMBINATION 'A' 'SW + Live Load' -
    COMBINE 'SW' 1.0 'LL' 1.0
LOADING COMBINATION 'B' '0.75(SW + Live Load + Wind Load from Right)' -
    COMBINE 'SW' 0.75 'LL' 0.75 'WLRX' 0.75
LOADING COMBINATION 'C' '0.75(SW + Live Load + Wind Load from Left)' -
    COMBINE 'SW' 0.75 'LL' 0.75 'WLLX' 0.75
LOADING COMBINATION 'D' '0.75(SW + Live Load + Wind Load from Back)' -
    COMBINE 'SW' 0.75 'LL' 0.75 'WLBZ' 0.75
LOADING COMBINATION 'E' '0.75(SW + Live Load + Wind Load from Front)' -
    COMBINE 'SW' 0.75 'LL' 0.75 'WLFZ' 0.75
```

\$ Service load combinations for deflection check

```
LOADING COMBINATION 'Def-F' 'SW + Live Load + Wind Load from Right' -
    COMBINE 'SW' 1.0 'LL' 1.0 'WLRX' 1.0
LOADING COMBINATION 'Def-G' 'SW + Live Load + Wind Load from Left' -
    COMBINE 'SW' 1.0 'LL' 1.0 'WLLX' 1.0
LOADING COMBINATION 'Def-H' 'SW + Live Load + Wind Load from Back' -
    COMBINE 'SW' 1.0 'LL' 1.0 'WLBZ' 1.0
```

```
LOADING COMBINATION 'Def-I' 'SW + Live Load + Wind Load from Front' -
COMBINE 'SW' 1.0 'LL' 1.0 'WLFZ' 1.0

....
....
....

PARAMETERS
CODE          ASD9          ALL MEMBERS
STEELGRD      A572-G50      ALL MEMBERS
DefCheck      YES-Y         MEMBERS 5 TO 8      $ Check Y direction deflection
DefLoads      YES           LOADS 'A' 'Def-F' 'Def-G' -
                                'Def-H' 'Def-I'      $ Deflection loads

$ Activate loads for code check

LOAD LIST 'A' 'B' 'C' 'D' 'E'

CHECK ALL MEMBERS
```

This page intentionally left blank.

5.2.7 Brazilian Table**Brazilian Standard Tables, NBR 5884 2000**

| | |
|-----|---|
| CS | I shapes from Brazilian Standard, ABNT, NBR 5884:2000 |
| CVS | I shapes from Brazilian Standard, ABNT, NBR 5884:2000 |
| VS | I shapes from Brazilian Standard, ABNT, NBR 5884:2000 |

The above tables are from Brazilian Standard, Perfil I estrutural de aço soldado por arco elétrico - Especificação, ABNT, NBR 5884:2000. Brazilian Standard, Structural profile type I welded steel joint by electric arc - Specification, ABNT, NBR 5884:2000.

The profiles which are available in the above tables are shown on the pages which follow.

Table CS

Profile Names

| | | | | |
|-----------|-----------|-----------|-----------|-----------|
| CS150x25 | CS300x122 | CS450x154 | CS550x290 | CS650x409 |
| CS150x29 | CS300x131 | CS450x165 | CS550x345 | CS650x425 |
| CS150x31 | CS300x138 | CS450x175 | CS550x358 | CS650x437 |
| CS150x37 | CS300x149 | CS450x188 | CS550x368 | CS650x468 |
| CS150x45 | CS350x89 | CS450x198 | CS550x395 | CS650x484 |
| CS200x29 | CS350x93 | CS450x209 | CS550x407 | CS650x496 |
| CS200x34 | CS350x108 | CS450x216 | CS550x417 | CS650x525 |
| CS200x39 | CS350x112 | CS450x227 | CS550x441 | CS650x593 |
| CS200x41 | CS350x119 | CS450x236 | CS550x498 | CS700x389 |
| CS200x50 | CS350x128 | CS450x280 | CS600x250 | CS700x426 |
| CS200x61 | CS350x135 | CS450x291 | CS600x281 | CS700x441 |
| CS250x43 | CS350x144 | CS450x321 | CS600x294 | CS700x458 |
| CS250x49 | CS350x153 | CS450x331 | CS600x305 | CS700x471 |
| CS250x52 | CS350x161 | CS500x172 | CS600x318 | CS700x505 |
| CS250x63 | CS350x175 | CS500x195 | CS600x332 | CS700x522 |
| CS250x66 | CS350x182 | CS500x207 | CS600x377 | CS700x535 |
| CS250x76 | CS350x216 | CS500x221 | CS600x391 | CS700x567 |
| CS250x79 | CS400x106 | CS500x233 | CS600x402 | CS700x640 |
| CS250x84 | CS400x128 | CS500x253 | CS600x432 | CS750x417 |
| CS250x90 | CS400x137 | CS500x263 | CS600x446 | CS750x457 |
| CS250x95 | CS400x146 | CS500x312 | CS600x456 | CS750x473 |
| CS250x108 | CS400x155 | CS500x324 | CS600x483 | CS750x492 |
| CS300x62 | CS400x165 | CS500x333 | CS600x546 | CS750x506 |
| CS300x76 | CS400x176 | CS500x369 | CS650x305 | CS750x542 |
| CS300x92 | CS400x185 | CS500x378 | CS650x319 | CS750x560 |
| CS300x95 | CS400x201 | CS550x228 | CS650x330 | CS750x574 |
| CS300x102 | CS400x209 | CS550x257 | CS650x345 | CS750x608 |
| CS300x109 | CS400x248 | CS550x269 | CS650x361 | CS750x687 |
| CS300x115 | CS450x144 | CS550x279 | CS650x395 | |

Table CVS

Profile Names

| | | | | |
|-----------|------------|------------|------------|-------------|
| CVS150X15 | CVS300X55 | CVS450X177 | CVS550X361 | CVS700X327 |
| CVS150X18 | CVS300X66 | CVS450X188 | CVS550X370 | CVS700X342 |
| CVS150X20 | CVS300X80 | CVS450X206 | CVS600X156 | CVS750X284 |
| CVS150X22 | CVS300X83 | CVS450X216 | CVS600X190 | CVS750X301 |
| CVS150X24 | CVS300X94 | CVS500X123 | CVS600X210 | CVS750X334 |
| CVS200X21 | CVS300X100 | CVS500X134 | CVS600X226 | CVS750X350 |
| CVS200X24 | CVS300X113 | CVS500X150 | CVS600X239 | CVS800X288 |
| CVS200X28 | CVS350X73 | CVS500X162 | CVS600X278 | CVS800X310 |
| CVS200X27 | CVS350X87 | CVS500X180 | CVS600X292 | CVS800X328 |
| CVS200X30 | CVS350X98 | CVS500X194 | CVS600X328 | CVS800X365 |
| CVS200X36 | CVS350X105 | CVS500X204 | CVS600X339 | CVS800X382 |
| CVS200X38 | CVS350X118 | CVS500X217 | CVS600X369 | CVS850X336 |
| CVS200X46 | CVS350X128 | CVS500X238 | CVS650X211 | CVS850X355 |
| CVS250X30 | CVS350X136 | CVS500X250 | CVS650X234 | CVS850X396 |
| CVS250X33 | CVS400X82 | CVS500X259 | CVS650X252 | CVS850X414 |
| CVS250X40 | CVS400X87 | CVS500X281 | CVS650X266 | CVS900X342 |
| CVS250X47 | CVS400X103 | CVS500X317 | CVS650X282 | CVS900X362 |
| CVS250X56 | CVS400X116 | CVS550X184 | CVS650X310 | CVS900X402 |
| CVS250X64 | CVS400X125 | CVS550X204 | CVS650X326 | CVS900X422 |
| CVS250X72 | CVS400X140 | CVS550X220 | CVS650X351 | CVS950X368 |
| CVS300X47 | CVS400X152 | CVS550X232 | CVS650X366 | CVS950X389 |
| CVS300X57 | CVS400X162 | CVS550X245 | CVS650X413 | CVS950X433 |
| CVS300X67 | CVS450X116 | CVS500X270 | CVS650X461 | CVS950X454 |
| CVS300X70 | CVS450X130 | CVS550X283 | CVS700X214 | CVS1000X394 |
| CVS300X79 | CVS450X141 | CVS550X293 | CVS700X232 | CVS1000X416 |
| CVS300X85 | CVS450X156 | CVS550X319 | CVS700X278 | CVS1000X464 |
| CVS300X95 | CVS450X168 | CVS550X329 | CVS700X293 | CVS1000X486 |

Table VS

Profile Names

| | | | | |
|-----------|-----------|-----------|------------|------------|
| VS150x15 | VS350x28 | VS550x88 | VS900x142 | VS1400x478 |
| VS150x18 | VS350x33B | VS550x100 | VS900x159 | VS1500x270 |
| VS150x20 | VS350x36 | VS600x81 | VS900x177 | VS1500x293 |
| VS150x19 | VS350x30B | VS600x95 | VS900x191 | VS1500x319 |
| VS150x21 | VS350x35 | VS600x111 | VS950x127 | VS1500x339 |
| VS200x19 | VS350x39 | VS600x125 | VS950x146 | VS1500x388 |
| VS200x22 | VS350x38 | VS600x140 | VS950x162 | VS1500x434 |
| VS200x25 | VS350x42 | VS600x152 | VS950x180 | VS1500x488 |
| VS200x20 | VS350x51 | VS650x84 | VS950x194 | VS1600x328 |
| VS200x23 | VS400x28 | VS650x98 | VS1000x140 | VS1600x348 |
| VS200x26 | VS400x32 | VS650x114 | VS1000x161 | VS1600x398 |
| VS250x21 | VS400x35 | VS650x128 | VS1000x180 | VS1600x444 |
| VS250x24 | VS400x30 | VS650x143 | VS1000x201 | VS1600x498 |
| VS250x27 | VS400x34 | VS650x155 | VS1000x217 | VS1700x338 |
| VS250x23 | VS400x38 | VS700x105 | VS1100x159 | VS1700x358 |
| VS250x26 | VS400x32B | VS700x122 | VS1100x180 | VS1700x408 |
| VS250x30 | VS400x37 | VS700x137 | VS1100x199 | VS1700x454 |
| VS250x25 | VS400x41 | VS700x154 | VS1100x219 | VS1700x507 |
| VS250x29 | VS400x39 | VS700x166 | VS1100x235 | VS1800x348 |
| VS250x32 | VS400x44 | VS750x108 | VS1200x200 | VS1800x368 |
| VS300x23 | VS400x53 | VS750x125 | VS1200x221 | VS1800x418 |
| VS300x26 | VS450x51 | VS750x140 | VS1200x244 | VS1800x464 |
| VS300x28 | VS450x60 | VS750x157 | VS1200x262 | VS1800x517 |
| VS300x25 | VS450x71 | VS750x170 | VS1200x307 | VS1800x465 |
| VS300x28B | VS450x80 | VS800x111 | VS1300x237 | VS1800x511 |
| VS300x31 | VS450x59 | VS800x129 | VS1300x258 | VS1800x564 |
| VS300x27 | VS450x70 | VS800x143 | VS1300x281 | VS1900x429 |
| VS300x31B | VS450x83 | VS800x160 | VS1300x299 | VS1900x478 |
| VS300x34 | VS450x95 | VS800x173 | VS1300x344 | VS1900x524 |
| VS300x33 | VS500x61 | VS850x120 | VS1400x260 | VS1900x577 |
| VS300x37 | VS500x73 | VS850x139 | VS1400x283 | VS2000x461 |
| VS300x46 | VS500x86 | VS850x155 | VS1400x309 | VS2000x515 |
| VS350x26 | VS500x97 | VS850x174 | VS1400x329 | VS2000x566 |
| VS350x30 | VS550x64 | VS850x188 | VS1400x378 | VS2000x624 |
| VS350x33 | VS550x75 | VS900x124 | VS1400x424 | |

5.2.8 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.

$$\begin{array}{c}
 \text{METHOD} \left(\left\{ \begin{array}{l} \rightarrow \text{ULTIMATE (STRENGTH)} \\ \text{WORKING (STRESS)} \end{array} \right\} \right) \left\{ \begin{array}{l} \text{ACI318-99} \\ \text{ACI318-89} \\ \text{ACI318-83} \\ \text{ACI318-77} \\ \text{ACI318-63} \\ \text{(BSI) CP110-72} \\ \text{(BSI) BS8110} \end{array} \right\} - \\
 \\
 \left(\text{BARS} \left\{ \begin{array}{l} \rightarrow \text{ASTM} \\ \text{CANADIAN (STANDARD)} \\ \text{UNESCO} \\ \text{KOREAN (STANDARD)} \end{array} \right\} \right) \left(\left\{ \begin{array}{l} \rightarrow \text{NONSEISMIC} \\ \text{SEISMIC} \\ \text{MODERATE SEISMIC} \end{array} \right\} \right)
 \end{array}$$

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) \sqrt{FCP} |
| EU | Ult. strain in concrete at extreme comp. fiber | 10.2.3 | 0.003 |

Notes:

1. The constant 'DENSITY' is the GTSTRUDL constant of the same name which has been set to a value of 150 pcf for reinforced concrete.
2. VU is multiplied by PHISH internally.
3. Calculations for V_c and T_c are modified by replacing $\sqrt{f'_c}$ with $RFSP/6.7(\sqrt{f'_c})$ as per Section 11.2.1.1.
4. The assumed value of FSC is also limited to 30,000 psi maximum.
5. This value is defined only at the time of stirrup design.

This page intentionally left blank.

5.2.9 Rectangular and Circular Concrete Cross-Section Tables

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

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

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

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

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

| | | | | | |
|----------|----------|----------|----------|----------|----------|
| REC18X12 | REC20X12 | REC22X12 | REC24X12 | REC26X12 | REC28X12 |
| REC18X14 | REC20X14 | REC22X14 | REC24X14 | REC26X14 | REC28X14 |

Design Prerelease Features

GT STRUDL

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

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

5.2.10 ASD9-E Code

A special Ninth Edition AISC allowable stress design code for W shapes has been implemented. The code name is ASD9-E. This code is based on the Ninth Edition AISC ASD except the equations have been modified to include modulus of elasticity (constant E). ASD9-E is applicable to W shapes only. This code is useful for structures where E and possibly other material data must be modified to account for high temperature. Parameters for the ASD9-E Code are shown below:

Table ASD9-E.1-1

ASD9-E Code Parameters

| <u>Parameter Name</u> | <u>Default Value</u> | <u>Meaning</u> |
|----------------------------------|---------------------------------|--|
| CODE | Required | Identifies the code to be used for member checking or member selection. Specify ASD9-E for code name. |
| TBLNAM | WSHAPES9 | Identifies the table of profiles to be used during selection. See Table ASD9-E.1-2 for choices. |
| CODETOL | 0.0 | Percent variance from 1.0 for compliance with the provisions of a code. The ratio of Actual/Allowable must be less than or equal to $[1.0 + \text{CODETOL}/100]$. |
| PF | 1.0 | Area reduction factor for holesout in members subject to axial tension. |
| A/H | 10000.0 | Ratio of clear span between transverse stiffeners to clear distance between flanges. Used in computing allowable shear stress. Default approximates infinity. |
| ALSTRINC | 0.0 | Allowable stress increase value. This parameter can be used to specify the 1/3 allowable stress increase for the wind or seismic loads. The user specified value for this parameter must be followed by the load list. An example for this parameter is to specify a value of 33.3333 followed by a load list. |

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

| Parameter Name | Default Value | Meaning |
|----------------------------|--------------------------|---|
| <u>Material Properties</u> | | |
| STEELGRD | A36 | Identifies the grade of steel from which a member is made. See Table ASD9-E.1-3 for steel grades and their properties. |
| FYLD | Computed | Yield stress of member. Computed from STEELGRD if not given. |
| FTS | Computed | Minimum tensile strength of member. Computed from STEELGRD if not given. |
| REDFYLD | 1.0 | Reduction factor for FYLD. This factor times FYLD gives the FY value used by the code. Used to account for property changes at high temperatures. |
| REDFTS | 1.0 | Reduction factor for FTS. Similar to REDFYLD. |
| REDE | 1.0 | Reduction factor for E, the modulus of elasticity. Similar to REDFYLD. |

Slenderness Ratio

| | | |
|----------|----------|--|
| SLENCOMP | Computed | Maximum permissible slenderness ratio (KL/r) for member subjected to axial compression. When no value is specified for this parameter, the value of 200 is used for the maximum slenderness ratio. |
| SLENTEN | Computed | Maximum permissible slenderness ratio (L/r) for member subjected to axial tension. When no value is specified for this parameter, the value of 300 is used for the maximum slenderness ratio. |

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|---|
| <u>K-Factors</u> | | |
| COMPCK | NO | <p>Parameter to request the computation of the effective length factors KY and KZ (Sections 2.2 and 2.3 of Volume 2A).</p> <p>YES = Compute KY and KZ factors. See the COLUMN/BEAM LINE command (Section 2.3 of Volume 2A).</p> <p>KY = Compute KY only.</p> <p>KZ = Compute KZ only.</p> <p>NO = Use default or specified values for KY and KZ.</p> |
| KY | 1.0 | Effective length factor for buckling about the local Y axis of the profile. See Sections 2.2 and 2.3 of Volume 2A for GTSTRUDL computation of effective length factor, KY. |
| KZ | 1.0 | Effective length factor for buckling about the local Z axis of the profile. See Sections 2.2 and 2.3 of Volume 2A for GTSTRUDL computation of effective length factor, KZ. |
| Print-K | YES | <p>Parameter to print the computed K-factor values after the default code check or select command output (TRACE 4 output). The default value of 'YES' for this parameter indicates that the computed K-factor values should be printed after the code check or select command output. The column names attached to the start and end of the code checked member is also printed. This printed information allows the user to inspect the automatic detection of the columns attached to the start and end of the designed member. A value of 'NO' indicates that K-factor values and the names of the attached columns to the start and end of the designed member should not be printed.</p> |

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

| Parameter <u>Name</u> | Default <u>Value</u> | <u>Meaning</u> |
|----------------------------------|---------------------------------|---|
| <u>K-Factors</u> (continued) | | |
| SDSWAYY | YES | Indicates the presence or absence of sidesway about the local Y axis. YES = sidesway permitted. NO = sidesway prevented. |
| SDSWAYZ | YES | Indicates the presence or absence of sidesway about the local Z axis. YES = sidesway permitted. NO = sidesway prevented. |
| GAY | Computed | G-factor at the start joint of the member. GAY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A). |
| GAZ | Computed | G-factor at the start joint of the member. GAZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A). |
| GBY | Computed | G-factor at the end joint of the member. GBY is used in the calculation of effective length factor KY (see parameter COMPK, KY, and Sections 2.2 and 2.3 of Volume 2A). |
| GBZ | Computed | G-factor at the end joint of the member. GBZ is used in the calculation of effective length factor KZ (see parameter COMPK, KZ, and Sections 2.2 and 2.3 of Volume 2A). |

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

| Parameter Name | Default Value | Meaning |
|------------------------------------|--------------------------|---|
| <u>Buckling Length</u> | | |
| LY | Computed | Unbraced length for buckling about the local Y axis of the profile. Computed as length of member. |
| LZ | Computed | Unbraced length for buckling about the local Z axis of the profile. Computed as length of member. |
| FRLY | 1.0 | Fractional form of the parameter LY, allows unbraced length to be specified as fractions of the total length. Used only when LY is computed. |
| FRLZ | 1.0 | Fractional form of the parameter LZ, similar to FRLY. Used only when LZ is computed. |
| <u>Flexural-Torsional Buckling</u> | | |
| FLTORBUK | YES | Indicates the consideration of flexural-torsional buckling check. YES = check flexural-torsional buckling. NO = do not check flexural-torsional buckling. |
| KX | 1.0 | Effective length factor for torsional buckling about the local X axis of the profile. This parameter is used in flexural-torsional buckling stress, F_e computations. |
| LX | Computed | Unbraced length for torsional buckling about the local X axis of the profile. Computed as length of member. This parameter is used in flexural-torsional buckling stress, F_e computations. |

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

| Parameter Name | Default Value | Meaning |
|--|--------------------------|---|
| <u>Flexural - Torsional Buckling</u> (continued) | | |
| FRLX | 1.0 | Fractional form of the parameter LX, allows unbraced length to be specified as fractions of the total length. Used only when LX is computed. |
| <u>Bending Stress</u> | | |
| CB | Computed | Coefficient used in computing allowable compressive bending stress (AISC ASD Ninth Ed. Section F1.3). |
| UNLCF | Computed | Unbraced length of the compression flange. Computed as length of member. In this parameter no distinction is made between the unbraced length for the top or bottom flange. See UNLCFTF or UNLCFBF. |
| FRUNLCF | 1.0 | Fractional form of the parameter UNLCF, allows unbraced length to be specified as fractions of the total length. Used only when UNLCF is computed. |
| UNLCFTF | Computed | Unbraced length of the compression flange for the top flange. When no value is specified, UNLCF and FRUNLCF is used for this parameter. |
| UNLCFBF | Computed | Unbraced length of the compression flange for the bottom flange. When no value is specified, UNLCF and FRUNLCF is used for this parameter. |

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

| Parameter Name | Default Value | Meaning |
|---------------------------|--------------------------|---|
| <u>Combined Stresses</u> | | |
| AXEFF | 0.0 | Axial stress reduction factor indicating the amount of the axial stress which is to be deducted from a corresponding bending stress acting in the opposite direction. |
| CMY | Computed | Coefficient which modifies Y axis bending stress in interaction equation (AISC ASD Ninth Ed. Section H1). |
| CMZ | Computed | Coefficient which modifies Z axis bending stress in interaction equation (AISC ASD Ninth Ed. Section H1). |
| <u>Force Limitation</u> | | |
| FXMIN | 0.5(lb) | Minimum axial force to be considered by the code; anything less in magnitude is taken as zero. |
| FYMIN | 0.5(lb) | Minimum Y-shear force to be considered by the code; anything less in magnitude is taken as zero. |
| FZMIN | 0.5(lb) | Minimum Z-shear force to be considered by the code; anything less in magnitude is taken as zero. |
| MYMIN | 20.0(in-lb) | Minimum Y-bending moment to be considered by the code; anything less in magnitude is taken as zero. |
| MZMIN | 20.0(in-lb) | Minimum Z-bending moment to be considered by the code; anything less in magnitude is taken as zero. |

Table ASD9-E.1-1 (continued)

ASD9-E Code Parameters

| <u>Parameter Name</u> | <u>Default Value</u> | <u>Meaning</u> |
|--|---------------------------------|---|
| <u>Output Processing and System Parameters</u> | | |
| MXTRIALS | 500.0 | Maximum number of profiles to be tried when designing a member. Default is larger than the number of profiles in most tables. |
| SUMMARY | NO | Indicates if 'SUMMARY' information is to be saved for the member. Choices are YES or NO; See Sections 2.9 and 7.2 of Volume 2A for explanation. |
| PrintStr | NO | Parameter to request to print the section stress values for allowable stress design codes. The default output from CHECK or SELECT command prints the section force values. A value of 'YES' for this parameter indicates that the section stress values should be printed instead of default section forces. |
| TRACE | 4.0 | Flag indicating when checks of code provisions should be output during design or code checking. See Section 7.2 of Volume 2A for explanation. 1 = never 2 = on failure 3 = all checks 4 = controlling Actual/Allowable values and section forces. |
| VALUES | 1.0 | Flag indicating if parameter or property values are to be output when retrieved. See Section 7.2 of Volume 2A for explanation. 1 = no output 2 = output parameters 3 = output properties 4 = output parameters and properties. |

GTSTRUDL I shape Profile Tables for the Design based on the ASD9-E Code*

(I shapes, Universal Beams, Universal Columns, Joists, Piles, etc.)

AISC Tables (American Institute of Steel Construction) (continued)

| <u>Table Name</u> | <u>Reference</u> |
|---|---|
| AISC Tables (American Institute of Steel Construction) | |
| W-LRFD3 | W shapes from 1999 AISC LRFD Third Edition (96) |
| M/S/HPL3 | M, S, and HP shape profiles from 1999 AISC LRFD Third Edition (96) |
| WSHAPES9 | W shapes from 1989 AISC ASD Ninth Edition (72) |
| M/S/HP9 | M, S, and HP shapes from 1989 AISC ASD Ninth Edition (72) |
| WBEAM9 | W shapes commonly used as beams from 1989 AISC ASD Ninth Edition (72) |
| WCOLUMN9 | W shapes commonly used as columns from 1989 AISC ASD Ninth Edition (72) |
| WSHAPESM | W shape from AISC Metric “WSHAPES” table (83) |
| M/S/HPM | M, S, and HP shape profiles from AISC Metric “M SHAPES, S SHAPES, and HP SHAPES” table (83) |
| WBEAMM | W shape profiles commonly used as beams from AISC Metric “WSHAPES” table (83) |
| WCOLUMNM | W shape profiles commonly used as columns from AISC Metric “WSHAPES” table (83) |
| STEELW78 | W shapes from 1978 AISC ASD Eighth Edition (33) |
| HP/S/M | HP, S, and M shapes from 1978 AISC ASD Eighth Edition (33) |
| W78BEAM | W shapes commonly used as beams from 1978 AISC ASD Eighth Edition (33) |
| W78COLUM | W shapes commonly used as columns from 1978 AISC ASD Eighth Edition (33) |
| STEELW | W shapes from 1969 AISC ASD Seventh Edition (16) |
| WCOLUMN | W shapes commonly used as columns from 1969 AISC ASD Seventh Edition (16) |

*See design code for applicable cross-sections

*See Appendix C of Volume 2A for Table description and profile names

Table ASD9-E.1-2 (continued)

GTSTRUDL I shape Profile Tables for the
Design based on the ASD9-E Code*
(I shapes, Universal Beams, Universal Columns, Joists, Piles, etc.)

| <u>Table Name</u> | <u>Reference</u> |
|-------------------|------------------|
|-------------------|------------------|

Brazilian Standard Tables, NBR 5884 2000

| | |
|-----|---|
| CS | I shapes from Brazilian Standard, ABNT, NBR 5884:2000 |
| CVS | I shapes from Brazilian Standard, ABNT, NBR 5884:2000 |
| VS | I shapes from Brazilian Standard, ABNT, NBR 5884:2000 |

British Standard Tables, BS 5950

| | |
|----------|--|
| UNIBEAMS | British Universal Beam profiles from 1996 BS 5950 Section Properties, 4th Edition (82) |
| UNICOL | British Universal Column profiles from 1996 BS 5950 Section Properties, 4th Edition (82) |
| JOISTS | British Joist profiles from 1996 BS 5950 Section Properties, 4th Edition (82) |
| UBPILES | I shape profiles from British "UNIVERSAL BEARING PILES" table (82) |

European Tables

| | |
|-----|---|
| HEA | H shaped (HE-A) profiles from Breite I-Träger, Reihe HE-A. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997" |
| HEB | H shaped (HE-B) profiles from Breite I-Träger, Reihe HE-B. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997" |
| HEM | H shaped (HE-M) profiles from Breite I-Träger, Reihe HE-M. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997" |

*See design code for applicable cross-sections

*See Appendix C of Volume 2A for Table description and profile names

Table ASD9-E.1-2 (continued)

GTSTRUDL I shape Profile Tables for the
Design based on the ASD9-E Code*
(I shapes, Universal Beams, Universal Columns, Joists, Piles, etc.)

| <u>Table Name</u> | <u>Reference</u> |
|-------------------|------------------|
|-------------------|------------------|

European Tables (Continued)

| | |
|----------|---|
| IPE | I shaped (IPE) profiles from Mittelbreite I-Träger, IPE-Reihe. The profiles are from "STAHLBAU-PROFILE, 21., neu bearbeitete und erweiterte Auflage, überarbeiteter Nachdruck 1997" |
| EUROPEAN | This table contains profiles from IPE, HEA, HEB, and HEM tables |

Indian Standard Tables from IS808:1989

| | |
|----------|--|
| ISBEAMS | I shape beam sections (medium flange beams, junior and light weight beams) from Tables 2.1 and 2.2 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision (93) |
| ISCOLUMN | I shape column/heavy weight beam sections (column and heavy weight beams) from Table 3.1 of the Indian Standard IS 808:1989, DIMENSIONS FOR HOT ROLLED STEEL BEAM, COLUMN, CHANNEL AND ANGLE SECTIONS, Third Revision (93) |

*See design code for applicable cross-sections

*See Appendix C of Volume 2A for Table description and profile names

This page intentionally left blank.

5.3 Analysis Prerelease Features

5.3.1 The CALCULATE ERROR ESTIMATE Command

The form of the command is as follows:

$$\begin{aligned} & \text{CALCULATE ERROR (ESTIMATE) (BASED ON) -} \\ & * \left\{ \begin{array}{l} \text{ENERGY (NORM)} \\ \text{MAX DIFFERENCE} \\ \text{DIFFERENCE FROM AVERAGE} \\ \text{PERCENT MAX DIFFERENCE} \\ \text{PERCENT DIFFERENCE FROM AVERAGE} \\ \text{NORMALIZED PERCENT MAX DIFFERENCE} \\ \text{NORMALIZED PERCENT DIFFERENCE FROM AVERAGE} \end{array} \right\} - \\ & (\text{AT}) * \left\{ \begin{array}{l} \text{TOP} \\ \text{MIDDLE} \\ \text{BOTTOM} \end{array} \right\} (\text{SURFACES}) (\text{FOR}) \left\{ \begin{array}{l} \rightarrow \text{ALL} \\ \text{ELEMENT list} \end{array} \right\} \end{aligned}$$

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:

$$\|e_s\|_{L_2} = \left(\int_{\Omega} (e_s)^T (e_s) d\Omega \right)^{1/2}$$

where e_s 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,

$$\|e_s\|_{L_2} = \left(\int_{\Omega} (\sigma^* - \sigma)^T N^T \cdot N (\sigma^* - \sigma) d\Omega \right)^{1/2}$$

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:

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

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

$$\mathbf{h} = \frac{\|\mathbf{e}_s\|}{\|\mathbf{s}\| + \|\mathbf{e}_s\|} \times 100$$

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

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

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

Difference from Average Method

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

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

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

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

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

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 Output of Response Spectrum Results

The LIST RESPONSE SPECTRUM command has been modified to include the output of BASE/STORY shear output. The section is numbered below as it is currently numbered in Volume 3 of the GTSTRUDL User Reference Manual.

2.4.6.7 Output of Response Spectrum Results

General form:

LIST RESPONSE SPECTRUM (* [PARTICIPATION (FACTORS)
SPECTRAL DISPLACEMENTS
SPECTRAL VELOCITIES
SPECTRAL ACCELERATIONS
(MODAL) COEFFICIENTS
[BASE]
[STORY] (SHEAR)]) -

* [DISPLACEMENTS
VELOCITIES
ACCELERATIONS
FORCES
LOADS
REACTIONS] -

((MODAL) COMBINATIONS * [RMS
ABS
PRMS
CQC
(NRC)TPM
(NRC)GRP
(NRC)DSM
ALL]) (list)

Elements:

list = see Section 2.4.6.5

Explanation:

This command is used to output results associated with active RESPONSE SPECTRA loading conditions (Section 2.4.4.6). The requested results must have been previously computed via the DYNAMIC ANALYSIS command (Section 2.4.5.4), the split solver commands (Section 2.4.5.5) or the COMPUTE RESPONSE SPECTRA command (Section 2.4.5.9).

Participation factors, spectral displacements, velocites, accelerations and modal coefficients are defined in Section 2.4.2.5. The participation factor output via the LIST DYNAMIC PART FACTOR command is a normalized value expressed as a percentage, while the value output via this command is the actual factor as computed in Equation 5-6 in Section 2.4.2.5.

The BASE/STORY SHEAR option lists the global X, Y, and Z components of the total response spectrum inertia forces computed for each active mode and summed over the joints specified in the list. Modal combinations are also computed and listed if specified. These results are computed and listed completely for each active response spectrum loading condition.

Finite element forces, generalized stresses and strains can be computed but cannot be output via the LIST RESPONSE SPECTRA command. However, these finite element results may be copied into static loading conditions via the CREATE PSEUDO STATIC LOADING command (Section 2.4.7) and then output via the LIST command (Section 2.1.14.4). Member section forces and stresses can be listed via a similar approach by using the CREATE PSEUDO STATIC LOAD and LIST SECTION commands (Section 2.1.14.6).

Other commands which affect LIST RESPONSE SPECTRA are as follows:

OUTPUT MODAL CONTRIBUTIONS (Section 2.4.6.3)

OUTPUT DECIMAL (Section 2.1.14.3)

OUTPUT FIELD (Section 2.1.14.3)

Examples:

- (1) LIST RESPONSE SPECTRA DISPL MODAL COMBINATIONS ALL. This command outputs previously computed peak displacements for all joints and all of the modal combination techniques.
- (2) COMPUTE RESPONSE SPECTRA STRESSES FORCES MODAL -
COMB ALL MEM 1 TO 10 OUTPUT MODAL CONTRIBUTIONS ON
LIST RESPONSE SPECTRA FORCES MODE COMB RMS CQC -
MEM 1 TO 10

The output generated by the LIST RESPONSE SPECTRA command consists of the modal contributions to member end forces and the RMS and CQC combinations of those contributions. Non-zero results will exist only for members 1 to 10.

- (3) LIST RESPONSE SPECTRA PARTICIPATION FACTORS SPECTRAL -
DISPLACEMENTS MODAL COEFFICIENT

This command lists the participation factors, spectral displacements and the modal coefficients associated with all active response spectra loadings.

- (4) LIST RESPONSE SPECTRUM STORY SHEARS -
MODE COMBINATION RMS CQC JOINTS 101 TO 140

This command lists the total response spectrum inertia forces, including the RMS and CQC modal combinations, summed over joints 101 to 140. If joints 101 through 140 represent all joints above the fifth floor of a structure, then the listed results may be interpreted as the modal and total story shear applied above the fifth floor.

This page intentionally left blank.

5.3.3 FORM STATIC LOAD Command -- Automatic Generation of Static Equivalent Earthquake Loads

General form:

$$\begin{array}{l} \text{FORM STATIC (EARTHQUAKE) LOAD } \left\{ \begin{array}{l} \text{'a}_{sl}\text{' } \\ \text{i}_{sl} \end{array} \right\} \text{ ('title}_{sl}\text{') -} \\ \text{FROM } \left\{ \begin{array}{l} \text{MASS [X] v}_x \text{ [Y] v}_y \text{ [Z] v}_z \\ \left[\begin{array}{l} \rightarrow \text{RMS} \\ \text{CQC} \\ \text{SUM} \end{array} \right] \end{array} \right\} \text{ (OF RESPONSE SPECTRUM) LOAD } \left\{ \begin{array}{l} \text{'a}_{RS}\text{' } \\ \text{i}_{RS} \end{array} \right\} \text{ (FACTOR v}_{RS}\text{) } \end{array}$$

Elements:

- 'a_{sl}'/i_{sl} = alphanumeric or integer name for the generated static earthquake load. This name must be unique among all current loading names and is limited to eight characters or digits.
- 'title_{sl}' = optional static load title of up to 64 characters in length.
- v_x = scaling factor for the mass load in the global X direction. v_x is taken as 0.0 by default.
- v_y = scaling factor for the mass load in the global Y direction. v_y is taken as 0.0 by default.
- v_z = scaling factor for the mass load in the global Z direction. v_z is taken as 0.0 by default.
- 'a_{RS}'/i_{RS} = name of the response spectrum load to be used for the calculation of static earthquake load 'a_{sl}'/i_{sl}.
- v_{RS} = scaling factor to be applied to the response spectrum static earthquake load 'a_{sl}'/i_{sl}.

Explanation:

The FORM STATIC LOAD command is used to compute an independent loading condition consisting of a static joint load representation of either the structural mass or a response spectrum load. The mass and response spectrum load options are described in greater detail as follows:

$$\text{MASS [X] v}_x \text{ [Y] v}_y \text{ [Z] v}_z$$

The MASS option generates an independent loading condition containing joint loads which are statically equivalent to any factored combination of the structural mass in the global X, Y, and Z directions. Because the global direction scaling factors v_x , v_y , and v_z are taken as 0.0 by default, it is necessary to specify a non-zero value for the appropriate scaling factor if joint load components are to be computed for a particular global direction.

The MASS-equivalent static joint load vector is computed by the following equation:

$$\{F_{EM}\} = [M] \{v_{XYZ}\} g \quad \text{Eq. 5.3.3-1}$$

where,

$$\begin{aligned} \{F_{EM}\} &= \text{MASS-equivalent joint load vector,} \\ [M] &= \text{system global mass matrix,} \\ \{v_{XYZ}\} &= \text{vector of global direction scaling factors } v_x, v_y, \text{ and } v_z, \\ &\quad \text{arranged in the appropriate joint degree-of-freedom} \\ &\quad \text{locations,} \\ g &= \text{acceleration due to gravity, taken as 386.0886 inches/} \\ &\quad \text{second}^2 \text{ by default.} \end{aligned}$$

According to Equation 5.3.3-1, it is necessary that the structural mass has been defined, and that, as a minimum, the PERFORM ASSEMBLY FOR DYNAMICS command (Section 2.4.5.5.1, Volume 3, GTSTRUDL User Reference Manual) has been executed prior the execution of this option.

The calculation of a MASS-equivalent static load conforms to the NEHRP guidelines for the calculation of the *uniform pattern* lateral load distribution described in Section 3.3.3.2.C of NEHRP Guidelines for the Seismic Rehabilitation of Buildings (FEMA Publication 273).

$$\left\{ \begin{array}{l} \rightarrow \underline{\text{RMS}} \\ \underline{\text{CQC}} \\ \underline{\text{SUM}} \end{array} \right\} (\underline{\text{OF}} \ \underline{\text{RESPONSE}} \ \underline{\text{SPECTRUM}}) \ \underline{\text{LOAD}} \left\{ \begin{array}{l} 'a_{RS}' \\ i_{RS} \end{array} \right\} (\underline{\text{FACTOR}} \ v_{RS})$$

The RESPONSE SPECTRUM LOAD option generates an independent loading condition consisting of joint loads that represent a measure of the total base shear computed for the response spectrum load ' a_{RS} '/ i_{RS} .

The additional RMS, CQC, and SUM options provide for the selection of the modal combination method to be used for the computation of the global joint loads from the modal joint load components. RMS and CQC indicate the Root Mean Square and Complete Quadratic Combination methods, respectively, as described in Section 2.4.2.5, Volume 3 of the GTSTRUDL User Reference Manual. The SUM option indicates a direct algebraic summation of the modal joint load components.

The equivalent response spectrum static joint loading in the i^{th} active mode is computed by the following equation:

$$\{f_{RS}\}_i = -v_{RS} \Gamma_i S_{ai} [M] \{M_i\} \quad \text{Eq. 5.3.3-2}$$

where,

$$\begin{aligned} \{f_{RS}\}_i &= \text{response spectrum static joint load vector for the } i^{\text{th}} \text{ mode,} \\ v_{RS} &= \text{scaling factor as defined above,} \\ [M] &= \text{the global system mass matrix,} \\ \Gamma_i &= \text{the response spectrum participation factor for the } i^{\text{th}} \text{ mode,} \\ S_{ai} &= \text{the response spectrum spectral acceleration for the } i_{\text{th}} \text{ mode,} \\ \{M_i\} &= \text{mode shape displacement vector for the } i^{\text{th}} \text{ mode.} \end{aligned}$$

The total response spectrum static joint load vector is computed by combining the $\{f_{RS}\}_i$ for each active mode using the selected RMS, CQC, or SUM procedure.

Because the values for Γ_i and S_{ai} are determined from the direction and response spectrum data of response spectrum load a_{RS}/i_{RS} , a response spectrum analysis for this load must have been performed prior to the execution of FORM STATIC LOAD RESPONSE SPECTRUM option. However, a COMPUTE RESPONSE SPECTRUM command execution subsequent to the response spectrum analysis is not required.

The calculation of a response spectrum static load conforms to the NEHRP guidelines for the calculation of the *modal pattern* lateral load distribution using a Response Spectrum Analysis as described in Section 3.3.3.2.C of NEHRP Guidelines for the Seismic Rehabilitation of Buildings (FEMA Publication 273).

The independent loading conditions generated by the FORM STATIC LOAD command are conventional independent static loading conditions, and as such, may be used and manipulated in the same manner as independent loads defined by other means.

Errors:

The following messages indicate error or warning conditions that can occur during the execution of the FORM STATIC LOAD command:

```
**** ERROR_STGELL -- System mass matrix does not exist.  
                    SCAN MODE entered.
```

This message indicates that the mass matrix had not been assembled prior to the execution of the FORM STATIC LOAD command. SCAN MODE is set and may be removed by giving the SCAN OFF command. The minimum requirement for the MASS option is that the PERFORM ASSEMBLY FOR DYNAMICS command must be executed.

```
**** ERROR_STGELL -- Specified response spectrum loading 1-G.2 does not  
                    exist. SCAN MODE entered.
```

This error message indicates that the specified response spectrum load has not been defined. SCAN MODE is set and may be removed by giving the SCAN OFF command.

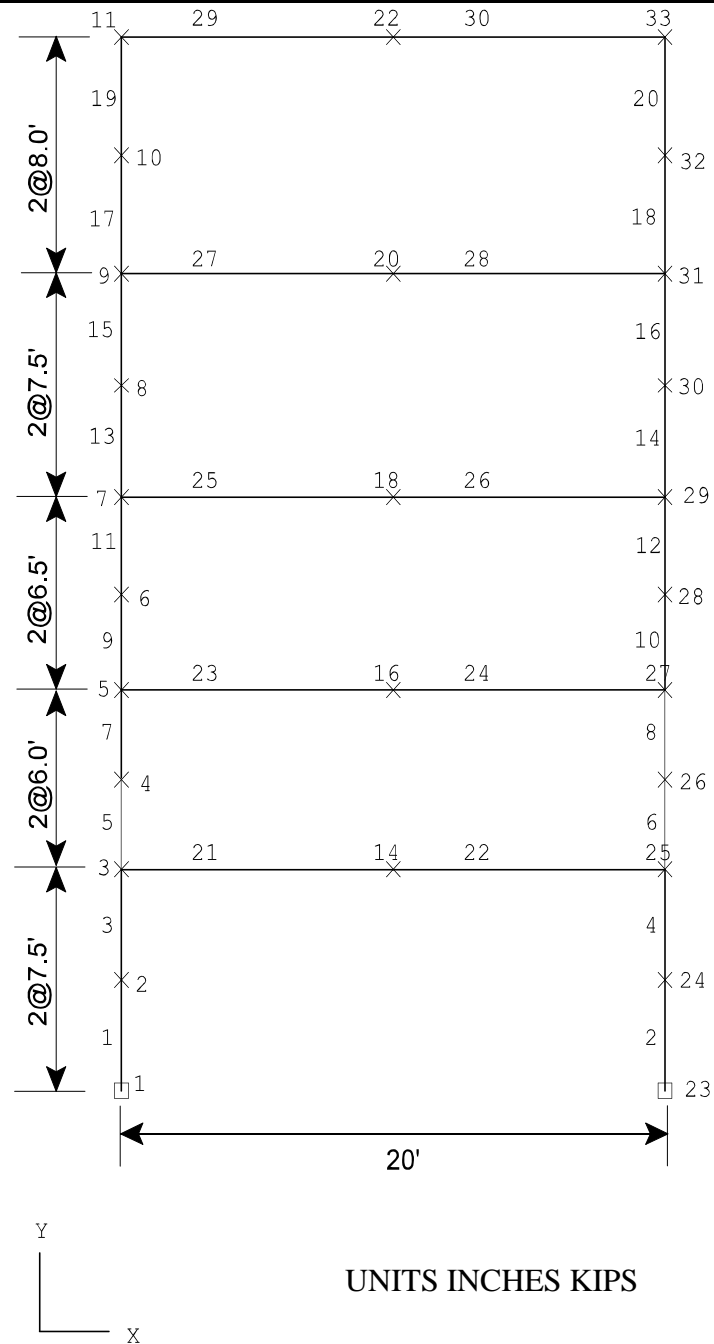
```
**** ERROR_STGELL -- Results do not exist for response spectrum loading  
                    1-G.2. Response spectrum analysis has not yet been  
                    run or the specified loading is not a response  
                    spectrum load. SCAN MODE entered.
```

This error message indicates that while the specified response spectrum load is valid, the required response spectrum analysis for this load has not yet been executed. SCAN MODE is set and may be removed by giving the SCAN OFF command.

Example:

Figure 5.3.3-1 shows the plane frame structure of example SEL-1 which illustrates the use of the FORM STATIC LOAD command to create two static lateral loads based on the structural mass and on a response spectrum load. Note that the structure model includes mid-member joints to insure that the effects of fundamental member modes are not overlooked in the response spectrum analysis. The effects of such modes may arise due to the presence of the added joint inertia at joint 14.

The complete command input for this example is shown in Figure 5.3.3-2.



AX 33.6 IZ 2000.0 Members 1 to 4
 AX 33.6 IZ 1000.0 Members 5 to 20
 AX 240.0 IZ 18000.0 Members 21 to 30

Figure 5.3.3-1 Example SEL-1, Geometry, Structural, and Material Properties

```
STRUDL 'SEL-1' 'Example of static earthquake load generation'
$
$ 4-story plane frame for static earthquake load generation.
$ Geometry.
$
PRINT GEN OFF
UNITS FEET
GEN 11 JOINTS ID 1 1 X 0.0 Y DIFF 0.0 2 AT 7.5 2 AT 6.0 2 AT 6.5 2 AT 7.5 2 AT 8.0
REPEAT 2 TIMES ID 11 X 20.0
TYPE PLANE FRAME
GEN 10 MEMB ID 1 2 FROM 1 1 TO 2 1
REPEAT 1 TIME ID 1 FROM 22 TO 22
GEN 2 MEMB ID 21 1 FROM 3 11 TO 14 11
REPEAT 4 TIMES ID 2 FROM 2 TO 2

DELETIONS
JOINTS 12 13 TO 21 BY 2
ADDITIONS

STATUS SUPPORTS 1 23

$
$ Structural and material properties.
$
UNITS INCHES KIPS
MEMBER PROP PRISMATIC
1 TO 4 AX 33.6 IZ 2000.0
5 TO 20 AX 33.6 IZ 1000.0
21 TO 30 AX 240 IZ 18000

CONSTANTS
E 3000000. ALL
DENSITY 0.00615

$
$ Lumped mass plus added joint masses.
$
INERTIA OF JOINTS LUMPED
INERTIA OF JOINTS MASS
14 TRANSL X 2.0 Y 2.0

$
$ For the static response spectrum load generation, use a
$ constant 1-g acceleration spectrum.
$
```

Figure 5.3.3-2 Command Listing for Example SEL-1

UNITS INCHES SECONDS

STORE RESPONSE SPECTRA ACCEL LIN VS NAT FREQ LIN 'ONE-G'

DAMPING 0.05 FACTOR 386.0886

1.0 0.0 1.0 10000.0

RESPONSE SPECTRA LOAD '1-G.1'

SUPPORT ACCELERATIONS

TRANS X FILE 'ONE-G'

END OF RESPONSE SPECTRA LOADING

DAMPING RATIOS 0.05 100

\$

\$ Perform eigenvalue analysis and response spectrum analysis.

\$

UNITS CYCLES SECS

EIGENPROBLEM PARAMETERS

PRINT MAX

END

ASSEMBLE FOR DYNAMICS

PERFORM EIGENVALUE ANALYSIS

LIST DYNAMIC PARTICIPATION FACTORS

LOAD LIST '1-G.1'

PERF RESPONSE SPECTRUM ANALYSIS

\$

\$ Generate the static earthquake loads and print the joint load contents.

\$

FORM STATIC EARTHQUAKE LOAD 'ERS1-G.1' -

'Equivalent STATIC EARTHQUAKE load 1-G.1, RS load 1-G.1' -

FROM RMS OF RESPONSE SPECTRUM LOAD '1-G.1'

FORM STATIC EARTHQUAKE LOAD 'EM1-G.1' -

'Equivalent STATIC EARTHQUAKE load 1-G.1 from total mass' FROM MASS X 1.0

PRINT APPLIED JOINT LOADS

\$

FINISH

Figure 5.3.3-2 Command Listing for Example SEL-1 (Continued)

Figure 5.3.3-3 contains the text output from the PRINT APPLIED JOINT LOADS command, showing the joint load contents of the loads generated by the FORM STATIC EARTHQUAKE LOAD examples.

```
{ 78} > $
{ 79} > $ Generate the static earthquake loads and print the joint load contents.
{ 80} > $
{ 81} > FORM STATIC EARTHQUAKE LOAD 'ERS1-G.1' 'Equivalent STATIC EARTHQUAKE load 1-G.1, RS load 1-G.1' -
{ 82} >_ FROM RMS OF RESPONSE SPECTRUM LOAD '1-G.1'
```

Time to create equivalent static earthquake load = 0.00 Seconds

```
{ 83} > FORM STATIC EARTHQUAKE LOAD 'EM1-G.1' 'Equivalent STATIC EARTHQUAKE load 1-G.1 from total mass' -
{ 84} >_ FROM MASS X 1.0
```

Time to create equivalent static earthquake load = 0.00 Seconds

```
{ 85} >
{ 86} > PRINT APPLIED JOINT LOADS
```

```
*****
* PROBLEM DATA FROM INTERNAL STORAGE *
*****
```

JOB ID - SEL-1 JOB TITLE - Example of static earthquake load generation

| ACTIVE UNITS - | LENGTH | WEIGHT | ANGLE | TEMPERATURE | TIME |
|----------------|--------|--------|-------|-------------|------|
| | INCH | KIP | CYC | DEGF | SEC |

***** LOADING DATA *****

LOADING - ERS1-G.1 Equivalent STATIC EARTHQUAKE load 1-G.1, RS load 1-G.1 STATUS - ACTIVE

JOINT LOADS-----/

| JOINT | STEP | FORCE X | Y | Z | MOMENT X | Y | Z |
|-------|------|---------|-------|-------|----------|-------|-------|
| 1 | | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 2 | | 9.713 | 0.189 | 0.000 | 0.000 | 0.000 | 0.000 |
| 3 | | 104.472 | 3.722 | 0.000 | 0.000 | 0.000 | 0.017 |
| 4 | | 10.172 | 0.239 | 0.000 | 0.000 | 0.000 | 0.000 |
| 5 | | 151.872 | 4.293 | 0.000 | 0.000 | 0.000 | 0.006 |
| 6 | | 14.246 | 0.373 | 0.000 | 0.000 | 0.000 | 0.000 |
| 7 | | 192.343 | 5.818 | 0.000 | 0.000 | 0.000 | 0.006 |
| 8 | | 19.907 | 0.514 | 0.000 | 0.000 | 0.000 | 0.001 |
| 9 | | 240.328 | 6.715 | 0.000 | 0.000 | 0.000 | 0.006 |
| 10 | | 26.654 | 0.624 | 0.000 | 0.000 | 0.000 | 0.001 |
| 11 | | 288.704 | 6.211 | 0.000 | 0.000 | 0.000 | 0.004 |
| 14 | | 608.142 | 0.000 | 0.000 | 0.000 | 0.000 | 0.018 |
| 16 | | 279.441 | 0.000 | 0.000 | 0.000 | 0.000 | 0.005 |
| 18 | | 350.463 | 0.000 | 0.000 | 0.000 | 0.000 | 0.005 |
| 20 | | 433.694 | 0.000 | 0.000 | 0.000 | 0.000 | 0.004 |
| 22 | | 546.897 | 0.000 | 0.000 | 0.000 | 0.000 | 0.002 |
| 23 | | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 24 | | 9.713 | 0.189 | 0.000 | 0.000 | 0.000 | 0.000 |

Figure 5.3.3-3 PRINT APPLIED JOINT LOAD Results for Example SEL-1

Analysis Prerelease Features

GT STRUDL

| | | | | | | |
|----|---------|-------|-------|-------|-------|-------|
| 25 | 104.472 | 3.722 | 0.000 | 0.000 | 0.000 | 0.017 |
| 26 | 10.172 | 0.239 | 0.000 | 0.000 | 0.000 | 0.000 |
| 27 | 151.872 | 4.293 | 0.000 | 0.000 | 0.000 | 0.006 |
| 28 | 14.246 | 0.373 | 0.000 | 0.000 | 0.000 | 0.000 |
| 29 | 192.343 | 5.818 | 0.000 | 0.000 | 0.000 | 0.006 |
| 30 | 19.907 | 0.514 | 0.000 | 0.000 | 0.000 | 0.001 |
| 31 | 240.328 | 6.715 | 0.000 | 0.000 | 0.000 | 0.006 |
| 32 | 26.654 | 0.624 | 0.000 | 0.000 | 0.000 | 0.001 |
| 33 | 288.704 | 6.211 | 0.000 | 0.000 | 0.000 | 0.004 |

LOADING - EM1-G.1

Equivalent STATIC EARTHQUAKE load 1-G.1 from total mass

STATUS - ACTIVE

JOINT LOADS-----/

| JOINT | STEP | FORCE X | Y | Z | MOMENT X | Y | Z |
|-------|------|----------|-------|-------|----------|-------|-------|
| 1 | | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 2 | | 18.598 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 3 | | 193.858 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 4 | | 14.878 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 5 | | 192.618 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 6 | | 16.118 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 7 | | 194.478 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 8 | | 18.598 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 9 | | 196.338 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 10 | | 19.837 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 11 | | 187.039 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 14 | | 1126.417 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 16 | | 354.240 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 18 | | 354.240 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 20 | | 354.240 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 22 | | 354.240 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 23 | | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 24 | | 18.598 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 25 | | 193.858 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 26 | | 14.878 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 27 | | 192.618 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 28 | | 16.118 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 29 | | 194.478 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 30 | | 18.598 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 31 | | 196.338 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 32 | | 19.837 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 33 | | 187.039 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |

 * END OF DATA FROM INTERNAL STORAGE *

Figure 5.3.3-3 PRINT APPLIED JOINT LOAD Results for Example SEL-1 (Continued)

5.3.4 FORM UBC97 LOAD Command -- Automatic Generation of Seismic Loads According to 1997 UBC

General form:

$$\begin{aligned}
 & \text{FORM UBC97 (STATIC) (SEISMIC) LOAD } \left\{ \begin{array}{c} 'a_{sl}' \\ i_{sl} \end{array} \right\} ('title_{sl}') - \\
 & \text{DIRECTION } \left\{ \begin{array}{c} \rightarrow \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right\} \text{ HEIGHT (DIRECTION) } \left\{ \begin{array}{c} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right\} \text{ WEIGHT (LOAD) } \left\{ \begin{array}{c} 'a_w' \\ i_w \end{array} \right\} - \\
 & \text{ZONE } \left\{ \begin{array}{c} \underline{1} \\ \underline{2A} \\ \underline{2B} \\ \underline{3} \\ \underline{4} \\ \underline{Z} \ v_Z \end{array} \right\} * \left\{ \begin{array}{c} \underline{SA} \\ \underline{SB} \\ \underline{SC} \\ \underline{SD} \\ \underline{SE} \\ \underline{SF} \\ \underline{CA} \ v_{CA} \\ \underline{CV} \ v_{CV} \\ \underline{NA} \ v_{NA} \\ \underline{NV} \ v_{NV} \end{array} \right\} \text{ OCCUPANCY (CATEGORY) } \left\{ \begin{array}{c} \underline{ESSENTIAL} \ (\underline{FACILITIES}) \\ \underline{HAZARDOUS} \ (\underline{FACILITIES}) \\ \underline{SPECIAL} \ (\underline{STRUCTURES}) \\ \underline{STANDARD} \ (\underline{STRUCTURES}) \\ \underline{MISCELLANEOUS} \ (\underline{STRUCTURES}) \\ \underline{I} \ v_I \end{array} \right\} - \\
 & \text{CT } \left\{ \begin{array}{c} \underline{SMR} \\ \underline{RCMR} \\ \underline{OTHER} \\ v_{CT} \end{array} \right\} \underline{R} \ v_R \left((\underline{WITH}) \underline{TORSION} \left\{ \begin{array}{c} \rightarrow \underline{PLUS} \\ \underline{MINUS} \end{array} \right\} \right) (\underline{FLOOR} \ (\underline{TOLERANCE}) \ v_{TOL})
 \end{aligned}$$

Elements:

'a_{sl}'/i_{sl} = alphanumeric or integer name for the generated UBC 1997 static seismic load. This name must be unique among all current loading names and is limited to eight characters or digits.

'title_{sl}' = optional static load title of up to 64 characters in length.

'a_w'/i_w = alphanumeric or integer name of the independent loading that is used for the calculation of the weight distribution of the structure.

v_Z = decimal value for the UBC 1997 seismic zone factor Z. This specified value supersedes the calculated value based on the seismic zone specified by the ZONE option.

| | | |
|-----------|---|--|
| v_{CA} | = | decimal value for the UBC 1997 seismic coefficient C_a . This specified value supersedes the calculated value based on the seismic zone and soil profiles specified by the ZONE and SOIL options. |
| v_{CV} | = | decimal value for the UBC 1997 seismic coefficient C_v . This specified value supersedes the calculated value based on the seismic zone and soil profiles specified by the ZONE and SOIL options. |
| v_{NA} | = | decimal value for the UBC 1997 near-source factor N_a used in the calculation of C_a when seismic zone 4 is selected using the ZONE option. The default value is taken as 1.0. |
| v_{NV} | = | decimal value for the UBC 1997 near-source factor N_v used in the calculation of C_v when seismic zone 4 is selected using the ZONE option. The default value is taken as 1.0. |
| v_I | = | decimal value for the UBC 1997 importance factor I , to be used instead of the calculated value based on the selection of one of the OCCUPANCE CATAGORY options: ESSENTIAL, HAZARDOUS, SPECIAL, STANDARD, or MISCELLANEOUS. |
| v_{CT} | = | decimal value for the UBC 1997 numerical coefficient C_t , to be used instead of the calculated value based on the selection of one of the CT options: SMR, RCMR, or OTHER. |
| v_R | = | decimal value for the UBC 1997 numerical coefficient R . |
| v_{TOL} | = | All active joints within this specified HEIGHT DIRECTION tolerance of one another are assumed to define the geometry of a single floor. The input of this value assumes length units. The default value for v_{TOL} is taken as 6.0 inches (15.24 cm). |

Explanation:

The FORM UBC97 LOAD command is used to compute an independent loading condition consisting only of static joint loads in accordance with the provisions of Sections 1630.2, 1630.5 and 1630.6 of the 1997 Uniform Building Code, Vol.2 [ref 97]. A very important aspect regarding the execution of this command is that all load computations are performed only on the currently active joints.

The options used to define this loading condition are described as follows:

$$\underline{\text{DIRECTION}} \left\{ \begin{array}{c} \rightarrow \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right\}$$

The DIRECTION option is used to specify the global coordinate direction of computed joint load components contained in the loading condition. The global X direction is the default.

$$\underline{\text{HEIGHT (DIRECTION)}} \left\{ \begin{array}{c} \underline{X} \\ \rightarrow \underline{Y} \\ \underline{Z} \end{array} \right\}$$

The HEIGHT DIRECTION option specifies the global coordinate direction that defines the elevation coordinate for the structure. For example, HEIGHT DIRECTION Y specifies that the height of the building structure and the building floor elevations are defined with respect to the global Y axis. It is assumed that elevations are measured from 0 at the joint having the least HEIGHT DIRECTION joint coordinate value to the full height of the structure at the joint having the largest HEIGHT DIRECTION joint coordinate value.

$$\underline{\text{WEIGHT (LOAD)}} \left\{ \begin{array}{l} \text{'a}_w\text{' } \\ \text{i}_w \end{array} \right\}$$

The required WEIGHT LOAD command option is used to identify the static, independent loading from which the total weight of the building model is computed. The total weight is computed as the sum of the absolute values of all translation load components (FORCE X, FORCE Y, and FORCE Z). The specified loading must have been defined previous to the time that the FORM UBC97 command is given.

$$\underline{\text{ZONE}} \left\{ \begin{array}{l} \underline{1} \\ \underline{2A} \\ \underline{2B} \\ \underline{3} \\ \underline{4} \\ \underline{Z} \ v_z \end{array} \right\}$$

This optional command is used to specify data about the relevant earthquake zone for the load calculations. Use the ZONE option to select zone 1, 2A, 2B, 3, 4, or to directly specify a value v_z for the seismic zone factor Z. One of the ZONE options 1 through 4 must be selected if one of the SOIL options SA through SF is also selected as described below.

$$* \left\{ \begin{array}{l} \underline{\text{SOIL}} \left\{ \begin{array}{l} \underline{SA} \\ \underline{SB} \\ \underline{SC} \\ \underline{SD} \\ \underline{SE} \\ \underline{SF} \end{array} \right\} \\ \\ \underline{CA} \ v_{CA} \\ \underline{CV} \ v_{CV} \\ \underline{NA} \ v_{NA} \\ \underline{NV} \ v_{NV} \end{array} \right\}$$

These options are used to specify data about the seismic zone soil conditions. The SOIL option is used to specify the soil type: $SA = S_A$, $SB = S_B$, $SC = S_C$, $SD = S_D$, $SE = S_E$, and $SF = S_F$. The selected SOIL specification – SA through SF – is used in combination of the selected seismic ZONE

specification – 1 through 4 – to calculate the values for seismic coefficients C_a and C_v according to 1997 UBC Tables 16-Q and 16-R. Any values specified using the CA and CV parameters supersede the values computed from the ZONE and SOIL specifications. Values for CA and CV must be specified if the ZONE and/or SOIL specifications are not given. The values for NA and NV are taken as 1.0 unless otherwise specified. These values are the values for the near-source factors N_a and N_v respectively, which are used in the calculation of C_a and C_v respectively when the ZONE 4 specification above is given.

$$\text{OCCUPANCY (CATEGORY)} \left\{ \begin{array}{l} \text{ESSENTIAL (FACILITIES)} \\ \text{HAZARDOUS (FACILITIES)} \\ \text{SPECIAL (STRUCTURES)} \\ \text{STANDARD (STRUCTURES)} \\ \text{MISCELLANEOUS (STRUCTURES)} \\ \underline{I} \quad v_I \end{array} \right\}$$

The seismic importance factor I is calculated based on the OCCUPANCY CATEGORY specified using this required option. The value I is calculated automatically by specifying one of the categories ESSENTIAL, HAZARDOUS, SPECIAL, STANDARD, or MISCELLANEOUS. A value for I can also be specified directly by giving the value v_I .

$$\text{CT} \left\{ \begin{array}{l} \text{SMR} \\ \text{RCMR} \\ \text{OTHER} \\ \underline{v_{CT}} \end{array} \right\}$$

The numerical coefficient C_t used for the calculation of the structure period according the 1997 UBC Section 1630.2.2, Method A is calculated using the specifications of this required option. The value for C_t is calculated automatically by specifying one of the categories SMR, RCMR, or OTHER. SMR stands for Steel Moment-Resisting frames, RCMR stands for Reinforced Concrete Moment-Resisting frames and eccentrically braced frames, and OTHER stands for all OTHER buildings. The value C_t can also be specified directly by giving a value for v_{CT} .

R v_R

This required option is used to specify the over-strength/ductility factor R according to 1997 UBC Tables 16-N or 16-P.

(WITH) TORSION $\left\{ \begin{array}{l} \rightarrow \text{PLUS} \\ \text{MINUS} \end{array} \right\}$

The optional WITH TORSION specification is used to specify that the calculation of the UBC97 joint loads shall include the effects of the rigid diaphragm, torsional mass displacement described in 1997 UBC Sections 1630.6 and 1630.7. The default PLUS option indicates that each floor mass distribution for the load computation shall reflect the shift from the floor center of mass in a positive global direction perpendicular to the global load-application direction specified by the DIRECTION option. The MINUS option indicates that each floor mass distribution shall reflect the shift from the floor center of mass in a negative global direction perpendicular to the global load-application direction.

FLOOR (TOLERANCE) v_{TOL}

The optional FLOOR TOLERANCE specification is used to specify the elevation neighborhood within which groups of joints comprise a particular floor for the calculation of the UBC97 load. All joints whose elevation is within the specified tolerance are assumed to comprise a floor. The value for v_{TOL} must reflect active length units and is taken as 6 inches by default.

Because the UBC97 load computations are performed only for the active joints, any joints that shall not be considered as part of any floor can be inactivated prior to issuing the FORM UBC97 command and then re-activated following the command:

```
INACTIVE JOINTS . . .
FORM UBC97 LOAD . . .
ACTIVE JOINTS ALL
```

Errors:

The following messages indicate warning conditions that can occur during the execution of the FORM UBC97 LOAD command:

1. The following message is produced if the specified weight loading does not exist.

```
**** WARNING_STUBC9 -- Specified WEIGHT loading DL1      does not
                        exist.  Command ignored.
```

2. One or more of the following three messages are produced if seismic zone and soil profile data are not correctly specified:

```
**** WARNING_STUBC9 -- Value for seismic coefficient Cv incorrectly
                        specified.  Command ignored.
```

```
**** WARNING_STUBC9 -- Value for seismic coefficient Ca incorrectly
                        specified.  Command ignored.
```

```
**** WARNING_STUBC9 -- Value for seismic zone factor Z incorrectly
                        specified.  Command ignored.
```

The following is an example of a FORM UBC97 command that will produce these warning messages:

```
FORM UBC97 LOAD 'TESTUBCZ' DIR X WEIGHT LOAD 'DL' -
      SOIL SC CT SMR OCC I 1.5 R 5.5
```

While the SOIL SC specification is given, the ZONE/Z specification is missing, resulting in insufficient data for the calculation of the seismic coefficients C_a and C_v .

3. The following message is produced if the required occupancy data are incorrectly specified.

```
**** WARNING_STUBC9 -- Value for importance factor I incorrectly
                        specified.  Command ignored.
```

The following is an example of a FORM UBC97 command that will produce this warning message:

```
FORM UBC97 LOAD 'TESTUBCZ' DIR X WEIGHT LOAD 'DL' -
      CA 0.24 CV 0.32 CT SMR R 5.5
```

The required OCCUPANCY option is not given.

4. This warning message is given if a value for the over-strength/ductility factor R is not specified using the required R command option:

```
**** WARNING_STUBC9 -- Value for R factor incorrectly specified.
                        Command ignored.
```

5. The following warning message is given if the value for the numerical coefficient C_t is not correctly specified using the required CT command option:

```
**** WARNING_STUBC9 -- Value for period coefficient Ct incorrectly
                        specified. Command ignored.
```

Example:

The following UBC97 command example correctly defines a 1997 UBC static lateral load having the name UBCX:

```
FORM UBC97 LOAD 'UBCX' DIR X WEIGHT LOAD 'DL' -
      SOIL SC ZONE 2B CT SMR OCC I 1.5 R 5.5
```

The load is calculated using the following 1997 UBC parameter values:

| | | |
|-------|---|-------|
| C_t | = | 0.035 |
| Z | = | 0.2 |
| I | = | 1.5 |
| R | = | 5.5 |
| C_a | = | 0.24 |
| C_v | = | 0.32 |
| N_a | = | 1.0 |
| N_v | = | 1.0 |

No torsion effects are included in the computation of loading UBCX.

The following example is the same as the previous one, with the exception that the CV 0.35 parameter is added following the ZONE 2B option. The TORSION option is also added:

```
FORM UBC97 LOAD 'UBCXNEW' DIR X WEIGHT LOAD 'DL' -
      SOIL SC ZONE 2B CV 0.35 CT SMR OCC I 1.5 R 5.5
      TORSION MINUS
```

The 1997 UBC parameter values that are used to calculate loading UBCXNEW are the same as those that are used to calculate load UBCX in the previous example, with the exception that $C_v = 0.35$ rather than $C_v = 0.32$, by virtue of the fact that this is the value that is directly specified for C_v using the CV 0.35 parameter specification. The specified value for C_v takes precedence over the value that is computed according to the 1997 UBC provisions. Loading UBCXNEW also reflects a negative global Z offset of the floor masses with respect to each floor center of mass.

This page intentionally left blank.

5.3.5 FORM IS1893 LOAD Command - Automatic Generation of Static Seismic Loads According to IS 1893

General Form:

$$\begin{aligned}
 & \underline{\text{FORM IS1893}} \ (\underline{\text{STATIC}}) \ (\underline{\text{SEISMIC}}) \ \underline{\text{LOAD}} \ \left\{ \begin{array}{c} \text{'a}_{sl}\text{' } \\ \text{i}_{sl} \end{array} \right\} \ (\text{'title}_{sl}\text{'}) - \\
 & \underline{\text{DIRECTION}} \ \left\{ \begin{array}{c} \rightarrow \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{array} \right\} \ \underline{\text{HEIGHT}} \ \underline{\text{DIRECTION}} \ \left\{ \begin{array}{c} \rightarrow \underline{\text{X}} \\ \underline{\text{Y}} \\ \underline{\text{Z}} \end{array} \right\} \ \underline{\text{WEIGHT}} \ (\underline{\text{LOAD}}) \ \left\{ \begin{array}{c} \text{'a}_w\text{' } \\ \text{i}_w \end{array} \right\} - \\
 & \underline{\text{ZONE}} \ \left\{ \begin{array}{c} \underline{\text{II}} \\ \underline{\text{III}} \\ \underline{\text{IV}} \\ \underline{\text{V}} \\ \underline{\text{Z}} \ v_Z \end{array} \right\} \ \underline{\text{SOIL}} \ (\underline{\text{TYPE}}) \ \left\{ \begin{array}{c} \underline{\text{I}} \\ \underline{\text{II}} \\ \underline{\text{III}} \end{array} \right\} \ \underline{\text{IMPORTANCE}} \ (\underline{\text{FACTOR}}) \ v_I - \\
 & \underline{\text{PERIOD}} \ (\underline{\text{TYPE}}) \ \left\{ \begin{array}{c} \underline{\text{SMR}} \\ \underline{\text{RCMR}} \\ \underline{\text{OTHER}} \\ v_T \end{array} \right\} \ \underline{\text{R}} \ v_R \ (\underline{\text{FLOOR}} \ (\underline{\text{TOLERANCE}}) \ v_{\text{TOL}})
 \end{aligned}$$

Elements:

- ‘a_{sl}’/i_{sl} = alphanumeric or integer name for the generated IS 1893 static seismic load. This name must be unique among all current loading names and is limited to eight characters or digits.
- ‘title_{sl}’ = optional static load title of up to 64 characters in length.
- ‘a_w’/i_w = alphanumeric or integer name of the independent loading that is used for the calculation of the weight distribution of the structure.
- v_Z = decimal value for the IS 1893 seismic zone factor Z. This specified value supersedes the calculated value based on the seismic zone specified by the ZONE option.

| | | |
|-----------|---|---|
| v_I | = | decimal value for the IS 1893 importance factor I. |
| v_T | = | decimal value for the building fundamental period in seconds to be used for the calculation of the average response acceleration coefficient S_a/g . |
| v_R | = | decimal value for the IS 1893 response reduction factor R. |
| v_{TOL} | = | all joints within this specified HEIGHT DIRECTION tolerance of one another are assumed to define the geometry of a single floor. The input of this value assumes length units. The default value for v_{TOL} is taken as 6.0 inches (15.24 cm). |

Explanation:

The FORM IS1893 LOAD command is used to compute an independent loading condition consisting of static joint loads only in accordance with the provisions of Sections 6 and 7, Indian Standard IS 1893 (Part I) : 2.2. A very important aspect regarding the execution of this command is that all computations are performed only on the currently active joints.

The options used to define this loading condition are described as follows:

$$\underline{\text{DIRECTION}} \quad \left\{ \begin{array}{c} \rightarrow \underline{\underline{X}} \\ \underline{\underline{Y}} \\ \underline{\underline{Z}} \end{array} \right\}$$

The DIRECTION option is used to specify the global coordinate direction of the computed joint load components contained in the loading condition. The global X direction is the default.

$$\underline{\text{HEIGHT}} \quad \underline{\text{DIRECTION}} \quad \left\{ \begin{array}{c} \underline{\underline{X}} \\ \rightarrow \underline{\underline{Y}} \\ \underline{\underline{Z}} \end{array} \right\}$$

The HEIGHT DIRECTION option specifies the global coordinate direction that defines the elevation coordinate for the structure. For example, HEIGHT DIRECTION Y specifies that the height of the building structure and the building floor elevations are defined with respect to the global Y

axis. It is assumed that elevations are defined with respect to the global Y axis. It is assumed that elevations are measured from 0 at the joint having the least HEIGHT DIRECTION joint coordinate value to the full height of the structure at the joint having the largest HEIGHT DIRECTION joint coordinate value.

$$\underline{\text{WEIGHT}} \ (\underline{\text{LOAD}}) \ \left\{ \begin{array}{c} 'a_w' \\ i_w \end{array} \right\}$$

The required WEIGHT LOAD command option is used to identify the static, independent loading from which the total weight of the building model is computed. The total weight is computed as the sum of the absolute values of all translation load components (FORCE X, FORCE Y, and FORCE Z). The specified loading must have been defined previous to the time that the FORM IS1893 LOAD command is given.

$$\underline{\text{ZONE}} \ \left\{ \begin{array}{c} \underline{\text{II}} \\ \underline{\text{III}} \\ \underline{\text{IV}} \\ \underline{\text{V}} \\ \underline{\text{Z}} \ v_z \end{array} \right\}$$

This required command is used to specify data about the relevant earthquake zone for the load calculations. Use the ZONE option to select zone II, III, IV, V or to directly specify a value v_z for the seismic zone factor Z.

$$\underline{\text{SOIL}} \ (\underline{\text{TYPE}}) \ \left\{ \begin{array}{c} \underline{\text{I}} \\ \underline{\text{II}} \\ \underline{\text{III}} \end{array} \right\}$$

These options are used to specify data about the seismic zone soil conditions for the calculation of the average response acceleration coefficient S_a/g .

IMPORTANCE (FACTOR) v_I

This required command is used to specify the value for the seismic importance factor I.

$$\underline{\text{PERIOD}} \ (\underline{\text{TYPE}}) \ \left\{ \begin{array}{l} \underline{\text{SMR}} \\ \underline{\text{RCMR}} \\ \underline{\text{OTHER}} \\ v_T \end{array} \right\}$$

The required PERIOD command is used to select the equation for the calculation of the approximate building natural period T_a . The SMR, RCMR and OTHER options are used to select an empirical equation to compute the building natural period, where SMT stands for Steel Moment-Resisting frames, RCMR stands for Reinforced Concrete Moment-Resisting frames and eccentrically braced frames, and OTHER stands for all OTHER buildings. The period in seconds also can be specified directly by specifying a decimal value v_T .

R v_R

This required option is used to specify the value for the response reduction factor.

FLOOR (TOLERANCE) v_{TOL}

The option FLOOR TOLERANCE specification is used to specify the elevation neighborhood within which groups of joints comprise a particular floor for the calculations of the IS1893 load. All joints whose elevation is within the specified tolerance are assumed to comprise a floor. The value for v_{TOL} must reflect active length units and is taken as 6 inches by default.

Because the IS1893 load computations are performed only for the active joints, any joints that shall not be considered as part of any floor can be inactivated prior to issuing the FORM IS1893 command and then re-activated following the command:

```
INACTIVE JOINTS ...
FORM IS1893 LOAD ...
ACTIVE JOINTS ALL
```

Errors:

The following messages indicate warning conditions that can occur during the execution of the FORM IS1893 LOAD command.

1. The following message is produced if the specified weight loading does not exist.

```
**** WARNING_STIS93 - Specified WEIGHT loading DL1 does not
                        exist. Command ignored.
```

2. The following message is produced if seismic zone data are not correctly specified:

```
**** WARNING_STIS93 - Specified WEIGHT loading DL1 does not
                        exist. Command ignored.
```

The following is an example of a FORM IS1893 command that will produce this warning message:

```
FORM IS1893 LOAD 'TESTISX' DIRE X WEIGHT LOAD 'DL' -
      SOIL II PERIOD TYPE SMR IMP 1.5 R 4.0
```

The ZONE option is not specified in the command.

3. The following message is produced if the soil type is not correctly specified:

```
**** WARNING_STIS93 - Soil type not correctly specified. Command
                        ignored.
```

The following is an example of a FORM IS1893 command that will produce this warning message:

```
FORM IS1893 LOAD 'TESTISX' DIR X WEIGHT LOAD 'DL' ZONE -
      III PERIOD TYPE SMR IMP 1.5 R 4.0
```

The SOIL TYPE option is not specified in the command.

4. The following message is produced if the importance factor I is incorrectly specified.

```
**** WARNING_STIS93 - Specified WEIGHT loading DL1 does not
                        exist. Command ignored.
```

The following is an example of a FORM IS1893 command that will produce this warning message:

```
FORM IS1893 LOAD 'RESTISX' DIR X WEIGHT LOAD 'DL' ZONE -
      III SOIL TYPE III PERIOD TYPE SMR R 4.0
```

The required value of the importance factor I is not specified.

5. This warning message is given if a value for the response reduction factor R is not specified using the required R command option:

```
**** WARNING_STIS93 - R factor incorrectly specified. Command
                        ignored.
```

6. The following warning message is given if the PERIOD option is not specified.

```
**** WARNING_STIS93 - R factor incorrectly specified. Command
                        ignored.
```

Example

The following FORM IS1893 command example correctly defines an IS 1893 static lateral load having the name IS1893X:

```
FORM IS1893 LOAD 'IS1893X' DIR X WEIGHT LOAD 'DL' -
      SOIL TYPE II ZONE V PERIOD 1.00 -
      IMPORT 1.5 R 4.0
```

The period is directly specified as 1.0 seconds.

5.3.6 Element Properties Command for Nonlinear Hysteretic Spring Element

The Element Properties command for the hysteretic version of the nonlinear spring element is shown below and is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.5.3.5 The ELEMENT PROPERTIES Command for the NLS4PH Element

The complete syntax of the ELEMENT PROPERTIES command is described in Section 2.3.5.2, Volume 3 of the GTSTRUDL User Reference Manual. The properties for the NLS4PH element, a four-parameter, hysteretic version of the nonlinear spring element (type NLS), are described by the following additional ELEMENT PROPERTIES command syntax elements:

General form:

ELEMENT PROPERTIES
 list (NLS4PH specs)
 .
 .
 .
 list (NLS4PH specs)

where,

NLS4PH specs = TYPE 'NLS4PH' (force specs) (moment specs) (orientation specs)

$$\begin{aligned}
 \text{force specs} &= \underline{\text{FORCE}} * \left\{ \begin{array}{l} \underline{\text{X}} [\underline{\text{UX0}}] \text{v}_{\text{ux0}} [\underline{\text{FX0}}] \text{v}_{\text{fx0}} [\underline{\text{UXMAX}}] \text{v}_{\text{uxm}} [\underline{\text{FXMAX}}] \text{v}_{\text{fxm}} \\ \underline{\text{Y}} [\underline{\text{UY0}}] \text{v}_{\text{uy0}} [\underline{\text{FY0}}] \text{v}_{\text{fy0}} [\underline{\text{UYMAX}}] \text{v}_{\text{uym}} [\underline{\text{FYMAX}}] \text{v}_{\text{fym}} \\ \underline{\text{Z}} [\underline{\text{UZ0}}] \text{v}_{\text{uz0}} [\underline{\text{FZ0}}] \text{v}_{\text{fz0}} [\underline{\text{UZMAX}}] \text{v}_{\text{uzm}} [\underline{\text{FZMAX}}] \text{v}_{\text{fzm}} \end{array} \right\} \\
 \text{moment specs} &= \underline{\text{MOMENT}} * \left\{ \begin{array}{l} \underline{\text{X}} [\underline{\text{THX0}}] \text{v}_{\text{thx0}} [\underline{\text{MX0}}] \text{v}_{\text{mx0}} [\underline{\text{THXMAX}}] \text{v}_{\text{thxm}} [\underline{\text{MXMAX}}] \text{v}_{\text{mxm}} \\ \underline{\text{Y}} [\underline{\text{THY0}}] \text{v}_{\text{thy0}} [\underline{\text{MY0}}] \text{v}_{\text{my0}} [\underline{\text{THYMAX}}] \text{v}_{\text{thym}} [\underline{\text{MYMAX}}] \text{v}_{\text{mym}} \\ \underline{\text{Z}} [\underline{\text{THZ0}}] \text{v}_{\text{thz0}} [\underline{\text{MZ0}}] \text{v}_{\text{mz0}} [\underline{\text{THZMAX}}] \text{v}_{\text{thzm}} [\underline{\text{MZMAX}}] \text{v}_{\text{mzm}} \end{array} \right\} \\
 \text{orientation specs} &= * \left\{ \begin{array}{l} \underline{\text{TH1}} \text{v}_1 \\ \underline{\text{TH2}} \text{v}_2 \\ \underline{\text{TH3}} \text{v}_3 \end{array} \right\}
 \end{aligned}$$

Elements:

| | | |
|-----------------|---|---|
| v_{ux0} | = | displacement in the spring X direction at curve point 0, |
| v_{fx0} | = | spring X force corresponding to v_{ux0} , |
| v_{uxm} | = | displacement in the spring X direction at the point of maximum displacement on the curve, |
| v_{fxm} | = | spring X force corresponding to v_{uxm} , |
| v_{uy0} | = | displacement in the spring Y direction at curve point 0, |
| v_{fy0} | = | spring Y force corresponding to v_{uy0} , |
| v_{uym} | = | displacement in the spring Y direction at the point of maximum displacement on the curve, |
| v_{fym} | = | spring Y force corresponding to v_{uym} , |
| v_{uz0} | = | displacement in the spring Z direction at curve point 0, |
| v_{fz0} | = | spring Z force corresponding to v_{uz0} , |
| v_{uzm} | = | displacement in the spring Z direction at the point of maximum displacement on the curve, |
| v_{fzm} | = | spring Z force corresponding to v_{uzm} , |
| v_{thx0} | = | rotation about the spring X axis at curve point 0, |
| v_{mx0} | = | spring X axis moment corresponding to v_{thx0} , |
| v_{thxm} | = | rotation about the spring X axis at the point of maximum rotation on the curve, |
| v_{mxm} | = | spring X axis moment corresponding to v_{thxm} , |
| v_{thy0} | = | rotation about the spring Y axis at curve point 0, |
| v_{my0} | = | spring Y moment corresponding to v_{thy0} , |
| v_{thym} | = | rotation about the spring Y axis at the point of maximum rotation on the curve, |
| v_{mym} | = | spring Y axis moment corresponding to v_{thym} , |
| v_{thz0} | = | rotation about the spring Z axis at curve point 0, |
| v_{mz0} | = | spring Z axis moment corresponding to v_{thz0} , |
| v_{thzm} | = | rotation about the spring Z axis at the point of maximum rotation on the curve, |
| v_{mzm} | = | spring Z axis moment corresponding to v_{thzm} , |
| v_1, v_2, v_3 | = | values in current angle units of the rotation angles 2_1 , 2_2 , and 2_3 respectively, as shown in Figure 2.1.7-1 (Section 2.1.7.1, Volume 1), that define the rotated orientation of the spring axes with respect to the global coordinate system. |

All displacement, rotation, force, and moment values assume the appropriate active length, angular, and force units.

Explanation:

The NLS4PH element is a special version of the nonlinear spring element (Section 2.5.3, Volume 3 of the GTSTRUDL User Reference Manual), where the stiffness properties are described by bilinear, symmetric, hysteretic, force-displacement and moment-rotation curves. A symmetric curve is defined by a total of three data points: the first point at the 0, 0 origin of the curve is assumed; the second and third points are defined by the specified data values.

The specified data points, for example v_{uy0} , v_{fy0} , v_{uym} , and v_{fym} , must always be positive, and the slope of the first line segment connecting point 0,0 and the point defined by the first pair of data values (e.g. v_{uy0} and v_{fy0}) must be greater than the slope of the second line segment connecting the points defined by the first pair (v_{uy0} and v_{fy0}) and the second pair (v_{uym} and v_{fym}) of values. In addition, the slope of the second line segment must be greater than or equal to zero. The symmetric negative portion of the force-displacement or moment-rotation curve is defined by taking the negative of the specified pairs of data values.

Figure 2.5.3.5-1 illustrates a force-displacement curve specified by the four data point values v_{uy0} , v_{fy0} , v_{uym} , and v_{fym} . The initial curve template and the assumed hysteretic behavior are shown.

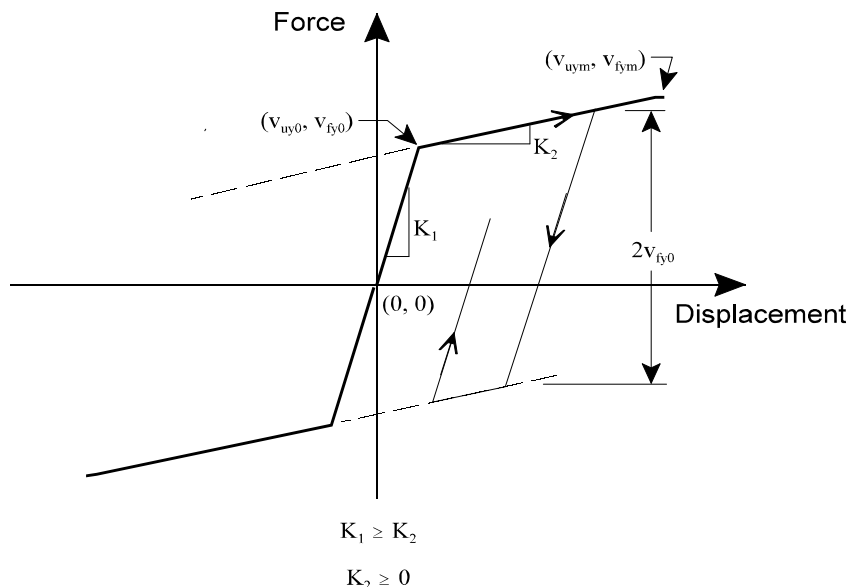


Figure 2.5.3.5-1 NLS4PH Force-Displacement Spring Curve with Hysteretic Behavior

The active degrees-of-freedom of the NLS4PH element are defined by the selection of the force/moment components for which the data point values are specified. A total of six degrees-of-freedom can be specified, and all degrees of freedom are uncoupled.

The default orientation of the degrees-of-freedom of the NLS4PH element is the global coordinate system. FORCE X means global force X, FORCE Y, global force Y, etc. This orientation can be rotated with respect to the global coordinate system by specifying values v_1 , v_2 , and v_3 for the angles TH1, TH2, and TH3. The sign convention for these angles is identical to that of the local release orientation angles of the JOINT RELEASE command in Section 2.1.7.1, Volume 1 of the GTSTRUDL User Reference Manual.

The NLS4PH element is applicable for both nonlinear static and dynamic analyses.

Modifications:

In CHANGES mode, existing data are replaced with the newly specified data. Unspecified data is left unchanged.

In DELETIONS mode, only the list of elements need to be specified (see Section 2.3.5.5). All properties data pertaining to the specified elements are deleted.

Example:

The following example illustrates the specification of NLS4PH properties for element 8. The specified degrees-of-freedom are force X and Moment Z. The local orientation of the spring is defined by the TH2 value of -30.0° (a negative left-hand rotation about the global Y axis).

UNITS DEGREES KIPS INCHES

ELEMENT PROPERTIES

8 TYPE 'NLS4PH' -

FORCE X UX0 0.5 FX0 20.0 UXM 3.0 FXM 22.0 -

MOM Z THZ0 1.2 MZ0 650.0 THZMAX 10.0 MZMAX 1500.0 TH2 -30.0

Error Messages:

The following messages indicate error conditions that can occur because of the incorrect specification of the element properties data for the NLS4PH element:

```
{ 63} > UNITS KIPS INCHES
{ 64} > ELEMENT PROPERTIES TYPE 'NLS4PH'
```

```
**** ERROR_XYPROP - Command syntax incorrect for NLS4PH element.
                   Scan mode entered.
```

```
{ 65} > 7 FORCE X UX0 0.5 FX0 20.0 UXM 3.0 FXM 22.0
CI-w-cmdmiss, ERROR: Command name missing from line.
```

These messages are general and are produced when any incorrect syntax is used in the ELEMENT PROPERTIES command for the NLS4PH element. In this case, the TYPE 'NLS4PH' specification in the ELEMENT PROPERTIES command header is not permitted. Scan mode is entered, precluding the execution of any time-consuming and resource-expensive operation such as analysis or the output of analysis results.

```
{ 70} > ELEMENT PROPERTIES
{ 71} > 7 TYPE 'NLS4PH' FORCE X UX0 0.5 FX0 -10.0 UXM 3.0 FXM
22.0
```

```
**** ERROR_STNS4P -- All NLS4PH element property values must be
>= 0.
```

```
Scan mode entered.
```

This message also is given when the specified data do not conform to the requirements described above. In this example, a negative value is specified for FX0; it must be greater than 0.0. Scan mode is also entered as a result of this error condition.

