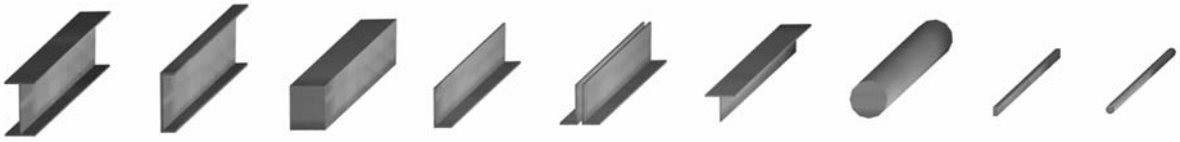
STRUCTURAL DESKTOP ®

Users Manual

Version 4.0



Structural Desktop® Users Manual

Structural Desktop® Users Manual

STRUCTURAL DESKTOP®
Users Manual

Contents

INTRODUCTION

The Program

The purpose of ***Structural Desktop*** is to convert structural information from a single model to Structural Contract Documents. Since different structural engineering offices have different flows of information, ***Structural Desktop*** works either with or without analytical/design software. Designs, drawings and material reports are placed firmly in the hands of the engineer.

Structural Desktop can be used to:

- Move models created in CAD packages into analytical files
- Move models from analytical files into ***Structural Desktop***
- Create drawings, models, and material reports from the Structural Desktop Model

This independent approach permits maximum flexibility in the flow of engineering information in your office. ***Structural Desktop*** was written by Structural Desktop, Incorporated and, with the release of Version 4, is designed to run within three different generations of AutoCAD. There are three separate installation programs on each disk sold. One installation will install a version of ***Structural Desktop*** compatible with AutoCAD 2000, 2000i, and 2002. Another is compatible with AutoCAD 2004, 2005 and 2006. The final compilation of the program is compatible with AutoCAD 2007 and later versions using the same compilation criteria.

Structural Desktop contains structural members from the American Institute of Steel Construction (**AISC**), Steel Joist Institute (**SJI**), Light Gage Structural Institute (**LGSI**), American Iron and Steel Institute (**AISI**) and the Prestressed Concrete Institute (**PCI**). Prismatic sections and plate elements are also supported.

Key aspects of ***Structural Desktop*** are the ability to automate the drawing of structural objects and to create and read Analytical/Design files. These allow the Structural Engineer to utilize all the information created in the Analytical/Design phase with speed and accuracy and eliminates the manual transfer of design information to a drawing.

Structural Desktop **LT** serves the drafting needs of the designers, draftspersons, Architects, and Engineers. ***Structural Desktop*** **LT** does not read or write analytical files, but contains all of the other features and functionality of ***Structural Desktop*** to run within AutoCAD. ***Structural Desktop*** **LT** is fully compatible with ***Structural Desktop*** files, and vice versa. Both will work with DXF files from other software and with DWG files from AutoCAD, AutoCAD LT, or Architectural Desktop as a starting point.

The final products of both programs are 100% AutoCAD drawings, the output drawings of either may be inserted, x-referenced, or combined with other files in any way that any other AutoCAD drawing may be used.

Book's Intent

The intent of the ***Structural Desktop*** manual is to give you reference information on the procedures, functions, and commands of ***Structural Desktop***. As you become more familiar with the program, more advanced features will begin to be easier to understand. ***Structural Desktop*** was created by structural engineers to be utilized by structural engineers, designers and draftsmen in the development of structural contract documents.

The ***Structural Desktop Tutorial*** will take you through a step-by-step process of a structural project from analysis to the development of structural contract documents. We strongly recommend following the step-by-step procedure established in the tutorial until you are comfortable with the program. In AutoCAD there are many ways to complete a set of drawings for a project, and this is also true of ***Structural Desktop***. Once you have completed the tutorial, you will be ready to experiment with different commands and procedures for creating contract documents utilizing the power of AutoCAD combined with ***Structural Desktop*** to fully compliment the workflow of your office standards.

Object Modeling Features

The objects that are automatically drawn by Structural Desktop include the following structural entities:

TABLE SECTIONS

- Structural Steel Sections (**AISC or British Standard Sections**)
  - Wide Flanges and similar sections (W, M, S, HP or UC, UB, UP)
  - Steel Tee Sections (WT, ST, MT or UCT, UBT, UPT)
  - Channels and Misc. Channels
  - Angles
  - Double Angles
  - Rectangular and Square Tubes
  - Hollow Steel Structural Sections
  - Pipe, from a Table or Custom

- Joist and Joist Girders (**SJI**) (American Tables Only)

- Light-Gage Sections (**LGSI & AISI**) (American Tables Only)
  - C, CS, CU
  - Z, ZS, ZU
  - HU

- Pre-stress Concrete (**PCI**) (American Tables Only)
  - L's
  - Inverted Tees

CUSTOM SECTIONS

- Prismatics (round or rectangular) in steel, aluminum, concrete, and wood
- Three or Four edged Elements in steel, aluminum, concrete, and wood.
- Tapered I-Sections defined by depth at start and end, with custom depth and width of top and bottom flanges and web thickness.
- Tapered Tubes defined by depth at start and end, with custom thickness and with 4, 6, 8, 12, and 16 sides, or round.
- Custom Piles, defined by depth at start and end, solid) in steel, aluminum, concrete, and wood.
- Double Tees created from Elements, with variable Leg depth and spacing.

CHAPTER 1 - Fundamental Concepts

The objective of this chapter is to help you understand the following concepts:

- The basic process of ***Structural Desktop***.
- The terminology used in ***Structural Desktop***.
- Commands in the Shortcut Menu.
- The AutoCAD World Coordinate System.
- The files used and needed in ***Structural Desktop***.

The *Structural Desktop* Process

The objective of Structural Desktop is to assist you, the engineer, to connect your analytical/design model to final drawings and contract documents.

The **analytical/design** model contains the physical information required for analysis. The ***Structural Desktop*** model represents actual relationships of the structural components required for construction. The ***Structural Desktop Program*** links the two, giving you many options, simple commands, and a powerful program.

You can begin by:

- Creating a model in ***Structural Desktop*** using AutoCAD and any information and drawing files you have. You can create analytical files from that model.

  - OR -

- Creating a model in your familiar analytical program through their methods.

Performing your analysis consists of adding information on loads, design codes, and other criteria specific to an analytical package. Once you have completed your structural analysis and your data file fulfills your design requirements:

- Create a ***Structural Desktop*** model from your analytical file.
- Modify the analytical file to true-length members in their correct relative positions.
- Create the plan views, elevations and sections required for construction documents.
- Complete dimensions, annotations and notes in AutoCAD and plot.

Structural Desktop can be used before and after your analysis phase. By applying the full power of AutoCAD, you can create complex models more quickly and accurately using only the LINE command, or every tool AutoCAD provides.

After you have performed your analysis, ***Structural Desktop*** is an aid to getting finished drawings from the information you have in the analytical/design file. It does not finish drawings for you and it does not perform the analysis. What ***Structural Desktop does*** do for you is create sharp, professional, and accurate drawings from a model. ***Structural Desktop*** gives you the tools to create and adjust that model, as quickly as possible, from the centroid model you create for analysis/design OR from simple lines and faces drawn in AutoCAD.

For use before analysis, Structural Desktop includes:

- A line generator that works to create lines in a rectilinear building pattern, that can be used again and again to lay out the basic outlines of a building with different modules.

- The full freedom to adjust the model and add any lines, stretch, copy and mirror lines until your model is ready to become a ***Structural Desktop*** model.

- Output of the model in STAAD.Pro, GT STRUDL, RISA 3D, or SAP 2000 formats...any one or all four.

- Provided tables with Joists, C's and Z's, and concrete members that can be used for analysis and loading within STAAD.Pro, GT STRUDL, and RISA 3D and for drawing purposes when you return the data file to ***Structural Desktop.***

After analysis, Structural Desktop includes:

- Top of Member commands to establish finished floor levels.

- Sloping commands to line up roof members.

- Viewing commands to isolate the members by member number or location as well as through layer techniques integral to AutoCAD.

- Creation of 3D and 2D drawings of all or part of a structure, ready to be completed or passed on to a draftsman using AutoCAD alone.

- Creation of Material Reports grouping and summarizing Steel and Concrete as simple text files for use in any word processor.

- Retention of additional analytical data (e.g. Loads) to return to your analysis file when you have made changes in the ***Structural Desktop*** model.

Structural Desktop **Terminology**

1. A **JOINT** or **NODE** is a set of three-dimensional coordinates, X Y and Z, with Z as the elevation axis. From the analytical/design software, ***Structural Desktop*** will automatically perform the transformation of coordinates from Y-axis vertical to Z-axis vertical in order to function with all other AutoCAD-based applications.

2. **MEMBERS** are objects defined by a line connecting two joints/nodes. The line representing the members is referred to as the **Centroid** of the member. A member has a set of properties such as type, density, depth, and width that are copied from the analysis file or can be added in ***Structural Desktop***. All properties can be edited within the available dialog boxes. In differentiating between the ends of the member, we refer to the first joint of the ordered pair that create the member as the **Start** or **J** joint, and the second as the **End** or **K** joint. A line can also represent a **Complex Member** that is representative of more than one section such as the various joist members, which have an additional dialog box for editing special features.

3. An **ELEMENT** connects three or four points, as a plate surface. Elements have properties of density and thickness and are represented by an AutoCAD entity called a 3d Face in their simplest view. **Complex Elements** are currently represented only by the ability to change elements to **Double Tees**, and edit special values associated with them through a special dialog box.

4. An **OFFSET** is a value that may be added or subtracted from the coordinate-location of a member or element to change its length or position in space. Joints never move; once placed they remain fixed in their location. The actual end of the member or corner of an element, however, can be moved with relation to the joint that supports it.

5. A **BETA ANGLE** is a measurement of the rotation of a member around its axis. In the case of a wide flange, for example, a zero Beta Angle on a vertical column will align the web with the X-axis. Any horizontal or sloped member will have the web aligned with the vertical axis. ***Structural Desktop*** will reflect changes in the Beta Angle by rotating a member around its own axis, and Beta Angles of 0 degrees and all multiples of 90 degrees will draw properly in all two dimensional representational output drawings. In the special case of Angles, the back edge of the angle will point upward in a horizontal member with a 0 degree beta, and angles will properly represent all multiples of 45 degrees. It is recommended the user experiment with various structural objects to understand how the Beta Angle is applied.

6. **Units -** If you are using American tables, all units are imported/changed to inches in ***Structural Desktop***. All other sets of tables assume metric units, and these units will be changed to centimeters as they are read from your input file. For example, if your coordinates are in Meters, all measurements will be multiplied to the correct corresponding measurement in Centimeters within AutoCAD and ***Structural Desktop.***

The Shortcut Menu

The commands and dialog boxes within ***Structural Desktop*** are made available by using the Shortcut Menu in AutoCAD or by typing the command on the AutoCAD command line. You can access the Shortcut Menu by right clicking the mouse key while the cross hairs are on the graphic screen. Then you select the menu item **Structural Desktop** with the pointer. This displays the commands found in ***Structural Desktop*** as shown in **Figure 1 - 1.**

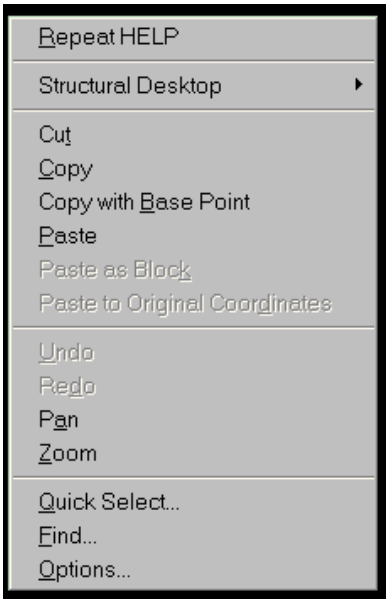


Figure 1 - 1

If you do not see the ***Structural Desktop*** menu item on your computer, refer to the ***Structural Desktop: Installation and Getting Started*** document. The program is optimized to run through this menu, but the Command Line commands to access ***Structural Desktop*** functionality are listed in **Appendix A**.

The *Structural Desktop* Coordinate System

Structural Desktop uses the World Coordinate System from AutoCAD. Many analytical programs use a coordinate system where the Y-axis is vertical and, in plan view, the Z-axis is positive toward the bottom of your monitor screen and the X-axis is positive toward the right.

A plan view of the ***Structural Desktop*** model looks the same as a plan view in your analytical model. To maintain this orientation from your analytical file, ***Structural Desktop*** automatically changes the coordinates to the World Coordinate System at the time the files are transferred from the analysis program. The X-axis is unchanged, the Z-

axis is now the elevation axis, and the former Z-axis from your analytical file will have the same numeric value, conserving your coordinate system, but will be opposite in sign.

A member in an analytical file between the joint coordinates (**100, 120, 240**) and (**344.5, 120, 240**) is 20' 4-1/2" in length and at an elevation of 10' - 0".

Figure 1 - 2 at right shows the member in analytical coordinates at the bottom and in AutoCAD's coordinate system at the top.

Structural Desktop coordinates of the member will be from (**100, -240, 120**) to (**344.5, -240, 120**), with the same length, elevation, and orientation on the screen. You still have the same coordinate, values and the orientation and relative positions and distances will be unchanged.

Software that uses vertical Z as AutoCAD does is converted unchanged.

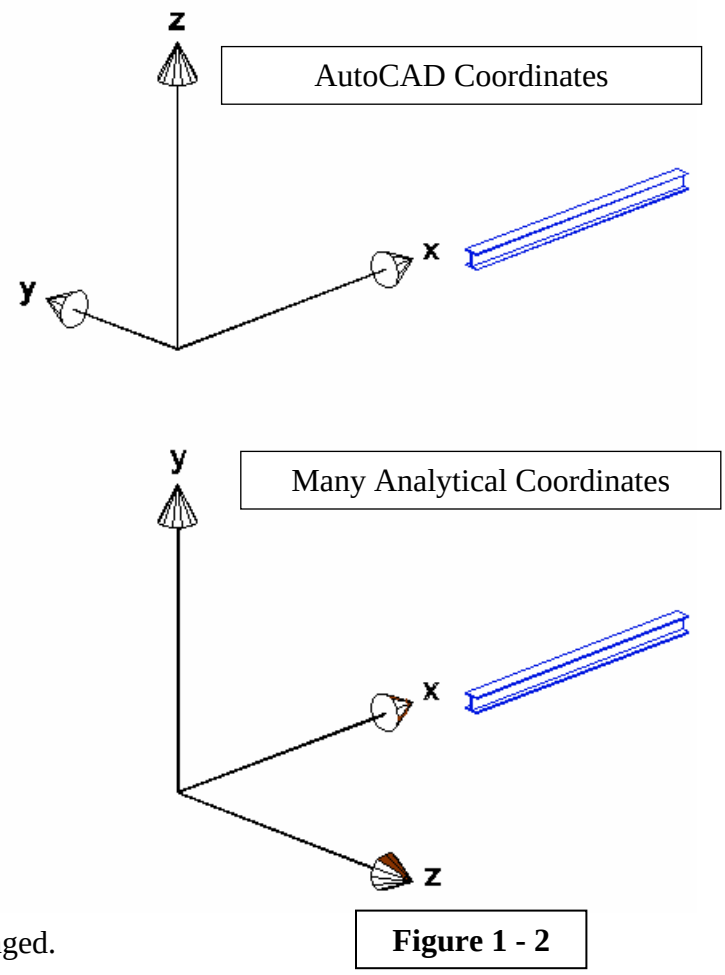


Figure 1 - 2

Working With Files

Structural Desktop saves all of its database information in a file format that we have given the extension HLD, and refer to as a HOLD file (as in "Here, HOLD this for me"). Working with the input files (analytical), HOLD files (***Structural Desktop***), and output files (AutoCAD DWG files.) is extremely important. You are integrating three software packages; managing the files is the key to success.

You have three-phases in the process as shown in **Figure 1 - 3** below.

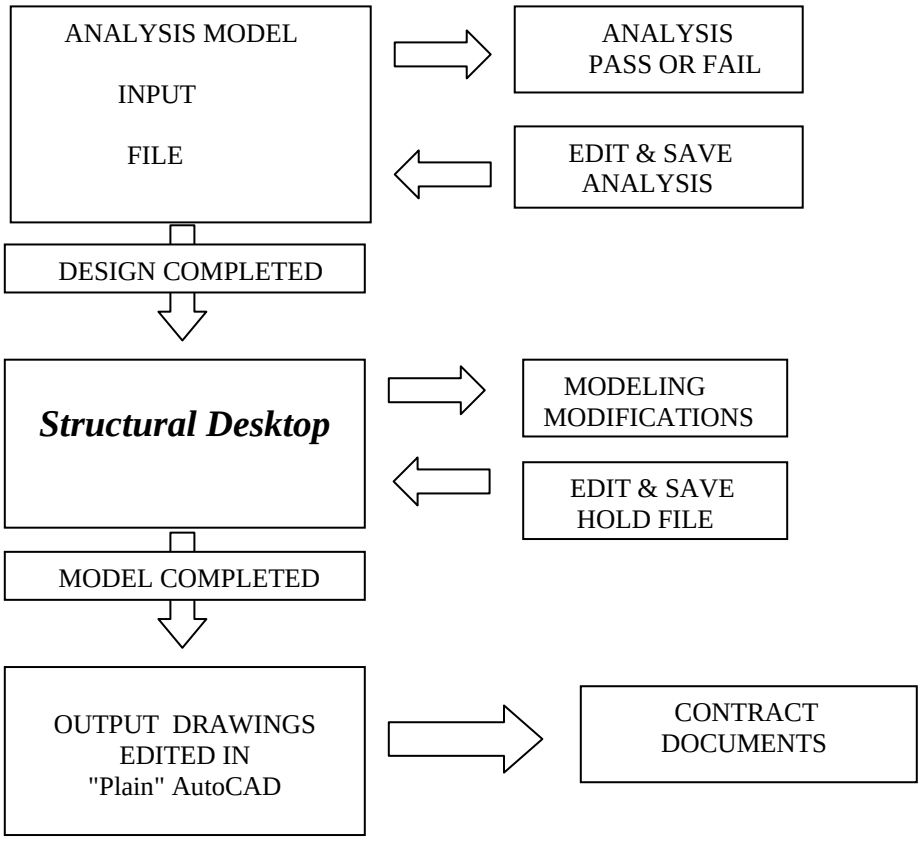


Figure 1 - 3

The most important detail to remember in working with the files is that ***Structural Desktop*** **WILL NOT** save **DATA** in a drawing file. The ***Structural Desktop*** Model must be saved in a **HOLD** file, and that file can be opened and edited in ***Structural Desktop*** as many times as you have need. The final, resultant drawings that you create from the model are AutoCAD drawings and they must be saved in **Drawing** files using the **SAVE** or **SAVE AS** command from the AutoCAD menu.

In addition to the files that ***Structural Desktop*** uses that we have discussed, you should also be aware that ***Structural Desktop*** also has some data files on your hard drive. These files must be loaded into the directory ***Structural Desktop*** in a non-network installation and consist of data files in the Structural Desktop folder under Program Files on your C drive. If you ever wish to remove ***Structural Desktop*** from a system, this can be accomplished by using the un-install function or by deleting the entire ***Structural Desktop*** directory under the Program Files folder on your C drive. Please note, however, that each installation of ***Structural Desktop*** requires a new authentication code from Structural Desktop, Inc.

CHAPTER 2 - Model Creation

The objective of this chapter is to help you understand the following concepts:

- How to load an analytical/design file into ***Structural Desktop***.
- How to create an analytical file from a Model.
- How to create a Model with AutoCAD without an analytical file.
- How to save your work and return to it at another time.

Loading an Analytical File

Before you attempt to load an analytical file, refer to **Figure 2 - 1** below.

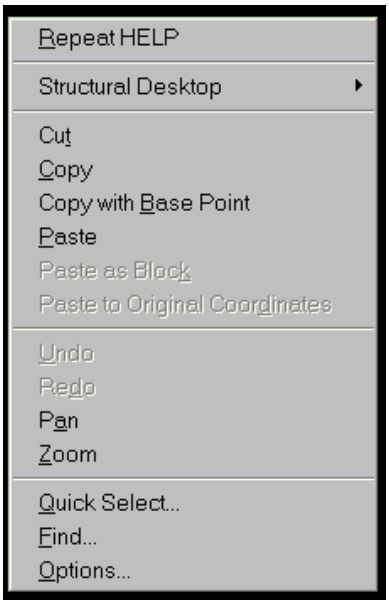


Figure 2 - 1

If you do not see the ***Structural Desktop*** menu item on your computer, refer to the ***Structural Desktop: Installation and Getting Started*** document. The program is optimized to run through this menu, but the Command Line commands to access ***Structural Desktop*** functionality are listed in **Appendix A**.

When you first load ***Structural Desktop,*** the options provided are limited to those that are useful in context. (For example, until you load a file or start a file from scratch, you cannot create a hold file.) ***Structural Desktop*** will not accept a **DRAWING** file. The standard AutoCAD menu will tempt you to use the file input routine provided for AutoCAD drawings, but you must begin your session with the **File** menu section under ***Structural Desktop*** in order to **USE *Structural Desktop***.

If you never click on or type any of the ***Structural Desktop*** commands, your environment continues to be as if you were in an ordinary AutoCAD session. You must select a ***Structural Desktop*** menu item to begin a session with a ***Structural Desktop*** model, and then further functions will be made available.

Figure 2 - 2 below shows the menu as it appears at the beginning of a session. The **FILE** menu item contains the commands and sub-menus to begin working with Structural Desktop.

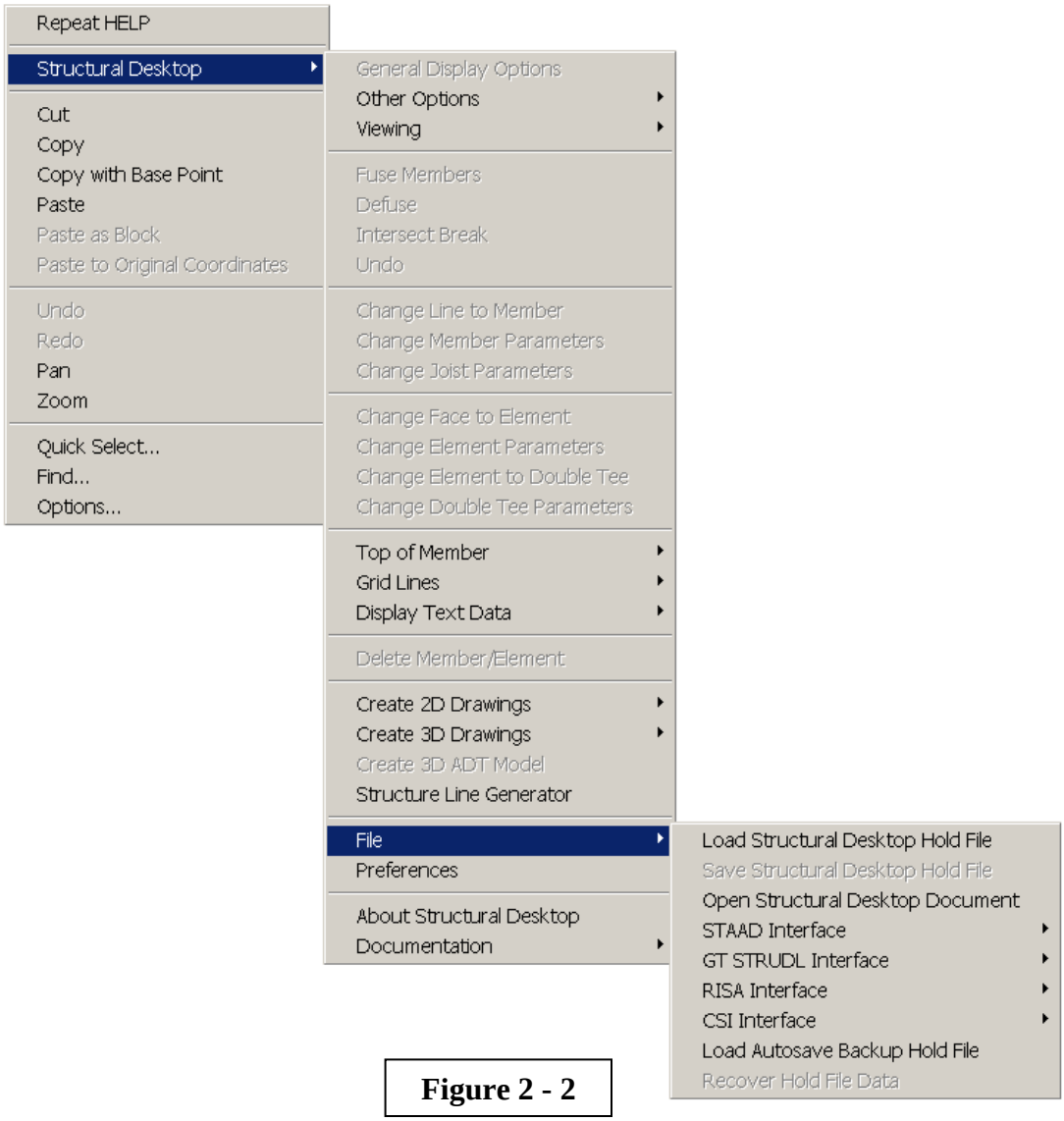
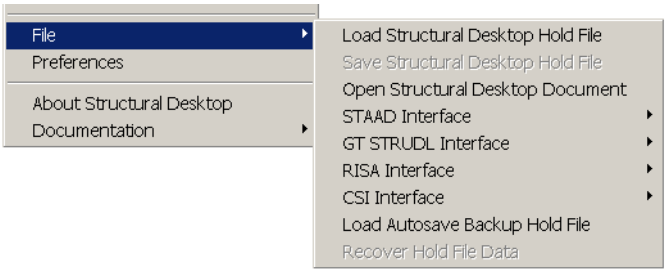


Figure 2 - 2

The menu options that are active at the start of a session are shown on the file menu in **Figure 2.2** above and **Figure 2.3** below. The top menu item, **Load Structural Desktop Hold File,** is covered in the final section of Chapter 2 that details the process of saving and returning to your ***Structural Desktop*** work session at a later time.

The next item shown as active, **Open Structural Desktop Document**, is used to initialize a session when you wish to build a model within SDT from a blank drawing or from some starting point, such as a floorplan. This command is covered in the section on creating a **Structural Desktop Model** without an analytical file.

Figure 2 - 3



The four **Interface Menus** contain commands to load and create files from:

- The Research Engineers, Inc. program **STAAD.Pro**
- The Georgia Tech program **GT STRUDL**
- The RISA Technologies, Inc. program **RISA 3D**
- The Computers and Structures, Inc program **SAP 2000**

Once you have loaded a file from an analytical program, loaded a hold file, or opened a new document, the other menu options will become available that are relevant to model and contract document manipulation and generation. Items that are unavailable are shown on the menu as a dark gray against the gray menu, and are refered to as being "grayed out".

Structural Desktop **LT** does not have functions to interface with analytical files, but can be used to create a model and create drawings, or to read a HOLD file from the unlimited version of ***Structural Desktop*** to create drawings and Material Reports.

The bottom options, **Load Autosave Backup Hold File** and the grayed out **Recover Hold File Data** are also covered in the section on saving and recovering your work.

The next sections deal with the Interface Menus. Each function to **Load** an analytical file is examined, followed by the methods and commands to **Create** an analytical file.

Loading a STAAD Input File

If you do not use STAAD III or STAAD-Pro from Research Engineers, Inc, you may skip this section.

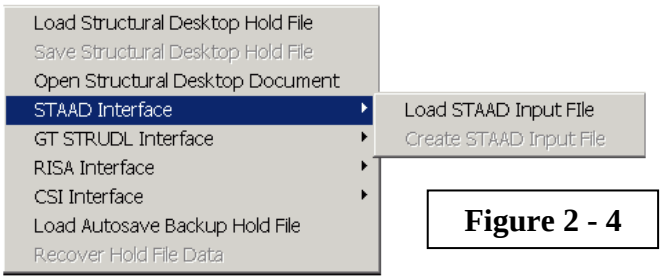


Figure 2 - 4

Structural Desktop uses the input file from Staad III and Staad-Pro from Research Engineers, Inc. as a starting point for model creation. This file ends in the three-letter extension **STD.** When you select this option the program will open the dialog box shown in **Figure 2 - 5** below.

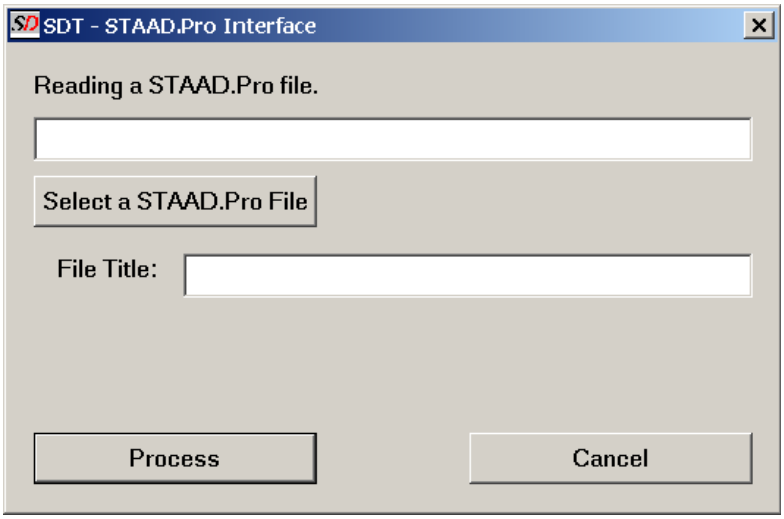


Figure 2 - 5

This is a new program exclusive to ***Structural Desktop*** Version 4 that is invoked by the ***Structural Desktop*** module from within AutoCAD. This program will either read or create a STAAD.Pro file, translating through a proprietary neutral format to the SDT database within AutoCAD.

Loading a STAAD.Pro file is a two step process; you click on the button to [**Select a STAAD.Pro File**] which uses a standard file-selection box primed to view only files ending in the **STD** extension as shown in **Figure 2-6** below.

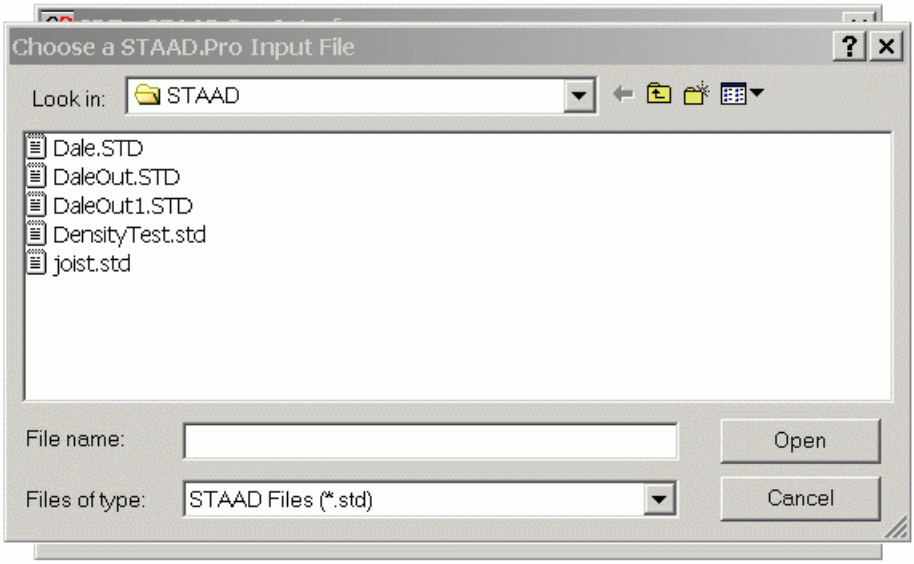


Figure 2 - 6

You may use the buttons at the top of the dialog box to search for your file in other folders. When you double-click the desired file, or select the file and click **Open**, the program returns to the first dialog box:

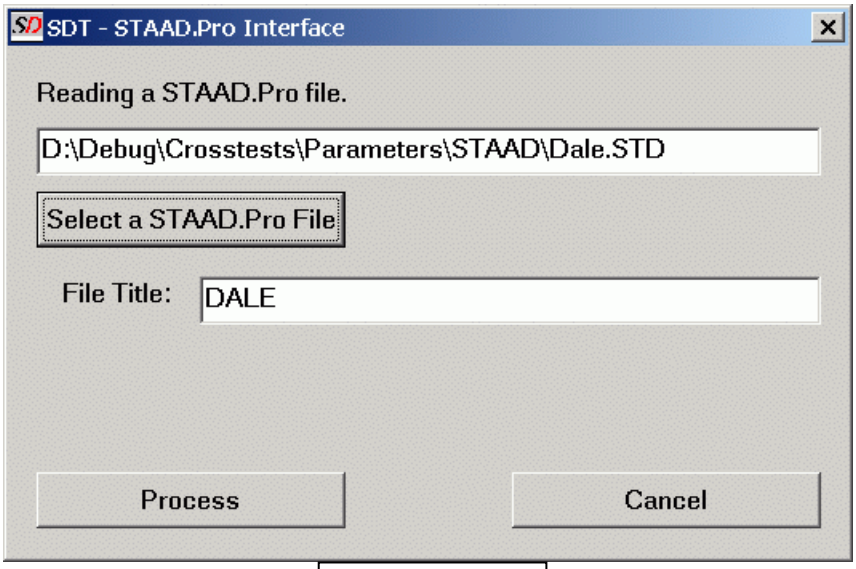


Figure 2 - 7

The STAAD file reader will display the title information from the first line of the STAAD.Pro file as a File Title and displays the complete path to the file as shown in **Figure 2 - 7**. If this is the file you wish to open and use, click on the [**Process**] button. If not, you can again [**Select a STAAD.Pro File**] as many times as you wish, or [**Cancel**] at any time.

When you click the [**Process**] button, the program will load the title of the file for future reference (it will be incorporated into your **Material Report**, for example) and will proceed to read the Joint Numbers and Coordinates, Member Incidences, and Element Incidences. The program will read the Member Properties, Element Thicknesses, and constants such as Beta and Density.

With the introduction of Version 4, Structural Desktop will continue to read and record your STAAD.Pro file with a much greater degree of completeness than in previous versions. It will read and retain Surfaces, Holes in Surfaces, and Solids as additional types of fundamental objects. Structural Desktop will then read your Load Statements, the Joint, Member, and Element loads that follow them, and your Load Combinations, custom materials, and even the Print and Perform Analysis statements. When you choose to create a STAAD.Pro file from within Structural Desktop, the program will completely re-create and write all of the Load, Load Combination, and other lines as it read them from the original STAAD.Pro file. We refer to this attempt to capture all possible analytical data and return it to you as the **ROUNDTURN**.

Please note, Structural Desktop works within AutoCAD with an attempt to carry enough accuracy and precision to be sufficient for all Structural needs. All units within your STAAD.Pro file will be converted into Inches for Length and Kips for force, and the appropriate combinations thereof for torque and other combined forces.

Once you have read the file into Structural Desktop, your Joints will display as small circles, the Members will display as Lines, and the Elements will display as 3d Faces on layers created for you by the program. Surfaces and Solids will also be displayed.

Creating a STAAD Input File

Structural Desktop can create a STAAD.Pro input file from the model at any time. For this to be effective, you must have a Structural Desktop Model open in AutoCAD that was created by reading an analytical file, by Opening a Structural Desktop Document and changing Lines or Faces to Members or Elements, or by reading a HOLD file. This will not be effective if you have simply read a Drawing file (*.DWG) and not performed transformation options; AutoCAD drawing files do not contain joint coordinates, member incidences, element incidences, member properties, element thicknesses, and constant values. The Structural Desktop model contains this information when the proper sequence has been performed. If this is confusing, please read the Tutorial which will lead you through a session that should demonstrate this process in more detail.

If members are FUSED in your Structural Desktop Model, the fused members will be ignored, but the sub-members that they are composed of will be written to the STAAD file with the current Member Properties, Constants, and Beta Angles.

When you have created a model, either by reading a file or by **Open a Structural Desktop Document**, the file menu will activate the function to **Create STAAD Input File** as shown in **Figure 2 – 8**.

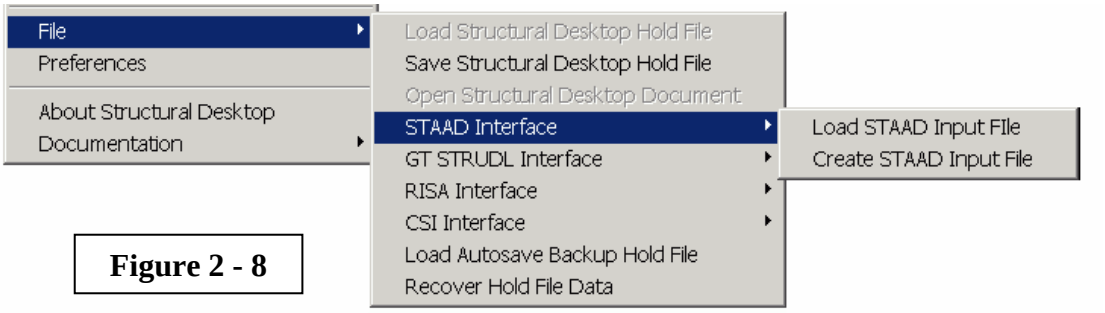


Figure 2 - 8

Clicking the **Create STAAD Input File** menu option will again invoke the separate Structural Desktop satellite program to read/write STAAD.Pro Files. The dialog will appear as shown in **Figure 2 - 9** at right. Note that this dialog has a label that displays "Creating a STAAD.Pro file" whereas the same dialog states "Reading a STAAD.Pro file" in **Figure 2 – 5.**

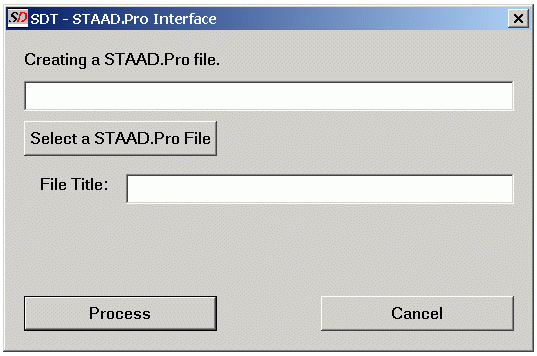


Figure 2 - 9

Clicking on the button marked [**Select a STAAD.Pro File**] will invoke a standard windows file dialog as shown in **Figure 2 – 10** at left and typing in "AnewFile" (shown) will result in the dialog in **Figure 2 –10** at right..

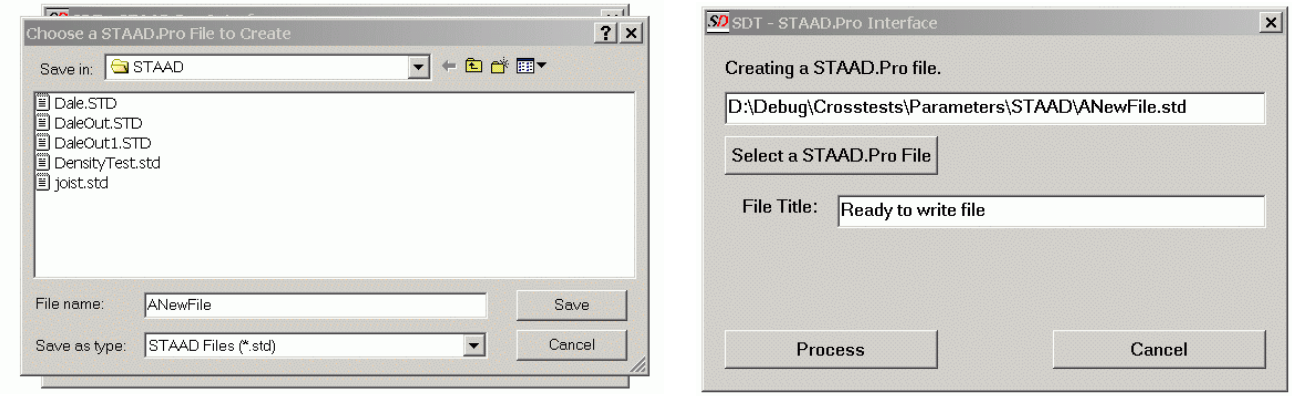


Figure 2 - 10

The dialog at right displays the path and filename selected. You may change this before creating your file if you wish. When you click the [**Process**] button, your file will be created and displayed in an editor.

Structural Desktop User Provided Tables

Structural Desktop provides support for a series of User Provided Tables that come with the program. The files are located in:

C:\Program Files\Structural Desktop\STAAD User Files

This is an example of a part of a file created using the Structural Desktop tables:

Examples of USER TABLES in a STAAD file

```
START USER TABLE
TABLE 1 L_T_DATA.DAT
TABLE 2 CEEZEE.DAT
TABLE 3 K_DATA.DAT
TABLE 4 LH_DATA.DAT
TABLE 5 JG_DATA.DAT
END
MEMBER PROPERTY
1 PRISMATIC YD 24 ZD 12
2 TA SD L60355 SP 0.375
3 UPT 1 R18LB24
4 UPT 1 L18LB24
5 UPT 1 T30IT24
6 TA T M14X18
7 UPT 2 C9X2512
8 UPT 2 CS123710
9 UPT 2 CU121214
10 UPT 2 Z8X314
11 UPT 2 ZS123210
12 UPT 2 ZU361220
13 UPT 2 HU10X514
14 UPT 3 K1216
15 UPT 4 LH281
16 UPT 5 JG326
UNIT FEET POUND
CONSTANTS
E STEEL MEMB 1 TO 2
DEN STEEL MEMB 1 TO 2
POI STEEL MEMB 1 TO 2
E CONCRETE MEMB 3 TO 5
DEN CONCRETE MEMB 3 TO 5
POI CONCRETE MEMB 3 TO 5
E STEEL MEMB 6 TO 16
DEN STEEL MEMB 6 TO 16
POI STEEL MEMB 6 TO 16
```

Please note: The user provided tables are designed to work with the American tables and are currently unavailable for use in metric files or with the tables from other countries.

At this time, Structural Desktop cannot support **USER PROVIDED TABLES that you create yourself.** The tables shown have internal descriptions within ***Structural Desktop*** that allow the Joists, Light-Gage, and Concrete members provided to be drawn and their weights (and volumes, where appropriate) to be included in Material Reports.