```
{ 87} > ELEMENT PROPERTIES
{ 88} > 8 TYPE 'NLS4PH' -
{ 89} >_ MOM Z THZ0 1.2 MZ0 650.0 THZMAX 10.0 MZMAX 1500.0 TH2
-30.0
```

```
**** ERROR_STNS4P -- Properties already specified for NLS4PH
element 8
```

```
Scan mode entered.
```

This message is given when the NLS4PH properties for a specified element previously have been given. Scan mode is entered. CHANGES mode is required to modify existing NLS4PH properties.

This page intentionally left blank.

5.3.7 Nonlinear Dynamic Analysis

Sections 2.5.6.1 and 2.5.6.2 below describe the commands used to execute a nonlinear dynamic analysis. Section 2.5.6.1 describes extensions to the optional DYNAMIC PARAMETERS command (Section 2.4.5.3) which are used to set nonlinear dynamic analysis control parameters. Section 2.5.6.2 describes the DYNAMIC ANALYSIS NONLINEAR command.

Section 2.5.6.3 provides an example problem illustrating a complete nonlinear dynamic analysis execution, including the use of the DYNAMIC PARAMETERS extensions and the DYNAMIC ANALYSIS NONLINEAR command. The sections are numbered as they will appear when they are added to Volume 3 of the GTSTRUDL User Reference Manual.

2.5.6.1 Extensions to the DYNAMIC PARAMETERS Command

General form:

DYNAMIC PARAMETERS

BLOCK (SIZE) i_{BS}

UPDATE (STIFFNESS EVERY) i_U (TIME STEPS)

MAXIMUM (NUMBER OF EQUILIBRIUM CYCLES) i_{MAX}

CONVERGENCE (TOLERANCE) ENERGY v_{TOL}

INITIAL (STRESS LOAD) $\left\{ \begin{array}{l} \rightarrow \text{OFF} \\ i_{ISL} \\ 'a_{ISL}' \end{array} \right\}$

RESULTS (FILE NAME) 'fn'

END (OF DYNAMIC PARAMETERS)

Elements:

- i_{BS} = size of equation blocks in double words if the out-of-core equation solver is to be used,
- i_U = time step interval at which complete updating of the effective global tangent stiffness matrix is to occur,
- i_{MAX} = maximum number of equilibrium correction iterations permitted within each time increment, (default $i_{MAX} = 15$),

| | | |
|---------------------|---|---|
| v_{TOL} | = | a decimal number much less than 1.0, representing the incremental energy convergence tolerance, (default $v_{TOL} = .001$), |
| $i_{ISL}/'a_{ISL}'$ | = | integer or alphanumeric id of the initial stress loading, |
| 'fn' | = | alphanumeric string to be used in the construction of the dynamic analysis results file names. The length of the alphanumeric string is limited to 24 characters. |

Explanation:

The DYNAMIC PARAMETERS command includes the following options which are used to control a nonlinear dynamic analysis:

BLOCK SIZE i_{BS}

By default, GTSTRUDL attempts to perform the nonlinear dynamic analysis solution entirely within the virtual memory (RAM plus page file) of the computer. If this is not possible or practical, the BLOCK SIZE parameter may be specified, indicating that an out-of-core solution procedure is to be used. The out-of-core solution process divides the equations of motion into blocks of maximum double word size i_{BS} , which are stored in scratch files. The blocks are transferred back and forth between the scratch files and virtual memory during the solution process, and the scratch files are deleted from the system when the solution is complete. The block transfer process results in longer solution times, and while the blocked solution process requires less virtual memory, sufficient hard disk space is needed for the storage of the blocks. If a value of 0 is given for i_{BS} , the solution procedure is reset to the in-core method.

UPDATE STIFFNESS EVERY i_U TIME STEPS

The nonlinear dynamic analysis is a modified Newton-Raphson process whereby the an effective dynamic stiffness matrix is updated, i.e. re-computed, only at the beginning of a time step and only at the time step interval specified by i_U . If this option is not specified or a value of 0 is specified for i_U , the effective dynamic stiffness matrix is computed at the beginning of the first time step and remains constant thereafter.

MAXIMUM NUMBER OF EQUILIBRIUM CYCLES i_{MAX}

Equilibrium correction cycles are executed for every time step. This option is used to specify the maximum permissible number of these cycles in any given time step. If a value for i_{MAX} is not specified, 15 is assumed. If the maximum number of cycles is reached and convergence has not been achieved, then the entire analysis is terminated.

CONVERGENCE TOLERANCE ENERGY v_{TOL}

The CONVERGENCE TOLERANCE ENERGY option specifies the value for the convergence tolerance used in the incremental energy convergence check. If not specified, a convergence tolerance value of 0.001 is assumed.

INITIAL STRESS LOAD OFF i_{ISL}/a_{ISL}

This option identifies a loading condition, i_{ISL}/a_{ISL} , from which an initial displacement and stress state is established for the subsequent nonlinear dynamic analysis. The analysis starts with the structure in this state, including the applied loading that produced it. The initial stress load induces no acceleration at the start of the analysis, and is carried throughout the entire analysis.

The initial stress load is typically a self-weight load applied to the structure, for which a static stiffness or nonlinear analysis has been previously executed. The initial stress load also may be the prestress loading from the previous prestress analysis of a cable structure (see Section 2.6.3, Volume 3, GTSTRUDL User Reference Manual).

The INITIAL STRESS LOAD OFF command is used to switch off the inclusion of an initial stress load for a nonlinear dynamic analysis in the case where an initial stress load was specified for a previous nonlinear dynamic analysis.

RESULTS FILE NAME 'fn'

Nonlinear dynamic analysis uses the external file solver procedure described in Section 2.4.5.3 (Section 5.10 of this Release Guide), whereby the results of the dynamic analysis – displacements, velocities, accelerations, member forces, etc. – are permanently stored at the end of each time step into external save files rather than in virtual memory. The RESULTS FILE NAME option specifies an alphanumeric string to be used as a file name prefix in the creation of the results file names. If this option is not given, then the file name prefix string 'fn' is taken as the problem id given in the STRUDL command (Section 2.1.2.3, Volume 1, GTSTRUDL User Reference Manual)

or the CHANGE ID command (Section 2.1.2.5, Volume 1, GTSTRUDL User Reference Manual). If a problem id is not specified in either of these two commands, then 'fn' is taken as 'DyJob'.

It is recommended that the RESULTS FILE NAME command be given only once in a given job, regardless of the number of nonlinear dynamic analyses performed for different transient loading conditions. Giving a new RESULTS FILE NAME command prior to each nonlinear dynamic analysis for a new transient loading is permitted; however, doing so will make it cumbersome later to access the results from the different loading conditions. Prior to accessing the results from a particular transient loading condition, it will be necessary to re-issue the RESULTS FILE NAME command with the value of 'fn' initially used to create the results for that loading condition.

Modifications:

The DYNAMIC PARAMETERS command functions identically in ADDITIONS, CHANGES, and DELETIONS modes. To change the value of a dynamic parameter, simply re-specify the new value, regardless of input mode.

2.5.6.2 DYNAMIC ANALYSIS NONLINEAR Command

General form:

$$\text{DYNAMIC ANALYSIS NONLINEAR} \quad \left\{ \begin{array}{l} \text{BETA} \quad v_b \\ \text{NJP} \quad i_{\text{NJP}} \end{array} \right\}^*$$

Elements:

v_b = decimal value of \$ in the Newmark-\$ formulation of direct integration. The default value is 0.25, corresponding to the constant average acceleration method.

i_{NJP} = integer number of joints per partition.

Explanation:

The DYNAMIC ANALYSIS NONLINEAR command is used to execute a nonlinear dynamic analysis, using the Newmark-\$ formulation as described in Section 2.4.2.4.2 of Volume 3, GTSTRUDL User Reference Manual.

The BETA option is used to change the value of \$ in the Newmark-\$ formulation from the default value of 0.25, which corresponds to the unconditionally stable constant average acceleration method.

The NJP option is used to specify the number of joints per partition, which has only a minimal effect on the efficiency of the nonlinear dynamic analysis procedure. The default NJP value is 36/JF, where JF = the number of degrees-of-freedom associated with the type of member or element specified in the last TYPE command (Section 2.1.5.2.2, Volume 1, GTSTRUDL User Reference Manual, Section 2.3.4, Volume 3, GTSTRUDL User Reference Manual).

Nonlinear dynamic analysis supports all available TRANSIENT LOADING options.

The nonlinear dynamic analysis calculates and stores nodal displacements, velocities, accelerations, and element nodal forces for nonlinear geometric frame and truss members, tension/compression-only members, friction damper members, nonlinear spring elements, NLS4PH elements, and IPCABLE elements. Member end forces and finite element nodal forces, and finite element stresses and strains may be

computed and stored for linear members and finite elements by using the COMPUTE TRANSIENT command described in Section 2.4.5.7, Volume 3, GTSTRUDL User Reference Manual.

All currently available dynamic analysis results processing functions support the results from nonlinear dynamic analysis.

Nonlinear dynamic analysis does not presently support the following capabilities:

1. Superelements
2. Joint constraints, i.e. joint ties and rigid bodies (Section 2.6.5, Volume 3, GTSTRUDL Reference Manual).
3. Dynamic degrees of freedom condensation using the DYNAMIC DEGREES OF FREEDOM command (Section 2.4.5.1, Volume 3, GTSTRUDL User Reference Manual).

If superelements are detected, the following error message is printed and the analysis is terminated:

```
**** ERROR_STDNL1 --      Superelement(s) detected.  
                          Analysis terminated and SCAN mode  
                          entered.
```

If joint constraints are detected, the following error message is printed and the analysis is terminated:

```
**** ERROR_STDNL1 --      Joint constraints specified but not  
                          presently supported by nonlinear  
                          dynamic analysis.  
                          Analysis terminated and SCAN mode  
                          entered.
```

If dynamic condensation of dynamic degrees of freedom is detected, the following error message is printed and the analysis is terminated.

```
**** ERROR_STDNL1 --      Dynamic DOF condensation requested but  
                          not presently supported by nonlinear  
                          dynamic analysis.  
                          Analysis terminated and SCAN mode  
                          entered.
```

2.5.6.3 Nonlinear Dynamic Analysis Example

Figures 2.5.6.3-1 and 2.5.6.3-2 show sketches of a simple two-story space frame structure which is the subject structure of the nonlinear dynamic analysis example problem command listing given in Figure 2.5.6.3-3. Figure 2.5.6.3-1 shows the structure with joints labeled and Figure 2.5.6.3-2 shows the structure with beam and column members and floor finite elements labeled. Figure 2.5.6.3-3 contains the command input file for this example, including comments which describe the use of the nonlinear dynamic analysis commands described in the previous sections.

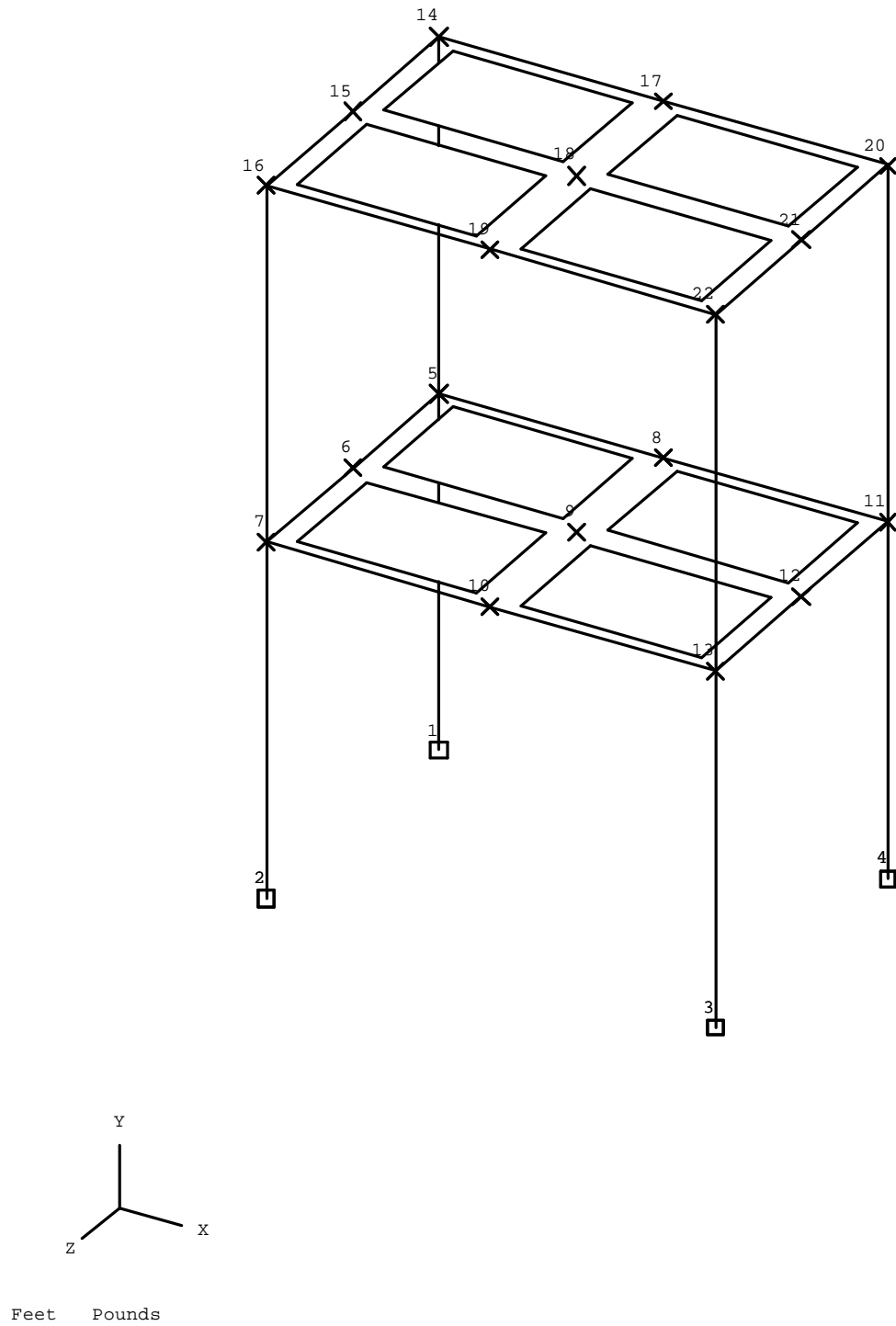
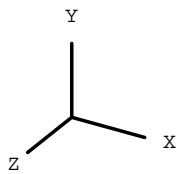
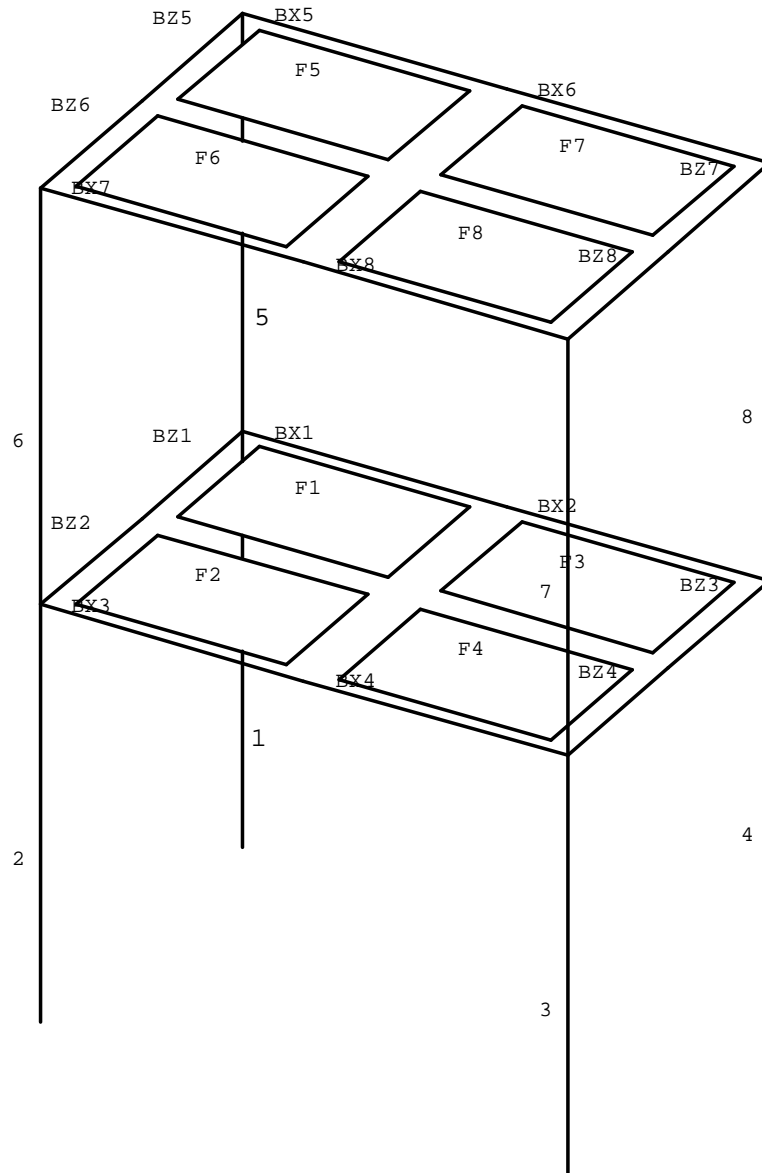


Figure 2.5.6.3-1 Nonlinear Dynamic Analysis Example Structure – Joints Labeled



Feet Pounds

Figure 2.5.6.3-2

Nonlinear Dynamic Analysis Example Structure – Members and Finite Elements Labeled

```

STRUDL 'NlDyEx' 'Nonlinear Dynamic Analysis Example Problem'
$* **
$*****
$*****
$* **
$
$
$ Geometry
$
UNITS FEET
JOINT COORD
  1      0.0      0.0      0.0 S
  2      0.0      0.0     10.0 S
  3     15.0      0.0     10.0 S
  4     15.0      0.0      0.0 S

GENERATE 3 JOINTS ID 5 1 X 0.0 Y 12.0 Z 0.0 5.0
REPEAT 2 TIMES ID 3 X 7.5
REPEAT 1 TIME ID 9 Y 12.0

TYPE SPACE FRAME
MEMBER INCIDENCES
1 1 5
2 2 7
3 3 13
4 4 11
GEN 4 MEMBERS ID 5 1 FROM LIST 5 7 13 11 TO LIST 14 16 22 20
GEN 2 MEMBERS ID 'BX1' 1 FROM 5 3 TO 8 3
REPEAT 1 TIME ID 2 FROM 2 TO 2
REPEAT 1 TIME ID 4 FROM 9 TO 9
GEN 2 MEMBERS ID 'BZ1' 1 FROM 5 1 TO 6 1
REPEAT 1 TIME ID 2 FROM 6 TO 6
REPEAT 1 TIME ID 4 FROM 9 TO 9
TYPE PLATE
GEN 2 ELEMENTS ID 'F1' 1 FROM 5 1 TO 6 1 TO 9 1 TO 8 1
REPEAT 1 TIME ID 2 FROM 3 TO 3 TO 3 TO 3
REPEAT 1 TIME ID 4 FROM 9 TO 9 TO 9 TO 9

MATERIAL CONC
UNITS INCHES
MEMBER DIMENSIONS
  1 TO 8 RECT B 8.0 H 12.0
  'BX1' TO 'BX8' 'BZ1' TO 'BZ8' RECT B 8.0 H 18.0
ELEMENT PROPERTIES
  'F1' TO 'F8' TYPE 'SBHQ6' THICK 4.0

$
$ Define self weight and dead load 1. This load is used as
$ the initial stress loading for the nonlinear dynamic analysis.
$
UNITS FEET LBS
DEAD LOADS 1 DIR -Y MEMBERS 1 TO 8 -
  'BX1' TO 'BX8' 'BZ1' TO 'BZ8'
MEMBER LOADS

```

```
'BX1' 'BX2' FORCE Y GLOB UNI FR W -500.0 LA 0.0 LB 1.0
ELEMENT LOADS
  'F1' TO 'F8' BODY FORCES GLOBAL BY -150.0

$
$ Define nonlinear geometric behavior for column
$ members 1 to 8. All other members and finite
$ elements remain linear.
$
NONLINEAR EFFECTS
  GEOMETRY MEMBERS 1 TO 8

$
$ Execute a nonlinear analysis for loading 1 to
$ establish the initial stress conditions.
$
MAXIMUM NUMBER OF CYCLES 10
CONVERGENCE TOLERANCE DISPL 0.001
LOAD LIST 1
NONLINEAR ANALYSIS

UNITS INCHES CYCLES SECS
LIST DISPLACEMENTS
LIST FORCES
LIST SUM REACTIONS

$
$ Add dynamic analysis data including inertia and
$ dynamic loading data. This data is specified
$ in the same manner as for linear dynamic analysis.
$
UNITS LBS
INERTIA OF JOINTS LUMPED
INERTIA OF JOINTS WEIGHT
  3 4 5 6 TRANSLATION ALL 10000.0

TRANSIENT LOAD 'EQ-X'
SUPPORT ACCELERATION
  TRANSL X FILE 'ELCENTRO'
  INTEGRATE FROM 0.0 TO 10.0 AT 0.01
END TRANS LOAD

TRANSIENT LOAD 'EQ-Z'
SUPPORT ACCELERATION
  TRANSL Z FILE 'ELCENTRO' FACTOR 0.5
  INTEGRATE FROM 0.0 TO 10.0 AT 0.01
END TRANS LOAD

$
$ This block of DYNAMIC PARAMETERS commands contains the most
$ commonly used control data specifications for nonlinear
$ dynamic analysis including the MAXIMUM, CONVERGENCE TOLERANCE,
$ INITIAL STRESS, UPDATE, and RESULTS FILE commands.
$
```

```
DYNAMIC PARAMETERS
  MAXIMUM NUMBER OF EQUILIBRIUM CYCLES 30
  CONVERGENCE TOLERANCE 0.0001
  INITIAL STRESS LOAD 1
  UPDATE STIFFNESS EVERY 3 TIME STEPS
  PRINT MAX
  RESULTS FILE NAME 'N1DyEX'
END DYNAMIC PARAMS

$
$ Execute the nonlinear dynamic analysis
$
DYNAMIC ANALYSIS NONLINEAR

$
$ Perform normal backsubstitution and results processing
$ operations
$
COMPUTE TRANSIENT FORCES
COMPUTE TRANSIENT STRESSES
COMPUTE TRANSIENT REACTIONS LOADS

LIST TRANSIENT FORCES TIMES FROM 1 TO 10 MEMBER 1 2 'BX1' 'BZ1'
LIST TRANSIENT MAX FORCES MEMBERS 1 2 'BX1' 'BZ1'
LIST TRANSIENT MAX DISPL JOINT 5

CREATE PSEUDO STATIC LOAD 3 FROM TIME 2 OF LOAD 'EQ-X'
CREATE PSEUDO STATIC LOAD 4 FROM MAX OF LOAD 'EQ-X'
CREATE PSEUDO STATIC LOAD 5 FROM TIME 2 OF LOAD 'EQ-Z'
CREATE PSEUDO STATIC LOAD 6 FROM MAX OF LOAD 'EQ-Z'
CREATE PSEUDO STATIC LOAD 7 FROM TIME 2 OF LOADS 'EQ-X' 'EQ-Z'
CREATE PSEUDO STATIC LOAD 8 FROM MAX OF LOADS 'EQ-X' 'EQ-Z'

LOAD LIST 1 3 TO 8
UNITS INCHES KIPS
OUTPUT BY MEMBER
LIST DISPL JOINT 5
LIST FORCES MEMBERS 'BX1' 'BZ1'

SECTION FR NS 3 0.0 0.5 1.0
LIST SECTION FORCES MEMBERS 'BX1' 'BZ1'

FIN
```

Figure 2.5.6.3 Command Input File for Nonlinear Dynamic Analysis Example

5.3.8 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

The Sections shown below are numbered as they will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.4.3.7 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

This section describes the commands that are used to incorporate the viscous damper element (dash pot) into a structural model that is used for linear and nonlinear dynamic analysis by the direct integration procedure. The commands that are used for this purpose include:

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

2.4.3.7.1 The DAMPER ELEMENT DATA Command

Tabular form:

DAMPER ELEMENT (DATA)

$$\begin{array}{l} \left\{ \begin{array}{c} i_D \\ 'a_D' \end{array} \right\} \underline{\text{INCIDENCES}} \left\{ \begin{array}{c} i_S \\ 'a_S' \end{array} \right\} \left(\left\{ \begin{array}{c} i_E \\ 'a_E' \end{array} \right\} \right) \left\{ \begin{array}{c} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{array} \right\} - \\ \quad [\underline{\text{CTX}}] \ v_{\text{CTX}} \quad [\underline{\text{CTY}}] \ v_{\text{CTY}} \quad [\underline{\text{CTZ}}] \ v_{\text{CTZ}} \quad [\underline{\text{CRX}}] \ v_{\text{CRX}} \quad [\underline{\text{CRY}}] \ v_{\text{CRY}} \quad [\underline{\text{CRZ}}] \ v_{\text{CRZ}} \\ \bullet \\ \bullet \\ \bullet \\ \left\{ \begin{array}{c} i_D \\ 'a_D' \end{array} \right\} \underline{\text{INCIDENCES}} \left\{ \begin{array}{c} i_S \\ 'a_S' \end{array} \right\} \left(\left\{ \begin{array}{c} i_E \\ 'a_E' \end{array} \right\} \right) \left\{ \begin{array}{c} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{array} \right\} - \\ \quad [\underline{\text{CTX}}] \ v_{\text{CTX}} \quad [\underline{\text{CTY}}] \ v_{\text{CTY}} \quad [\underline{\text{CTZ}}] \ v_{\text{CTZ}} \quad [\underline{\text{CRX}}] \ v_{\text{CRX}} \quad [\underline{\text{CRY}}] \ v_{\text{CRY}} \quad [\underline{\text{CRZ}}] \ v_{\text{CRZ}} \end{array}$$

END (OF DAMPER ELEMENT DATA)

Elements:

- i_D/a_D = integer or alphanumeric name of the new damper element. The name must be unique among all previously defined damper elements and is restricted to no more than eight digits or alphanumeric characters.
- i_S/a_S = integer or alphanumeric name of a previously defined joint to be the starting incident joint of the new damper element.
- i_E/a_E = optional integer or alphanumeric name of the previously defined joint to be the ending incident joint of the new damper element. The starting joint and ending joint names must be different.
- V_{CTX} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL X direction. Active force, length, and time units apply [force/(length/time)].
- V_{CTY} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL Y direction. Active force, length, and time units apply [force/(length/time)].
- V_{CTZ} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL Z direction. Active force, length, and time units apply [force/(length/time)].
- V_{CRX} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].
- V_{CRY} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].
- V_{CRZ} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].

Explanation:

The DAMPER ELEMENT DATA command is used to create new viscous damper elements and define their joint connectivity and damping force and moment properties. The viscous damper element data are entered by giving the DAMPER ELEMENT DATA command header first, followed by one or more tabular element data entry lines of the form:

$$\left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\} \underline{\text{INCIDENCES}} \left\{ \begin{matrix} i_s \\ 'a_s' \end{matrix} \right\} \left(\left\{ \begin{matrix} i_E \\ 'a_E' \end{matrix} \right\} \right) \left\{ \begin{matrix} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{matrix} \right\} -$$

$$[\underline{\text{CTX}}] \ v_{\text{CTX}} \quad [\underline{\text{CTY}}] \ v_{\text{CTY}} \quad [\underline{\text{CTZ}}] \ v_{\text{CTZ}} \quad [\underline{\text{CRX}}] \ v_{\text{CRX}} \quad [\underline{\text{CRY}}] \ v_{\text{CRY}} \quad [\underline{\text{CRZ}}] \ v_{\text{CRZ}}$$

for each new damper element. This data entry line consists of the element name, the element incidences, the element orientation, and the element viscous damping coefficients, which are described in greater detail as follows:

Element name $\left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\}$

Each new damper element must be given an integer or alphanumeric name that is unique among all other existing damper element names. The name may not exceed eight digits or alphabetic characters. The name may be a duplicate of a previously defined member or finite element name.

$\underline{\text{INCIDENCES}} \left\{ \begin{matrix} i_s \\ 'a_s' \end{matrix} \right\}$

The damper element connectivity is defined by one or two incident joints. The first incident joint, $i_s/'a_s'$, defines the start of the element. The second incident joint, $i_E/'a_E'$, is optional and defines the end of the element. If only one joint is given, the second joint is taken as a totally fixed support joint; it is fictitious and invisible. The specified joints must have been previously defined and if two are specified, they must be different. However, they may be coincident. The only restriction on the selection of incident joints is that they may not be slave joints.

$$\left\{ \begin{array}{l} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{array} \right\}$$

The GLOBAL and LOCAL options are used to specify the coordinate reference frame for the damper element. The GLOBAL option, which is the default, means that the element is a global element and that the six element damping degrees-of-freedom are defined with respect to the global coordinate system. The LOCAL option means that the element damping degrees-of-freedom are defined with respect to the element local coordinate system, which is identical to the local joint-to-joint coordinate system for frame members. The only difference between the frame member and damper element local coordinate systems is that the damper element does not support the Beta angle. If the LOCAL option is specified, but the joint-to-joint length of the element is equal to 0 (# 10⁻⁵ inches), then GLOBAL is assumed. In addition, GLOBAL is automatically assumed for any damper element for which only one incident joint is specified.

$$[\underline{\text{CTX}}] v_{\text{CTX}} [\underline{\text{CTY}}] v_{\text{CTY}} [\underline{\text{CTZ}}] v_{\text{CTZ}} [\underline{\text{CRX}}] v_{\text{CRX}} [\underline{\text{CRY}}] v_{\text{CRY}} [\underline{\text{CRZ}}] v_{\text{CRZ}}$$

These decimal data values represent the damping coefficient values on the diagonal of the uncoupled element damping matrix, which has the following form:

$$\begin{bmatrix} \text{CTX} & 0 & 0 & 0 & 0 & 0 \\ & \text{CTY} & 0 & 0 & 0 & 0 \\ & & \text{CTZ} & 0 & 0 & 0 \\ & (\text{sym}) & & \text{CRX} & 0 & 0 \\ & & & & \text{CRY} & 0 \\ & & & & & \text{CRZ} \end{bmatrix}$$

These values refer to the element damping translational and rotational degrees-of-freedom with respect to the specified coordinate system, GLOBAL, the default, or LOCAL. Only non-zero values need be specified.

Command processing is completed when the END option is given.

The damping properties from the viscous damper elements are assembled into the total global system damping matrix of the equations of motion that are solved using the direct integration methods executed by the DYNAMIC ANALYSIS PHYSICAL and DYNAMIC ANALYSIS NONLINEAR commands. The viscous damper element data are used only by the execution of these two commands

Modifications:

The DAMPER ELEMENT DATA command operates only in the ADDITIONS mode. If the command is given when the active input mode is CHANGES or DELETIONS, then the command execution is terminated and the command data are ignored. If it is necessary to change the data for an existing damper element, then use the DELETE DAMPER ELEMENT command described in Section 2.4.3.7.3 to delete the damper element to be changed, followed by the re-specification of the new data in the DAMPER ELEMENT DATA command. All of these steps are performed in ADDITIONS mode.

Example:

The following example illustrates the creation of two damper elements DAMP1 and DAMP2. DAMP1 spans from joint 2 to joint 10 and has one damping coefficient equal to 10^7 kips/(inches/second) corresponding to translation in the local y direction of the element. DAMP2 spans from joint 1 to joint 2 and has global damping factors $CTX = 100$ kips/(inches/second) and $CRZ = 1000$ kip-inches/(radians/second). The damping coefficients for element DAMP2 are referenced with respect to the global coordinate system because the GLOBAL/LOCAL option was not given. The execution of this example depends on DAMP1 and DAMP2 not having been previously defined and joints 1, 2, and 10 being valid joints.