Members created in ***Structural Desktop*** that rely on the **USER PROVIDED TABLES** are defined in terms of those table names and the above STAAD file was created automatically. To use these members for design within STAAD.Pro, you must copy the relevant USER files from the directory on your hard disk (also available on the supplied installation disk) to the same directory the STAAD file is in, so that the STAAD.Pro program can find and use them.

Releases in a STAAD Input File

Structural Desktop Version 2.0 introduced a new feature regarding Member Releases. ***Structural Desktop*** will create a STAAD input file that includes releases if those releases are applied, either through the methods that follow below, or within your STAAD.Pro graphical user interface.

Structural Desktop loads linetypes from a file in the ***Structural Desktop*** program directory when you open a file. These linetypes are as shown in **Figure 2 - 11** taken from the Linetype Manger in **AutoCAD,** at right:

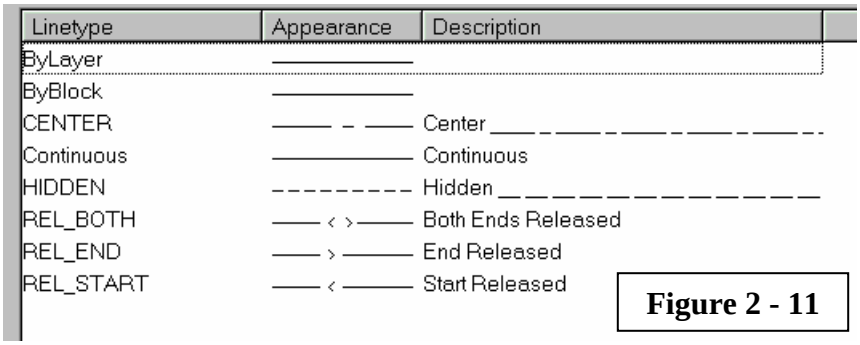


Figure 2 - 11

The releases that were considered to be the most common were MY MZ releases. This version of ***Structural Desktop*** will apply those releases to any member that has had the linetype of the centroid of the member changed to one of the last three linetypes above. The program will also show releases OF ANY TYPE that apply to a member read from a STAAD file with the same linetype display.

Selecting the members centroids to be released using the general selection method...such as is used with grips, or as is shown in the ***Structural Desktop Tutorial*** for changing the layer of a member...select all members that you wish to have releases. You can do this to one or many members. With the grips on the member lines visible, click on the linetype bar at the top of your AutoCAD session, and click on the linetype that represents the releases you wish to apply, either at the start, the end, or both ends of the member.

The member will then be displayed with the centroid changed to the linetype desired. The arrows within the line point at the released end of the member. Start and End for this use are defined as the direction in which the original line was drawn, or the sequence in which the nodes are given for the Member Incidences.

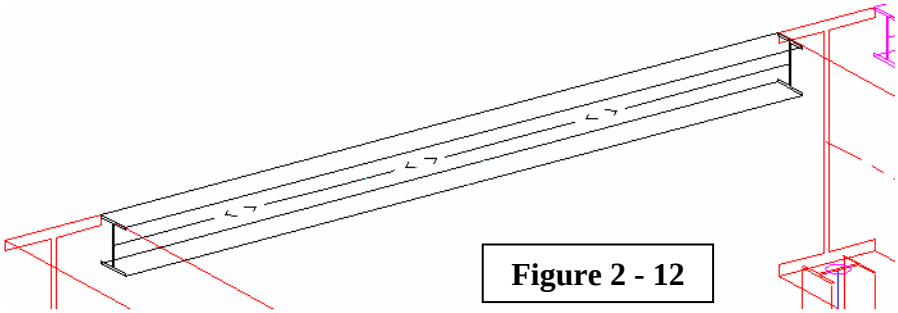


Figure 2 - 12

A member with releases at both ends will therefore be displayed as shown in **Figure 2 - 12** above. If the member is fused, the releases will be for the Start and the End of the appropriate "piece" of the member when the STAAD.Pro input file is written.

Load GT STRUDL Input File

If you do not use GT STRUDL from Georgia Tech then you may skip this section.

Loading a GT STRUDL file has changed in Version 4 of Structural Desktop. Previous versions read a series of data files created by GT STRUDL referred to as DBX files and wrote the Input files designated with the extension *.GTI. Now Structural Desktop interfaces with GT STRUDL through the GTI files, both for input and output. There are a large number of programs that create GTI files in several different formats that GT STRUDL itself can read. Structural Desktop has focussed on the file structure as it is generated by GT STRUDL itself when that file is created from within the graphical user interface. If you have problems reading a GT STRUDL input file then you should load the file into GT STRUDL, invoke the Graphic Modeler, and re-write the input file from that module. This will create a standard input file that Structural Desktop is calibrated to read.

When you first open AutoCAD and Structural Desktop is loaded, the file menu for GT STRUDL will appear as shown in **Figure 2 – 13**.

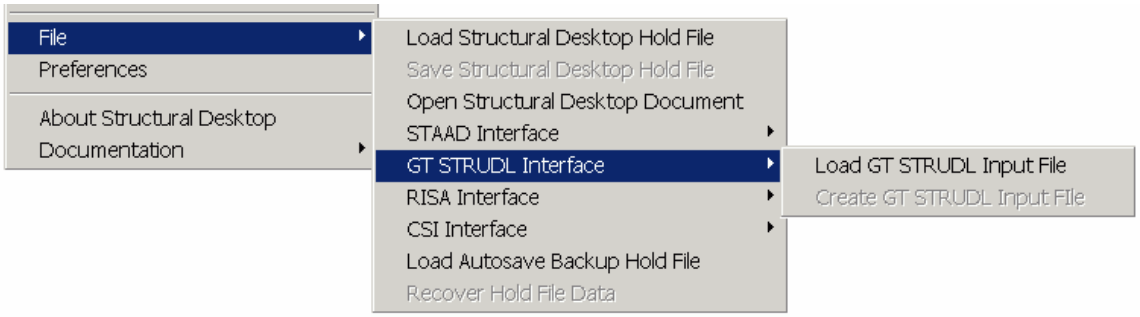


Figure 2 - 13

Invoking this command will display the dialog box at right as shown in **Figure 2 - 14**. Note that the dialog displays the notation “Reading a GT STRUDL file” near the top of the dialog box.

In order to load your file, click the button marked [**Select GT STRUDL File**].

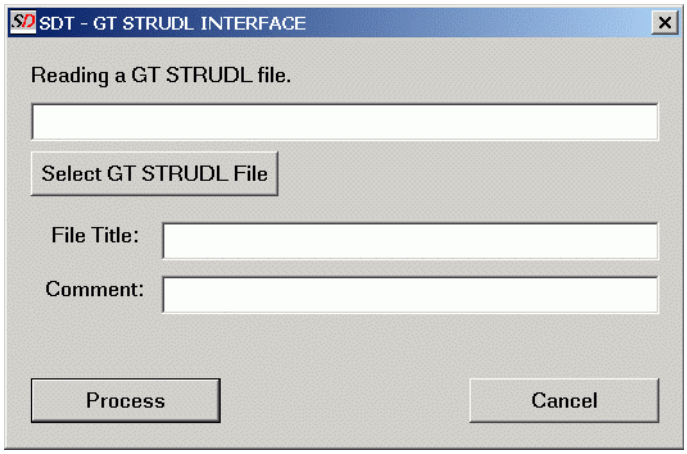


Figure 2 - 14

Selecting this option will invoke the dialog box as seen in **Figure 2 – 15,** to the left. Double-clicking on a file, or clicking a file and then [**Open**] will return to the dialog box in **Figure 2 – 15** to the right.

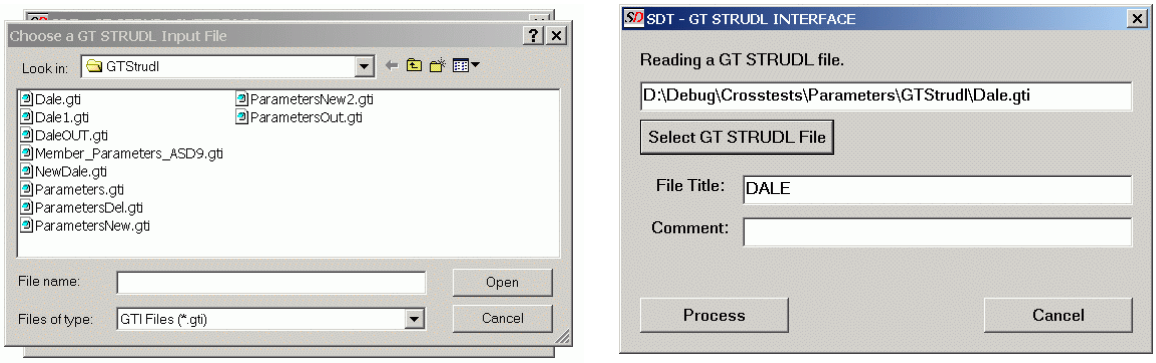


Figure 2 - 15

The program will display the Title and Comment from the first line of the GT STRUDL input file, if the file has them. You may return to the selection process as many times as you wish to change the displayed file, and when you have the file that you want, you may click the [**Process**] button. Structural Desktop will then read the GT STRUDL inputs and generate a model displaying nodes, lines for members, 3D Faces for elements, and it will store the Member Properties, Element Thicknesses, Material Properties, and Loads and Load Information for use in re-creating a GT STRUDL file from the model.

Creating a GT STRUDL Input File

Structural Desktop can create a GT STRUDL input file from the model at any time. For this to be effective, you must have a Structural Desktop Model open in AutoCAD that was created by reading an analytical file, by Opening a Structural Desktop Document and changing Lines or Faces to Members or Elements, or by reading a HOLD file. This will not be effective if you have simply read a Drawing file (*.DWG) and not performed transformation options; AutoCAD drawing files do not contain joint coordinates, member incidences, element incidences, member properties, element thicknesses, and constant values. The Structural Desktop model contains this information when the proper sequence has been performed. If this is confusing, please read the Tutorial which will lead you through a session that should demonstrate this process in more detail.

If members are FUSED in your Structural Desktop Model, the fused members will be ignored, but the sub-members that they are composed of will be written to the GT STRUDL input file with the current Member Properties, Constants, and Beta Angles. ***Structural Desktop*** also writes **Offsets** to a GT STRUDL file in order to preserve this information, and to adjust members to real-world relative positions for creation of CIS/2 files compatible with other software. GT STRUDL labels the information contained in ***Structural Desktop*** offsets as **ECCENTRICITIES**.

When you have created a model, either by reading a file or by **Open a Structural Desktop Document**, the file menu will activate the function to **Create GT STRUDL Input File** as shown in **Figure 2 – 16**.

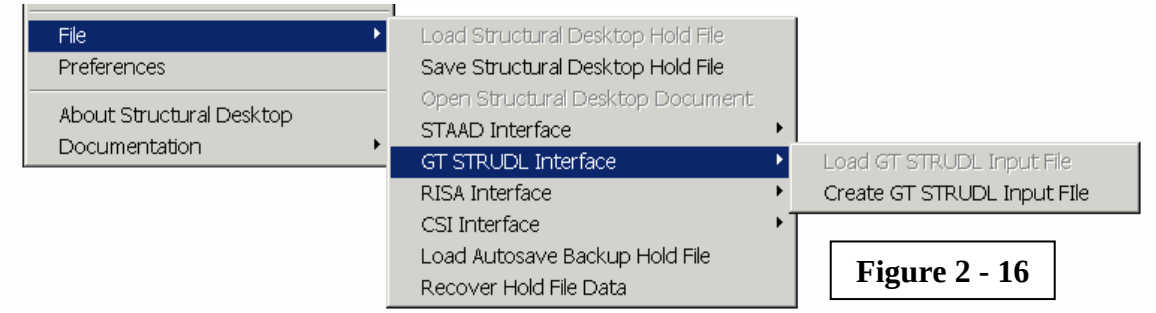


Figure 2 - 16

When you select this function, you will invoke the GT STRUDL read/write program that is included with your installation of ***Structural Desktop.*** The dialog will appear as shown in **Figure 2 - 17** at right. Note that this dialog has a label that displays "Creating a GT STRUDL file" whereas the same dialog states "Reading a GT STRUDL file" in **Figure 2 – 14.**

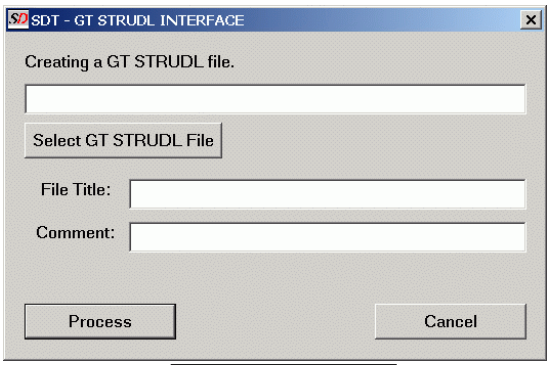


Figure 2 - 17

Clicking on the button marked [**Select GT STRUDL File**] will invoke a standard windows file dialog as shown in **Figure 2 – 18** at left and typing in "AnotherNewFile" (shown) will result in the dialog in **Figure 2 –18** at right..

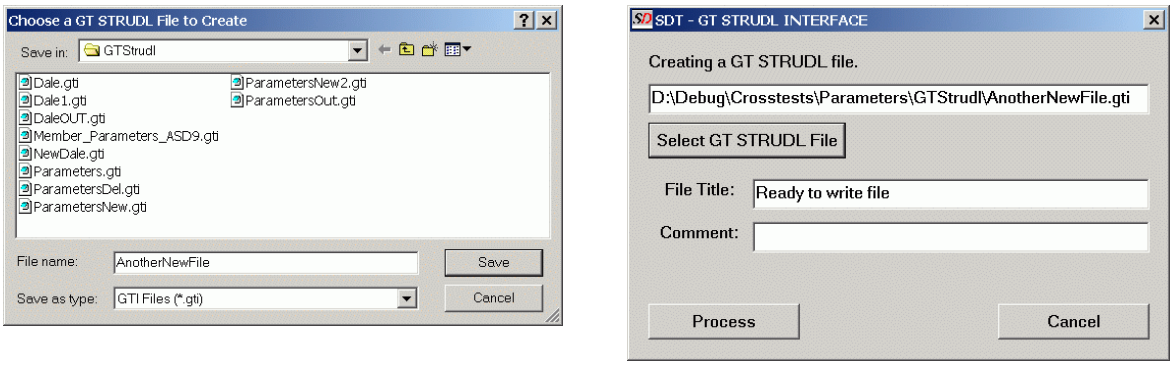


Figure 2 - 18

Structural Desktop User Provided Tables

Members created in ***Structural Desktop*** that rely on the **USER PROVIDED TABLES** are defined in terms of those table names. A section of a GT STRUDL file created using these members is provided below. In order for these members to be used within GT STRUDL, you must copy the relevant USER file from the directory on your hard disk or on the supplied installation disk to the temp directory which you indicate to GT STRUDL when you load the input file.

The table provided is designed to work with American tables and is in Imperial units. At that time, you must specify the data file StructDT.ds which is provided on your hard drive in the directory

C:\Program Files\Structural Desktop\GT STRUDL User Files

The section of a GTI below shows how Structural Desktop calls for rectangular prismatic members and a wide flange section. It also demonstrates a joist girder, a light-gage C-section and a concrete Inverted T called from our provided table, which permits design with these members.

```
MEMBER DIMENSION
  1   RECT B 24.000000 H 12.000000
$$
MEMBER PROPERTIES    TABLE 'WBEAM9 '  'W36X135 '
  2
$$
MEMBER PROPERTIES    TABLE 'STRUCTDT'  'JG326  '
  3
$$
MEMBER PROPERTIES    TABLE 'STRUCTDT'  'C9X2512  '
  4
$$
MEMBER PROPERTIES    TABLE 'STRUCTDT'  'T30IT24  '
  5
$$
UNITS  INCHES  KIPS  DEG   FAH
$$
CONSTANTS
E    3.6000002E+03  ALL
G    1.4400001E+03  ALL
POI  1.7000000E-01  ALL
DEN  8.6800006E-05  ALL
CTE  5.5000000E-06  ALL
E    2.9000002E+04    1    5
G    1.1000001E+04    1    5
POI  3.0000001E-01    1    5
DEN  2.8330003E-04    1    5
CTE  6.4999999E-06     1    5
```

Load a RISA 3D Input File

If you do not use RISA 3D or RISAFloor from RISA Technologies, Inc, you may skip this section.

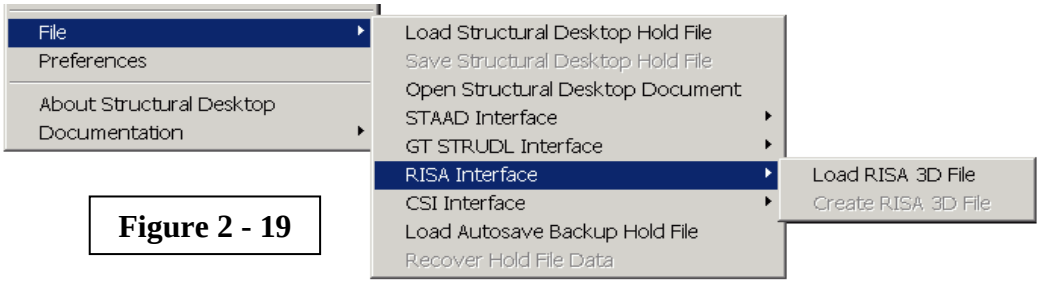


Figure 2 - 19

Structural Desktop uses the input file from RISA 3D from RISA Technologies, Inc. as a starting point for model creation. This file ends in the three-letter extension **R3D.** When you select this option the program will open the dialog box shown in **Figure 2 - 20** below.

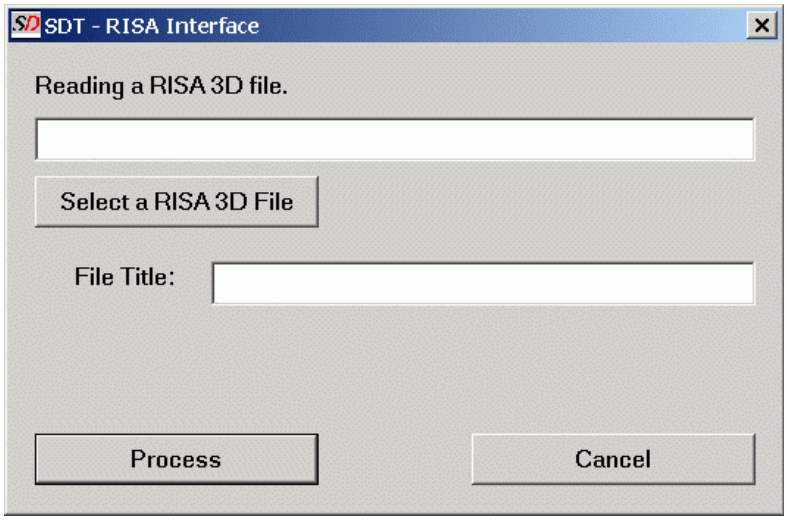


Figure 2 - 20

This is a new program exclusive to ***Structural Desktop*** Version 4 that is invoked by the ***Structural Desktop*** module from within AutoCAD. This program will either read or create a RISA 3D file, translating through a proprietary neutral format to the SDT database within AutoCAD.

Loading a RISA 3D file is a two step process; you click on the button to [**Select a RISA 3D File**] which uses a standard file-selection box primed to view only files ending in the **STD** extension as shown in **Figure 2 - 21** below.

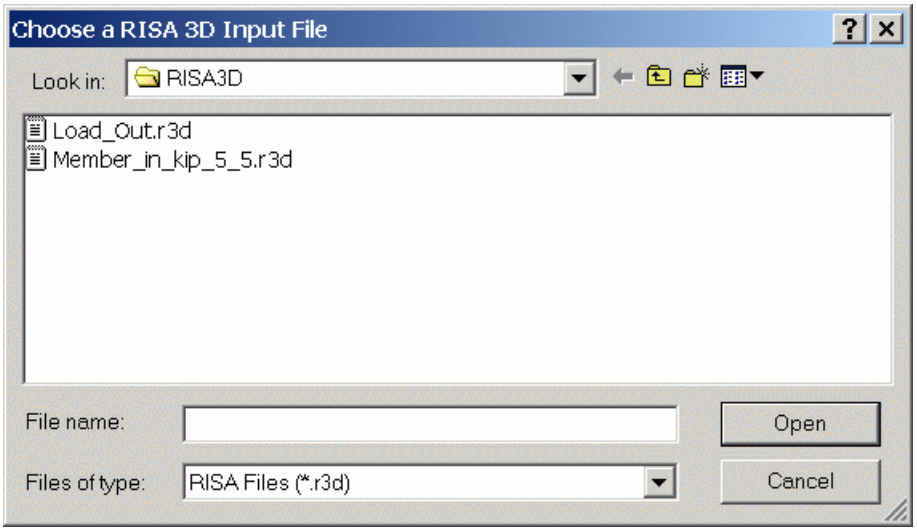


Figure 2 - 21

You may use the buttons at the top of the dialog box to search for your file in other folders. When you double-click the desired file, or select the file and click **Open**, the program returns to the first dialog box:

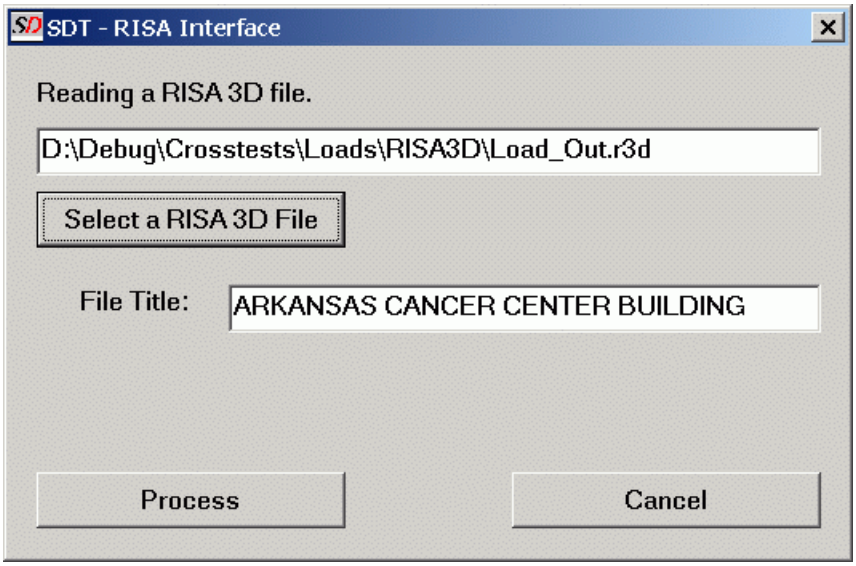


Figure 2 - 22

The RISA 3D file reader will display the title information from the MODEL TITLE data block of the RISA 3D file as a File Title and displays the complete path to the file as shown in **Figure 2 - 22**. If this is the file you wish to open and use, click on the [**Process**] button. If not, you can again [**Select a RISA 3D File**] as many times as you wish, or [**Cancel**] at any time.