```

UNITS KIPS INCHES RADIANS
DAMPING ELEMENT DATA
  'DAMP1' INC 2 10 LOCAL CTY 1.E7
  'DAMP2' INC 1 2 CTX 100.0 CRZ 1000.0
END

```

Errors:

1. When two or more damper elements are defined with the same name, the following warning message is printed. Command processing is terminated for the offending element and continues for subsequent elements.

```

{ 10} > DAMPING ELEMENT DATA
{ 11} >  'DAMP1' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{ 12} >  'DAMP1' INC 2 4 GLOBAL CTY 1.E7

**** WARNING_STDELD -- Damper element DAMP1      previously defined.  Command ignored.

{ 13} >  'DAMP3' INC 3 3 GLOBAL CTY 1.E7
{ 14} > END

```

Element DAMP1 is successfully created by the first tabular command entry. The warning message for DAMP1 is printed for the second tabular entry for DAMP1. Command processing continues with the tabular entry for DAMP3.

2. The following warning message is printed if one or both of the specified element incidence joints are not defined. Command processing continues with the tabular entry for the next element.

```
{ 10} > DAMPING ELEMENT DATA
{ 11} > 'DAMP1' INC 2 10 LOCAL CTY 1.E7

**** WARNING_STDELD -- Damper element incidence joint not defined.
                        Command ignored.

{ 12} > 'DAMP2' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{ 13} > END
```

The warning message indicates that one or both of the specified element incidences for element DAMP1 are not defined.

3. The following warning message is printed when the starting and ending element incidence joints are the same. Command processing continues with the tabular entry for the next element.

```
{ 10} > DAMPING ELEMENT DATA
{ 12} > 'DAMP1' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{ 13} > 'DAMP2' INC 2 4 GLOBAL CTY 1.E7
{ 14} > 'DAMP3' INC 3 3 GLOBAL CTY 1.E7

**** WARNING_STDELD -- Damper element starting and ending incident joints are the
                        same. Command ignored.

{ 15} > 'DAMP4' INC 4 5 CTY 1.E7
{ 16} > END
```

2.4.3.7.2 The PRINT DAMPER ELEMENT DATA Command

General form:

PRINT DAMPER (ELEMENT DATA)

Explanation:

The PRINT DAMPER ELEMENT DATA is used to print a table of the damper element data for all existing damper elements. The following is an example of the printed output from this command:

Example:

The following example illustrates the format for the output from the PRINT DAMPER ELEMENT command.

```
{ 17} > PRINT DAMPING ELEMENT DATA
*****
* RESULTS FROM LATEST ANALYSIS *
*****

ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
      LENGTH      WEIGHT      ANGLE      TEMPERATURE      TIME
      FEET        LB         RAD         DEGF          SEC

Damping Element Data
=====
Element      Start Jnt  End Jnt  CTX      CTY      CTZ      CRX      CRY      CRZ
-----
DAMP1  LOC   1         2      100.0    0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  1000.
DAMP2  GLO   2         4    0.0000E+00  0.1000E+08  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
```

Errors:

The following warning message is printed when no damper element data exists.

```
{ 9} > PRINT DAMPING ELEMENT DATA
*****
* RESULTS FROM LATEST ANALYSIS *
*****
```

```

ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
      LENGTH      WEIGHT      ANGLE      TEMPERATURE      TIME
      FEET        LB         RAD         DEGF          SEC

Damping Element Data
=====

Element      Start Jnt   End Jnt      CTX      CTY      ...      CRZ
-----
**** INFO_STPDED -- Damper element data have not been defined.

```

2.4.3.7.3 The DELETE DAMPER ELEMENT DATA Command

General form:

$$\underline{\text{DELETE}} \underline{\text{DAMPER}} (\underline{\text{ELEMENT}} \underline{\text{DATA}}) \left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\} \dots \left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\}$$

Elements:

$i_D / 'a_D'$ = integer or alphanumeric name of damper element to be deleted. The name is limited to no more than eight digits or characters.

Explanation:

This command is used to delete previously defined damper elements. The names of the elements to be deleted are given in the list of individually named damper elements. No other list construct, such as "1 TO 10" is permitted. Specified damper elements that are not defined are ignored.

5.4 General Prerelease Features

5.4.1 Align Command

The ALIGN command will align joints in a line of members in a column to ensure that the column is parallel to the global Y axis. The section below is numbered as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.12.16 The ALIGN Command

This command will move joints at the "top" (top if the Y axis is vertical) of members parallel to the global Y axis, so that the "top" joint has the same X and Z coordinates as the "bottom" joint. This is to ensure that all these members follow the 'Special Case' for local reference frame orientation. See Section 1.10.4 in Volume 1 of the GTSTRUDL User Reference Manual or Section 8.5, "The BETA Angle", in the GTSTRUDL Analysis Users Guide. This command is useful for situations where coordinates are calculated to a precision that does not ensure alignment with the Y axis, such as coordinates created by an outside program, or use of the GENERATE or OBJECT COPY commands with many copies or large coordinates. Since the check for being parallel is very exacting in GTSTRUDL (within 0.01%), it is possible for a column line to have some columns that are 'Special Case' members and others that are not, resulting in inconsistent default orientations.

Notes:

This "out of alignment" problem can also occur for beams parallel to the Y global axis in "Z up" structures.

MEMBER ECCENTRICITIES are not considered during the alignment check because they are not used to determine the default member orientation.

Syntax:

ALIGN (MEMBERS list) (REPORT (ONLY)) (ALONG Y AXIS) -
(TOLERANCE v_1 (RATIO)) (ITEMIZE (WITH DETAIL))

Where,

list is an optional list of members, as defined by Section 2.1.2.2 of this volume.

v_1 is the specified TOLERANCE used to define "parallel to global Y"

Explanation:

MEMBERS If no list is given, all members will be checked.

REPORT Don't change joint coordinates, only report detected possible alignment problems. Since the adjustment procedure changes joints from the "bottom" of the structure (starting at the most negative Y location) and works toward the "top", 'REPORT ONLY' alignment checks may be slightly different because "lower" joints will not have been changed.

TOLERANCE

Members which have an X or Z projection equal to or less than this value will be considered parallel to Y and will be checked for alignment. The default value is 0.1 inch (2.5 mm).

RATIO This option specifies that the tolerance specified be used as a ratio: (X or Z projection)/(member length). This option allows you to have more control in special situations. To avoid possible problems the TOLERANCE value is not permitted to exceed 0.05.

ITEMIZE This option will generate a message for each joint that is changed.

DETAIL This option will also print the old coordinates and the new coordinates of each changed joint.

The **ALIGN MEMBERS** command will change joint coordinates to keep members parallel to the global Y axis, unless the **REPORT** option is specified. A list of all members that meet the criteria for being considered "parallel to global Y" is built, respecting the default or user-specified values for **TOLERANCE** and **RATIO**. Members are then 'aligned' (joint coordinates changed) from the "bottom", or most negative Y coordinate, to the "top".

Examples:

```
ALIGN REPORT ONLY ITEMIZE
```

```
****INFO_STALGN - The following joints need to be adjusted to align members  
                  in the global Y direction:
```

```
Joint 5          needs to be aligned with joint 1  
Joint 9          needs to be aligned with joint 5
```

ALIGN ITEMIZE WITH DETAIL

```
****INFO_STALGN - The following joints were adjusted to align members
                   in the global Y direction:
      Joint 5      was aligned with joint 1
        Old (X,Z): 0.10000E+00 0.10000E+00 New (X,Z): 0.00000E+00 0.00000E+00
      Joint 9      was aligned with joint 5
        Old (X,Z): 0.20000E+00 0.20000E+00 New (X,Z): 0.00000E+00 0.00000E+00
      Joint 13     was aligned with joint 9
        Old (X,Z): -0.30000E+00 -0.30000E+00 New (X,Z): 0.00000E+00 0.00000E+00
      Joint 16     was aligned with joint 12
        Old (X,Z): 0.45100E+03 0.10000E+01 New (X,Z): 0.45000E+03 0.00000E+00

****INFO_STALGN - A total of      4 joints were adjusted.
```

ALIGN ITEMIZE

```
****INFO_STALGN - The following joints were adjusted to align members
                   in the global Y direction:
      Joint 5      was aligned with joint 1
      Joint 9      was aligned with joint 5
      Joint 13     was aligned with joint 9
      Joint 16     was aligned with joint 12

****INFO_STALGN - A total of      4 joints were adjusted.
```

ALIGN

```
****INFO_STALGN - A total of      4 joints were adjusted.
```

If no members were determined to be misaligned:

ALIGN

```
****INFO_STALGN - No joints required adjustment.
```

This page intentionally left blank.

5.4.2 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

$$\text{ROTATE } \text{LOADING} \left\{ \begin{array}{c} i_R \\ 'a_R' \end{array} \right\} (\text{ANGLES}) [\underline{T1}] r_1 [\underline{T2}] r_2 [\underline{T3}] r_3$$

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 $2_1, 2_2, 2_3$ as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

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

The ROTATE LOADING command is used to take the global applied loading components from an existing loading condition, rotate them through a set of rotation angles, and copy the new rotated global components to a new or modified different destination loading condition. The existing independent loading condition, the ROTATE load, from which the rotated global load components are computed is specified by the

loading name i_R/a_R . The angles of rotation are specified by the values r_1, r_2, r_3 . These rotation angles are defined according to the same conventions as those that define the local support release directions in the JOINT RELEASE command described in Section 2.1.7.2, Volume 1 of the GTSTRUDL User Reference Manual, and illustrated in Figure 2.1.7-1.

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

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

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

Examples:

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

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

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

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

Error Messages:

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

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

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

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

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

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

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

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

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

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

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

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

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

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


5.4.3 RUN Command

The RUN command allows you run external programs or DOS batch (cmd) files with a GTSTRUDL command. This is useful for automating procedures that rely on GTSTRUDL generated data, such as a user created design program that needs member ends forces from GTSTRUDL.

The RUN command has been improved to allow new options and longer commands. Implementation has changed from the "C" system library to a Microsoft API, which is more robust way to run external programs.

Syntax

RUN ((BATCH) (KEEP)) (WAIT) 'program'

where program = a ".exe", ".bat" or ".cmd" file or DOS command, along with arguments. The total length of 'program' is limited to 255 characters. You cannot use the quote/apostrophe character (') in 'program', but double quotes (") are acceptable.

Explanation:

- | | |
|-------|---|
| BATCH | Indicates that a ".bat" or ".cmd" file or DOS command will be executed. ".bat" or ".cmd" file are DOS script files, containing DOS commands or other programs. The starting directory for the DOS environment is the current GTSTRUDL working directory. The Microsoft API now being used needs to know that a DOS environment is needed before it runs. |
| KEEP | If BATCH is used, the KEEP option leaves the created DOS window active after the requested action is completed. You can then type new commands in the DOS window. You can kill the DOS window with the "X" in the upper, right hand corner when you are finished. |
| WAIT | Indicates that GTSTRUDL command processing should stop until 'program' has completed. If this option is not used, GTSTRUDL will continue and process the next command, i.e. if you are running a macro or an input file. Use the WAIT option if the results of 'program' are required for subsequent GTSTRUDL commands. |

Examples:

`RUN BATCH KEEP 'DIR/W'`

This command will open a DOS window, display the contents of the GTSTRUDL working directory in the "/W" format, and leave the DOS window open so you can type more commands. The DOS window is a separate program, so you can continue GTSTRUDL operations while the DOS window is open.

`RUN 'NOTEPAD myfile.txt'`

This command will open Notepad with file 'myfile.txt'. Again, Notepad is a separate process, so you can continue GTSTRUDL operations while Notepad is open.

`RUN BATCH WAIT 'find "$" input.gti > comments.gti'`
`CINPUT 'comments.gti'`

The first command will use the DOS command "find" to locate the comments in the input file "input.gti" and put them into a file named "comments.gti". The WAIT option tells the GTSTRUDL command processor to wait until the DOS operation is complete before trying to CINPUT the generated file.

5.4.4 COUTPUT Command

The COUTPUT command now can replace (overwrite) an existing output file. Previously, an existing file could be appended only.

$$\underline{\text{COUTPUT}} \left(\begin{array}{c} \rightarrow \underline{\text{APPEND}} \\ \underline{\text{REPLACE}} \\ \underline{\text{STANDARD}} \end{array} \right) ('file_name')$$

where,

'file_name' is a new or existing text file. 'file_name' is limited to 256 characters and must be enclosed in quotes (apostrophes).

Explanation:

APPEND is the default action, so "COUTPUT 'file1'" and "COUTPUT APPEND 'file1'" are equivalent. APPEND tells GTSTRUDL to add subsequent output to the end of the specified file. If APPEND is requested, 'file_name' must be given.

REPLACE tells GTSTRUDL to delete the contents of the specified file and the write subsequent output to the specified file. If REPLACE is requested, 'file_name' must be given.

APPEND and REPLACE act identically when 'file_name' does not already exist. While GTSTRUDL is in the APPEND or REPLACE state, only input (commands) are echo printed in the text window - all generated output will be placed in the specified output file.

STANDARD tells GTSTRUDL to stop writing to the specified output file and direct subsequent output to the text window. This is the output state when GTSTRUDL starts.

Usage:

COUTPUT APPEND 'file1'

All subsequent output, from PRINT, LIST, etc., will be written to 'file1' and will not appear in the text window, although the actual command will be displayed in the text window. If 'file1' existed previously to this COUTPUT request, the new output will appear at the end of the existing contents.

COUTPUT REPLACE 'file2'

All subsequent output, from PRINT, LIST, etc., will be written to 'file2' and will not appear in the text window, although the actual command will be displayed in the text window. If 'file2' existed previously to this COUTPUT request, the existing contents will be deleted and only the new output will appear in 'file2'.

COUTPUT STANDARD

Stop writing output to an output file and write all output to the text window.

5.4.5 Reference Coordinate System Command

General form:

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

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

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 GTSTRUDL Release Guide) to facilitate the creation of the structural model. Reference Coordinate systems created using the above command will be available as Local systems in GTMenu. In a future release, the user will be able to output results such as joint displacements and reactions in a Reference Coordinate System.

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

- (1) Define the origin and rotation of coordinate axes of the reference system with respect to the global coordinate system, and
- (2) define three joints or the coordinates of three points in space.

In either case, i_1 or ' a_1 ' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x , v_y , and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x , v_y , and v_z , are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive

directions of these axes; v_1 , v_2 , and v_3 are the rotation angles R_1 , R_2 , and R_3 in active angular units between the orthogonal axes of this system and the axes of the overall global coordinate system. The description of these angles is the same as given in Section 2.1.7.2 of Volume 1 of the GTSTRUDL User Reference Manuals for rotated joint releases (2_1 , 2_2 , and 2_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 'a₂' 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 'a₃' 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 'a₄' or v_{10} , v_{11} , and v_{12}) is used to define the XY-plane of the reference system. The positive Y-axis is directed toward the third joint. The Z-axis then is determined by the right-hand rule.

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

Modifications of Reference Systems:

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

Examples:

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

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

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

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

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

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

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

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

- e) DELETIONS
 REFERENCE SYSTEM 2
 ADDITIONS

The above command deletes Reference System 2.

5.4.5-1 Printing Reference Coordinate System Command

General form:

$$\text{PRINT REFERENCE (COORDINATE) (SYSTEM) } \left\{ \begin{array}{l} \rightarrow \text{ALL} \\ \text{list} \end{array} \right\}$$

Explanation:

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

5.4.6 Hashing Algorithm to Accelerate Input Processing

An advanced data-structuring technique called HASHING can now be used when storing and searching lists of joints and/or elements. The command to control this feature is as follows:

$$\text{SET ELEMENTS} \left\{ \begin{array}{l} \text{HASHED} \\ \text{SEQUENTIAL} \end{array} \right\}$$

The following points concern HASHING:

- 1) The benefit of HASHING is that it GENERATES large structures faster. The disadvantage is that it is more complex internally.
- 2) HASHING is disabled by GTMenu. The GTSTRUDL database is usually not modified extensively in GTSTRUDL after invoking GTMenu, so this has minimal affect. However, the SET ELEMENTS HASHED command, when given with an existing database, builds hashing data structures for the existing database.
- 3) The order of a joint and/or element listing is the same for HASHED and SEQUENTIAL unless the structural database has been edited in DELETIONS mode and then in ADDITIONS mode again. Then SEQUENTIAL will place the latest addition in the deleted slot whereas HASHING will append the addition to the end of the list.

This page intentionally left blank.

5.4.7 Add/Delete Without Commands

2.1.3.7 Delete Joints Without Joint Coordinates

General form:

DELETE JOINTS WITHOUT (JOINT) COORDINATES

Explanation:

The DELETE JOINTS WITHOUT JOINT COORDINATES command can be used to delete the joints that do not have joint coordinates. There are times when a user may have accidentally specified data for a joint that does not exist or accidentally specified a non-existent joint name in the PRINT or LIST commands. Once a joint name is specified in a command, the joint name is stored in the database. If joint coordinates have not been specified for a joint and STIFFNESS ANALYSIS is requested, the following error message is output:

```
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 30
```

When the DELETE JOINTS WITHOUT JOINT COORDINATES command is specified, the program searches through all the joint names and deletes the joint names that do not have JOINT COORDINATES.

Example:

```
{ 100} >
{ 101} > STIFFNESS ANALYSIS
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 30
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 31
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 32
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 33
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 34
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 35
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 40
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 45
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 50
**** WARN_STDSY1 -- SCAN mode is ON. Analysis will be halted.

{ 102} >
{ 103} > SCAN OFF
{ 104} >
{ 105} > DELETE JOINTS WITHOUT JOINT COORDINATES
```

```
Following joints have been deleted: -----
30          31          32          33          34          35
40          45          50
```

In the above example, joints 30 to 35, 40, 45, and 50 accidentally were specified in the PRINT JOINT COORDINATES command. The command DELETE JOINTS WITHOUT JOINT COORDINATES deletes the joints that do not have joint coordinates.

2.1.3.8 Delete Members or Finite Elements Without Member or Element Incidences

General form:

DELETE MEMBERS WITHOUT (MEMBER) INCIDENCES

or

DELETE (FINITE) ELEMENTS WITHOUT (ELEMENT) INCIDENCES

Explanation:

The above two commands can be used to delete the members or finite elements that do not have member or element incidences. There are times when a user may have accidentally specified data for a member or a finite element that does not exist or accidentally specified a member or element name in the PRINT or LIST commands. Once a member or a finite element name has been specified in a command, the member or finite element name is stored in the database. If member or element incidences have not been specified for a member or element and STIFFNESS ANALYSIS is requested, the following error message is output.

```
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 20
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 45
```

When the DELETE MEMBERS WITHOUT MEMBER INCIDENCES command is specified, the program searches through all the member names and deletes the members that do not have MEMBER INCIDENCES. When the DELETE FINITE ELEMENTS WITHOUT ELEMENT INCIDENCES command is specified, the program searches through all the finite element names and deletes the elements that do not have ELEMENT INCIDENCES.

Example:

```

{ 97} >
{ 98} > STIFFNESS ANALYSIS
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 20
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 21
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 22
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 23
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 24
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 30
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 33
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 51
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 52
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 53
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 54
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 55
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 27
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 32
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 38
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 45
**** WARN_STDSY1 -- SCAN mode is ON. Analysis will be halted.

{ 99} >
{ 100} > SCAN OFF
{ 101} >
{ 102} > DELETE MEMBERS WITHOUT MEMBER INCIDENCES

```

Following members have been deleted: -----

| | | | | | |
|----|----|----|----|----|----|
| 20 | 21 | 22 | 23 | 24 | 30 |
| 33 | 51 | 52 | 53 | 54 | 55 |
| 27 | 32 | 38 | 45 | | |

In the above example, some of the members that are listed were accidentally specified in the MEMBER PROPERTIES command and some in the PRINT MEMBER INCIDENCES command. The command DELETE MEMBERS WITHOUT MEMBER INCIDENCES deletes the members that do not have member incidences.

This page intentionally left blank.

5.4.8 The List Sum Forces Command

The LIST SUM FORCES command is used to perform a general computation of resultant forces for a specified set of joints, members, and finite elements, and is described in the section below, which is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.3.7.4 The LIST SUM FORCES Command

General form:

$$\text{LIST SUM FORCES (TITLE 'a_T') (ABOUT } \left\{ \begin{array}{l} \text{JOINT} \left\{ \begin{array}{l} i_c \\ 'a_c' \end{array} \right\} \\ [X] v_X [Y] v_Y [Z] v_Z \end{array} \right\}) -$$

$$(\text{ORIENTATION } \left\{ \begin{array}{l} [TH1] v_1 [TH2] v_2 [TH3] v_3 \\ \left\{ \begin{array}{l} \text{MEMBER} \\ \text{ELEMENT} \end{array} \right\} \left\{ \begin{array}{l} i_o \\ 'a_o' \end{array} \right\} \end{array} \right\}) \text{ JOINTS list}_j \text{ MEMBERS list}_m$$

Elements:

| | | |
|---------------|---|---|
| 'a_T' | = | optional 16-character title. The default title is "None Given." |
| i_c/'a_c' | = | integer or alphanumeric name of the optional joint about which moment summation is computed. |
| v_X, v_Y, v_Z | = | decimal values of the global X, Y, and Z coordinates of the optional point about which moment summation is computed. A value of 0.0 is assumed for coordinates not specified. |
| v_1, v_2, v_3 | = | decimal values of the rotation angles that define the orientation of the optional local reference frame in which the force summation results are to be reported. The rotation angles are the same ones used to define the local released support reference frame described in Section 2.1.7.1, Volume 1 of the GTStrudl Reference Manual and illustrated by Figure 2.1.7.1 of that same Reference Manual section. |
| i_o/'a_o' | = | integer or alphanumeric name of the member or finite element that is to be used as the basis of the local reference frame in which the force summation results are to be reported. |
| list_j | = | list of joints at which forces and moments from the members and finite elements of list_m are accumulated for the force summation computation. |

list_m = list of members and finite elements that contribute forces at the joints of list_j for the force summation computation.

Explanation:

The LIST SUM FORCES command is used to calculate and report the summation of forces and moments at a specified point from a specified set of member and finite element nodal forces for all active static loading conditions. This command is similar to, but more general than, the CALCULATE RESULTANT command described in Section 2.3.7.3, Volume 3 of the GTSTRUDL Reference Manual. The LIST SUM FORCES command options are described as follows:

$$\underline{\text{ABOUT}} \left\{ \begin{array}{l} \underline{\text{JOINT}} \left\{ \begin{array}{l} i_c \\ 'a_c' \end{array} \right\} \\ [\underline{X}] \ v_X \ [\underline{Y}] \ v_Y \ [\underline{Z}] \ v_Z \end{array} \right\}$$

By default, the moment summation is computed by summing the member and finite element forces and moments about the geometric center of the joints specified in list_j , assuming the joints are equally weighted. The ABOUT specs provide for the identification of an optional JOINT or point, v_X , v_Y , v_Z , about which the moment summation is performed. The specified joint must be defined, but may be inactive.

$$\underline{(\text{ORIENTATION}} \left\{ \begin{array}{l} [\underline{\text{TH1}}] \ v_1 \ [\underline{\text{TH2}}] \ v_2 \ [\underline{\text{TH3}}] \ v_3 \\ \left\{ \begin{array}{l} \underline{\text{MEMBER}} \\ \underline{\text{ELEMENT}} \end{array} \right\} \left\{ \begin{array}{l} i_o \\ 'a_o' \end{array} \right\} \end{array} \right\})$$

By default, the result of the force and moment summation is a list of three global resultant force components and three global resultant moment components for each active loading condition. The ORIENTATION specs provide a way to specify an optional alternative local reference frame for the summation report. In the first method, three rotation angles v_1 , v_2 , and v_3 define the orientation of the local reference frame with respect to the global reference frame. The rotation angles are the same ones used to define the local released support reference frame described in Section 2.1.7.1, Volume 1 of the GTSTRUDL Reference Manual and illustrated by Figure 2.1.7.1 of that same Reference Manual section. According to the MEMBER/ELEMENT option, the summation results are reported in the local or planar reference frame of the specified MEMBER or finite ELEMENT.

JOINTS list_j MEMBERS list_m

The LIST SUM FORCES command computes the force and moment summation from the nodal forces and moments at the joints specified in list_j from the members and finite elements specified in list_m. list_m may contain any mix of frame members, truss members, 2-D, and 3-D finite elements and list_j may contain any set of joints, with the only qualification that each element in list_m must be incident on at least one joint in list_j. In addition, every joint and element in list_j and list_m must be defined and active, and list_m may not contain rigid bodies. list_j may contain master joints and slave joints.

The force and moment summation is computed for all active static loading conditions for which member end forces and finite element nodal forces have been computed by a prior STIFFNESS ANALYSIS or by the CREATE PSEUDO STATIC, COMBINE, and CREATE LOAD COMBINATION commands. In the case of a pseudo static loading condition created from a response spectrum modal combination such as RMS or CQC, a force and moment summation is computed first for each mode of the root response spectrum load. The final mode combination force and moment summation is then computed by applying the mode combination rule of the pseudo static loading to the individual modal force and moment summations. In the case of a pseudo static loading condition created from the maximum of a transient loading condition, a force and moment summation is computed for each individual time point, from which the maximum force and moment summation component is then selected.

Error and Warning conditions

The LIST SUM FORCES command issues error and warning messages when the following conditions are encountered. In each case, command processing is terminated, but scan mode is not entered.

```
**** WARNING_STLFM -- Joint and/or member list not defined. LIST SUM
FORCES command ignored.
```

This message is issued when either the JOINT and/or MEMBER list are missing from the LIST SUM FORCES command.

```
**** WARNING_STLFM -- Specified joint 500 does not exist or is
inactive. LIST SUM FORCES command ignored.
**** WARNING_STLFM -- Specified element 500 does not exist or is
inactive. LIST SUM FORCES command ignored.
```

These messages mean that the indicated joint or element in list_j or list_m respectively has not been defined or is inactive.

```
**** WARNING_STLFISM -- Joint 500      for center of force summation does
                        not exist.  LIST SUM FORCES command ignored.
```

This message indicates that the specified joint in the ABOUT JOINT option is not defined.

```
**** WARNING_STLFISM -- None of the specified members are incident on
                        joint 24 .  LIST SUM FORCES command ignored.
```

```
**** WARNING_STLFISM -- Member 40     not incident on any of the specified
                        joints.  LIST SUM FORCES command ignored.
```

These messages are issued when the indicated joint is disconnected from the elements named in list_m, or when the indicated member is disconnected from the joints named in list_j respectively.

```
**** WARNING_STLFISM -- There are no active loading conditions.  LIST SUM
                        FORCES command ignored.
```

This message indicates that no active loading conditions were found.

```
**** WARNING_STLFISM -- Member/element force results not available for
                        member 25 loading 2.  LIST SUM FORCES command
                        ignored.
```

This message indicates that static analysis results do not exist for the specified member and loading condition. The output of this message obeys the OUTPUT WARNING MESSAGE LIMIT command.