When you click the [**Process**] button, the program will load the title of the file for future reference (it will be incorporated into your **Material Report**, for example) and will proceed to read the Joint Numbers and Coordinates, Member Incidences, and Element Incidences. The program will read the Member Properties, Element Thicknesses, and constants such as Beta and Density.

With the introduction of Version 4, Structural Desktop will continue to read and record your RISA 3D file with a much greater degree of completeness than in previous versions. It will read and retain Diaphragms and Solids as additional types of fundamental objects. Structural Desktop will then read your Load Statements, the Joint, Member, and Element loads that follow them, and your Load Combinations, and additional Materials. When you choose to create a RISA 3D file from within Structural Desktop, the program will completely re-create and write all of the Load, Load Combination, and other lines as it read them from the original RISA 3D file. We refer to this attempt to capture all possible analytical data and return it to you as the **ROUNDTURN**.

Please note, Structural Desktop works within AutoCAD with an attempt to carry enough accuracy and precision to be sufficient for all Structural needs. All units within your RISA 3D file will be converted into Inches for Length and Kips for force, and the appropriate combinations thereof for torque and other combined forces.

Once you have read the file into Structural Desktop, your Joints will display as small circles, the Members will display as Lines, and the Elements will display as 3d Faces on layers created for you by the program. Diaphrams, Solids, and indicators of Supports and members that have been Released will also be displayed.

RISA 3D File Creation

Structural Desktop can create a RISA 3D data file from the model at any time. For this to be effective, you must have a Structural Desktop Model open in AutoCAD that was created by reading an analytical file, by Opening a Structural Desktop Document and changing Lines or Faces to Members or Elements, or by reading a HOLD file. This will not be effective if you have simply read a Drawing file (*.DWG) and not performed transformation options; AutoCAD drawing files do not contain joint coordinates, member incidences, element incidences, member properties, element thicknesses, and constant values. The Structural Desktop model contains this information when the proper sequence has been performed. If this is confusing, please read the Tutorial which will lead you through a session that should demonstrate this process in more detail.

If members are FUSED in your Structural Desktop Model, the fused members will be ignored, but the sub-members that they are composed of will be written to the RISA 3D data file with the current Member Properties, Constants, and Beta Angles.

Once you have created members or elements in a ***Structural Desktop*** session, the option to **Create RISA 3D file** (*.r3d) will be available as shown in the menu section illustrated in **Figure 2 - 23** below.

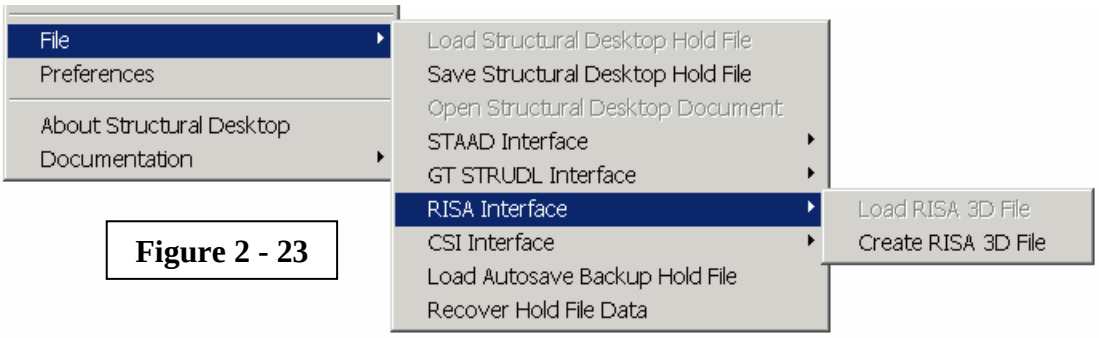


Figure 2 - 23

When you select this function, you will invoke the RISA 3D read/write program that is included with your installation of ***Structural Desktop.*** The dialog will appear as shown in **Figure 2 - 24** at right. Note that this dialog has a label that displays "Creating a RISA 3D file" whereas the same dialog states "Reading a RISA 3D file" in **Figure 2 – 20.**

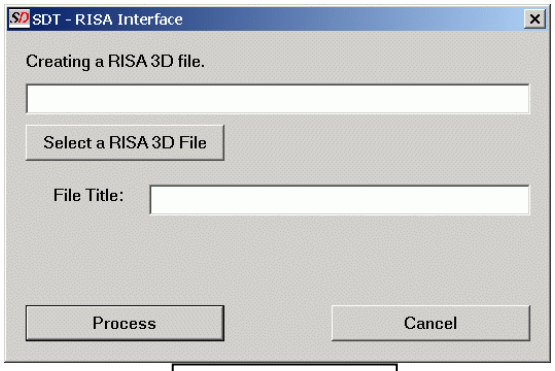


Figure 2 - 24

Clicking on the button marked [**Select a RISA 3D File**] will invoke a standard windows file dialog as shown in **Figure 2 – 25** at left and typing in "ANewFile" (shown) will result in the dialog in **Figure 2 –25** at right..

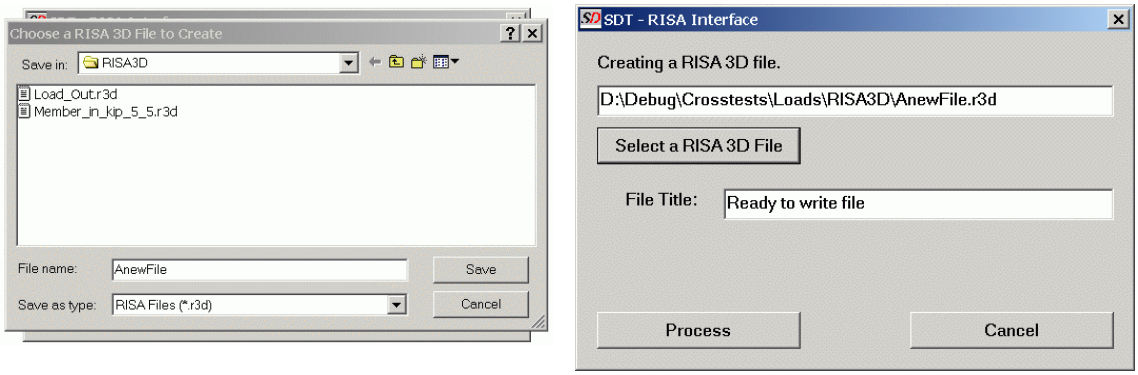


Figure 2 - 25

Structural Desktop User Provided Tables

Structural Desktop produces members derived from standard tables in the format and with the parametric values that would be produced if you created the same member from the RISA 3D tables within the RISA 3D program. Custom members can be produced within RISA 3D and assigned to members created with a temporary "standard" definition within ***Structural Desktop***, but ***Structural Desktop*** does not support custom member shapes and definitions. Please make certain that what you have within RISA 3D for analysis purposes as a member definition is the member that you intended.

Structural Desktop will produce LongSpan Joists, Joist Girders, K Joists, certain C's and Z's, and Concrete L's and inverted T's for use with **RISA 3D**. This use requires **RISA 3D 4.5b version** or later and the SDT table file (**SDT.fil**) must be copied from the ***Structural Desktop*** directory to the RISA directory for **RISA 3D** to recognize the members. Please check the ***Structural Desktop*** documentation section on reading **RISA 3D** files for more information, or the help file in **RISA 3D version 4.5b**.

Structural Desktop has a single definition for wood members, defined as the Wood material type applied to Prismatic rectangular and cylindrical members. This material has been mapped in the RISA format to reflect Southern Pine Select. The user may change the wood type for any and all groups of members as soon as the file has been read into the RISA 3D program. (A future version of ***Structural Desktop*** is planned to permit variations in defining wood members to match the diversity of the RISA 3D program.)

Structural Desktop will not save loading and design information from the RISA 3D file format, nor will RISA 3D retain layer and other AutoCAD related information from a ***Structural Desktop*** Model. Providing this full round-turn capability is a high priority with the ***Structural Desktop*** design team, but in the interim care must be taken to update changes in both models in order to maintain current information on your structure.

Load CSI SAP 2000 Input File

If you do not use SAP 2000 from Computers & Structures, Inc, you may skip this section.

Structural Desktop uses an input file for SAP 2000 from Computers & Structures, Inc. as a starting point for model creation. The file format used ends in the three-letter extension **S2K.** These files are generated from SAP 2000 through the "Export" function on the SAP 2000 menu. Please note that files manually generated through this format for SAP 2000 (such as some of the examples that come with SAP 2000) or generated by other programs may use a "shorthand" version of the input format, including "macro-like" statements that expand to represent multiple nodes or elements. ***Structural Desktop*** will read these files if they are first **IMPORTED** into SAP 2000, and then **EXPORTED** again. When SAP 2000 exports an **S2K** file, all compound statements are expanded into a basic form that ***Structural Desktop*** is designed to read.

When you select the **Load SAP 2000 File** option the program will assume that you have a file with the **S2K** extension on a local or network drive that you can access through an **Open File** dialog box. When you click on the menu item **Load SAP 2000 File**, the dialog box in **Figure 2 - 14** below will appear.

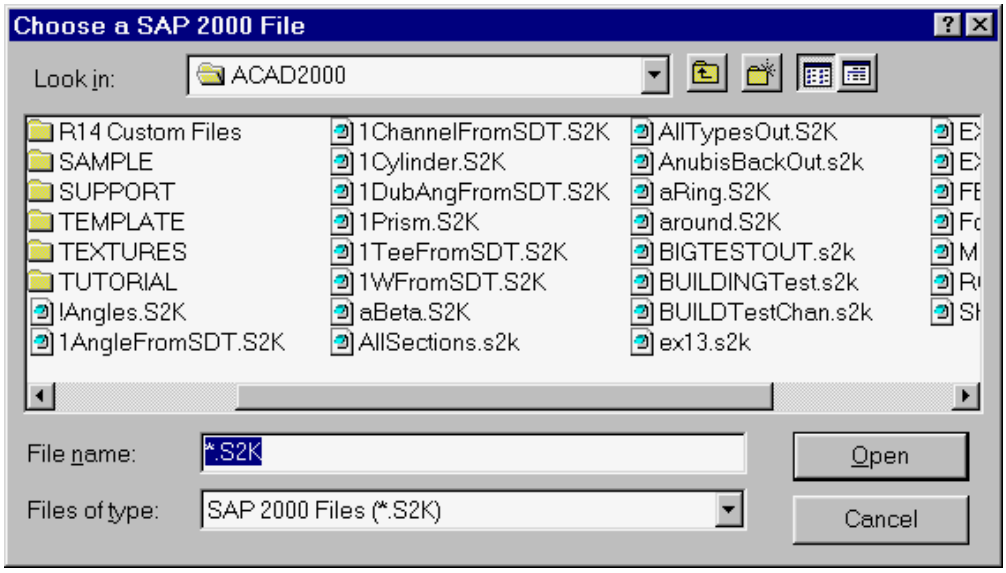


Figure 2 - 14

This is a standard file selection dialog box. It is primed to show files that have the **S2K** extension and will allow you to type the name of your file or click on the file. You may also use the buttons at the top of the dialog box to search for your file in other folders. Both SAP 2000 Version 8 and SAP 2000 Version 7 files can be loaded.

When you select the file and click **Open**, the following steps will occur:

The program will access and open the SAP 2000 S2K file, if another program does not have the file locked and inaccessible. The program will distinguish between the earlier format S2K files created by SAP 2000 Version 7 and the more recent format used by SAP 2000 Version 8 by the location and structure of the Units as recorded in the file. ***Structural Desktop*** will then read the remainder of the file as described below. Currently, units of either feet or inches will be read; all measurements of coordinates or prismatic sections dimensions will be converted into inches as necessary.

The program will read the title of the file for future reference (it will be incorporated into your **Material Report**, for example) and will proceed to read the Joint Numbers and Coordinates, Member information, and Plate Information. The program expects a letter in front of each value, but you should renumber your file so that no two joints, members, or elements have the SAME number. The program will read the Section Properties, Material Properties, Beta angles (if given as an angle, not to a point) and the thicknesses of the Plates. The Joints will display as small circles, the Members will display as Lines, and the Elements will display as 3d Faces on layers created for you by the program.

Structural Desktop will read CSI Members drawn from the AISC tables provided with the CSI program and these members will automatically be assigned the "STEEL" material and density. ***Structural Desktop*** requires that the NAME of the section as defined in SAP 2000 is unchanged from the name given from the table. (***Structural Desktop*** will not assign the proper Member Type values to a W14x99 that has been given the name "Column1", for example, but will properly process the geometry of the member. For all the relevant properties to apply, permit the SAP 2000 program to assign the default name of "W14X99" from the table.

Prismatic sections can be created with any name (e.g. FSEC1, BEAM1) and will be converted into prismatic sections in ***Structural Desktop***. For your ease in identifying the member, the Description field will display the Name as you have defined it in SAP 2000.

Material types labeled "CONC" or "CONCRETE" and "STL" or "STEEL" will be applied to prismatic sections from a SAP file. Other Material definitions can be corrected within ***Structural Desktop*** using the Change Member Parameters or Change Element Parameters dialog boxes to set Aluminum, Lightweight Concrete, or Wood or to set incompatibly labeled Steel or Concrete sections.

Create CSI SAP 2000 Input File

Structural Desktop will create a Version 7 SAP 2000 Input File from the menu item **Create SAP 2000 File**. This file can be read into Version 8 by using the translator provided by CSI, and will support Prismatic Sections and AISC members.

Working With *Structural Desktop* from Scratch

Structural Desktop is also designed to work without an analysis file, and this is the primary method for working with ***Structural Desktop* LT**. When there is no data file from an analysis program available, ***Structural Desktop*** will allow a user to create a model in AutoCAD. This model can be used to create an input file for analytical/design software using ***Structural Desktop***, or you can proceed directly to the output drawings with either ***Structural Desktop*** or ***Structural Desktop* LT**.

Structural Desktop creates input files for analytical programs that describe the geometry and basic properties of members and elements. ***Structural Desktop*** creates the joint numbers and coordinates, member incidences, and element incidences.

Structural Desktop will create the geometry for an analytical model without a pre-existing analytical model. You may then export the input file for analysis and design and return the result to ***Structural Desktop***. Alternately, you can create a model and create your drawings completely within ***Structural Desktop***.

On the ***Structural Desktop*** menu, under **File**, the last menu selection is **Open Structural Desktop Document**. (See **Figure 2 - 3**) This selection will set all the necessary layers and create a ***Structural Desktop Model*** environment. This environment enables you to create a structural model with access to the full library of sections.

Any method that will place LINES and 3D FACES within AutoCAD is valid to create a ***Structural Desktop Model.*** DXF files from any version of AutoCAD or AutoCAD LT (or any other CAD software or analytical design program that produces DXF files), or any drawing file that is compatible with your version of AutoCAD can be inserted as a block and exploded. Lines and faces to be changed to ***Structural Desktop*** objects will retain their layer information and color. However, do NOT try to use lines that are only inserted through XREF or that are still part of a BLOCK. Simple lines and simple 3D faces are the rule.

You can use existing drawings to create lines and insert them into this model, or you can draw lines, copy lines, array lines, or mirror lines to create a structure. Any AutoCAD tool such as trim, extend or offset can be applied to these lines BEFORE they are changed to MEMBERS to get them precisely where you want them. Since lines do not become members until changed, you may use any construction lines or reference lines you wish. Lines that are not selected at the time you Change Lines to Members will be erased by ***Structural Desktop*** as part of the database management cycle.

See the section on **Changing Lines to Members** for more information. In like manner, you can create or import 3d Face objects that can be changed to elements and that is covered in the section on **Changing Faces to Elements**.

The Structure Line Generator

Structural Desktop Version 1.9 included a structure generator. This function permits the quick generation of simple, repetitive bay structures in AutoCAD, but can be used several times in succession to create more complex building outlines.

The function is accessed from the right-click menu. The Structure Line Generator command is shown highlighted at the right in **Figure 2 - 15.**

This function is designed to be used in any AutoCAD drawing. The line generator may be used in any AutoCAD session or in a ***Structural Desktop*** Model drawing and it may be used to add structural members to an existing model as many times as you wish.

When you select this function, the dialog box shown in **Figure 2 - 16** below will appear.

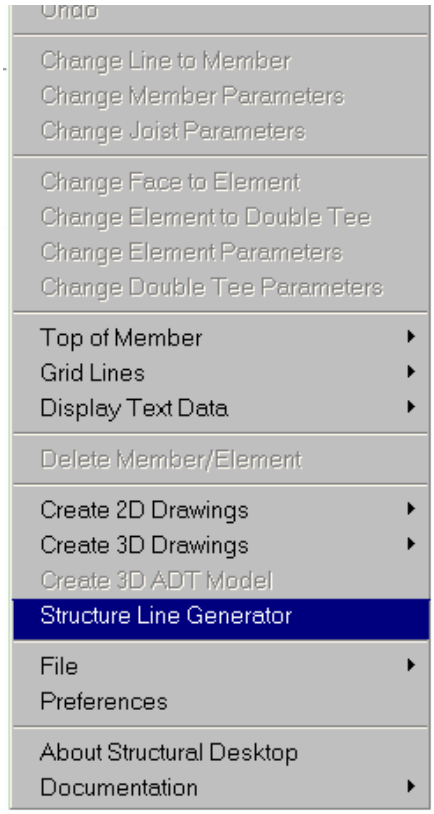


Figure 2 - 15

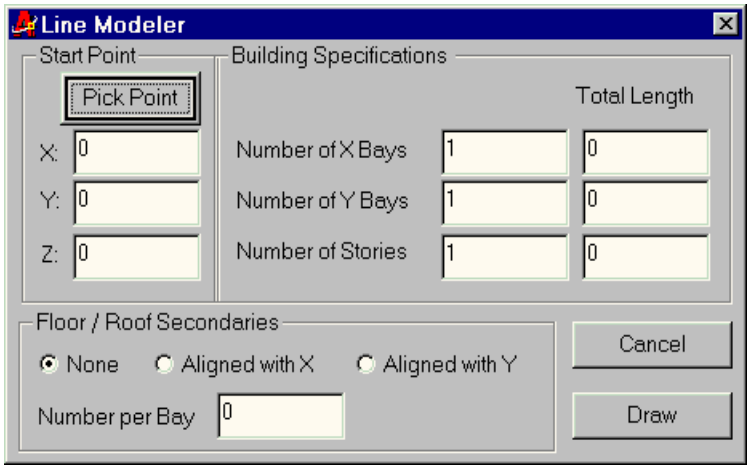


Figure 2 - 16

The "Start Point" button does not have to be used unless you wish to begin at a point you select from an existing drawing. You may also enter values for coordinates in each of the X, Y, and Z boxes. All measurements and coordinates for this dialog box are in inches and decimal fractions or centimeters depending upon your AutoCAD default measurements.

Some non-zero value, **in inches or centimeters**, must be entered into each of the "Total Length" edit boxes at the right of the dialog box for the function to create a line model. You may select the "Number of X Bays", "Number of Y Bays", and "Number of Stories" by entering these values into the boxes to the left of the Length values. Each bay created will be the total length divided by the number of bays in the direction of that axis.

No "secondaries" will be created in the floors or roof structures unless you select a "Number per Bay" at the bottom of the dialog box AND chose a direction for alignment of the secondary members. An example of a dialog box properly filled out is shown in **Figure 2 - 17**, at the right, and the results of this dialog box are shown in **Figure 2 - 18** below.