```
**** WARNING_STLFISM -- Specified element RB1 is an invalid rigid body.
                        LIST SUM FORCES command ignored.
```

This message is issued when a rigid body is detected in list_m.

Example:

Example LSF01 demonstrates the use of the LIST SUM FORCES command for the simple steel-concrete composite beam shown in Figure 2.3.7.4-1 below. The nodal forces from Joints 41 to 45 at X = 40 ft. from plate elements 29 to 32 and beam member 56 (the more darkly-shaded elements) are used to compute the force summation at the cross section X = 40 ft. The command listing is shown in Figure 2.3.7.4-2. The report from the LIST SUM FORCES command is shown in Figure 2.3.7.4-3.

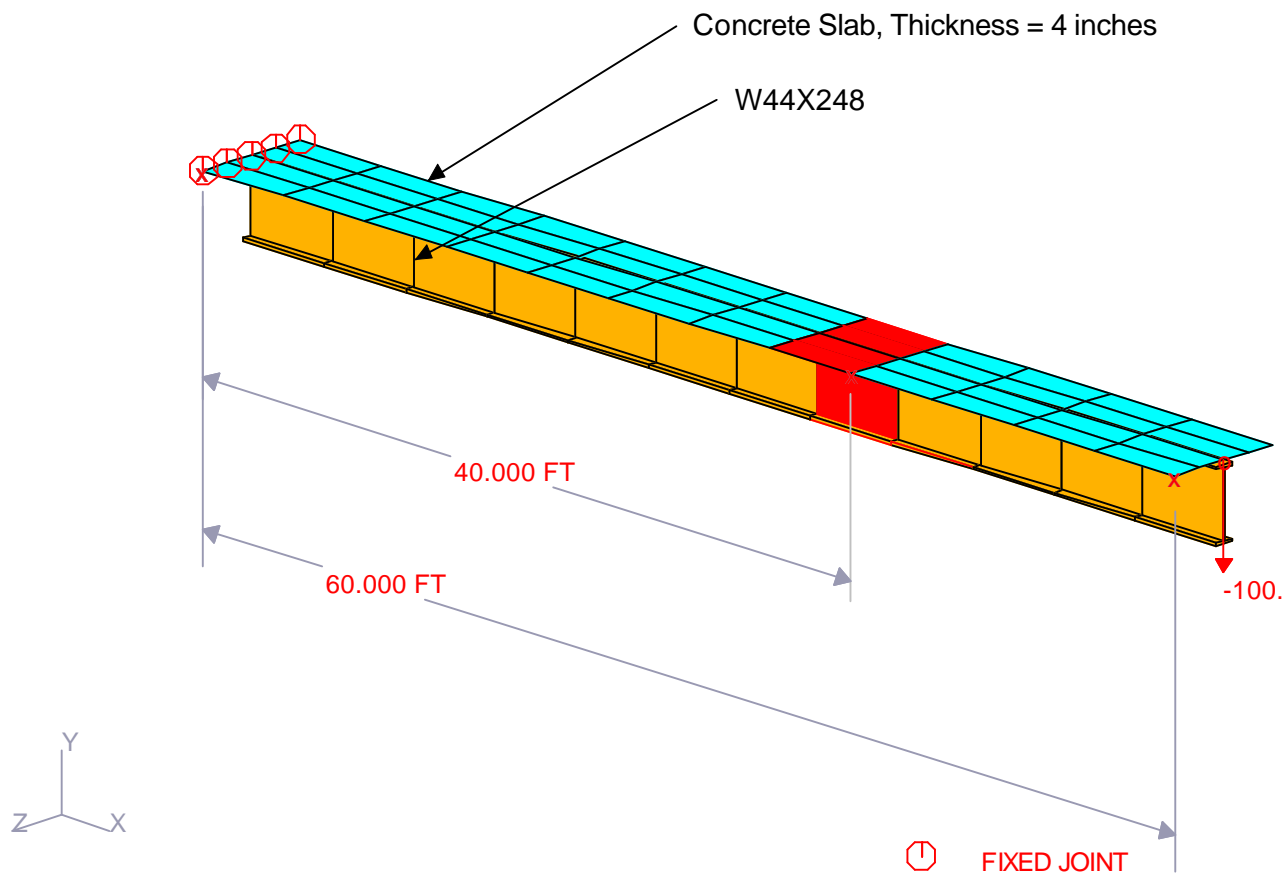


Figure 2.3.7.4-1 Example LFS01, Concrete-Steel Composite Cantilever Beam

```

$$$ **
$$$ ** This job demonstrates the LIST SUM FORCES command for a
$$$ ** steel-concrete composite beam
$$$ **
UNITS  FEET  KIPS  DEG  FAH
$$
$$
GTMenu POINT COORDINATES
  'P1'      0.0000      0.0000     -3.0000
  'P2'     60.0000      0.0000     -3.0000
  'P3'      0.0000      0.0000      3.0000
$$
$$

```

General Prerelease Features

GT STRUDL

GTMenu LINE INCIDENCES

| | | |
|------|------|------|
| 'C1' | 'P2' | 'P1' |
| 'C2' | 'P1' | 'P3' |

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

\$\$

JOINT COORDINATES GLOBAL

| | | | |
|----|---------|--------|---------|
| 1 | 0.0000 | 0.0000 | -3.0000 |
| 2 | 0.0000 | 0.0000 | -1.5000 |
| 3 | 0.0000 | 0.0000 | 0.0000 |
| 4 | 0.0000 | 0.0000 | 1.5000 |
| 5 | 0.0000 | 0.0000 | 3.0000 |
| 6 | 5.0000 | 0.0000 | -3.0000 |
| 7 | 5.0000 | 0.0000 | -1.5000 |
| 8 | 5.0000 | 0.0000 | 0.0000 |
| 9 | 5.0000 | 0.0000 | 1.5000 |
| 10 | 5.0000 | 0.0000 | 3.0000 |
| 11 | 10.0000 | 0.0000 | -3.0000 |
| 12 | 10.0000 | 0.0000 | -1.5000 |
| 13 | 10.0000 | 0.0000 | 0.0000 |
| 14 | 10.0000 | 0.0000 | 1.5000 |
| 15 | 10.0000 | 0.0000 | 3.0000 |
| 16 | 15.0000 | 0.0000 | -3.0000 |
| 17 | 15.0000 | 0.0000 | -1.5000 |
| 18 | 15.0000 | 0.0000 | 0.0000 |
| 19 | 15.0000 | 0.0000 | 1.5000 |
| 20 | 15.0000 | 0.0000 | 3.0000 |
| 21 | 20.0000 | 0.0000 | -3.0000 |
| 22 | 20.0000 | 0.0000 | -1.5000 |
| 23 | 20.0000 | 0.0000 | 0.0000 |
| 24 | 20.0000 | 0.0000 | 1.5000 |
| 25 | 20.0000 | 0.0000 | 3.0000 |
| 26 | 25.0000 | 0.0000 | -3.0000 |
| 27 | 25.0000 | 0.0000 | -1.5000 |
| 28 | 25.0000 | 0.0000 | 0.0000 |
| 29 | 25.0000 | 0.0000 | 1.5000 |
| 30 | 25.0000 | 0.0000 | 3.0000 |
| 31 | 30.0000 | 0.0000 | -3.0000 |
| 32 | 30.0000 | 0.0000 | -1.5000 |
| 33 | 30.0000 | 0.0000 | 0.0000 |
| 34 | 30.0000 | 0.0000 | 1.5000 |
| 35 | 30.0000 | 0.0000 | 3.0000 |
| 36 | 35.0000 | 0.0000 | -3.0000 |
| 37 | 35.0000 | 0.0000 | -1.5000 |
| 38 | 35.0000 | 0.0000 | 0.0000 |
| 39 | 35.0000 | 0.0000 | 1.5000 |
| 40 | 35.0000 | 0.0000 | 3.0000 |
| 41 | 40.0000 | 0.0000 | -3.0000 |
| 42 | 40.0000 | 0.0000 | -1.5000 |
| 43 | 40.0000 | 0.0000 | 0.0000 |
| 44 | 40.0000 | 0.0000 | 1.5000 |
| 45 | 40.0000 | 0.0000 | 3.0000 |
| 46 | 45.0000 | 0.0000 | -3.0000 |
| 47 | 45.0000 | 0.0000 | -1.5000 |
| 48 | 45.0000 | 0.0000 | 0.0000 |
| 49 | 45.0000 | 0.0000 | 1.5000 |
| 50 | 45.0000 | 0.0000 | 3.0000 |
| 51 | 50.0000 | 0.0000 | -3.0000 |

GT STRUDL

The LIST SUM FORCES Command

| | | | |
|----|---------|--------|---------|
| 52 | 50.0000 | 0.0000 | -1.5000 |
| 53 | 50.0000 | 0.0000 | 0.0000 |
| 54 | 50.0000 | 0.0000 | 1.5000 |
| 55 | 50.0000 | 0.0000 | 3.0000 |
| 56 | 55.0000 | 0.0000 | -3.0000 |
| 57 | 55.0000 | 0.0000 | -1.5000 |
| 58 | 55.0000 | 0.0000 | 0.0000 |
| 59 | 55.0000 | 0.0000 | 1.5000 |
| 60 | 55.0000 | 0.0000 | 3.0000 |
| 61 | 60.0000 | 0.0000 | -3.0000 |
| 62 | 60.0000 | 0.0000 | -1.5000 |
| 63 | 60.0000 | 0.0000 | 0.0000 |
| 64 | 60.0000 | 0.0000 | 1.5000 |
| 65 | 60.0000 | 0.0000 | 3.0000 |

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

\$\$

\$\$

TYPE SPACE FRAME

MEMBER INCIDENCES

| | | |
|----|----|----|
| 49 | 3 | 8 |
| 50 | 8 | 13 |
| 51 | 13 | 18 |
| 52 | 18 | 23 |
| 53 | 23 | 28 |
| 54 | 28 | 33 |
| 55 | 33 | 38 |
| 56 | 38 | 43 |
| 57 | 43 | 48 |
| 58 | 48 | 53 |
| 59 | 53 | 58 |
| 60 | 58 | 63 |

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

MEMBER ECCENTRICITIES GLOBAL

49 TO 60 START Y -2.0000 END Y -2.0000

\$\$

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

MEMBER PROPERTIES TABLE 'WBEAM9 ' 'W44x248

49 TO 60

\$\$

STATUS SUPPORT -

| | | | | |
|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 |
|---|---|---|---|---|

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

CONSTANTS

BETA 0.00000 ALL

\$\$

\$\$

```

TYPE PLATE
GENERATE 4 ELEMENTS ID 1 1 FROM 1 1 TO 2 1 TO 7 1 TO 6 1
REPEAT 11 TIMES ID 4 FROM 5 TO 5 TO 5 TO 5
$$
$$
UNITS FEET KIPS DEG FAH
$$
CONSTANTS
      E 5.1839997E+05 ALL
      G 2.0735998E+05 ALL
      POI 1.7000000E-01 ALL
      DEN 1.4999039E-01 ALL
      CTE 5.5000000E-06 ALL
MATERIAL STEEL MEMBERS 49 TO 60
$$
UNITS FEET KIPS DEG FAH
$$
$$
ELEMENT PROPERTIES
      1 TO 48 TYPE 'SBHQ6' THICK 0.3333
$$
UNITS FEET KIPS DEG FAH
$$
LOADING 1
JOINT LOADS
      63 FORCE Y -100.0

STIFFNESS ANALYSIS

UNITS KIPS FEET
LIST FORCES MEMBER 56
LIST ELEMENT FORCES ELEMENTS 29 TO 32

$$* **
$$* ** Resultant sum of forces in the composite beam at
$$* ** X = 40 ft., using the member and finite element nodal
$$* ** forces at joints 41 to 45 of elements 29 to 32 and
$$* ** member 56
$$* **
LIST SUM FORCES TITLE 'Example LSF01' ELEMENTS 29 TO 32 56 -
      JOINTS 41 TO 45
FINISH

```

Figure 2.3.7.4-2 List of Commands for Example LSF01

```

{ 177} > $$* **
{ 178} > $$* ** Resultant sum of forces in the composite beam at
{ 179} > $$* ** X = 40 ft., using the member and finite element nodal
{ 180} > $$* ** forces at joints 41 to 45 of elements 29 to 32 and
{ 181} > $$* ** member 56
{ 182} > $$* **
{ 183} > LIST SUM FORCES TITLE 'LSF01' ELEMENTS 29 TO 32 56 JOINTS 41 TO 45

*****
* RESULTS FROM LATEST ANALYSIS *
*****

ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
      LENGTH      WEIGHT      ANGLE      TEMPERATURE      TIME
      FEET        KIP        DEG        DEGF        SEC

Member and Finite Element Force Resultants
=====

Title: Example LSF01

Sum of Forces and Moments about      40.0000      0.0000      0.0000
Local Axis Orientation Angles:  Z      0.0000  Y      0.0000  X      0.0000

      Loading      FX      FY      FZ      MX      MY      MZ
      -----      -      -      -      -      -      -
1          0.0000     -100.0000      0.0000      0.0000      0.0000     -2000.0001

```

Figure 2.3.7.4-3 Report From the LIST SUM FORCES Command, Example LSF01

This page intentionally left blank.

5.4.9 GTMenu Point and Line Incidences Commands

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

(1) GTMenu POINT COORDINATES

General Form:

GTMenu POINT COORDINATES

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

$$\begin{array}{c} A \\ A \\ A \end{array}$$

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

Elements:

$$\text{coordinate-specs} = [\underline{X}] v_1 [\underline{Y}] v_2 [\underline{Z}] v_3$$

Where,

$$i_1, i_2, \dots, i_n = \text{unsigned integer Point identifiers.}$$

$$'a_1', 'a_2', \dots, 'a_n' = \text{1 to 8 character alphanumeric Point identifiers.}$$

$$v_1, v_2, v_3 = \text{Cartesian Point coordinates (integer or real).}$$

(2) **GTMenu LINE INCIDENTS****General Form:**GTMenu LINE INCIDENTS

$$\begin{array}{c}
 \left\{ \begin{array}{c} i_1 \\ 'a_1' \end{array} \right\} \text{ type}_1 \text{ incidence-specs}_1 \\
 \text{A} \\
 \text{A} \\
 \text{A} \\
 \\
 \left\{ \begin{array}{c} i_n \\ 'a_n' \end{array} \right\} \text{ type}_n \text{ incidence-specs}_n
 \end{array}$$

Elements:

$$\text{type} = \left\{ \begin{array}{l} \rightarrow \text{LINE} \\ \text{POLYNOMINAL (CURVE)} \\ \text{ARC (TEMPLATE)} \\ \text{CENTERED (ARC) PERCENT } v_1 \\ \text{BEZIER (CURVE)} \\ \text{SPLINE (CURVE) (ORDER } k_2) \end{array} \right\}$$

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

Where,

$$i_1, i_2, \dots, i_n = \text{unsigned integer Line/Curve identifiers.}$$

'a₁', 'a₂', ..., 'a_n' = 1 to 8 character alphanumeric Line/Curve identifiers.

i₁, i₂, ..., i_p = unsigned integer Point identifiers used.

'a₁', 'a₂', ..., 'a_p' = 1 to 8 character alphanumeric Point identifiers.

v₁ = positive number (integer or real).

k₂ = 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.

5.3.7 Nonlinear Dynamic Analysis

Sections 2.5.6.1 and 2.5.6.2 below describe the commands used to execute a nonlinear dynamic analysis. Section 2.5.6.1 describes extensions to the optional DYNAMIC PARAMETERS command (Section 2.4.5.3) which are used to set nonlinear dynamic analysis control parameters. Section 2.5.6.2 describes the DYNAMIC ANALYSIS NONLINEAR command.

Section 2.5.6.3 provides an example problem illustrating a complete nonlinear dynamic analysis execution, including the use of the DYNAMIC PARAMETERS extensions and the DYNAMIC ANALYSIS NONLINEAR command. The sections are numbered as they will appear when they are added to Volume 3 of the GTSTRUDL User Reference Manual.

2.5.6.1 Extensions to the DYNAMIC PARAMETERS Command

General form:

DYNAMIC PARAMETERS

BLOCK (SIZE) i_{BS}

UPDATE (STIFFNESS EVERY) i_U (TIME STEPS)

MAXIMUM (NUMBER OF EQUILIBRIUM CYCLES) i_{MAX}

CONVERGENCE (TOLERANCE) ENERGY v_{TOL}

INITIAL (STRESS LOAD) $\left\{ \begin{array}{l} \rightarrow \text{OFF} \\ i_{ISL} \\ 'a_{ISL}' \end{array} \right\}$

RESULTS (FILE NAME) 'fn'

END (OF DYNAMIC PARAMETERS)

Elements:

- i_{BS} = size of equation blocks in double words if the out-of-core equation solver is to be used,
- i_U = time step interval at which complete updating of the effective global tangent stiffness matrix is to occur,
- i_{MAX} = maximum number of equilibrium correction iterations permitted within each time increment, (default $i_{MAX} = 15$),

- v_{TOL} = a decimal number much less than 1.0, representing the incremental energy convergence tolerance, (default $v_{TOL} = .001$),
- i_{ISL}/a_{ISL} = integer or alphanumeric id of the initial stress loading,
- 'fn' = alphanumeric string to be used in the construction of the dynamic analysis results file names. The length of the alphanumeric string is limited to 24 characters.

Explanation:

The DYNAMIC PARAMETERS command includes the following options which are used to control a nonlinear dynamic analysis:

BLOCK SIZE i_{BS}

By default, GTSTRUDL attempts to perform the nonlinear dynamic analysis solution entirely within the virtual memory (RAM plus page file) of the computer. If this is not possible or practical, the BLOCK SIZE parameter may be specified, indicating that an out-of-core solution procedure is to be used. The out-of-core solution process divides the equations of motion into blocks of maximum double word size i_{BS} , which are stored in scratch files. The blocks are transferred back and forth between the scratch files and virtual memory during the solution process, and the scratch files are deleted from the system when the solution is complete. The block transfer process results in longer solution times, and while the blocked solution process requires less virtual memory, sufficient hard disk space is needed for the storage of the blocks. If a value of 0 is given for i_{BS} , the solution procedure is reset to the in-core method.

UPDATE STIFFNESS EVERY i_U TIME STEPS

The nonlinear dynamic analysis is a modified Newton-Raphson process whereby the an effective dynamic stiffness matrix is updated, i.e. re-computed, only at the beginning of a time step and only at the time step interval specified by i_U . If this option is not specified or a value of 0 is specified for i_U , the effective dynamic stiffness matrix is computed at the beginning of the first time step and remains constant thereafter.

MAXIMUM NUMBER OF EQUILIBRIUM CYCLES i_{MAX}

Equilibrium correction cycles are executed for every time step. This option is used to specify the maximum permissible number of these cycles in any given time step. If a value for i_{MAX} is not specified, 15 is assumed. If the maximum number of cycles is reached and convergence has not been achieved, then the entire analysis is terminated.

CONVERGENCE TOLERANCE ENERGY v_{TOL}

The CONVERGENCE TOLERANCE ENERGY option specifies the value for the convergence tolerance used in the incremental energy convergence check. If not specified, a convergence tolerance value of 0.001 is assumed.

INITIAL STRESS LOAD OFF $i_{ISL}/'a_{ISL}'$

This option identifies a loading condition, $i_{ISL}/'a_{ISL}'$, from which an initial displacement and stress state is established for the subsequent nonlinear dynamic analysis. The analysis starts with the structure in this state, including the applied loading that produced it. The initial stress load induces no acceleration at the start of the analysis, and is carried throughout the entire analysis.

The initial stress load is typically a self-weight load applied to the structure, for which a static stiffness or nonlinear analysis has been previously executed. The initial stress load also may be the prestress loading from the previous prestress analysis of a cable structure (see Section 2.6.3, Volume 3, GTSTRUDL User Reference Manual).

The INITIAL STRESS LOAD OFF command is used to switch off the inclusion of an initial stress load for a nonlinear dynamic analysis in the case where an initial stress load was specified for a previous nonlinear dynamic analysis.

RESULTS FILE NAME 'fn'

Nonlinear dynamic analysis uses the external file solver procedure described in Section 2.4.5.3 (Section 5.10 of this Release Guide), whereby the results of the dynamic analysis – displacements, velocities, accelerations, member forces, etc. – are permanently stored at the end of each time step into external save files rather than in virtual memory. The RESULTS FILE NAME option specifies an alphanumeric string to be used as a file name prefix in the creation of the results file names. If this option is not given, then the file name prefix string 'fn' is taken as the problem id given in the STRUDL command (Section 2.1.2.3, Volume 1, GTSTRUDL User Reference Manual)

or the CHANGE ID command (Section 2.1.2.5, Volume 1, GTSTRUDL User Reference Manual). If a problem id is not specified in either of these two commands, then 'fn' is taken as 'DyJob'.

It is recommended that the RESULTS FILE NAME command be given only once in a given job, regardless of the number of nonlinear dynamic analyses performed for different transient loading conditions. Giving a new RESULTS FILE NAME command prior to each nonlinear dynamic analysis for a new transient loading is permitted; however, doing so will make it cumbersome later to access the results from the different loading conditions. Prior to accessing the results from a particular transient loading condition, it will be necessary to re-issue the RESULTS FILE NAME command with the value of 'fn' initially used to create the results for that loading condition.

Modifications:

The DYNAMIC PARAMETERS command functions identically in ADDITIONS, CHANGES, and DELETIONS modes. To change the value of a dynamic parameter, simply re-specify the new value, regardless of input mode.

2.5.6.2 DYNAMIC ANALYSIS NONLINEAR Command

General form:

$$\text{DYNAMIC ANALYSIS NONLINEAR} \quad \left\{ \begin{array}{l} \text{BETA} \quad v_b \\ \text{NJP} \quad i_{\text{NJP}} \end{array} \right\}^*$$

Elements:

- v_b = decimal value of β in the Newmark- β formulation of direct integration. The default value is 0.25, corresponding to the constant average acceleration method.
- i_{NJP} = integer number of joints per partition.

Explanation:

The DYNAMIC ANALYSIS NONLINEAR command is used to execute a nonlinear dynamic analysis, using the Newmark- β formulation as described in Section 2.4.2.4.2 of Volume 3, GTSTRUDL User Reference Manual.

The BETA option is used to change the value of β in the Newmark- β formulation from the default value of 0.25, which corresponds to the unconditionally stable constant average acceleration method.

The NJP option is used to specify the number of joints per partition, which has only a minimal effect on the efficiency of the nonlinear dynamic analysis procedure. The default NJP value is 36/JF, where JF = the number of degrees-of-freedom associated with the type of member or element specified in the last TYPE command (Section 2.1.5.2.2, Volume 1, GTSTRUDL User Reference Manual, Section 2.3.4, Volume 3, GTSTRUDL User Reference Manual).

Nonlinear dynamic analysis supports all available TRANSIENT LOADING options.

The nonlinear dynamic analysis calculates and stores nodal displacements, velocities, accelerations, and element nodal forces for nonlinear geometric frame and truss members, tension/compression-only members, friction damper members, nonlinear spring elements, NLS4PH elements, and IPCABLE elements. Member end forces and finite element nodal forces, and finite element stresses and strains may be computed and stored for linear members and finite elements by using the COMPUTE TRANSIENT command described in Section 2.4.5.7, Volume 3, GTSTRUDL User Reference Manual.

All currently available dynamic analysis results processing functions support the results from nonlinear dynamic analysis.

Nonlinear dynamic analysis does not presently support the following capabilities:

1. Superelements
2. Joint constraints, i.e. joint ties and rigid bodies (Section 2.6.5, Volume 3, GTSTRUDL Reference Manual).
3. Dynamic degrees of freedom condensation using the DYNAMIC DEGREES OF FREEDOM command (Section 2.4.5.1, Volume 3, GTSTRUDL User Reference Manual).

If superelements are detected, the following error message is printed and the analysis is terminated:

```
**** ERROR_STDNL1 --      Superelement(s) detected.  
                        Analysis terminated and SCAN mode  
                        entered.
```

If joint constraints are detected, the following error message is printed and the analysis is terminated:

```
**** ERROR_STDNL1 --      Joint constraints specified but not  
                        presently supported by nonlinear  
                        dynamic analysis.  
                        Analysis terminated and SCAN mode  
                        entered.
```

If dynamic condensation of dynamic degrees of freedom is detected, the following error message is printed and the analysis is terminated.

```
**** ERROR_STDNL1 --      Dynamic DOF condensation requested but
                           not presently supported by nonlinear
                           dynamic analysis.
                           Analysis terminated and SCAN mode
                           entered.
```

2.5.6.3 Nonlinear Dynamic Analysis Example

Figures 2.5.6.3-1 and 2.5.6.3-2 show sketches of a simple two-story space frame structure which is the subject structure of the nonlinear dynamic analysis example problem command listing given in Figure 2.5.6.3-3. Figure 2.5.6.3-1 shows the structure with joints labeled and Figure 2.5.6.3-2 shows the structure with beam and column members and floor finite elements labeled. Figure 2.5.6.3-3 contains the command input file for this example, including comments which describe the use of the nonlinear dynamic analysis commands described in the previous sections.

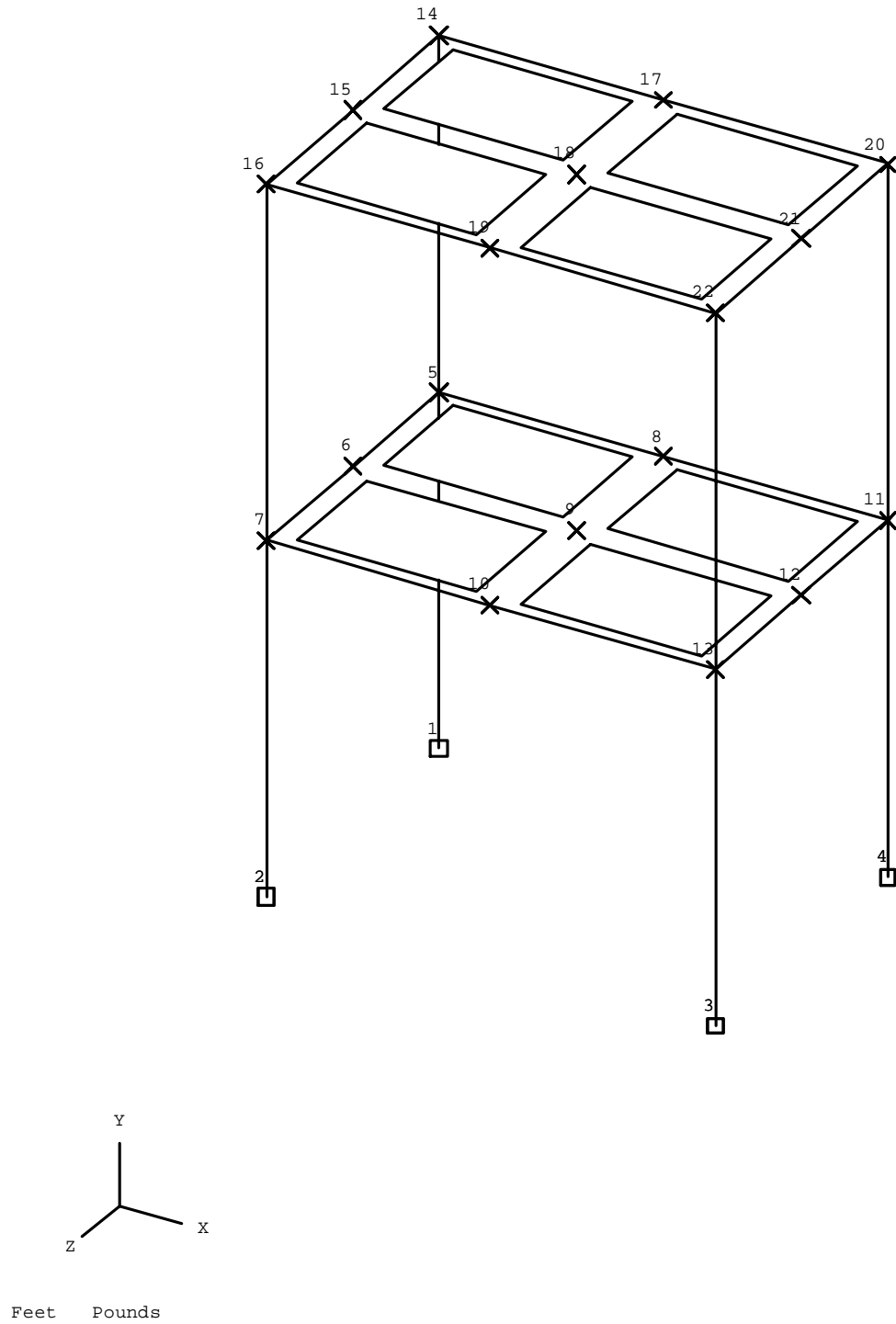


Figure 2.5.6.3-1 Nonlinear Dynamic Analysis Example Structure – Joints Labeled

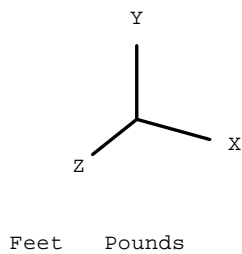
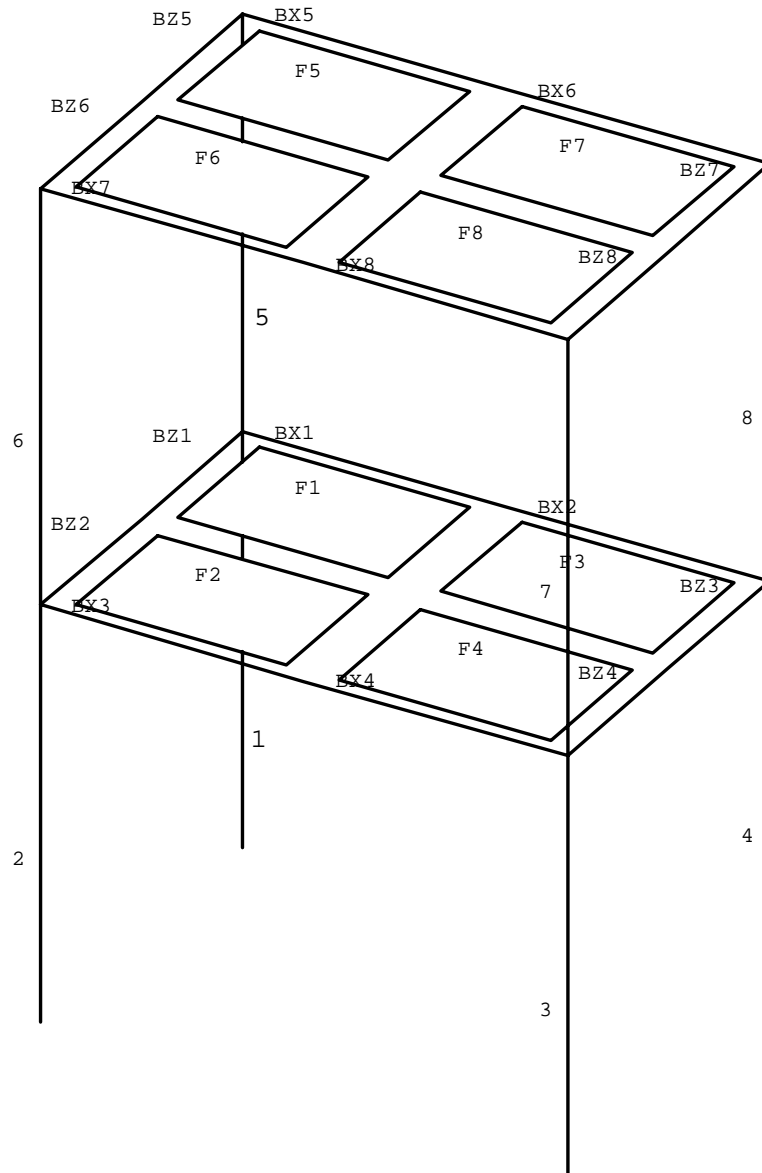


Figure 2.5.6.3-2

Nonlinear Dynamic Analysis Example Structure – Members
and Finite Elements Labeled