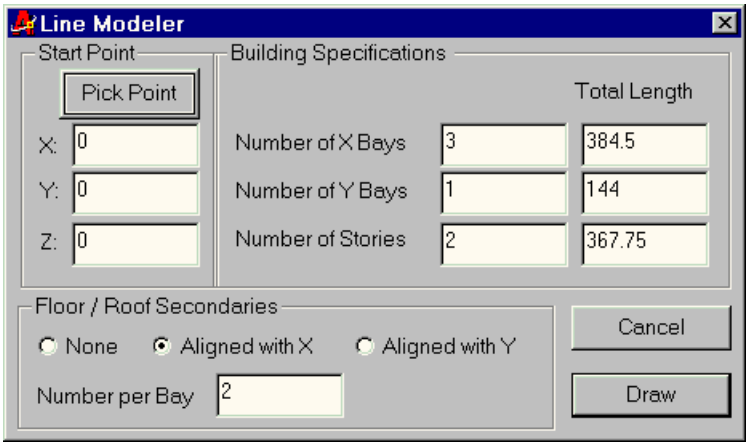


Figure 2 - 17

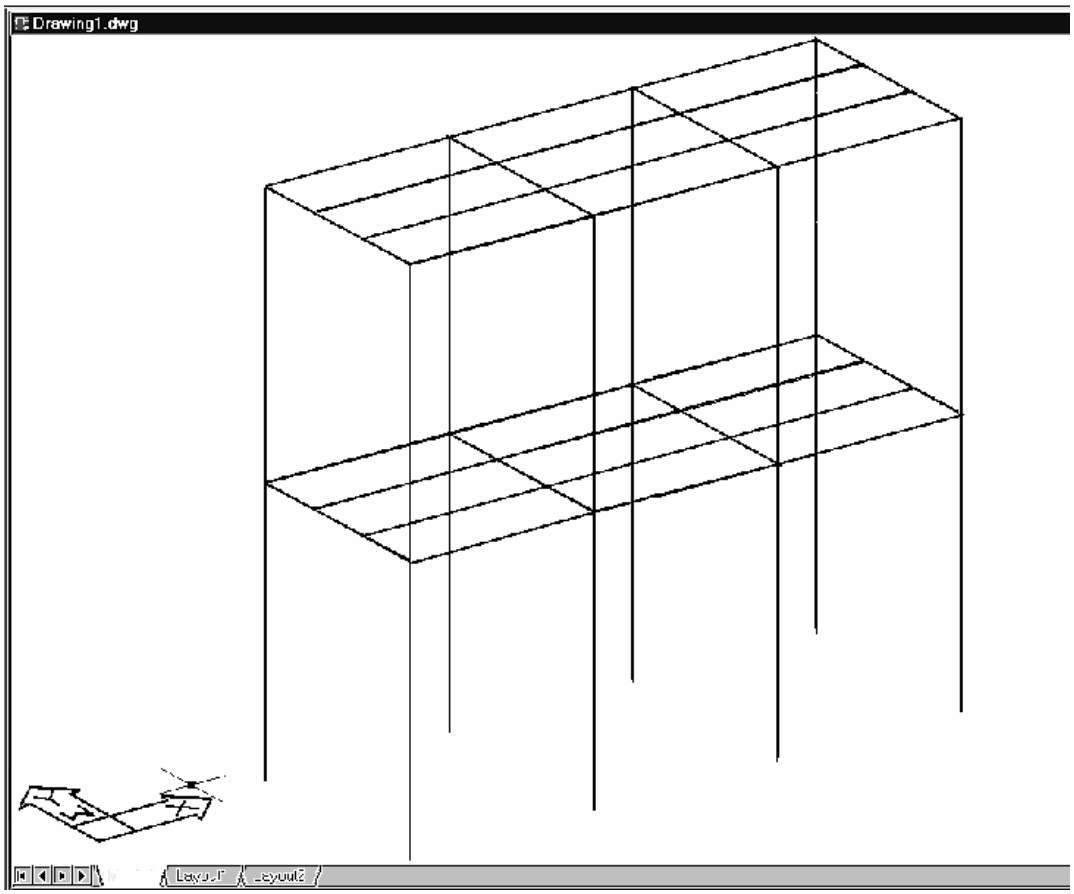


Figure 2 - 18

In this example, the secondary members are drawn horizontal, aligned with or parallel to the X axis as was checked in the dialog box. As further shown in the example, an entry of 2 in the **Number Per Bay** edit box for secondaries will actually place two secondaries and divide a bay into 3 equal divisions.

As soon as you have created the structure, you can add additional bays with different characteristics by running the function again and again, and setting the start point to the appropriate part of the existing model. You can rotate, copy, stretch, erase and modify the basic structures produced to fit your needs. If you require secondary members at the lowest elevation level of the model, you can copy them from the floor above; any members or groups of members you do not need can be simply deleted while they are in AutoCAD line form and have no consequences on your analytical model or numbering schemes.

To give you the greatest amount of freedom in placing members, the secondaries created do not divide the primary members they intersect. Using the ***Structural Desktop*** command **Intersect Break** (once the lines have been turned into members) sets the intersection points between the secondaries and primaries in analytical files created from the model. This permits you to move, change or erase a single secondary member or groups of secondaries and have the joints/nodes precisely at the intersection point for the generation of an accurate analytical model.

Because the lines generated are not ***Structural Desktop*** Members until you use the **Change Line to Member** command, you can experiment freely with these lines without actually causing any change to a ***Structural Desktop*** internal database. You can also add 3d Face objects to this model, that will become ***Structural Desktop*** elements when you use either the **Change Line to Member** or **Change Face to Element** command**.**

When you are satisfied with the combination of successive uses of this generator and basic AutoCAD functions and, you can use the ***Structural Desktop*** commands **Open Structural Desktop Document** and **Change Line to Member** to change your model of simple lines into ***Structural Desktop*** members.

Saving Your Work - The *Structural Desktop* Hold File

Once you have loaded a file to create a document or used the **Open Structural Desktop Document** menu option, you have the responsibility to save your work in a ***Structural Desktop*** **Hold File**. This file format is the ONLY way to resume work on a ***Structural Desktop*** Model and is your best protection against loss of your work from any cause. To resume work on a saved model, use the **Load Structural Desktop Hold File** menu option. When you save and load an AutoCAD drawing file, the "background" information permitting manipulation of the ***Structural Desktop*** objects as database objects is not saved in that drawing file. This is done so that the final output of ***Structural Desktop***, your drawings, should be 100% compatible with every drafting package that can utilize a 2D AutoCAD drawing file or DXF file.

Structural Desktop has a function that performs an automatic backup of your model every five minutes. This backup is saved on your hard drive in the directory C:\Program Files\Structural Desktop\Backup. If you wish to load this file, you may do so by clicking on **Load Autosave Backup Hold File** as the first action you take after running the program. Once you have loaded another file, waiting for any length of time will cause ***Structural Desktop*** to overwrite the existing backup with a backup of the open document. We can not assure you that the Autosave function will always do exactly what you want it to do, but it is the second best method of recovering data in the event of a loss or error. The best method is to save your work, frequently, and under a number of names that represent incremental progress of your model, and make backups of these files periodically to other media.

CHAPTER 3 - Display, Viewing, and Selection

The objective of this chapter is to help you understand the following concepts:

- How to use the Display Option dialog box.
- The function of each option in the dialog box.
- Select members and elements using selection sets.
- Use Viewing options to view parts of a model.
- Use Layers to selectively view and group members and elements.
- Display input data and all member/element modifications.

The Display Option Dialog Box

Once a data file or hold file has been loaded or a new file created, the program is fully initialized and the shortcut menu should appear as shown in **Figure 3-1** below. The display and viewing options provide functions necessary to construct a ***Structural Desktop*** model viewing the object, rather than just lines or faces. The General Display Options menu item is the top of the command list.

Figure 3-1

Once you have selected this menu item, the Display Options dialog box for Geometric Data will appear in the graphics screen area as shown in **Figure 3 - 2** below. Each of these groups of options will be explained.

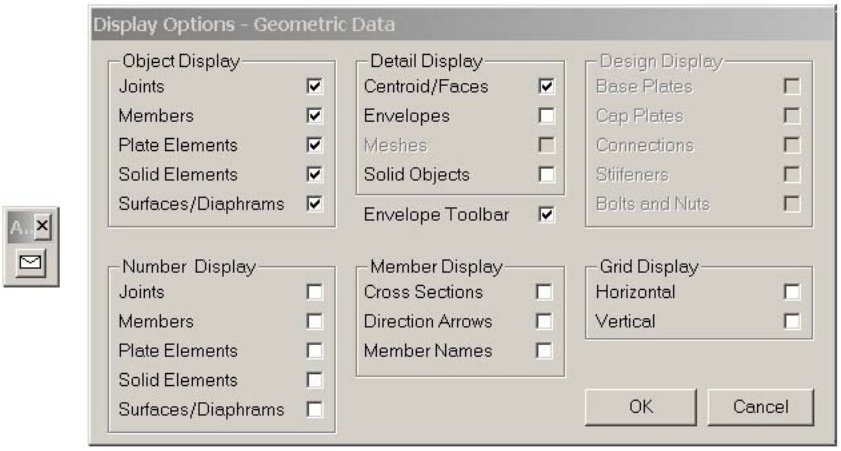


Figure 3-2

Object Display

The Object Display section of the Display Options dialog box determines whether or not a specific type of entity will be displayed, and is complementary to the layering system employed by AutoCAD. Unlike the layering options in AutoCAD, when an object is turned off in this box, it simply does not exist in AutoCAD at all. For the user, this means that a large, complicated model will update, regenerate, and process much more quickly if the display is made as simple as possible through this option. If you do not need to see elements or joints, turn them off here for maximum performance. Also, if objects do not display when you have turned on all the layers that they should be on, check if members or elements have been turned off.

Joints will display as small circles on their own layer, [!SD_Joints]. Members will display as lines (with the CENTER linetype, if you have loaded that linetype into your model) on the layer [!SD_Centroids]. Elements will display as 3d Faces on the layer [!SD_Faces]. If you change the layer of a member or element, it will remain on that layer and display accordingly. If a layer is turned off or frozen, elements or members on that layer will not display because AutoCAD has turned them off, but they will still exist and be updated each time you perform an operation that changes this display.

Between layering options, Object Display options in Structural Desktop, and the Viewing Options (discussed below) you have three very powerful tools to control what is on the screen at any given time. Likewise, you should always remember that there are at least three different places that you may have to check to display an absent member!

Member Display

The **Member Display** options are designed to give you graphic access to information about members.

The **Cross Sections** option displays a cross section of the member's shape on the Centroid line representing the member and is oriented to display the beta angle of the object. This is the fastest and least complicated method to determine what type of object you have and the orientation. You could use this option to determine the orientation of a channel, for example.

The **Direction Arrows** option displays an arrow on the Centroid Line indicating the direction the member was created from the J (START) to K (END) of the member. This orientation information is very important to know when you are modifying one or the other end of the member. All of the member commands that refer to one or the other ends of the member will do so using the START/END or J/K nomenclature.

The **Member Names** option will permit you to see the DESCRIPTION associated with the member. It identifies the member type (W18x35, TS8x8x1/2) when the member is first read from an input file. You have the option of changing this "name" to ANY text string you want, which will be covered in the Change Member Parameters section of Chapter 4 on Modifying Members.

Show Toolbar

The **Show Toolbar** option will create the toolbar shown to the left of the dialog box in **Figure 3 - 2** that has an icon of an "envelope" on it. This Toolbar can be placed anywhere on your screen and will toggle your envelopes on and off without having to return to the display dialog box. This button/toolbar is the first button/toolbar to be created for the program, and was chosen by our beta testers as the single most used function. Turning the envelopes on and off can assist greatly in visualizing complex models, so this button is designed to permit this action with a single mouse-click.

Detail Display

The **Detail Display** options are designed to let you control the level of detail about a member or element that shows on the graphic screen at any given time. Higher levels of detail are not always desirable. If, for example, you wish to enter a totally new member, it is easier to see where you are putting it if the Centroid lines are all that are currently being displayed. The more complicated the display, the more power your computer will have to have to handle it at a comfortable speed.

The **Centroid/Face** option is there to remind you that the Centroid line is the heart of a member and a 3d Face is the basis of an element's representation. Turning this option on and off will not actually affect the display!

The **Envelopes** option turns on and off a representation of the "Outline" of a member or element. Envelopes were created to increase the speed of the computer processing while working in ***Structural Desktop***. We recommend using envelopes as your first choice.

The envelopes, technically, are lines and polylines that are **GROUPED** with the actual representation of the Centroid or Face objects. What this means to the user is that a selection of any line in an envelope selects the entire object. This also requires that the setting for **Object Grouping** be checked. This setting in AutoCAD can be found by selecting **Options** from the AutoCAD menu, clicking on the **Selection** tab, and looking at the left side of the Options tabbed dialog box. This option can also be accessed through the AutoCAD System Variable **PICKSTYLE**, which should be set to "1".

The reliance on grouping can result in selections that are not quite what you expect if you are using an AutoCAD **WINDOW** to perform a selection. A **WINDOW** selection selects only objects that lie entirely within the selection window. In ***Structural Desktop***, when a window encloses any part of a group representing a member, it will select that member.

A little experimentation makes the meaning of this clear to users who are advanced enough in AutoCAD use to need this information. A remedy for this is to perform **WINDOW** selections using the centroid/face viewing option, and reserve envelope viewing for those occasions that you need the extra information or orientation that envelopes provide. Envelopes are quick and useful, but Centroid/Faces are even quicker to update when you are doing a large number of changes.

The **Meshes** option is not active at this time, and is grayed out. This option will permit a display using AutoCAD meshes to represent the outlines of objects, and is aimed at users with high-end rendering requirements.

The **Solid Objects** option turns on and off the generation of AutoCAD 3d solid objects. This option will display the best view that ***Structural Desktop*** can create with that object. While displaying solids, you can shade and hide to view the respective locations of members and elements; something that you cannot do as well with envelopes. In the event of a complex object such as a joist girder, all of the aspects of that object will display and can be viewed with an eye to how the final 2d drawing of that object will appear.

This option requires the most computer power, and many ***Structural Desktop*** functions will automatically turn the 3d **Solid Objects** mode into **Envelope** mode for you, before you are allowed to select members. Selecting any part of an Envelope selects the member object and connects to all the data for that object, but you must select the Centroid of the member if you are in **Solid Objects** mode to accomplish the same thing.

Selecting just a solid for a function, without selecting the centroid of the object, will cause an AutoCAD non-fatal error.

We do **not** recommend that you completely **avoid** using the Solid mode, but use it sparingly to find the information you want, such as interference or location of an edge. You can then return to the **Envelope** or **Centroid/Face** settings.

Once you have completed your model, you can create a similar 3d Model for manipulation in regular AutoCAD. The AutoCAD 3d Model the program is designed to create, unlike the ***Structural Desktop*** Model, has no special database or other baggage to carry. The **exported** 3d model is purely AutoCAD and purely a drawing file for you to use as you see fit. More information about creating a 3d Model for use outside of ***Structural Desktop*** (in AutoCAD or compatible programs) can be found in Chapter 9.

Numbers Display

The five check boxes under the Numbers Display section of the Display Options dialog box are for Joints, Members, Plate Elements, Solid Elements, and Surfaces/Diaphrams. Each toggles on or off a text display of the numbers associated with the objects in question. The numbers are those from the original input file. When members are **Fused**, a new member is created and, for consistency, is given a new number higher than any other used in the model. The same holds true for any new member or element you create.

The Joint Numbers Option displays numbers identifying each joint. The Member Numbers Option displays numbers identifying each member. The Element Numbers Option displays numbers identifying each element.

Grid Display

When you create grid lines in ***Structural Desktop*** they display on a layer of their own, [!SD_Gridlines]. For some views that you may wish to take of your model, a simple horizontal gridline is sufficient. When you create gridlines, the **Horizontal Grid** option is checked for you automatically. This displays the gridlines running horizontally and parallel to either the AutoCAD X or Y-axis, with the letters or numbers labeling the gridlines displayed at the appropriate ends of the gridline.

For some views, you may wish to see the gridlines on your model as a vertical line projected "behind" the model. For this option, you would select **Vertical Grid.** Once you have created the grid lines, you can banish them at will by checking these boxes "off". Gridlines will always appear on 2d output drawings, regardless of how these boxes are checked. They will appear in the output drawings on the [!SD_Gridlines] layer and can be deleted or isolated and edited easily. Creating and editing gridlines is covered in Chapter 8.

Design Display (Inactive)

The Design Display options are inactive in this version of the software. A future version is currently being developed that handles the objects named under this section and will permit their visibility through these options. This list includes, but is not limited to **Base Plates**, **Cap Plates**, **Connections**, **Stiffeners**, and **Bolts and Nuts**.

Structural Desktop **Regeneration**

When you have made your selections from the Display Options, click on the [OK] button with your mouse. At this time, ***Structural Desktop*** performs a ***Structural Desktop*** **Regeneration** of the entire model.

Everything on the AutoCAD screen is erased.

Structural Desktop then recreates the model from the database. This assures the user that what he sees on the screen is what the model has in the database, and is the data that will be used to save a hold file or create output drawings and Material Reports.

This method wipes everything from the AutoCAD session that is NOT a part of the ***Structural Desktop*** database. Every line, circle, polyline, xref, or block is erased. Only Joints, Members and Elements remain.

With planning, this can be very useful. You can draw construction lines and use them to create other lines. These can be changed to new members and everything but the actual members (old and new) will disappear. A block can be inserted, used to locate a physical change in a member, and will disappear...but will still be available to be inserted again as long as the current session lasts. This operational method is a consequence of ***Structural Desktop's*** design to deal with structures as collections of objects, rather than as lines. This also promotes the goal of ***Structural Desktop*** to produce drawings as quickly as possible, which can then be used in unmodified AutoCAD in any way you wish, with lines, circles, polylines, xrefs, or blocks as the user desires.

A ***Structural Desktop*** **Regeneration** will occur every time you click [OK] on the Display Options dialog box. It will also occur if you begin a function such as Change Member Properties while in the Solid Objects display mode, or when you create a new member or element from a line or 3d face. Lines and circles and other, ordinary AutoCAD objects are not a part of the database for ***Structural Desktop.***

There is now a function, new to Version 4.0, that performs the ***Structural Desktop*** **Regeneration** directly from the menu. This is covered in the next section on **Other Options**.

Other Options

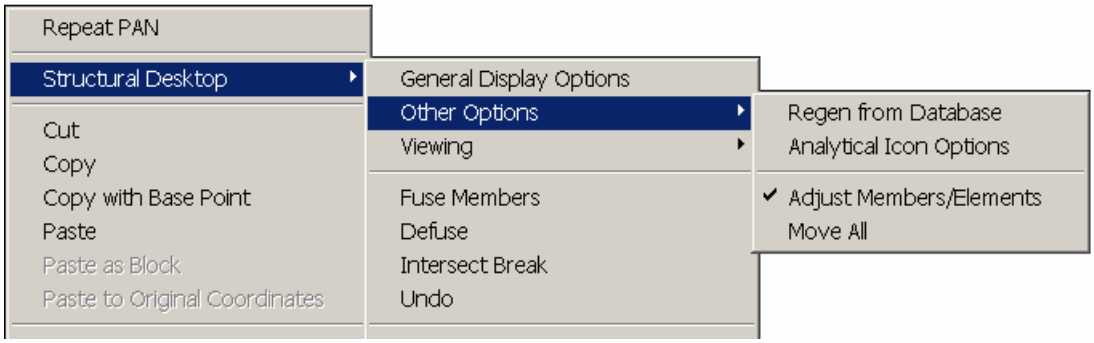


Figure 3-3

The second item on the ***Structural Desktop*** menu is the **Other Options** menu as shown in **Figure 3–3** above. This sub-menu contains two commands and settings for two special preferences.

The command **Regen from Database** is a shortcut to the ***Structural Desktop*** **Regeneration** function. In practise, we have found this function too useful to bury under too many levels, so we have exposed it rather than requiring the user to select the **General Display Options** and click **[OK]** as the only method of regenerating the drawing from the SDT database.

Clicking the second item in the menu, **Analytical Icon Options**, exposes a dialog box that permits you to select whether **Supports** and **Releases** are displayed with icons. As future versions of ***Structural Desktop*** will permit editing of analytical information such as Member Loads or Joint Loads, this dialog box will expand in these future versions to include the new display objects and permit the user to organize the screen display to hide or reveal analytical icons not relevant to the drafting functions of ***Structural Desktop*** but necessary for the display of information when you create or edit the analytical data for your file.

The final two items on the sub-menu are **Adjust Members/Elements** and **Move All**. This is an entirely new function introduced in Version 4.0. **Adjust Members/Elements** is the **default state** for ***Structural Desktop,*** as the program will always begin each session in that mode. In this state, moving **Members** or **Elements** results in an adjustment of the relative position with respect to the **Joint Nodes**, or an **Offset**, and this is how ***Structural Desktop*** has functioned in the past.

Clicking on the **Move All** menu item changes that. Selecting Joints while this item is checked permits you to actually move Joints. This power is sufficient to totally destroy a model if you are uncautious. On the other hand, the power to rotate an entire model 10, 30, 45 or 90 degrees for drawing or analytical purposes, or to actually adjust nodes that

are in the wrong place by using the AutoCAD stretch command to slide a bay a foot one way or the other makes this command worth mastering.

Please note, in order to Move Joints, the circle objects that represent the Joints must be visible and on an unlocked layer. Selecting a joint alone will NOT move the member associated with it; you must select every entity that is to be moved. Items on hidden or locked layers will not move.

However, you can select the entire structure, all objects, and move them to a new base elevation in your model. You can rotate the building around any point, and the analytical file produced will reflect the rotated joint coordinates. You can rotate the building in order to produce true-length elevations when the building contains walls at angles to the orthogonal.

We strongly suggest that you practise with the **Move All** setting, and make a clear habit of turning it off as soon as you have accomplished the movements you wish to make. For this reason, opening a hold file or analtyical file will always begin a session with the selection set to **Adjust Members/Elements.**

Viewing Commands

The **Viewing** commands permit you to selectively view (or remove from view) any members or elements. These members remain stored in the ***Structural Desktop*** database, but are not "in" AutoCAD while they are not visible. Judicious use of Viewing commands can speed up your session significantly. The Viewing commands are available from the Shortcut Menu as shown in **Figure 3-4** below.

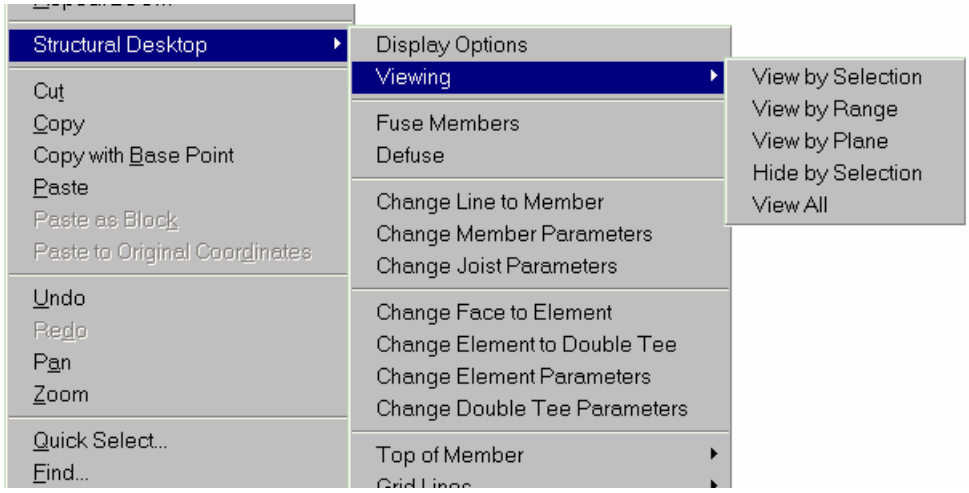


Figure 3-4

The **Viewing** commands will invoke a ***Structural Desktop*** **Regeneration** when they are used. This complete flush of the AutoCAD database contributes to the speed with which you can then manipulate the objects.

The **View by Selection** command asks you to use the AutoCAD selection process. You will see the instruction **Select objects:** on the Command Line. You can use any of the AutoCAD options for selection at this time. If these are unfamiliar to you, please use the AutoCAD help or review the material in the AutoCAD manual.

Once you have made your selection, the program performs a ***Structural Desktop*** **Regeneration** and presents you with only those members and elements that you have selected. When the model is very complex, this function permits you to simplify what you see before you, quickly and easily. At any time, you can return to the full view of your model by clicking on the **Viewing** command **View All**.

You may also use AutoCAD layering techniques with this function to further refine what you see on the screen. **The only way** to return objects to a usable visibility is to select the **View All** function from the **Viewing** menu (See **Figure 3-4** above) which is the last option in the **Viewing** menu.

Remember that the corresponding **Type** of object must be clicked on the **Display Options** menu. You can turn all members off at once by turning them off at the **Display Options** menu. Further, the AutoCAD layer the object resides upon must not be frozen if it is to be visible. With practice, these three options give you a great deal of power in what you select to view from the model.

The **View by Range** command opens a dialogue box as shown at the right in **Figure 3-5** below.

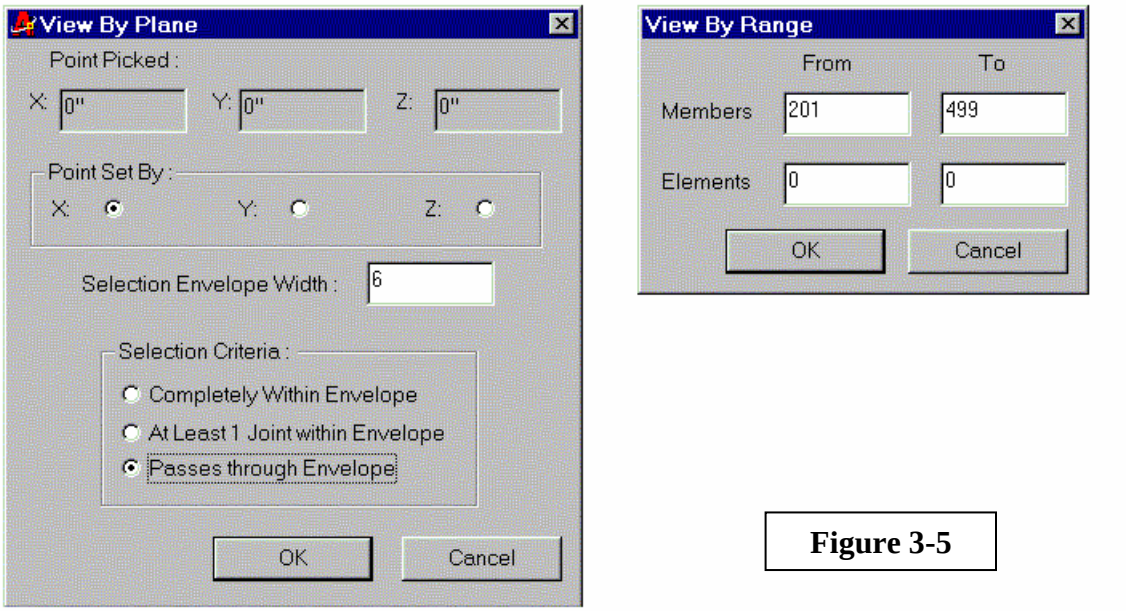


Figure 3-5

The **View by Range** command uses the numbering sequence of your original input file to an advantage. Members numbered in sequence can be parceled off from the rest of the members in your model. You can use a range of member numbers and a range of element numbers if you wish, but elements will not be affected if you do not enter a range for them; they will remain visible. Likewise, selecting a range of elements but leaving the members range untouched will only remove the elements not in the numeric range selected.

By selecting the **View by Plane** command, the Command Line requests you to select a point on the screen. This selection can be done by using any snap point of a member, or by entering a coordinate. The **View by Plane** dialog box appears as shown in **Figure 3-4** above, on the left.

The concept for **View by Plane** is that you want to view a single floor of a structure with or without attendant members that pass through the horizontal plane of the floor, or that you wish to sequester a set of members and elements associated with one or another grid lines or elevations with any rationale you wish. All that is required to use this function is that you have an orthogonal plane in mind.

The dialog box displays the three coordinates of the point you have selected. Directly beneath these boxes are three checkpoints associated with the controlling coordinate axis for your plane. If you want a horizontal plane, the Z-axis controls the elevation so you select the Z button. If you want a vertical plane parallel to the X-axis, the Y-axis determines the distance of your plane FROM the X-axis, so the Y-axis controls the plane and you would check that.

If you wish to select members that are not precisely in line, but only close to the plane, the **Select Envelope Width** box permits you to enter a value for the amount of "forgiveness" you want...if you enter "12" then any member or element that satisfies the criteria within a foot will be selected and remain visible.

The rigor of selection is determined by the three check boxes at the bottom of the dialog box. If you want to select only the members and elements that have all points in the plane the top selection **Completely Within Envelope** should be checked with a value of zero for the envelope. Checking the next box, **At Least 1 Joint Within Envelope**, will select and make visible MORE members and elements. The last selection, **Passes through Envelope**, will select any member or element that is in or passes through the space that is described; this is the most numerous selection set of the three.

As an example, if you have a building where all the members on the roof are at an elevation of 184 inches and one wall is parallel to the X axis at the Y coordinate of 240 inches, we can consider the consequences of different selection criteria once the point (0,240,184) has been selected. If you select Z controlling, zero envelope, and completely within envelope then you will get the roof-framing plan only. Selecting "At least one joint" will show the columns also. Finally, you would select "passes through envelope"

to pick up and view a column member that began below the roof and ended above...part of a parapet, perhaps.

With the same structure and point selected, with the Y coordinate controlling the members remaining visible would be candidates for a grid line elevation. Zero envelope and completely within envelope would show only those columns and beams in that narrow slice of the building, while "one point within" would show adjacent beams framed into that wall slice. Widening the envelope would show columns or beams close, but not on the precisely on the plane.

This command uses the current, actual location of the member or element. This "actual location" includes any change of the Centroid or Face location due to offsets. Where members are moved to take up their actual location with respect to where the member will be in the final position, an envelope width may be required to pick that member up. An example of this would be a floor with beams 18 inches deep as primary beams and 10 inch beams as secondaries, offset to place all of the top of steel elevations in the same plane. Where the analytical model had all members at the same centroid elevation, the centroids of the smaller members would need to be offset four inches higher than the centroids of the deeper members to place all of the tops of the members in the same plane. (See the **Top of Member** command for how to do this automatically!)

You would want to perform this operation to make the output drawings draw the members in final position; that is the value to you of ***Structural Desktop***. With the **View by Plane** command, members offset in this fashion can be accessed by widening the envelope setting.

The **Hide by Selection** command uses the **Select object:** prompt. Any member or element you select will go away. This is useful to refine a selection made with any of the earlier commands by selectively removing an object or two to get a better view.

The **View All** command resets the graphic screen to show all of the members/elements that have been hidden by prior view commands. As explained earlier, the layer options of AutoCAD and the **Display Options** must also be taken into account.

Using Selection

Most ***Structural Desktop*** commands, such as the Viewing commands outlined above, require you to use a selection set during the command operation. The command line will request that you **Select object:** on the AutoCAD command line.

Text, arrows and solids will not select a member or element; only the centroid line, 3D Face, or envelope around a member or element will. When objects are selected they become highlighted: displayed by dashed lines that shift slightly in color. This indicates the current selection set. Pressing the [Enter] key indicates the selection is complete, and you are ready to proceed with the command.

Some of the basic methods for generating and controlling a selection set are:

- Pickbox
- Auto
- Window
- Crossing Window
- Fence

Reference your AutoCAD manual for a complete listing of the selection set commands and their functions.

CHAPTER 4 - Member Modification Commands

The objective of this chapter is to help you understand how to use the following methods or functions to modify, add, or remove members:

- Fuse Members
- Defuse Members
- Change Line to Member
- Change Member Parameters/Undo
- The Intersect Break Command
- The Component Tables
- Top of Member -
  - Top of Member – Flat
  - Top of Member – Slope
- Delete Member/Element

Fuse

Analytical files contain members that are subdivided. A member that connects to another member somewhere along the actual members length must be connected at a joint. For forces to properly be distributed, this joint breaks the actual member into two. When a number of members are connected, there may be two or more analytical members that represent only one member in the structure. The **FUSE** command allows the fusion of selected segments of a member that has been subdivided for the analytical model. An example of this is found in **Figure 4 – 1** below showing members (in this case, bar joists) connected to a support beam. For this support beam to be drawn as a single member, the analytical segments need to be fused into a single member. The fuse command enables you to do this and creates a new member.

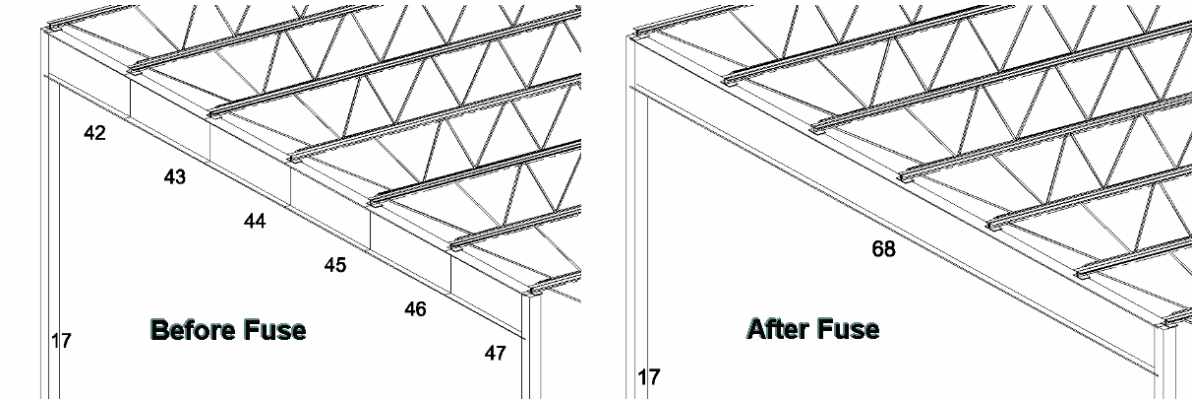


Figure 4 - 1

Members that are fused in ***Structural Desktop*** will be exported as a single member, both for 3D models and in plans and elevations. For reference, the newly created member is given a new member number but retains the properties of the members that were fused. The fuse command will reject members that do not have joints in common, have different sections or properties, different Beta Angles, or are not in a straight line. When writing an analytical file, the fused-result is ignored, the members making up the fused members are used to create a new analytical file.

Defuse

This command will turn a selected fused member back into the original segments that were fused to create it. When a fused member is defused, the individual members will have the original properties and member numbers that they had before fusing. Offsets or layer changes that are applied to a fused member are NOT applied to the original members, so defusing a member is a return to the original position for those members.

Defuse will perform its function on a fused member even after the model is saved in a hold file and reloaded. No matter how many times you open and save a hold file, the member that was fused will always retain the information required to extract the original members.

Change Line to Member

This ***Structural Desktop*** command enables you to add a member to your structure by using the line command from AutoCAD. To implement this feature:

1. Set the Current Layer in AutoCAD to the desired layer.

2. Create lines using the Line Command within AutoCAD. The first point you select with the line is the Start (**J**) of the member and the second point is the End (**K**). You can create as many lines as you wish, copy these lines, array them, offset them, or use any function that ends up with a Line on the screen in Layer 0. You may also insert drawings that contain lines and change those lines to the 0 layer. (Please note: Polylines, Arcs, Circles and other discrete AutoCAD entities are not lines for the purpose of this command.)

3. In the ***Structural Desktop*** Shortcut Menu select the **Change Line to Member** menu item. **Select object:** will appear on the command line. Select the lines or line drawn. When you are finished with your selection, lines on layer 0 will change to lines on the layer !SD_CENTROID. Where joints do not exist in your model at the ends of the members, new numbered joints will be created. The new member will use an existing joint if it is within one inch of the end of the member. The members will be created and numbered in the order you select them in, not the order they were drawn in.

4. In the ***Structural Desktop*** Shortcut Menu select **Change Member Parameters**. The newly created members have no properties, so you should assign a member type and a material type or density. This is covered in the next part of this chapter.

The ability to add a member lets you create additions to your model. When used with an empty document, you can create an entire model in AutoCAD with this command.

Change Line to Member has the side effect of eliminating any object which is not a joint, member, or element in ***Structural Desktop***. For this reason, the **Change Line to Member** command has been augmented in Version 1.6 to perform both the functions **Change Line to Member** and **Change Face to Element**. All of the lines you select will be changed to members and will be numbered in the order in which you selected them, and all faces you select will then be changed to elements and given numbers starting with the next number higher than the members. To minimize confusion, either command will perform all these functions identically. This change permits you to import a drawing with both lines and faces and change all of them that you want to change with one command. Prior to this change (Version 1.5 and previous) you would have had to insert the drawing twice to get all your objects, once for the members and a second time for the elements.

Change Member Parameters / Undo

After completing this section, you will be able to:

- Invoke the Change Member Parameters dialog box in ***Structural Desktop.***
- Use the dialog box to modify lengths and offset members.
- Determine the coordinates of a member.
- Change the type or material of members.
- View member information.
- Undo your changes

The Change Member Parameters menu option in ***Structural Desktop*** will display the dialog box shown in **Figure 4 - 2** on the next page. This dialog box is the focal point for the control of all aspects of a simple member. (Complex members, or joists, have additional information that is explained in Chapter 5.)

Every aspect of a member dealing with the parameters of that member in Structural Desktop is controlled through this dialog box and the boxes that depend from it. The material that follows **Figure 4 - 2** will explain each of the data entry methods associated with this dialog box.

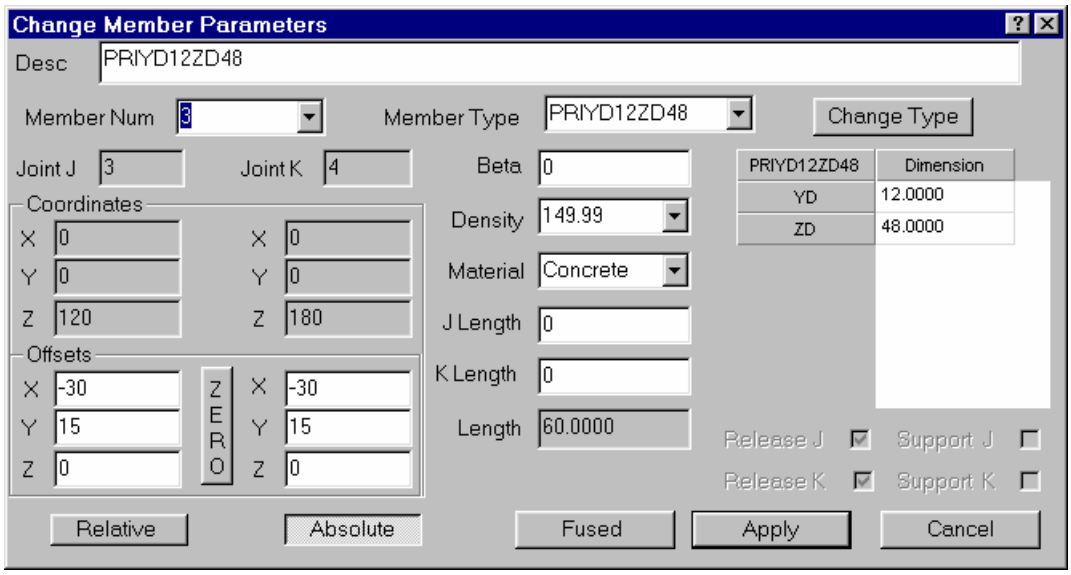


Figure 4 - 2

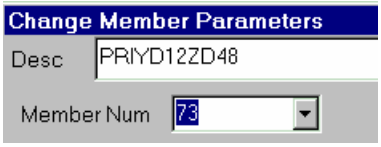


Figure 4 - 3

- **Desc.** – This box displays a label that may be unique for each member or shared by any number of members. When a member is read from the data file or assigned a new member type, this description defaults to the description of the member. In **Figure 4-3**, the Desc field contains **PRIYD12ZD48**, which indicates a prismatic rectangular member 12 x 48 in section. A wide flange might display **W14x99** in this box. You can change the contents of this box to be whatever you wish, or add a note at the end of the description to specify special treatment at a later time. This label will display on the screen when you click the **Display Options** Member Names checkbox, and will be the name displayed on your 2d drawings.

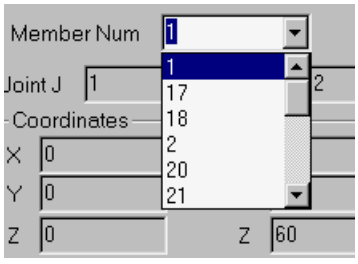


Figure 4 - 4

- **Member Num** – Indicates the member number originally extracted from the analytical file. In the event of a new member or a member that has been created by fusing members, this number will be the number assigned when the member was created. This is a drop-down box. This means that you may click on the box to see the list of members that have been selected. (See **Figure 4 - 4**)

  The details that display on the rest of the dialog box will derive from the member currently showing in this box. This is important to consider when some details may be different. Changes you make in the **Change Member Parameters** box will be applied to **ALL** of these members when **APPLY** is clicked. If you do not change the member type, for example, then they

will not all be changed to be the same member type. Similarly, if you select a dozen members of different types, different beta angles, and different offsets and change the beta angle to 45 degrees, the different types and offsets will not be changed but all the beta angles of all the members that have been selected will become 45 degrees.

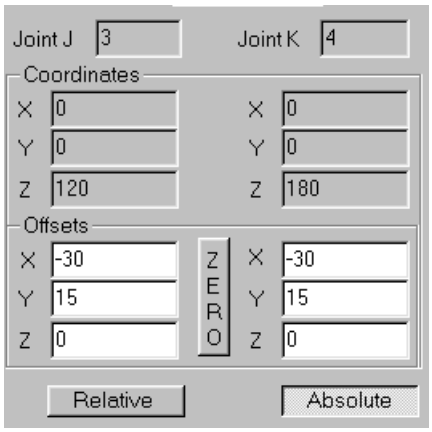


Figure 4 - 5

- **Joint J** – Represents the member **Start** joint number from the analytical file or from when the member was created or fused.

- **Joint K** – Represents the joint number at the **End** of the member.

- **Coordinates** – Displays the member coordinates of the **Start** joint (**J**) and the **End** joint (**K**) of the member. Joint coordinates cannot be changed in ***Structural Desktop.*** New joints can be created only by creating new members, and old joints cannot be changed. Members can be moved, rotated, or changed in any way imaginable by changing their offsets FROM these immutable points.

- **Offsets** – Display and permit adjustments of the member Start or End. An offset shows the displacement of the real member centroid from the analytical member endpoints. If you **MOVE** the member using the AutoCAD **MOVE** command, the offsets will be updated to respect the moved endpoints new offset situation. At any time, a member can be returned to the original position by entering zeros for this entry, so you can also move the member by changing the offsets.