```

STRUDL 'NlDyEx' 'Nonlinear Dynamic Analysis Example Problem'
$* **
$*****
$*****
$* **
$
$
$ Geometry
$
UNITS FEET
JOINT COORD
  1      0.0      0.0      0.0 S
  2      0.0      0.0     10.0 S
  3     15.0      0.0     10.0 S
  4     15.0      0.0      0.0 S

GENERATE 3 JOINTS ID 5 1 X 0.0 Y 12.0 Z 0.0 5.0
REPEAT 2 TIMES ID 3 X 7.5
REPEAT 1 TIME ID 9 Y 12.0

TYPE SPACE FRAME
MEMBER INCIDENCES
1 1 5
2 2 7
3 3 13
4 4 11
GEN 4 MEMBERS ID 5 1 FROM LIST 5 7 13 11 TO LIST 14 16 22 20
GEN 2 MEMBERS ID 'BX1' 1 FROM 5 3 TO 8 3
REPEAT 1 TIME ID 2 FROM 2 TO 2
REPEAT 1 TIME ID 4 FROM 9 TO 9
GEN 2 MEMBERS ID 'BZ1' 1 FROM 5 1 TO 6 1
REPEAT 1 TIME ID 2 FROM 6 TO 6
REPEAT 1 TIME ID 4 FROM 9 TO 9
TYPE PLATE
GEN 2 ELEMENTS ID 'F1' 1 FROM 5 1 TO 6 1 TO 9 1 TO 8 1
REPEAT 1 TIME ID 2 FROM 3 TO 3 TO 3 TO 3
REPEAT 1 TIME ID 4 FROM 9 TO 9 TO 9 TO 9

MATERIAL CONC
UNITS INCHES
MEMBER DIMENSIONS
  1 TO 8 RECT B 8.0 H 12.0
  'BX1' TO 'BX8' 'BZ1' TO 'BZ8' RECT B 8.0 H 18.0
ELEMENT PROPERTIES
  'F1' TO 'F8' TYPE 'SBHQ6' THICK 4.0

$
$ Define self weight and dead load 1. This load is used as
$ the initial stress loading for the nonlinear dynamic analysis.
$
UNITS FEET LBS
DEAD LOADS 1 DIR -Y MEMBERS 1 TO 8 -
  'BX1' TO 'BX8' 'BZ1' TO 'BZ8'
MEMBER LOADS

```

```
'BX1' 'BX2' FORCE Y GLOB UNI FR W -500.0 LA 0.0 LB 1.0
ELEMENT LOADS
  'F1' TO 'F8' BODY FORCES GLOBAL BY -150.0

$
$ Define nonlinear geometric behavior for column
$ members 1 to 8. All other members and finite
$ elements remain linear.
$
NONLINEAR EFFECTS
  GEOMETRY MEMBERS 1 TO 8

$
$ Execute a nonlinear analysis for loading 1 to
$ establish the initial stress conditions.
$
MAXIMUM NUMBER OF CYCLES 10
CONVERGENCE TOLERANCE DISPL 0.001
LOAD LIST 1
NONLINEAR ANALYSIS

UNITS INCHES CYCLES SECS
LIST DISPLACEMENTS
LIST FORCES
LIST SUM REACTIONS

$
$ Add dynamic analysis data including inertia and
$ dynamic loading data. This data is specified
$ in the same manner as for linear dynamic analysis.
$
UNITS LBS
INERTIA OF JOINTS LUMPED
INERTIA OF JOINTS WEIGHT
  3 4 5 6 TRANSLATION ALL 10000.0

TRANSIENT LOAD 'EQ-X'
SUPPORT ACCELERATION
  TRANSL X FILE 'ELCENTRO'
  INTEGRATE FROM 0.0 TO 10.0 AT 0.01
END TRANS LOAD

TRANSIENT LOAD 'EQ-Z'
SUPPORT ACCELERATION
  TRANSL Z FILE 'ELCENTRO' FACTOR 0.5
  INTEGRATE FROM 0.0 TO 10.0 AT 0.01
END TRANS LOAD

$
$ This block of DYNAMIC PARAMETERS commands contains the most
$ commonly used control data specifications for nonlinear
$ dynamic analysis including the MAXIMUM, CONVERGENCE TOLERANCE,
$ INITIAL STRESS, UPDATE, and RESULTS FILE commands.
$
```



```
DYNAMIC PARAMETERS
  MAXIMUM NUMBER OF EQUILIBRIUM CYCLES 30
  CONVERGENCE TOLERANCE 0.0001
  INITIAL STRESS LOAD 1
  UPDATE STIFFNESS EVERY 3 TIME STEPS
  PRINT MAX
  RESULTS FILE NAME 'N1DyEX'
END DYNAMIC PARAMS

$
$ Execute the nonlinear dynamic analysis
$
DYNAMIC ANALYSIS NONLINEAR

$
$ Perform normal backsubstitution and results processing
$ operations
$
COMPUTE TRANSIENT FORCES
COMPUTE TRANSIENT STRESSES
COMPUTE TRANSIENT REACTIONS LOADS

LIST TRANSIENT FORCES TIMES FROM 1 TO 10 MEMBER 1 2 'BX1' 'BZ1'
LIST TRANSIENT MAX FORCES MEMBERS 1 2 'BX1' 'BZ1'
LIST TRANSIENT MAX DISPL JOINT 5

CREATE PSEUDO STATIC LOAD 3 FROM TIME 2 OF LOAD 'EQ-X'
CREATE PSEUDO STATIC LOAD 4 FROM MAX OF LOAD 'EQ-X'
CREATE PSEUDO STATIC LOAD 5 FROM TIME 2 OF LOAD 'EQ-Z'
CREATE PSEUDO STATIC LOAD 6 FROM MAX OF LOAD 'EQ-Z'
CREATE PSEUDO STATIC LOAD 7 FROM TIME 2 OF LOADS 'EQ-X' 'EQ-Z'
CREATE PSEUDO STATIC LOAD 8 FROM MAX OF LOADS 'EQ-X' 'EQ-Z'

LOAD LIST 1 3 TO 8
UNITS INCHES KIPS
OUTPUT BY MEMBER
LIST DISPL JOINT 5
LIST FORCES MEMBERS 'BX1' 'BZ1'

SECTION FR NS 3 0.0 0.5 1.0
LIST SECTION FORCES MEMBERS 'BX1' 'BZ1'

FIN
```

Figure 2.5.6.3 Command Input File for Nonlinear Dynamic Analysis Example

This page intentionally left blank.

5.3.8 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

The Sections shown below are numbered as they will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.4.3.7 The Viscous Damper Element for Linear and Nonlinear Dynamic Analysis

This section describes the commands that are used to incorporate the viscous damper element (dash pot) into a structural model that is used for linear and nonlinear dynamic analysis by the direct integration procedure. The commands that are used for this purpose include:

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

2.4.3.7.1 The DAMPER ELEMENT DATA Command

Tabular form:

DAMPER ELEMENT (DATA)

$$\begin{array}{l} \left\{ \begin{array}{c} i_D \\ 'a_D' \end{array} \right\} \underline{\text{INCIDENCES}} \left\{ \begin{array}{c} i_S \\ 'a_S' \end{array} \right\} \left(\left\{ \begin{array}{c} i_E \\ 'a_E' \end{array} \right\} \right) \left\{ \begin{array}{c} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{array} \right\} - \\ \quad [\underline{\text{CTX}}] \ v_{\text{CTX}} \quad [\underline{\text{CTY}}] \ v_{\text{CTY}} \quad [\underline{\text{CTZ}}] \ v_{\text{CTZ}} \quad [\underline{\text{CRX}}] \ v_{\text{CRX}} \quad [\underline{\text{CRY}}] \ v_{\text{CRY}} \quad [\underline{\text{CRZ}}] \ v_{\text{CRZ}} \\ \bullet \\ \bullet \\ \bullet \\ \left\{ \begin{array}{c} i_D \\ 'a_D' \end{array} \right\} \underline{\text{INCIDENCES}} \left\{ \begin{array}{c} i_S \\ 'a_S' \end{array} \right\} \left(\left\{ \begin{array}{c} i_E \\ 'a_E' \end{array} \right\} \right) \left\{ \begin{array}{c} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{array} \right\} - \\ \quad [\underline{\text{CTX}}] \ v_{\text{CTX}} \quad [\underline{\text{CTY}}] \ v_{\text{CTY}} \quad [\underline{\text{CTZ}}] \ v_{\text{CTZ}} \quad [\underline{\text{CRX}}] \ v_{\text{CRX}} \quad [\underline{\text{CRY}}] \ v_{\text{CRY}} \quad [\underline{\text{CRZ}}] \ v_{\text{CRZ}} \end{array}$$

END (OF DAMPER ELEMENT DATA)

Elements:

- i_D/a_D = integer or alphanumeric name of the new damper element. The name must be unique among all previously defined damper elements and is restricted to no more than eight digits or alphanumeric characters.
- i_S/a_S = integer or alphanumeric name of a previously defined joint to be the starting incident joint of the new damper element.
- i_E/a_E = optional integer or alphanumeric name of the previously defined joint to be the ending incident joint of the new damper element. The starting joint and ending joint names must be different.
- V_{CTX} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL X direction. Active force, length, and time units apply [force/(length/time)].
- V_{CTY} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL Y direction. Active force, length, and time units apply [force/(length/time)].
- V_{CTZ} = decimal value for the damper force coefficient corresponding to translation velocity in the LOCAL or GLOBAL Z direction. Active force, length, and time units apply [force/(length/time)].
- V_{CRX} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].
- V_{CRY} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].
- V_{CRZ} = decimal value for the damper moment coefficient corresponding to angular velocity about the LOCAL or GLOBAL X axis. Active force, length, angle, and time units apply [force-length/(angle/time)].

Explanation:

The DAMPER ELEMENT DATA command is used to create new viscous damper elements and define their joint connectivity and damping force and moment properties. The viscous damper element data are entered by giving the DAMPER ELEMENT DATA command header first, followed by one or more tabular element data entry lines of the form:

$$\left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\} \underline{\text{INCIDENCES}} \left\{ \begin{matrix} i_s \\ 'a_s' \end{matrix} \right\} \left(\left\{ \begin{matrix} i_E \\ 'a_E' \end{matrix} \right\} \right) \left\{ \begin{matrix} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{matrix} \right\} -$$

$$[\underline{\text{CTX}}] v_{\text{CTX}} [\underline{\text{CTY}}] v_{\text{CTY}} [\underline{\text{CTZ}}] v_{\text{CTZ}} [\underline{\text{CRX}}] v_{\text{CRX}} [\underline{\text{CRY}}] v_{\text{CRY}} [\underline{\text{CRZ}}] v_{\text{CRZ}}$$

for each new damper element. This data entry line consists of the element name, the element incidences, the element orientation, and the element viscous damping coefficients, which are described in greater detail as follows:

Element name $\left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\}$

Each new damper element must be given an integer or alphanumeric name that is unique among all other existing damper element names. The name may not exceed eight digits or alphabetic characters. The name may be a duplicate of a previously defined member or finite element name.

$\underline{\text{INCIDENCES}} \left\{ \begin{matrix} i_s \\ 'a_s' \end{matrix} \right\}$

The damper element connectivity is defined by one or two incident joints. The first incident joint, $i_s/'a_s'$, defines the start of the element. The second incident joint, $i_E/'a_E'$, is optional and defines the end of the element. If only one joint is given, the second joint is taken as a totally fixed support joint; it is fictitious and invisible. The specified joints must have been previously defined and if two are specified, they must be different. However, they may be coincident. The only restriction on the selection of incident joints is that they may not be slave joints.

$$\left\{ \begin{array}{l} \rightarrow \underline{\text{GLOBAL}} \\ \underline{\text{LOCAL}} \end{array} \right\}$$

The GLOBAL and LOCAL options are used to specify the coordinate reference frame for the damper element. The GLOBAL option, which is the default, means that the element is a global element and that the six element damping degrees-of-freedom are defined with respect to the global coordinate system. The LOCAL option means that the element damping degrees-of-freedom are defined with respect to the element local coordinate system, which is identical to the local joint-to-joint coordinate system for frame members. The only difference between the frame member and damper element local coordinate systems is that the damper element does not support the Beta angle. If the LOCAL option is specified, but the joint-to-joint length of the element is equal to 0 ($\leq 10^{-5}$ inches), then GLOBAL is assumed. In addition, GLOBAL is automatically assumed for any damper element for which only one incident joint is specified.

$$[\underline{\text{CTX}}] v_{\text{CTX}} [\underline{\text{CTY}}] v_{\text{CTY}} [\underline{\text{CTZ}}] v_{\text{CTZ}} [\underline{\text{CRX}}] v_{\text{CRX}} [\underline{\text{CRY}}] v_{\text{CRY}} [\underline{\text{CRZ}}] v_{\text{CRZ}}$$

These decimal data values represent the damping coefficient values on the diagonal of the uncoupled element damping matrix, which has the following form:

$$\begin{bmatrix} \text{CTX} & 0 & 0 & 0 & 0 & 0 \\ & \text{CTY} & 0 & 0 & 0 & 0 \\ & & \text{CTZ} & 0 & 0 & 0 \\ & (\text{sym}) & & \text{CRX} & 0 & 0 \\ & & & & \text{CRY} & 0 \\ & & & & & \text{CRZ} \end{bmatrix}$$

These values refer to the element damping translational and rotational degrees-of-freedom with respect to the specified coordinate system, GLOBAL, the default, or LOCAL. Only non-zero values need be specified.

Command processing is completed when the END option is given.

The damping properties from the viscous damper elements are assembled into the total global system damping matrix of the equations of motion that are solved using the direct integration methods executed by the DYNAMIC ANALYSIS PHYSICAL and DYNAMIC ANALYSIS NONLINEAR commands. The viscous damper element data are used only by the execution of these two commands

Modifications:

The DAMPER ELEMENT DATA command operates only in the ADDITIONS mode. If the command is given when the active input mode is CHANGES or DELETIONS, then the command execution is terminated and the command data are ignored. If it is necessary to change the data for an existing damper element, then use the DELETE DAMPER ELEMENT command described in Section 2.4.3.7.3 to delete the damper element to be changed, followed by the re-specification of the new data in the DAMPER ELEMENT DATA command. All of these steps are performed in ADDITIONS mode.

Example:

The following example illustrates the creation of two damper elements DAMP1 and DAMP2. DAMP1 spans from joint 2 to joint 10 and has one damping coefficient equal to 10^7 kips/(inches/second) corresponding to translation in the local y direction of the element. DAMP2 spans from joint 1 to joint 2 and has global damping factors $CTX = 100$ kips/(inches/second) and $CRZ = 1000$ kip-inches/(radians/second). The damping coefficients for element DAMP2 are referenced with respect to the global coordinate system because the GLOBAL/LOCAL option was not given. The execution of this example depends on DAMP1 and DAMP2 not having been previously defined and joints 1, 2, and 10 being valid joints.

```

UNITS KIPS INCHES RADIANS
DAMPING ELEMENT DATA
  'DAMP1' INC 2 10 LOCAL CTY 1.E7
  'DAMP2' INC 1 2 CTX 100.0 CRZ 1000.0
END

```

Errors:

1. When two or more damper elements are defined with the same name, the following warning message is printed. Command processing is terminated for the offending element and continues for subsequent elements.

```

{ 10} > DAMPING ELEMENT DATA
{ 11} >  'DAMP1' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{ 12} >  'DAMP1' INC 2 4 GLOBAL CTY 1.E7

**** WARNING_STDELD -- Damper element DAMP1      previously defined.  Command ignored.

{ 13} >  'DAMP3' INC 3 3 GLOBAL CTY 1.E7
{ 14} > END

```

Element DAMP1 is successfully created by the first tabular command entry. The warning message for DAMP1 is printed for the second tabular entry for DAMP1. Command processing continues with the tabular entry for DAMP3.

2. The following warning message is printed if one or both of the specified element incidence joints are not defined. Command processing continues with the tabular entry for the next element.

```
{ 10} > DAMPING ELEMENT DATA
{ 11} > 'DAMP1' INC 2 10 LOCAL CTY 1.E7

**** WARNING_STDELD -- Damper element incidence joint not defined.
                        Command ignored.

{ 12} > 'DAMP2' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{ 13} > END
```

The warning message indicates that one or both of the specified element incidences for element DAMP1 are not defined.

3. The following warning message is printed when the starting and ending element incidence joints are the same. Command processing continues with the tabular entry for the next element.

```
{ 10} > DAMPING ELEMENT DATA
{ 12} > 'DAMP1' INC 1 2 LOCAL CTX 100.0 CRZ 1000.0
{ 13} > 'DAMP2' INC 2 4 GLOBAL CTY 1.E7
{ 14} > 'DAMP3' INC 3 3 GLOBAL CTY 1.E7

**** WARNING_STDELD -- Damper element starting and ending incident joints are the
                        same. Command ignored.

{ 15} > 'DAMP4' INC 4 5 CTY 1.E7
{ 16} > END
```


2.4.3.7.2 The PRINT DAMPER ELEMENT DATA Command

General form:

PRINT DAMPER (ELEMENT DATA)

Explanation:

The PRINT DAMPER ELEMENT DATA is used to print a table of the damper element data for all existing damper elements. The following is an example of the printed output from this command:

Example:

The following example illustrates the format for the output from the PRINT DAMPER ELEMENT command.

```
{ 17} > PRINT DAMPING ELEMENT DATA
*****
* RESULTS FROM LATEST ANALYSIS *
*****

ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
      LENGTH      WEIGHT      ANGLE      TEMPERATURE      TIME
      FEET        LB         RAD        DEGF          SEC

Damping Element Data
=====
Element      Start Jnt  End Jnt  CTX      CTY      CTZ      CRX      CRY      CRZ
-----
DAMP1  LOC   1         2      100.0    0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  1000.
DAMP2  GLO   2         4    0.0000E+00  0.1000E+08  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
```

Errors:

The following warning message is printed when no damper element data exists.

```
{ 9} > PRINT DAMPING ELEMENT DATA
*****
* RESULTS FROM LATEST ANALYSIS *
*****
```

```

ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
      LENGTH      WEIGHT      ANGLE      TEMPERATURE      TIME
      FEET        LB         RAD         DEGF          SEC

Damping Element Data
=====

Element      Start Jnt   End Jnt      CTX      CTY      ...      CRZ
-----
**** INFO_STPDED -- Damper element data have not been defined.

```

2.4.3.7.3 The DELETE DAMPER ELEMENT DATA Command

General form:

$$\underline{\text{DELETE}} \underline{\text{DAMPER}} (\underline{\text{ELEMENT}} \underline{\text{DATA}}) \left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\} \dots \left\{ \begin{matrix} i_D \\ 'a_D' \end{matrix} \right\}$$

Elements:

$i_D / 'a_D'$ = integer or alphanumeric name of damper element to be deleted. The name is limited to no more than eight digits or characters.

Explanation:

This command is used to delete previously defined damper elements. The names of the elements to be deleted are given in the list of individually named damper elements. No other list construct, such as "1 TO 10" is permitted. Specified damper elements that are not defined are ignored.

5.4 General Prerelease Features

5.4.1 Align Command

The ALIGN command will align joints in a line of members in a column to ensure that the column is parallel to the global Y axis. The section below is numbered as it will appear when added to Volume 1 of the GTSTRUDL User Reference Manual.

2.1.12.16 The ALIGN Command

This command will move joints at the "top" (top if the Y axis is vertical) of members parallel to the global Y axis, so that the "top" joint has the same X and Z coordinates as the "bottom" joint. This is to ensure that all these members follow the 'Special Case' for local reference frame orientation. See Section 1.10.4 in Volume 1 of the GTSTRUDL User Reference Manual or Section 8.5, "The BETA Angle", in the GTSTRUDL Analysis Users Guide. This command is useful for situations where coordinates are calculated to a precision that does not ensure alignment with the Y axis, such as coordinates created by an outside program, or use of the GENERATE or OBJECT COPY commands with many copies or large coordinates. Since the check for being parallel is very exacting in GTSTRUDL (within 0.01%), it is possible for a column line to have some columns that are 'Special Case' members and others that are not, resulting in inconsistent default orientations.

Notes:

This "out of alignment" problem can also occur for beams parallel to the Y global axis in "Z up" structures.

MEMBER ECCENTRICITIES are not considered during the alignment check because they are not used to determine the default member orientation.

Syntax:

ALIGN (MEMBERS list) (REPORT (ONLY)) (ALONG Y AXIS) -
(TOLERANCE v₁ (RATIO)) (ITEMIZE (WITH DETAIL))

Where,

list is an optional list of members, as defined by Section 2.1.2.2 of this volume.

v₁ is the specified TOLERANCE used to define "parallel to global Y"

Explanation:

MEMBERS If no list is given, all members will be checked.

REPORT Don't change joint coordinates, only report detected possible alignment problems. Since the adjustment procedure changes joints from the "bottom" of the structure (starting at the most negative Y location) and works toward the "top", 'REPORT ONLY' alignment checks may be slightly different because "lower" joints will not have been changed.

TOLERANCE

Members which have an X or Z projection equal to or less than this value will be considered parallel to Y and will be checked for alignment. The default value is 0.1 inch (2.5 mm).

RATIO This option specifies that the tolerance specified be used as a ratio: (X or Z projection)/(member length). This option allows you to have more control in special situations. To avoid possible problems the TOLERANCE value is not permitted to exceed 0.05.

ITEMIZE This option will generate a message for each joint that is changed.

DETAIL This option will also print the old coordinates and the new coordinates of each changed joint.

The **ALIGN MEMBERS** command will change joint coordinates to keep members parallel to the global Y axis, unless the **REPORT** option is specified. A list of all members that meet the criteria for being considered "parallel to global Y" is built, respecting the default or user-specified values for **TOLERANCE** and **RATIO**. Members are then 'aligned' (joint coordinates changed) from the "bottom", or most negative Y coordinate, to the "top".

Examples:

```
ALIGN REPORT ONLY ITEMIZE
```

```
****INFO_STALGN - The following joints need to be adjusted to align members  
                  in the global Y direction:
```

```
Joint 5          needs to be aligned with joint 1  
Joint 9          needs to be aligned with joint 5
```

ALIGN ITEMIZE WITH DETAIL

```
****INFO_STALGN - The following joints were adjusted to align members
                   in the global Y direction:
      Joint 5      was aligned with joint 1
        Old (X,Z): 0.10000E+00 0.10000E+00 New (X,Z): 0.00000E+00 0.00000E+00
      Joint 9      was aligned with joint 5
        Old (X,Z): 0.20000E+00 0.20000E+00 New (X,Z): 0.00000E+00 0.00000E+00
      Joint 13     was aligned with joint 9
        Old (X,Z): -0.30000E+00 -0.30000E+00 New (X,Z): 0.00000E+00 0.00000E+00
      Joint 16     was aligned with joint 12
        Old (X,Z): 0.45100E+03 0.10000E+01 New (X,Z): 0.45000E+03 0.00000E+00

****INFO_STALGN - A total of      4 joints were adjusted.
```

ALIGN ITEMIZE

```
****INFO_STALGN - The following joints were adjusted to align members
                   in the global Y direction:
      Joint 5      was aligned with joint 1
      Joint 9      was aligned with joint 5
      Joint 13     was aligned with joint 9
      Joint 16     was aligned with joint 12

****INFO_STALGN - A total of      4 joints were adjusted.
```

ALIGN

```
****INFO_STALGN - A total of      4 joints were adjusted.
```

If no members were determined to be misaligned:

ALIGN

```
****INFO_STALGN - No joints required adjustment.
```

This page intentionally left blank.

5.4.2 ROTATE LOAD Command

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

2.1.11.4.6 The ROTATE LOAD Command

General form:

$$\text{ROTATE } \text{LOADING} \left\{ \begin{array}{c} i_R \\ 'a_R' \end{array} \right\} (\text{ANGLES}) [\underline{T1}] r_1 [\underline{T2}] r_2 [\underline{T3}] r_3$$

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 $\theta_1, \theta_2, \theta_3$ as shown in Figure 2.1.7-1, Volume 1, GTSTRUDL User Reference Manual.

Explanation:

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

The ROTATE LOADING command is used to take the global applied loading components from an existing loading condition, rotate them through a set of rotation angles, and copy the new rotated global components to a new or modified different destination loading condition. The existing independent loading condition, the ROTATE load, from which the rotated global load components are computed is specified by the

loading name i_R/a_R . The angles of rotation are specified by the values r_1, r_2, r_3 . These rotation angles are defined according to the same conventions as those that define the local support release directions in the JOINT RELEASE command described in Section 2.1.7.2, Volume 1 of the GTSTRUDL User Reference Manual, and illustrated in Figure 2.1.7-1.

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

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

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

Examples:

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

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

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

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

Error Messages:

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

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

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

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

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

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

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

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

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

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

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

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

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

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

5.4.3 RUN Command

The RUN command allows you run external programs or DOS batch (cmd) files with a GTSTRUDL command. This is useful for automating procedures that rely on GTSTRUDL generated data, such as a user created design program that needs member ends forces from GTSTRUDL.

The RUN command has been improved to allow new options and longer commands. Implementation has changed from the "C" system library to a Microsoft API, which is more robust way to run external programs.

Syntax

RUN ((BATCH) (KEEP)) (WAIT) 'program'

where program = a ".exe", ".bat" or ".cmd" file or DOS command, along with arguments. The total length of 'program' is limited to 255 characters. You cannot use the quote/apostrophe character (') in 'program', but double quotes (") are acceptable.

Explanation:

- | | |
|-------|---|
| BATCH | Indicates that a ".bat" or ".cmd" file or DOS command will be executed. ".bat" or ".cmd" file are DOS script files, containing DOS commands or other programs. The starting directory for the DOS environment is the current GTSTRUDL working directory. The Microsoft API now being used needs to know that a DOS environment is needed before it runs. |
| KEEP | If BATCH is used, the KEEP option leaves the created DOS window active after the requested action is completed. You can then type new commands in the DOS window. You can kill the DOS window with the "X" in the upper, right hand corner when you are finished. |
| WAIT | Indicates that GTSTRUDL command processing should stop until 'program' has completed. If this option is not used, GTSTRUDL will continue and process the next command, i.e. if you are running a macro or an input file. Use the WAIT option if the results of 'program' are required for subsequent GTSTRUDL commands. |

Examples:

`RUN BATCH KEEP 'DIR/W'`

This command will open a DOS window, display the contents of the GTSTRUDL working directory in the "/W" format, and leave the DOS window open so you can type more commands. The DOS window is a separate program, so you can continue GTSTRUDL operations while the DOS window is open.

`RUN 'NOTEPAD myfile.txt'`

This command will open Notepad with file 'myfile.txt'. Again, Notepad is a separate process, so you can continue GTSTRUDL operations while Notepad is open.

`RUN BATCH WAIT 'find "$" input.gti > comments.gti'`
`CINPUT 'comments.gti'`

The first command will use the DOS command "find" to locate the comments in the input file "input.gti" and put them into a file named "comments.gti". The WAIT option tells the GTSTRUDL command processor to wait until the DOS operation is complete before trying to CINPUT the generated file.

5.4.4 COUTPUT Command

The COUTPUT command now can replace (overwrite) an existing output file. Previously, an existing file could be appended only.

$$\underline{\text{COUTPUT}} \left(\begin{array}{c} \rightarrow \underline{\text{APPEND}} \\ \underline{\text{REPLACE}} \\ \underline{\text{STANDARD}} \end{array} \right) ('file_name')$$

where,

'file_name' is a new or existing text file. 'file_name' is limited to 256 characters and must be enclosed in quotes (apostrophes).

Explanation:

APPEND is the default action, so "COUTPUT 'file1'" and "COUTPUT APPEND 'file1'" are equivalent. APPEND tells GTSTRUDL to add subsequent output to the end of the specified file. If APPEND is requested, 'file_name' must be given.

REPLACE tells GTSTRUDL to delete the contents of the specified file and the write subsequent output to the specified file. If REPLACE is requested, 'file_name' must be given.

APPEND and REPLACE act identically when 'file_name' does not already exist. While GTSTRUDL is in the APPEND or REPLACE state, only input (commands) are echo printed in the text window - all generated output will be placed in the specified output file.

STANDARD tells GTSTRUDL to stop writing to the specified output file and direct subsequent output to the text window. This is the output state when GTSTRUDL starts.

Usage:

COUTPUT APPEND 'file1'

All subsequent output, from PRINT, LIST, etc., will be written to 'file1' and will not appear in the text window, although the actual command will be displayed in the text window. If 'file1' existed previously to this COUTPUT request, the new output will appear at the end of the existing contents.

COUTPUT REPLACE 'file2'

All subsequent output, from PRINT, LIST, etc., will be written to 'file2' and will not appear in the text window, although the actual command will be displayed in the text window. If 'file2' existed previously to this COUTPUT request, the existing contents will be deleted and only the new output will appear in 'file2'.

COUTPUT STANDARD

Stop writing output to an output file and write all output to the text window.

5.4.5 Reference Coordinate System Command

General form:

$$\text{REFERENCE (COORDINATE) (SYSTEM) } \left\{ \begin{array}{l} i_1 \\ 'a_1' \end{array} \right\} -$$

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

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 GTSTRUDL Release Guide) to facilitate the creation of the structural model. Reference Coordinate systems created using the above command will be available as Local systems in GTMenu. In a future release, the user will be able to output results such as joint displacements and reactions in a Reference Coordinate System.

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

- (1) Define the origin and rotation of coordinate axes of the reference system with respect to the global coordinate system, and
- (2) define three joints or the coordinates of three points in space.

In either case, i_1 or ' a_1 ' is the integer or alphanumeric identifier of the reference coordinate system. For the first option, v_x , v_y , and v_z are the magnitude of translations in active length units of the origin of this system from the origin of the overall global coordinate system. The translations v_x , v_y , and v_z , are measured parallel to the orthogonal axes X, Y, and Z, respectively, of the global system and are positive in the positive

directions of these axes; v_1 , v_2 , and v_3 are the rotation angles R_1 , R_2 , and R_3 in active angular units between the orthogonal axes of this system and the axes of the overall global coordinate system. The description of these angles is the same as given in Section 2.1.7.2 of Volume 1 of the GTSTRUDL User Reference Manuals for rotated joint releases (2_1 , 2_2 , and 2_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 'a₂' 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 'a₃' 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 'a₄' or v_{10} , v_{11} , and v_{12}) is used to define the XY-plane of the reference system. The positive Y-axis is directed toward the third joint. The Z-axis then is determined by the right-hand rule.

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

Modifications of Reference Systems:

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

Examples:

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

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

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

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

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

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

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

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

- e) DELETIONS
 REFERENCE SYSTEM 2
 ADDITIONS

The above command deletes Reference System 2.

5.4.5-1 Printing Reference Coordinate System Command

General form:

$$\text{PRINT REFERENCE (COORDINATE) (SYSTEM) } \left\{ \begin{array}{l} \rightarrow \text{ALL} \\ \text{list} \end{array} \right\}$$

Explanation:

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

5.4.6 Hashing Algorithm to Accelerate Input Processing

An advanced data-structuring technique called HASHING can now be used when storing and searching lists of joints and/or elements. The command to control this feature is as follows:

$$\text{SET ELEMENTS} \left\{ \begin{array}{l} \text{HASHED} \\ \text{SEQUENTIAL} \end{array} \right\}$$

The following points concern HASHING:

- 1) The benefit of HASHING is that it GENERATES large structures faster. The disadvantage is that it is more complex internally.
- 2) HASHING is disabled by GTMenu. The GTSTRUDL database is usually not modified extensively in GTSTRUDL after invoking GTMenu, so this has minimal affect. However, the SET ELEMENTS HASHED command, when given with an existing database, builds hashing data structures for the existing database.
- 3) The order of a joint and/or element listing is the same for HASHED and SEQUENTIAL unless the structural database has been edited in DELETIONS mode and then in ADDITIONS mode again. Then SEQUENTIAL will place the latest addition in the deleted slot whereas HASHING will append the addition to the end of the list.

This page intentionally left blank.

5.4.7 Add/Delete Without Commands

2.1.3.7 Delete Joints Without Joint Coordinates

General form:

DELETE JOINTS WITHOUT (JOINT) COORDINATES

Explanation:

The DELETE JOINTS WITHOUT JOINT COORDINATES command can be used to delete the joints that do not have joint coordinates. There are times when a user may have accidentally specified data for a joint that does not exist or accidentally specified a non-existent joint name in the PRINT or LIST commands. Once a joint name is specified in a command, the joint name is stored in the database. If joint coordinates have not been specified for a joint and STIFFNESS ANALYSIS is requested, the following error message is output:

```
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 30
```

When the DELETE JOINTS WITHOUT JOINT COORDINATES command is specified, the program searches through all the joint names and deletes the joint names that do not have JOINT COORDINATES.

Example:

```
{ 100} >
{ 101} > STIFFNESS ANALYSIS
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 30
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 31
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 32
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 33
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 34
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 35
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 40
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 45
**** STRUDL ERROR 2.01 - COORDINATES MISSING FOR JOINT 50
**** WARN_STDSY1 -- SCAN mode is ON. Analysis will be halted.