  When you edit an offset, the default value is **Absolute**. You must click one the **Relative** button for it to override the **Absolute** setting. The difference is:

  - **Relative** coordinates add the new offset values to the existing values. If you want to move six inches MORE in X, Y, or Z, use this setting.
  - **Absolute** coordinates replace the existing offset.

Example: Both ends of a member are already offset six inches positive in the Z-axis. You decide that four inches would be the correct offset to place the member where it should be in the model to reflect reality. You could:

- Offset both ends, using a **Relative** offset of Minus 2 inches (-2.0) in the Z-axis. The offset is added to the existing 6 inch offset, resulting in a 4 inch offset.
- Offset both ends, using an **Absolute** offset of four inches (4.0) in the Z-axis. The offset replaces the 6-inch offset with a 4 inch offset.

- Exit from the dialog box and just move the member up two inches in AutoCAD. The next time you look at the dialog box, the offsets will have changed to reflect your action.

All of the above actions will accomplish the same results.

- **Zero** – Added in Version 1.95, the Zero Button on the dialog box will zero all offsets for every member currently selected. When you click on "APPLY" the members will be returned to the end-coordinates determined by the joint locations associated with the member. This is a very powerful and quick operation, and is a good reason to ***save your work frequently in a hold file***.

- **Member Type** – Displays the member type of the objects that have been selected. If you select another Member Number in the **Member Num** box, **Member Type** will change (if needed) to display the type of the member selected. This is a drop-down box. If you click on the arrow at the right side of the box, you will see other types as shown in Figure 4-6. You may click on one of these member types to change the current member.

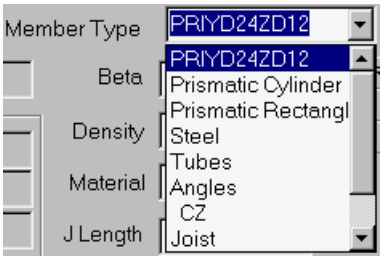


Figure 4 - 6

  If you click on a member type defined by parameters (e.g. a prismatic), you will be able to access the grid at the right of the Member Type box to edit the depth and width. If you select a type that derives from a table (e.g. Wide Flanges from the Steel table), the dialog box for a particular section of that type will appear. The same thing occurs when you click on the **Change Type** box (detailed next).

- **Change Type** – Moves directly to the dialog box for selecting a new member type from standard tables. Pressing this button displays the tables of the objects that are available from all of the standard and ***Structural Desktop*** tables. How to apply this is covered in detail in the **Component Tables** section below.

- **Beta Angle** – Defines the member's rotation around the Centroid Axis. There are several ways to move and modify where a member is through its end points, but the rotation of a member must be adjusted through this box. Rotating the envelope or solid in the model will not update this item or persist in the model.

- **Density** – Describes the material used in the structural design. ***Structural Desktop*** has set values it recognizes for Steel, Aluminum, Concrete, Lightweight Concrete, and Wood as shown at right (**Figure 4 - 7**). These values will be used in the Material Report. If you change the value of the density, the Material text box (below) will reflect that change.

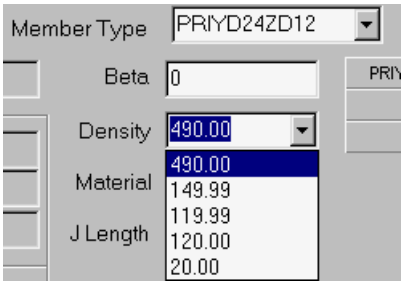


Figure 4 - 7

- **Material** – Describes the material used in the structural design. ***Structural Desktop*** has set values it recognizes for Steel, Aluminum, Concrete, Lightweight Concrete, and Wood. These values will be used in the Material Report. If you change the value of the material, the density text box (above) will reflect that change.

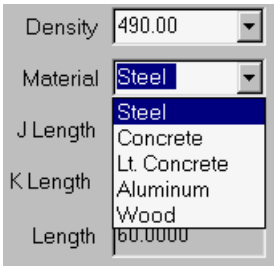


Figure 4 - 8

- **J Length and K Length** – These entries permit you to change the member length regardless of the orientation. For example, if you wish the member to be six inches longer at the start, a positive six in the J Length box will add six inches of length to the member at that end. If the member is not orthogonal, the program will calculate the necessary X, Y, and Z displacements to lengthen the member at the end desired and by the amount specified. A negative entry will subtract length from the member at the end changed. As always, these manipulations are reflected in the final offset values and can be undone by entering zero offsets to reset the member to original settings.

- **Release and Support Display** – These boxes will show as checked when the joints associated with a member were supports or the member end joints had release conditions in the analytical file. This section will be more involved when ***Structural Desktop*** uses this information in creating input files for analysis.

Figure 4 - 9

- **Fused, Apply, Cancel** – Fused will display a dialog box that in turn displays information about a member that has been fused, such as the member numbers that went to make it up. Apply is the button you must click to make changes take effect, and Cancel is how you change you mind before making a commitment to changes if you are unsure of them or their effects.

Component Tables Dialog Box

Clicking the **Change Type** button in the **Change Member Parameters** dialog box will display the **Component Tables**. The **Component Tables** enable you to select member types/sections from standard tables or from the ***Structural Desktop*** member tables. The **Component Tables** dialog box has changed significantly from version 1.9 to Version 1.95 to accommodate multiple country tables, and again in Version 4.0 to accommodate changes in the underlying database support of Windows.

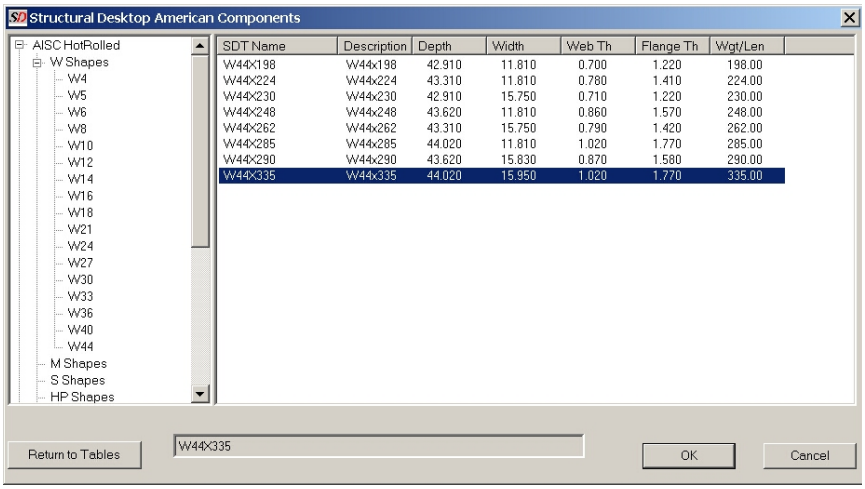


Figure 4 - 10

Each table displays the respective country in the top window margin of the dialog box. The window to the left is a drop-down tree that permits selection of types and groups within each type of member. The window to the right will display all members that are within that grouping with information labels across the top appropriate to that member. The A.I.S.C. tables are shown with the Wide Flange table displayed and the grouping of members with a nominal depth of 44 inches is shown in the window to the right.

- A.I.S.C. - Wide Flanges, Channels, Miscellaneous Channels, M-S-HP sections, or T sections as in WT, MT, and ST's.

- Tubes - Tubes include all of the HSS sections.

- Angles - Angles have buttons on the side for regular L angles, Reverse angles, and both varieties of Double Angle: Long Legs or Short Legs Back-to-Back.

- ColdFormed - The ColdFormed button will invoke two groups of buttons down the side, one for the AISI sections (CS, CU, ZS, ZU, and HU) and one for the LGSI sections (Z, C). The User Tables included in your installation of ***Structural Desktop*** can be used within STAAD, GT STRUDL or RISA 3D to designate members with these names, and have properties drawn from these members. You may also attach properties from these member designations after-the-fact within ***Structural Desktop***.

- Joists - The Joists button will invoke three tables. One is for Joist Girders, denoted by JG. The intermediate size is denoted as Long Span or LH joists. Short Span or Bar joists are denoted by K. These members also have User tables for use within your analysis program.

- Concrete - The Concrete button invokes two tables for Concrete L's and one for Inverted Tees. There are two L files for the same reason that unequal-leg angles are capable of being specified as Reverse Angles: These files contain members that have a "handedness" and that cannot be simply reversed to display the way you might wish them to. The Concrete members also have corresponding ***Structural Desktop*** user files for use within STAAD, GT STRUDL or RISA 3D.

- Pipe - The Pipe button denotes Strong, Double Strong, and Double Extra-Strong pipe sizes.

- Wood, Aluminum, FRP - These buttons are shown on the dialog box, but the tables are still under development for future versions of ***Structural Desktop***.

You may browse within the Component Tables until you find the member you wish to assign. Highlighting or double-clicking on the member and clicking **OK** will assign that member name and its properties to be brought into the **Change Member Parameter** box. You then must hit the **Apply** button on the previous dialog box to assure that these properties are distributed to the members selected.

Pressing **Cancel** in either dialog box will void the change.

For information purposes only, you can view the **Component Tables** from the main ***Structural Desktop*** Menu under the **Display Text Data / Display Tables** item. This can be useful to run a quick check on a dimension without recourse to the Steel Manual.

Structural Desktop has a built-in **Undo** function. (See **Figure 2 -2**) Due to the separation of the AutoCAD drawing and the actual ***Structural Desktop*** database elements, use of the regular AutoCAD undo function can have unpredictable consequences. Until this is rectified, a special undo function has been created that is linked to changes within **Change Member Parameters** and **Change Element Parameters**. Whenever invoked, this function will undo the changes done that pertain to the most recent use of **Change Member Parameters** and **Change Element Parameters**.

The Intersect Break Command

Structural Desktop version 1.7 introduced a new command for users who are creating geometry for analytical programs. The **Intersect Break** command is located on the Right-Click Menu under the **Fuse** and **Defuse Members** commands.

When you select this command, you are prompted to select members. Members selected will be checked against every other member selected for intersections. If the members intersect without endpoints at the intersection, the members that cross will be broken.

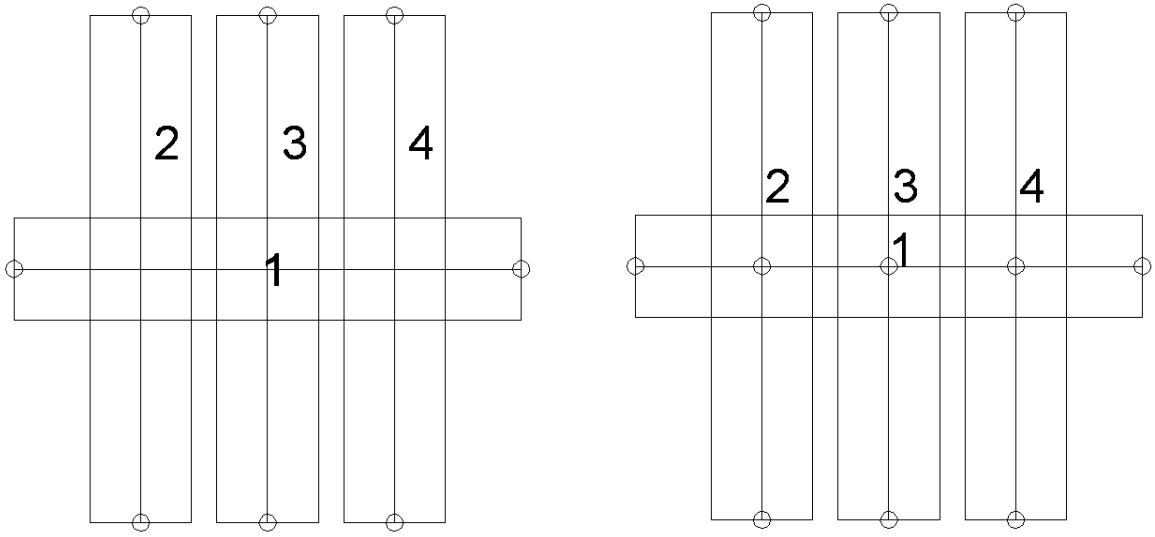


Figure 4 - 11

In the example shown in **Figure 4 - 11** above, the members at the left were drawn as lines and changed to members. The eight joints were created from the endpoints of the members.

The image at the right shows the same members after the Intersect Break command has been applied and all of the members selected. There are now three new joints across the drawing. In fact, 10 new members have been created, numbered 5 through 14. Members 2, 3 and 4 have been split where member 1 crosses them, then **FUSED** back into members 2, 3, and 4. Likewise, member 1 was split into four members, which were **FUSED** to recreate member 1. What this will mean to an analytical input file is that the "real" members that will be created in the file will be members numbered from 5 to 14 and the physical members, 1 through 4, will be screened out by ***Structural Desktop***.

Any changes (beta angle, member type, etc) to these four members will be automatically applied to the analytical members they are now built "from." If you wish, the **FUSED** members can be defused, but this is not necessary for the creation of an analytical file. When you create an analytical file, the members 5 through 14 will be the ones written to the file, but with the current member types and beta angles of the member they comprise applied to the analytical model.

If the break command had been issued in this model without selecting member number 1, no new members or joints would have been created. Only members that are selected can be broken, and only members that have been selected can do the breaking.

Top of Member

In the ***Structural Desktop*** Shortcut Menu there is a **Top of Member** menu item. This item contains two options:

- **Top of Member - Flat**
- **Top of Member - Slope**

The purpose of these two menu items is to perform the adjustment of the relative elevation of beams within a floor or roof so that all of the members have the same elevation for the top of the member relative to either a flat or sloped plane.

When you create your model, all of the members are represented by centroid lines that have no depth or thickness. Simply turning these lines into members will not place the members exactly where they should be in terms of elevation, so offsets must be applied. This function will perform the beam Z-axis offsets necessary to bring the tops of the selected members into a common elevation.

Top of Member - Flat

Top of Member Flat is a modifying command that manipulates offsets for a selected set of members. No matter what the depth of the member, the program already has this information in the database. By selecting this command and a group of members, you are given the dialog box below in **Figure 4 - 12**.

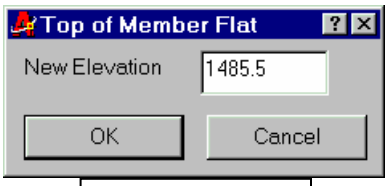


Figure 4 - 12

Before using this command, check the model or your notes to determine what the Top of Member should be for the part of the structure you are rectifying. If you are preparing to modify a significant number of members, save your work in a hold file before proceeding. You can always return the members you modify to their original position through the Change Member Parameters command and by zeroing out the offsets that this command produces.

Click the command on the menu, then select all of the members you wish to modify. (The program is designed to ignore columns in this instance, if the column is truly vertical.) In the **New Elevation** text box enter the true elevation of the top of the members in **INCHES (or CENTIMETERS)**. When you click **OK**, the screen will update and the members will be adjusted as necessary to place all of the tops of the members in the flat plane at the elevation you specified.

It should be noted that elements will also follow the **Top of Member** command. Since elements commonly rest on top of the members that support them, the program is constructed to place elements with the plate of the surface at the elevation specified. If the elements were created to extrude upward (counter-clockwise numbering) then the representation of the element will display that element with the bottom at the elevation given to the command. Elements that extrude in the opposite direction will have their tops at the elevation specified.

Top of Member - Slope

The **Top of Member - Slope** function enables you to:

Take an analytical model with a flat roof and modify the roof to a designated slope.

Take an analytical model with a designated roof slope and adjust members so that the tops of the members are in the same plane.

Alter the slope of the roof of a model.

Top of Member - Slope is designed to deal with roofs that have a slope in the direction of the **X** or **Y**-axis in AutoCAD. The dialog box that you will see after you select this command from the menu and select members to be modified appears in **Figure 4 - 13**.

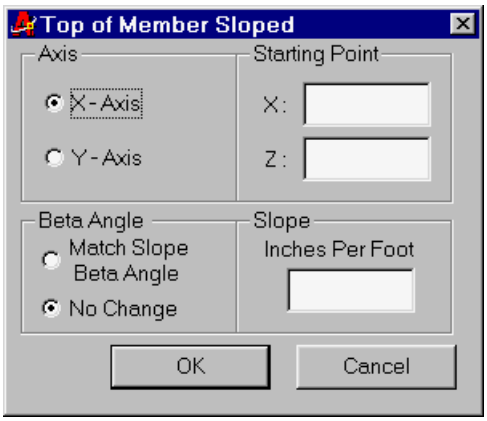


Figure 4 - 13

Using the dialog box to set a slope on selected members begins with the **Axis** box. Select the **Axis** (**X** or **Y**) in the dialogue box. The direction of the slope should be parallel to the direction of the **X** or **Y**-axis. Other slope directions are not supported at this time.

The program starts with the direction of the slope and a starting point. What you are basically telling the program is that the top of the roof plane slopes in a particular direction and passes through a particular point. Since there can only be one plane that passes through any given point and represents a roof with a set sloping angle, the program can calculate and place members based on the information in this dialog box.

The **Starting Point** values require two coordinates. If the X-axis is selected as the direction of the slope, you must provide an X coordinate as shown being requested in the dialog box of **Figure 4 - 13**. If the roof has a uniform slope, at that X value the elevations will all be equal for any value on the Y-axis. If the Y-axis is selected as the direction of the slope, you must provide a Y coordinate. The dialog box will change to request the proper coordinate from you, so this is a good check point to make certain you are directing your slope the direction that you intend to. In either case, the corresponding Z coordinate (elevation) needs to be filled in to place the top of member in the sloped plane.

The **Slope** text box on the dialog box requests the slope of the roof in inches per foot. This value is either positive or negative.

- A positive number entered into the slope dialog box increases the elevation of the plane of the roof in a positive "X" or "Y" direction.
- A negative number entered into the slope dialog box decreases the elevation of the plane of the roof in a positive "X" or "Y" direction.

To change the direction of the slope of the roof in a positive "X" or "Y" direction (such as a gable roof) you use a new starting point and a new slope. The illustration shown in **Figure 4 - 14** indicates such a setup for one such roof.

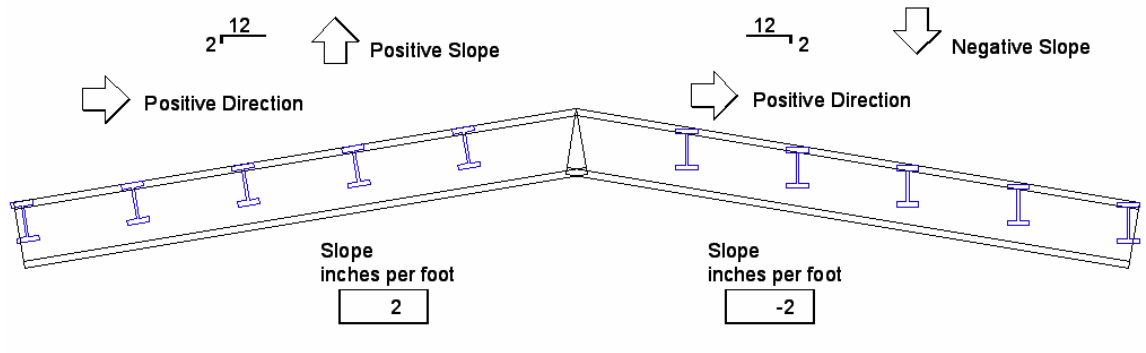


Figure 4 - 14

If you take a close look at **Figure 4 - 14**, you will see one other difference in the two halves of the sloped example roof. When the members on the left half of the building were selected, it was intended that they be sloped to match the slope of the building roof. This is accomplished with ***Structural Desktop*** members through the alteration of the Beta Angle of the members. Checking the checkbox **Match Slope Beta Angle** in the dialog box will set the **Beta Angle**, regardless of the current value, to match the slope of the roof as shown in **Figure 4 - 14**.

The other half of the roof had members with a **Beta Angle** of zero. Checking **No Change** in the **Beta Angle** section of the dialog box will simply move the members vertically until some corner touches the slope of the roof, thus establishing a stair-step effect in the right half of the illustration. The **No Change** box will not set the **Beta Angles** to zero, but will leave the angle as it is in the database.

Delete Member/Element

If you use the erase command in AutoCAD to remove a member or element in ***Structural Desktop***, you have removed the member from the screen, but not from the database. Any function that creates a **Special Regeneration** will show the member to still be a part of your structure, and it will come back.

Since a member created for an analysis is likely to be important to you, eliminating that member should require a special function at the very least. This function is available through the menu command **Delete Member/Element**. This command enables you to select members or elements. When you are though with the selection and permit the program to continue, this function will list the members and elements on the screen and ask you if you are certain that you want to delete them. If you confirm that you want them deleted, they are gone beyond retrieval.

We strongly suggest making a special hold file of your data at the point you are when you wish to delete members. A hold file can be created that has some derivative of the day's date, for example, so that you may return to your data in that configuration at any time in the future. You cannot make too many backups.

CHAPTER 5 - Complex Members

The objective of this chapter is to provide information on using Structural Desktop members that we refer to as complex members. Complex members may be analyzed in your analysis program through the use of the User files that we provide. Currently, the complex members are only available in Imperial Units with American Tables. When these single-centroid-line members are extracted to or created in Structural Desktop, they are composed of structures of smaller members that are arranged as a pictorial representation of joist girders, longspan joists, and shortspan or bar joists. This chapter will help you to understand in detail how to use the Change Joist Parameters function.