{ 102} >
```

```
{ 103} > SCAN OFF
{ 104} >
{ 105} > DELETE JOINTS WITHOUT JOINT COORDINATES
```

```
Following joints have been deleted: -----
      30          31          32          33          34          35
      40          45          50
```

In the above example, joints 30 to 35, 40, 45, and 50 accidentally were specified in the PRINT JOINT COORDINATES command. The command DELETE JOINTS WITHOUT JOINT COORDINATES deletes the joints that do not have joint coordinates.

2.1.3.8 Delete Members or Finite Elements Without Member or Element Incidences

General form:

DELETE MEMBERS WITHOUT (MEMBER) INCIDENCES

or

DELETE (FINITE) ELEMENTS WITHOUT (ELEMENT) INCIDENCES

Explanation:

The above two commands can be used to delete the members or finite elements that do not have member or element incidences. There are times when a user may have accidentally specified data for a member or a finite element that does not exist or accidentally specified a member or element name in the PRINT or LIST commands. Once a member or a finite element name has been specified in a command, the member or finite element name is stored in the database. If member or element incidences have not been specified for a member or element and STIFFNESS ANALYSIS is requested, the following error message is output.

```
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 20
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 45
```

When the DELETE MEMBERS WITHOUT MEMBER INCIDENCES command is specified, the program searches through all the member names and deletes the members that do not have MEMBER INCIDENCES. When the DELETE FINITE ELEMENTS WITHOUT ELEMENT INCIDENCES command is specified, the program searches through all the finite element names and deletes the elements that do not have ELEMENT INCIDENCES.

Example:

```
{ 97} >
{ 98} > STIFFNESS ANALYSIS
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 20
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 21
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 22
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 23
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 24
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 30
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 33
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 51
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 52
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 53
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 54
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 55
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 27
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 32
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 38
**** STRUDL ERROR 2.02 - INCIDENCES MISSING FOR MEMBER 45
**** WARN_STDSY1 -- SCAN mode is ON. Analysis will be halted.

{ 99} >
{ 100} > SCAN OFF
{ 101} >
{ 102} > DELETE MEMBERS WITHOUT MEMBER INCIDENCES
```

```
Following members have been deleted: -----
      20      21      22      23      24      30
      33      51      52      53      54      55
      27      32      38      45
```

In the above example, some of the members that are listed were accidentally specified in the MEMBER PROPERTIES command and some in the PRINT MEMBER INCIDENCES command. The command DELETE MEMBERS WITHOUT MEMBER INCIDENCES deletes the members that do not have member incidences.

This page intentionally left blank.

5.4.8 The List Sum Forces Command

The LIST SUM FORCES command is used to perform a general computation of resultant forces for a specified set of joints, members, and finite elements, and is described in the section below, which is numbered as it will appear when added to Volume 3 of the GTSTRUDL User Reference Manual.

2.3.7.4 The LIST SUM FORCES Command

General form:

$$\begin{aligned} \text{LIST SUM FORCES } (\text{TITLE 'a}_T\text{'}) (\text{ABOUT } \left\{ \begin{array}{l} \text{JOINT} \left\{ \begin{array}{l} i_c \\ \text{'a}_c\text{'} \end{array} \right\} \\ \left[\begin{array}{l} \underline{X} \\ \underline{Y} \\ \underline{Z} \end{array} \right] \begin{array}{l} v_X \\ v_Y \\ v_Z \end{array} \end{array} \right\}) - \\ (\text{ORIENTATION } \left\{ \begin{array}{l} \left[\begin{array}{l} \underline{TH1} \\ \underline{TH2} \\ \underline{TH3} \end{array} \right] \begin{array}{l} v_1 \\ v_2 \\ v_3 \end{array} \end{array} \right\} \left\{ \begin{array}{l} \text{MEMBER} \\ \text{ELEMENT} \end{array} \right\} \left\{ \begin{array}{l} i_O \\ \text{'a}_O\text{'} \end{array} \right\}) \text{ JOINTS list}_j \text{ MEMBERS list}_m \end{aligned}$$

Elements:

'a_T' = optional 16-character title. The default title is "None Given."

i_c/'a_c' = integer or alphanumeric name of the optional joint about which moment summation is computed.

v_X, v_Y, v_Z = decimal values of the global X, Y, and Z coordinates of the optional point about which moment summation is computed. A value of 0.0 is assumed for coordinates not specified.

v₁, v₂, v₃ = decimal values of the rotation angles that define the orientation of the optional local reference frame in which the force summation results are to be reported. The rotation angles are the same ones used to define the local released support reference frame described in Section 2.1.7.1, Volume 1 of the GTStrudl Reference Manual and illustrated by Figure 2.1.7.1 of that same Reference Manual section.

| | | |
|------------|---|--|
| i_o/a_o' | = | integer or alphanumeric name of the member or finite element that is to be used as the basis of the local reference frame in which the force summation results are to be reported. |
| $list_j$ | = | list of joints at which forces and moments from the members and finite elements of $list_m$ are accumulated for the force summation computation. |
| $list_m$ | = | list of members and finite elements that contribute forces at the joints of $list_j$ for the force summation computation. |

Explanation:

The LIST SUM FORCES command is used to calculate and report the summation of forces and moments at a specified point from a specified set of member and finite element nodal forces for all active static loading conditions. This command is similar to, but more general than, the CALCULATE RESULTANT command described in Section 2.3.7.3, Volume 3 of the GTSTRUDL Reference Manual. The LIST SUM FORCES command options are described as follows:

$$\underline{\text{ABOUT}} \left\{ \begin{array}{l} \underline{\text{JOINT}} \left\{ \begin{array}{l} i_c \\ 'a_c' \end{array} \right\} \\ [\underline{X}] \ v_X \ [\underline{Y}] \ v_Y \ [\underline{Z}] \ v_Z \end{array} \right\}$$

By default, the moment summation is computed by summing the member and finite element forces and moments about the geometric center of the joints specified in $list_j$, assuming the joints are equally weighted. The ABOUT specs provide for the identification of an optional JOINT or point, v_X , v_Y , v_Z , about which the moment summation is performed. The specified joint must be defined, but may be inactive.

$$\underline{(\text{ORIENTATION}} \left\{ \begin{array}{l} [\underline{\text{TH1}}] \ v_1 \ [\underline{\text{TH2}}] \ v_2 \ [\underline{\text{TH3}}] \ v_3 \\ \left\{ \begin{array}{l} \underline{\text{MEMBER}} \\ \underline{\text{ELEMENT}} \end{array} \right\} \left\{ \begin{array}{l} i_o \\ 'a_o' \end{array} \right\} \end{array} \right\})$$

By default, the result of the force and moment summation is a list of three global resultant force components and three global resultant moment components for each active loading condition. The ORIENTATION specs provide a way to specify an optional alternative local reference frame for the summation report.

In the first method, three rotation angles v_1 , v_2 , and v_3 define the orientation of the local reference frame with respect to the global reference frame. The rotation angles are the same ones used to define the local released support reference frame described in Section 2.1.7.1, Volume 1 of the GTSTRUDL Reference Manual and illustrated by Figure 2.1.7.1 of that same Reference Manual section. According to the MEMBER/ELEMENT option, the summation results are reported in the local or planar reference frame of the specified MEMBER or finite ELEMENT.

JOINTS list_j MEMBERS list_m

The LIST SUMFORCES command computes the force and moment summation from the nodal forces and moments at the joints specified in list_j from the members and finite elements specified in list_m. list_m may contain any mix of frame members, truss members, 2-D, and 3-D finite elements and list_j may contain any set of joints, with the only qualification that each element in list_m must be incident on at least one joint in list_j. In addition, every joint and element in list_j and list_m must be defined and active, and list_m may not contain rigid bodies. list_j may contain master joints and slave joints.

The force and moment summation is computed for all active static loading conditions for which member end forces and finite element nodal forces have been computed by a prior STIFFNESS ANALYSIS or by the CREATE PSEUDO STATIC, COMBINE, and CREATE LOAD COMBINATION commands. In the case of a pseudo static loading condition created from a response spectrum modal combination such as RMS or CQC, a force and moment summation is computed first for each mode of the root response spectrum load. The final mode combination force and moment summation is then computed by applying the mode combination rule of the pseudo static loading to the individual modal force and moment summations. In the case of a pseudo static loading condition created from the maximum of a transient loading condition, a force and moment summation is computed for each individual time point, from which the maximum force and moment summation component is then selected.

Error and Warning conditions

The LIST SUM FORCES command issues error and warning messages when the following conditions are encountered. In each case, command processing is terminated, but scan mode is not entered.

```
**** WARNING_STLFISM -- Joint and/or member list not defined. LIST SUM
                        FORCES command ignored.
```

This message is issued when either the JOINT and/or MEMBER list are missing from the LIST SUM FORCES command.

```
**** WARNING_STLFISM -- Specified joint 500          does not exist or is
                        inactive. LIST SUM FORCES command ignored.
**** WARNING_STLFISM -- Specified element 500         does not exist or is
                        inactive. LIST SUM FORCES command ignored.
```

These messages mean that the indicated joint or element in list_j or list_m respectively has not been defined or is inactive.

```
**** WARNING_STLFISM -- Joint 500          for center of force summation does
                        not exist. LIST SUM FORCES command ignored.
```

This message indicates that the specified joint in the ABOUT JOINT option is not defined.

```
**** WARNING_STLFISM -- None of the specified members are incident on
                        joint 24 . LIST SUM FORCES command ignored.
**** WARNING_STLFISM -- Member 40    not incident on any of the specified
                        joints. LIST SUM FORCES command ignored.
```

These messages are issued when the indicated joint is disconnected from the elements named in list_m, or when the indicated member is disconnected from the joints named in list_j respectively.

```
**** WARNING_STLFISM -- There are no active loading conditions. LIST SUM
                        FORCES command ignored.
```

This message indicates that no active loading conditions were found.

```
**** WARNING_STLFM -- Member/element force results not available for  
member 25 loading 2. LIST SUM FORCES command  
ignored.
```

This message indicates that static analysis results do not exist for the specified member and loading condition. The output of this message obeys the OUTPUT WARNING MESSAGE LIMIT command.

```
**** WARNING_STLFM -- Specified element RB1 is an invalid rigid body.  
LIST SUM FORCES command ignored.
```

This message is issued when a rigid body is detected in list_m.

Example:

Example LSF01 demonstrates the use of the LIST SUM FORCES command for the simple steel-concrete composite beam shown in Figure 2.3.7.4-1 below. The nodal forces from Joints 41 to 45 at $X = 40$ ft. from plate elements 29 to 32 and beam member 56 (the more darkly-shaded elements) are used to compute the force summation at the cross section $X = 40$ ft. The command listing is shown in Figure 2.3.7.4-2. The report from the LIST SUM FORCES command is shown in Figure 2.3.7.4-3.

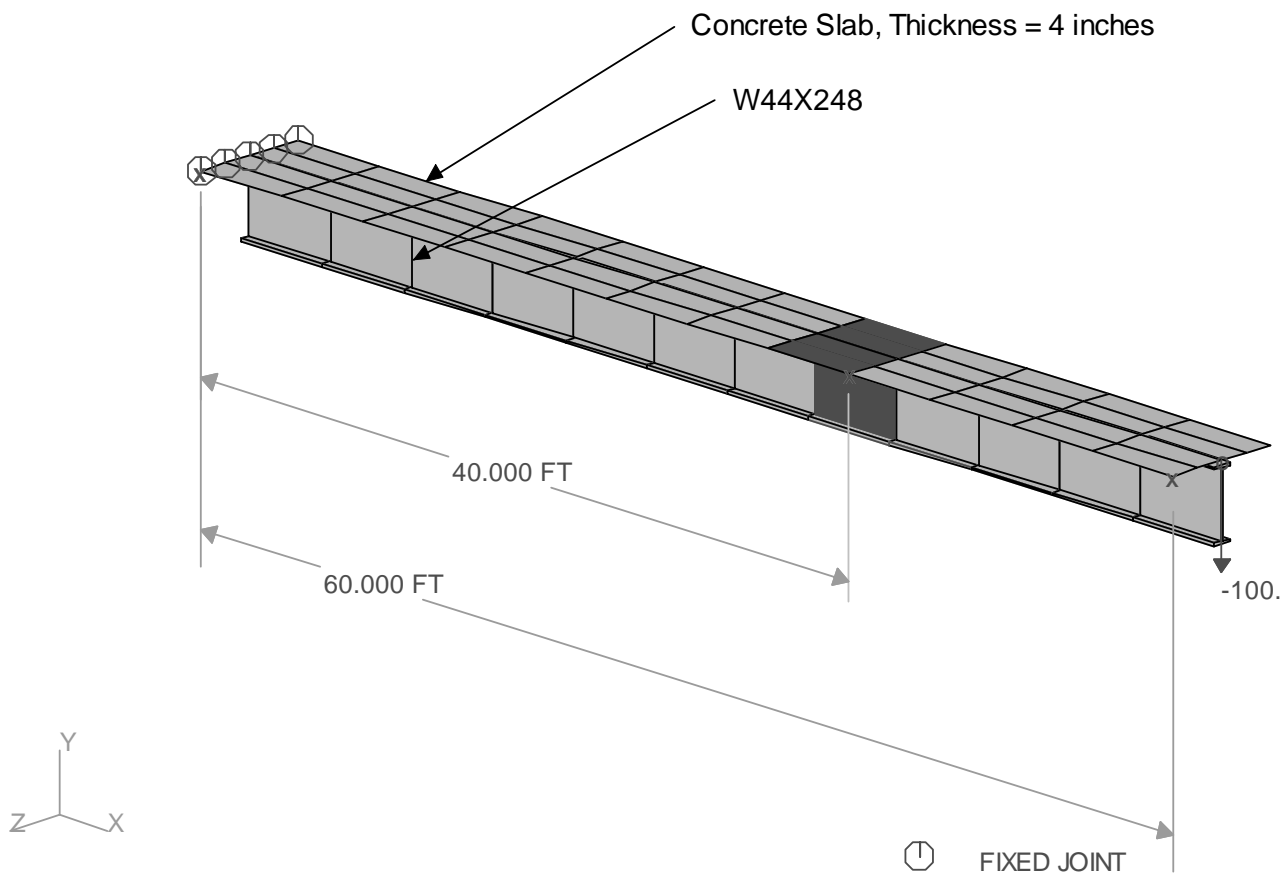


Figure 2.3.7.4-1 Example LFS01, Concrete-Steel Composite Cantilever Beam

```

$$$ **
$$$ ** This job demonstrates the LIST SUM FORCES command for a
$$$ ** steel-concrete composite beam
$$$ **
UNITS  FEET  KIPS  DEG  FAH
$$
$$
GTMenu POINT COORDINATES
  'P1'          0.0000      0.0000     -3.0000
  'P2'         60.0000      0.0000     -3.0000
  'P3'          0.0000      0.0000      3.0000
$$
$$
GTMenu LINE INCIDENCES

```

GT STRUDL

The LIST SUM FORCES Command

| 'C1 ' | 'P2 ' | 'P1 ' |
|-------|-------------|------------------------|
| 'C2 ' | 'P1 ' | 'P3 ' |
| \$\$ | | |
| \$\$ | | |
| UNITS | FEET | KIPS DEG FAH |
| \$\$ | | |
| \$\$ | | |
| JOINT | COORDINATES | GLOBAL |
| 1 | | 0.0000 0.0000 -3.0000 |
| 2 | | 0.0000 0.0000 -1.5000 |
| 3 | | 0.0000 0.0000 0.0000 |
| 4 | | 0.0000 0.0000 1.5000 |
| 5 | | 0.0000 0.0000 3.0000 |
| 6 | | 5.0000 0.0000 -3.0000 |
| 7 | | 5.0000 0.0000 -1.5000 |
| 8 | | 5.0000 0.0000 0.0000 |
| 9 | | 5.0000 0.0000 1.5000 |
| 10 | | 5.0000 0.0000 3.0000 |
| 11 | | 10.0000 0.0000 -3.0000 |
| 12 | | 10.0000 0.0000 -1.5000 |
| 13 | | 10.0000 0.0000 0.0000 |
| 14 | | 10.0000 0.0000 1.5000 |
| 15 | | 10.0000 0.0000 3.0000 |
| 16 | | 15.0000 0.0000 -3.0000 |
| 17 | | 15.0000 0.0000 -1.5000 |
| 18 | | 15.0000 0.0000 0.0000 |
| 19 | | 15.0000 0.0000 1.5000 |
| 20 | | 15.0000 0.0000 3.0000 |
| 21 | | 20.0000 0.0000 -3.0000 |
| 22 | | 20.0000 0.0000 -1.5000 |
| 23 | | 20.0000 0.0000 0.0000 |
| 24 | | 20.0000 0.0000 1.5000 |
| 25 | | 20.0000 0.0000 3.0000 |
| 26 | | 25.0000 0.0000 -3.0000 |
| 27 | | 25.0000 0.0000 -1.5000 |
| 28 | | 25.0000 0.0000 0.0000 |
| 29 | | 25.0000 0.0000 1.5000 |
| 30 | | 25.0000 0.0000 3.0000 |
| 31 | | 30.0000 0.0000 -3.0000 |
| 32 | | 30.0000 0.0000 -1.5000 |
| 33 | | 30.0000 0.0000 0.0000 |
| 34 | | 30.0000 0.0000 1.5000 |
| 35 | | 30.0000 0.0000 3.0000 |
| 36 | | 35.0000 0.0000 -3.0000 |
| 37 | | 35.0000 0.0000 -1.5000 |
| 38 | | 35.0000 0.0000 0.0000 |
| 39 | | 35.0000 0.0000 1.5000 |
| 40 | | 35.0000 0.0000 3.0000 |
| 41 | | 40.0000 0.0000 -3.0000 |
| 42 | | 40.0000 0.0000 -1.5000 |
| 43 | | 40.0000 0.0000 0.0000 |
| 44 | | 40.0000 0.0000 1.5000 |
| 45 | | 40.0000 0.0000 3.0000 |
| 46 | | 45.0000 0.0000 -3.0000 |
| 47 | | 45.0000 0.0000 -1.5000 |
| 48 | | 45.0000 0.0000 0.0000 |
| 49 | | 45.0000 0.0000 1.5000 |
| 50 | | 45.0000 0.0000 3.0000 |
| 51 | | 50.0000 0.0000 -3.0000 |

General Prerelease Features

GT STRUDL

| | | | |
|----|---------|--------|---------|
| 52 | 50.0000 | 0.0000 | -1.5000 |
| 53 | 50.0000 | 0.0000 | 0.0000 |
| 54 | 50.0000 | 0.0000 | 1.5000 |
| 55 | 50.0000 | 0.0000 | 3.0000 |
| 56 | 55.0000 | 0.0000 | -3.0000 |
| 57 | 55.0000 | 0.0000 | -1.5000 |
| 58 | 55.0000 | 0.0000 | 0.0000 |
| 59 | 55.0000 | 0.0000 | 1.5000 |
| 60 | 55.0000 | 0.0000 | 3.0000 |
| 61 | 60.0000 | 0.0000 | -3.0000 |
| 62 | 60.0000 | 0.0000 | -1.5000 |
| 63 | 60.0000 | 0.0000 | 0.0000 |
| 64 | 60.0000 | 0.0000 | 1.5000 |
| 65 | 60.0000 | 0.0000 | 3.0000 |

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

\$\$

\$\$

TYPE SPACE FRAME

MEMBER INCIDENCES

| | | |
|----|----|----|
| 49 | 3 | 8 |
| 50 | 8 | 13 |
| 51 | 13 | 18 |
| 52 | 18 | 23 |
| 53 | 23 | 28 |
| 54 | 28 | 33 |
| 55 | 33 | 38 |
| 56 | 38 | 43 |
| 57 | 43 | 48 |
| 58 | 48 | 53 |
| 59 | 53 | 58 |
| 60 | 58 | 63 |

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

MEMBER ECCENTRICITIES GLOBAL

49 TO 60 START Y -2.0000 END Y -2.0000

\$\$

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

MEMBER PROPERTIES TABLE 'WBEAM9 ' 'W44x248

49 TO 60

\$\$

STATUS SUPPORT -

| | | | | |
|---|---|---|---|---|
| 1 | 2 | 3 | 4 | 5 |
|---|---|---|---|---|

\$\$

\$\$

UNITS FEET KIPS DEG FAH

\$\$

CONSTANTS

BETA 0.00000 ALL

\$\$


```

$$
TYPE PLATE
GENERATE 4 ELEMENTS ID 1 1 FROM 1 1 TO 2 1 TO 7 1 TO 6 1
REPEAT 11 TIMES ID 4 FROM 5 TO 5 TO 5 TO 5
$$
$$
UNITS FEET KIPS DEG FAH
$$
CONSTANTS
      E 5.1839997E+05 ALL
      G 2.0735998E+05 ALL
      POI 1.7000000E-01 ALL
      DEN 1.4999039E-01 ALL
      CTE 5.5000000E-06 ALL
MATERIAL STEEL MEMBERS 49 TO 60
$$
UNITS FEET KIPS DEG FAH
$$
$$
ELEMENT PROPERTIES
      1 TO 48 TYPE 'SBHQ6' THICK 0.3333
$$
UNITS FEET KIPS DEG FAH
$$
LOADING 1
JOINT LOADS
      63 FORCE Y -100.0

STIFFNESS ANALYSIS

UNITS KIPS FEET
LIST FORCES MEMBER 56
LIST ELEMENT FORCES ELEMENTS 29 TO 32

$$* **
$$* ** Resultant sum of forces in the composite beam at
$$* ** X = 40 ft., using the member and finite element nodal
$$* ** forces at joints 41 to 45 of elements 29 to 32 and
$$* ** member 56
$$* **
LIST SUM FORCES TITLE 'Example LSF01' ELEMENTS 29 TO 32 56 -
      JOINTS 41 TO 45
FINISH

```

Figure 2.3.7.4-2 List of Commands for Example LSF01

```

{ 177} > $$* **
{ 178} > $$* ** Resultant sum of forces in the composite beam at
{ 179} > $$* ** X = 40 ft., using the member and finite element nodal
{ 180} > $$* ** forces at joints 41 to 45 of elements 29 to 32 and
{ 181} > $$* ** member 56
{ 182} > $$* **
{ 183} > LIST SUM FORCES TITLE 'LSF01' ELEMENTS 29 TO 32 56 JOINTS 41 TO 45

*****
* RESULTS FROM LATEST ANALYSIS *
*****

ACTIVE UNITS (UNLESS INDICATED OTHERWISE):
      LENGTH      WEIGHT      ANGLE      TEMPERATURE      TIME
      FEET        KIP        DEG        DEGF        SEC

Member and Finite Element Force Resultants
=====

Title: Example LSF01

Sum of Forces and Moments about      40.0000      0.0000      0.0000
Local Axis Orientation Angles:  Z      0.0000  Y      0.0000  X      0.0000

      Loading      FX      FY      FZ      MX      MY      MZ
      -----      -
1          0.0000     -100.0000      0.0000      0.0000      0.0000     -2000.0001

```

Figure 2.3.7.4-3 Report From the LIST SUM FORCES Command, Example LSF01

5.4.9 GTMenu Point and Line Incidences Commands

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

(1) GTMenu POINT COORDINATES

General Form:

GTMenu POINT COORDINATES

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

$$\begin{array}{c} A \\ A \\ A \end{array}$$

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

Elements:

$$\text{coordinate-specs} = [\underline{X}] v_1 [\underline{Y}] v_2 [\underline{Z}] v_3$$

Where,

$$i_1, i_2, \dots, i_n = \text{unsigned integer Point identifiers.}$$

$$'a_1', 'a_2', \dots, 'a_n' = \text{1 to 8 character alphanumeric Point identifiers.}$$

$$v_1, v_2, v_3 = \text{Cartesian Point coordinates (integer or real).}$$

(2) **GMenu LINE INCIDENCES****General Form:**GMenu LINE INCIDENCES

$$\begin{array}{c}
 \left\{ \begin{array}{c} i_1 \\ 'a_1' \end{array} \right\} \\
 \text{A} \\
 \text{A} \\
 \text{A} \\
 \\
 \left\{ \begin{array}{c} i_n \\ 'a_n' \end{array} \right\}
 \end{array}
 \begin{array}{l}
 \text{type}_1 \text{ incidence-specs}_1 \\
 \\
 \\
 \\
 \\
 \text{type}_n \text{ incidence-specs}_n
 \end{array}$$

Elements:

$$\text{type} = \left\{ \begin{array}{l} \rightarrow \underline{\text{LINE}} \\ \underline{\text{POLYNOMINAL}} (\underline{\text{CURVE}}) \\ \underline{\text{ARC}} (\underline{\text{TEMPLATE}}) \\ \underline{\text{CENTERED}} (\underline{\text{ARC}}) \underline{\text{PERCENT}} v_1 \\ \underline{\text{BEZIER}} (\underline{\text{CURVE}}) \\ \underline{\text{SPLINE}} (\underline{\text{CURVE}}) (\underline{\text{ORDER}} k_2) \end{array} \right\}$$

$$\text{incidence-specs} = \left\{ \begin{array}{c} i_1 \\ 'a_1' \end{array} \right\} \cdot \left\{ \begin{array}{c} i_2 \\ 'a_2' \end{array} \right\} \cdot \dots \cdot \left\{ \begin{array}{c} i_p \\ 'a_p' \end{array} \right\}$$

Where,

- i_1, i_2, \dots, i_n = unsigned integer Line/Curve identifiers.
- 'a₁', 'a₂', ..., 'a_n' = 1 to 8 character alphanumeric Line/Curve identifiers.
- i_1, i_2, \dots, i_p = unsigned integer Point identifiers used.
- 'a₁', 'a₂', ..., 'a_p' = 1 to 8 character alphanumeric Point identifiers.
- v_1 = positive number (integer or real).
- k_2 = integer between 2 and the number of incidences.
- 1, 2, ..., p = Point subscripts for a Line/Curve. The following table gives the number of Points used to specify different types of Line/Curve:

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

End of Document.