Change Joist Parameters Dialog Box

Just as the **Change Member Parameters** dialog box is the heart of making modifications to the usual members in ***Structural Desktop***, the **Change Joist Parameters** dialog box (**Figure 5 - 1**) enables you to make changes to Complex Members. These members have an internal structure and can be parametrically modified through this dialog box.

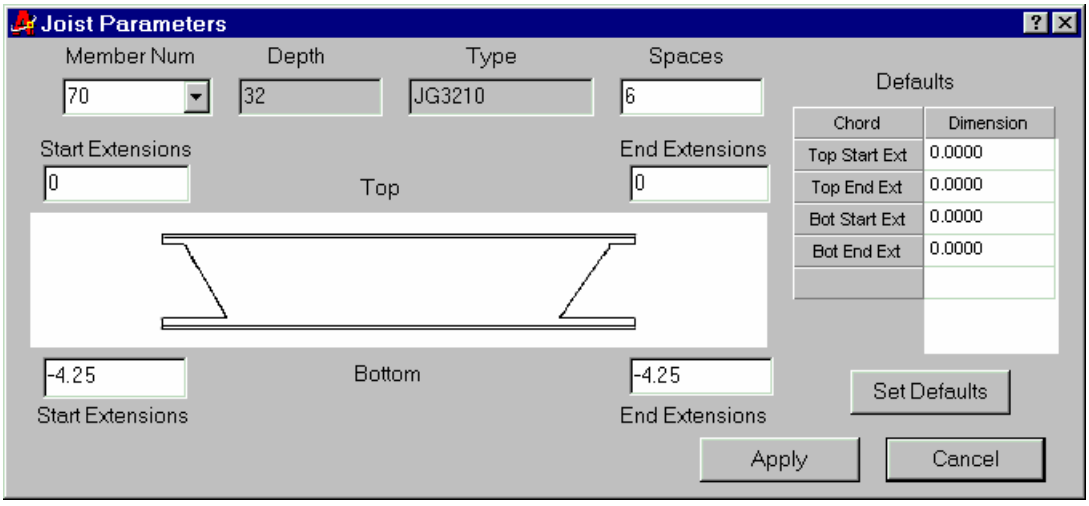


Figure 5 - 1

The dialog box breaks down into fairly simple components. With explanations, these are:

Member Num - As in the Change Member Parameters dialog box, this is a drop-down box that will display all the members in a current selection set. The other text boxes on the dialog box will display the information about the currently viewed number. Changing the number in the Member Num box can change the display.

Depth and Type - These boxes display the physical depth and the Member Name of the object indicated by the number in the Member Num box. As these values are not directly edited by the user, they are a dark gray in color to distinguish them. To change the TYPE of member, and thus, through that mechanism, its depth, you must user the Change Member Parameters box. A breakdown of the meanings of the Type Name can be found in the Appendix.

Spaces - When you model a Joist Girder using the members provided in the User Files, you have the option of also modeling the secondary members that frame into them such as LongSpan Joists. This option can be very useful.

First, you can generate accurate point loads for the Joist Girder by applying uniform and other loads to the secondaries. Second, you can have those members drawn for you in elevations and plan views, and labeled automatically. If you have taken this route, the Joist Girder will have nodes at each intersection.

Structural Desktop counts the nodes of the members you fuse to create a Joist Girder, and creates the Joist Girder with that number of panels for equally spaced support. In **Figure 5 - 2** you see two joist girders modeled in envelope form before and after fusing. Note that the image on the left is confusing; the program is trying to draw a complete joist girder for every sub-member.

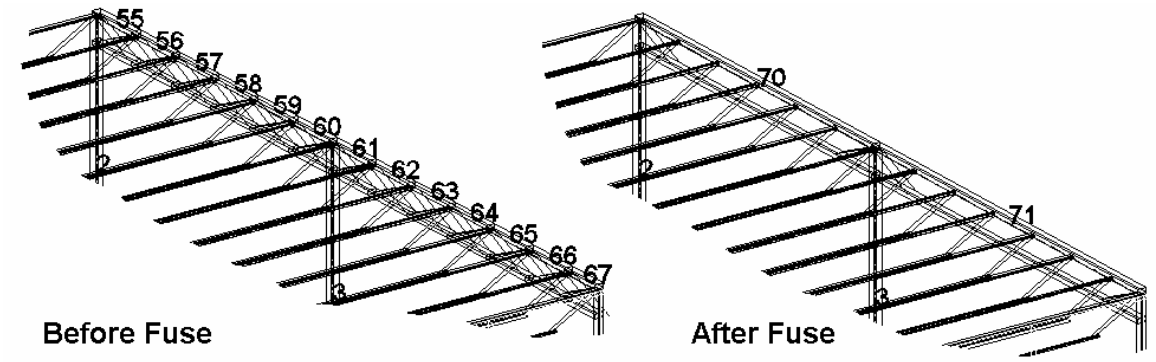


Figure 5 - 2

If you count the joists framing into members 70 and 71, you will see that member 70 should have five support points between the ends, or six panels (or spaces). Member 71 supports six joists between the ends, for seven panels. **Figure 5 - 3** shows how these members are drawn in elevation. Note the members shown sitting on top of the equally spaced support points between the panels.

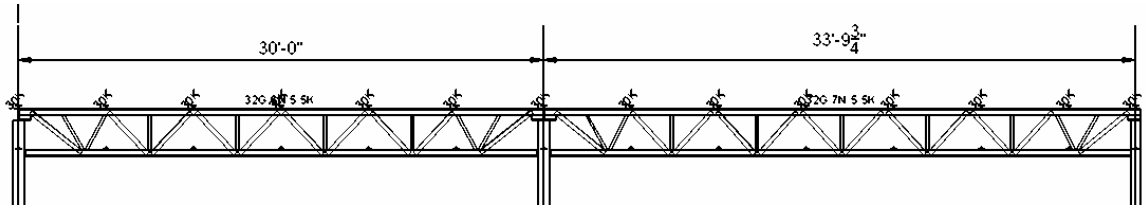


Figure 5 - 3

If you have not fused members or the fused members will not provide the proper equal spacing, you may set the database value for the number of spaces in the **Spaces** box on the **Change Joist Parameter** dialog box. As long as the number of spaces is correct, any inequality in the positioning of the panel points can be quickly adjusted in the final drawing in AutoCAD by you or your draftsman.

Top Extensions - The Start and End extensions at the top of the symbolic joist on the dialog box will add inches to the top chord of the member. The actual length of the member, as seen in the Change Member Properties box, is measured from the outside of bearing plate to outside of bearing plate; chord extensions are added to that length, in addition to it.

Once you have added an extension to the top chord, changing the length of the member (either parametrically or through an AutoCAD action such as trimming the member or stretching it) will not change the extension. The extension itself can only be edited through the dialog box, since any attempt to stretch or alter the line representing the member affects the member, not its internal structure.

Negative values cannot be accepted by the program for a top chord extension, since to do so would remove the top chord component from the support of the bearing plate component of the complex member. **Figure 5 - 4** shows a 3d solid of a top chord, before and after a 24-inch extension is made to the member.

Note that the dashed line representing the member ends at the bearing plate.

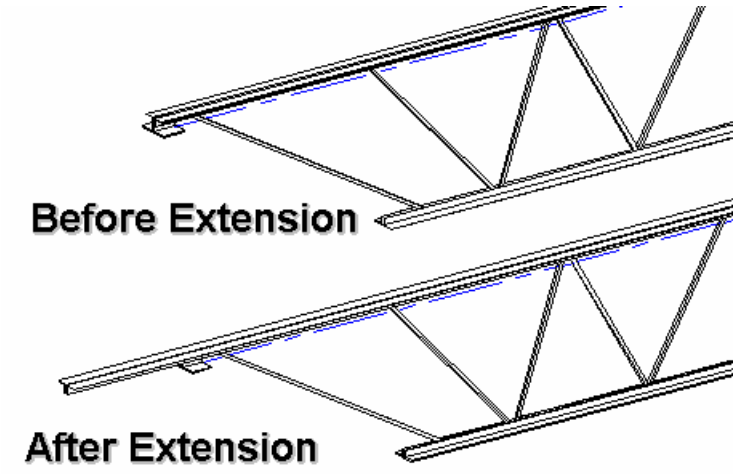


Figure 5 - 4

Bottom Extensions - Bottom extensions default to a value calculated from the depth of the member for Bar Joists and LongSpan Joists. This value is negative, and subtracts from the length of the bottom chord to "pull back" the bottom chord from being directly under the bearing plate. **Figure 5 - 4** shows a default bottom chord. A positive value in this dialog box entry box will extend the bottom chord past the bearing plate for the member. Joist Girders set their default value at zero, but you may edit that value to any value that is reasonable for the member referenced.

Defaults and Set Defaults - The text grid at the right of the **Change Joist Parameters** box is there to display the default values for the members you have selected. In the event that you wish to return to the simple default value for top and bottom chord extensions, click on the Set Default button. This will set the extensions to the values in the Defaults display.

Apply and Cancel - The **Apply** button must be clicked to make your changes to the members take effect. The **Cancel** button will close the dialog box without performing any changes to the members selected.

CHAPTER 6 - Element Modification Commands

The objective of this chapter is to help you use the following areas and functions:

- Element Concepts
- Change Face to Element
- Change Element Parameters

Element Concepts

Structural Desktop displays elements through an entity called a 3D Face. The **3DFACE** command in AutoCAD creates a three- or four-sided surface anywhere in 3D space. You can specify different Z coordinate values for each corner point of a 3D face. **3DFACE** differs from creating a **POLYLINE POLYGON**, which can create a three- or four-sided outline that is parallel to the current user coordinate system (UCS) and cannot use different Z coordinate values for each corner point. **3DFACE** also differs from creating a **SOLID (not an AutoCAD solid entity, but the 2 dimensional solid)**, which is also parallel to the current UCS, cannot use different Z coordinate values, and is entered with a criss-cross node sequence that is counter-intuitive for users of analytical programs.

For these reasons, the 3DFACE command creates an object most consistent with the object behavior of an element in analysis programs. You are not restricted as to the orientation of a 3D Face, and the object retains the order in which the nodes are specified, which permits the object to indicate a direction in which to **extrude** (extend by a thickness into the 3rd dimension) the element: counter-clockwise to the direction in which the element nodes are created. This enables the 3D Face to generate a solid or to retain the information so that the thickness of the element can be displayed in edge-views. The extrusion of a Face into an Element Object is shown in **Figure 6 - 1**.

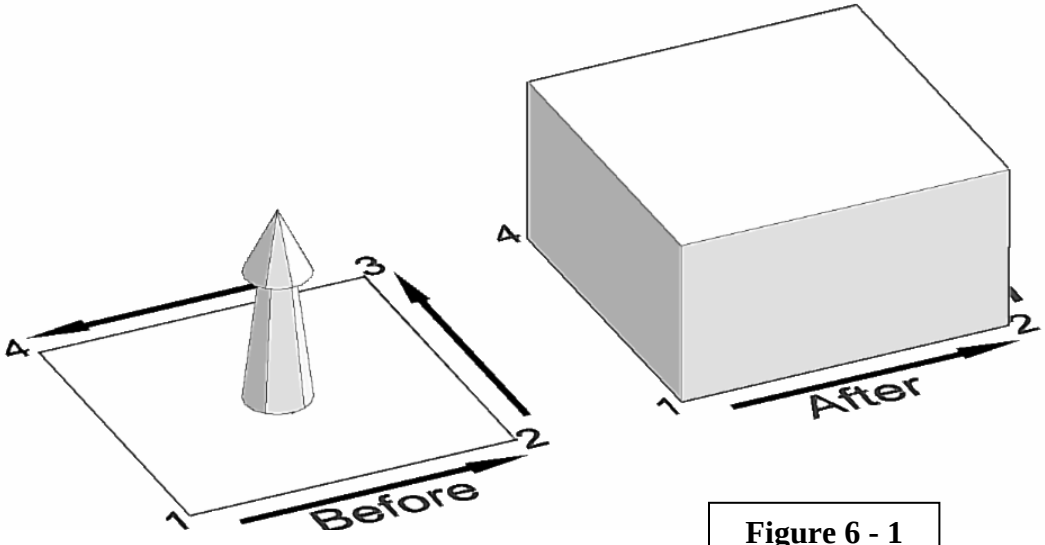


Figure 6 - 1

Change Face to Element

The **Change Face to Element** command is identical to the **Change Line to Member** command. ***Structural Desktop*** displays Members as Lines, which contain the necessary information to link to the database and permit all the modifications and displays that go with members. This command permits you to add element members to the database that were not in the original analytical file. You can also use this command to create elements for a model created from scratch.

Before you use the command, you are expected to create the 3D Faces. The 3D Faces must be on AutoCAD layer 0. The program will ignore any object that is not a 3D Face (or line) when executing this command, but it will also ERASE any AutoCAD object that is not a member, joint or element of ***Structural Desktop***. Since this includes any lines that you may wish to turn into members, you must remember to select lines and elements that you wish to change to members and faces. If you draw lines and faces and only select one or the other, the other type is going away!

Once you have all of the faces drawn that you wish to change into elements, right-clicking the mouse while the cross tee is on the graphic screen engages the Shortcut Menu. Selecting **Change Face to Element** displays the **Select objects:** prompt. Select the faces, or just type "all" on the command line...***Structural Desktop*** will ignore all ***Structural Desktop*** objects while performing this command. If you use the "all" command, however, the faces will not be changed to elements in any predictable numbered order. To create elements in a particular numbered sequence, select them in the order you wish them to be numbered by using a pickbox and picking them one at a time.

You should also remember that you can create lines or 3d faces inside a model at any time and change them into members and elements. If you wish, you may create a line and change it to a member, one at a time, until your entire model has been created, and add elements in the same way.

When each new element is created in ***Structural Desktop,*** it is assigned a thickness of one inch as a default value. To change the thickness, specify the material of an element, or parametrically modify offsets for the corners of an element, use the **Change Element Parameters** command.

Change Element Parameters

The Change Element Parameters menu item is found on the ***Structural Desktop*** Shortcut Menu. You are prompted to **Select objects:** (elements), which displays the dialog box as shown in **Figure 6 - 2** as soon as you have made your selection.

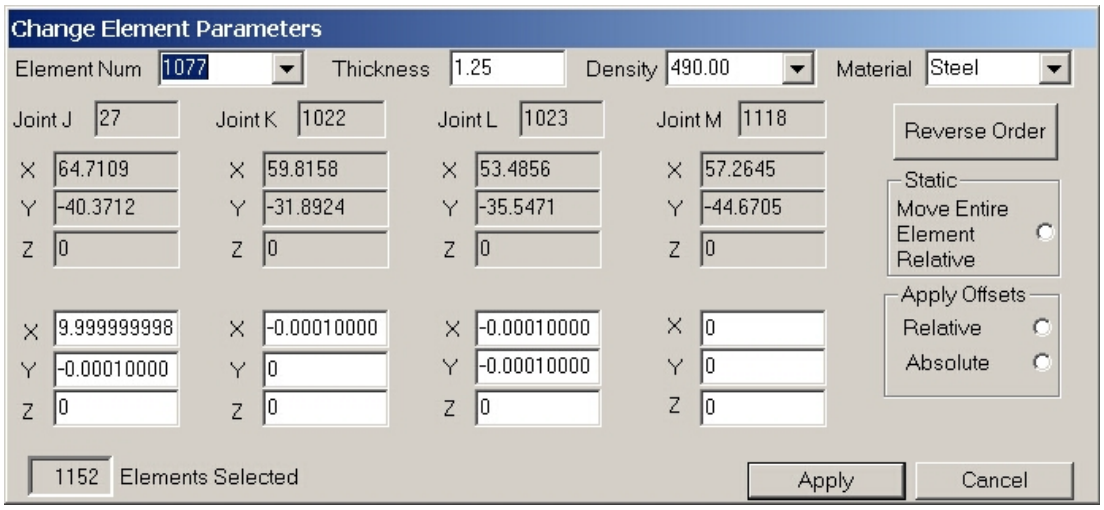


Figure 6 - 2

The different data-entry text boxes and features of this dialog box are detailed and explained as follows:

- **Element Num** – The Element Numbers of the selected elements. Just as in the Change Member Properties box, the different values displayed will be those of the element number displayed on this drop-down box. Changing the "current" element will update the display, but a change to any item in the display will be propagated across the entire set of selected elements.

- **Thickness** – This value displays the thickness of the elements. Change this value to change the thickness.

- **Density** – Describes the material used in the structural design. ***Structural Desktop*** has set values it recognizes for Steel, Aluminum, Concrete, Lightweight Concrete, and Wood. These values will be used in the Material Report. If you change the value of the density, the **Material** text box will reflect that change.

- **Material** – Describes the material used in the structural design. ***Structural Desktop*** has set values it recognizes for Steel, Aluminum, Concrete, Lightweight Concrete, and Wood. These values will be used in the Material Report. If you change the value of the material, the **Density** text box will reflect that change.

- **Joint and Joint Coordinate Boxes** – These boxes are gray in color because they cannot be edited. The values are for your reference and show the 3 or 4 joints that

are used to create the element. These are referred to in the order they are entered as **J, K, L** and **M**. Applying an offset does not move the **JOINT**, so these values will not change.

- **Offsets –** There are four sets of coordinate boxes for you to use to move or alter the shape of an element. If you move the corners of an element and end up with a non-planar shape, AutoCAD and ***Structural Desktop*** work together to provide the closest approximation to a planar slab that can be drawn from the data you provide. In order to simplify the function of moving an element in the X, Y, or Z-axis, ***Structural Desktop*** has two ways of using these boxes.

  - **Static –** ***Structural Desktop*** will interpret a click in the frame of the checkbox labeled **Move Entire Element Relative** as your intent to move as a unit. The **Offsets** boxes will change to appear as in **Figure 6 - 3** below:

Figure 6 - 3

With **Move Entire Element Relative** clicked, the changes made in the box at the left with an outline around it will apply to every node: **J**, **K**, **L** and **M**. This set of coordinate changes will move the element without changing the relative positions of the corners to each other, moving the element as a whole.

  - **Apply Offsets -** ***Structural Desktop*** will interpret a click in either of these checkboxes (**Relative** or **Absolute)** as your intent to move one or more corners of the element, which will change the shape of the element. In this instance, you must make whatever changes you wish to have occur for each corner in all three dimensions.

- **Reverse Order –** This button is a safety valve, in the event that you have created an element, only to find that the order of the joints cause that element to extrude the thickness in the direction opposite to that intended. Clicking the **Reverse Order** button will change the sequence of joints to cause an element to extrude in the opposite direction. The joint locations, and their offsets, will not change, only the order in which they reside in the element. The first joint number will not change.

CHAPTER 7 - Complex Elements

The objective of this chapter is to provide information on using ***Structural Desktop*** elements that we refer to as **Complex Elements**. **Complex Elements** represent precast concrete double tees, and may be analyzed in your analysis program through the use of regular elements. Currently, Complex Elements are only available within the Imperial units system and with American Tables active. When these elements are brought into or created in ***Structural Desktop***, they will be a thickness that satisfies you in regards to simulating whatever properties you wish to model. You can use the Change Element Parameters function to change the thickness of an element, but to display that element as a double tee with legs and internal parametrically determined structure, you must be able to:

- Change Element to Double Tee
- Change Double Tee Parameters

Change Element to Double Tee

The Shortcut Menu item **Change Element to Double Tee** will trigger the **Select Objects:** prompt**.** For this command, you wish to select ***Structural Desktop*** elements that should display and plot as double tees.

Do not select 3D Faces that are NOT elements to change into Double Tees.
To change 3D Faces to Double Tees is a two step process:
Change Faces to Elements
Change Elements to Double Tees.

When you have completed the command, the elements will display two additional 3D Faces as a part of the Centroid/Faces display in order to identify them as elements that represent double tees. The legs will display in envelope or solid mode (**Figure 7 - 1**), and will be drawn on plans and elevations.

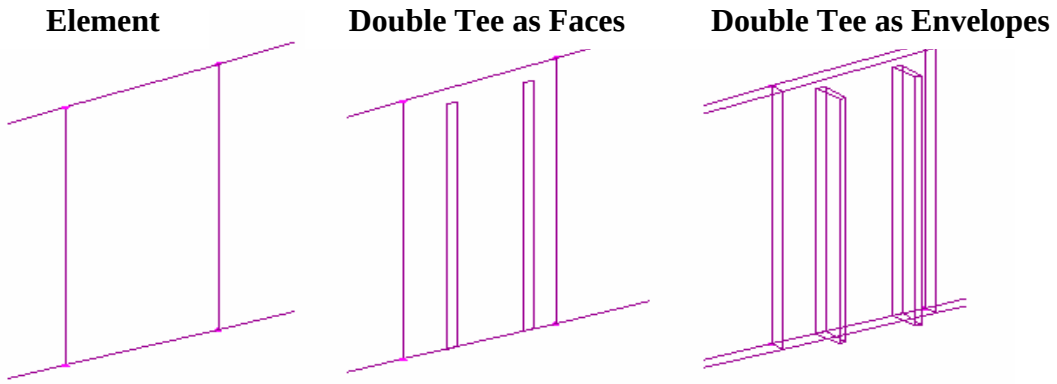


Figure 7 - 1

Change Double Tee Parameters

Just as the Change Joist Parameters dialog box lets you deal with the extra internal structure of a joist, the Change Double Tee Parameters dialog box (**Figure 7 - 2**) enables you to adjust the parameters of the Double Tee.

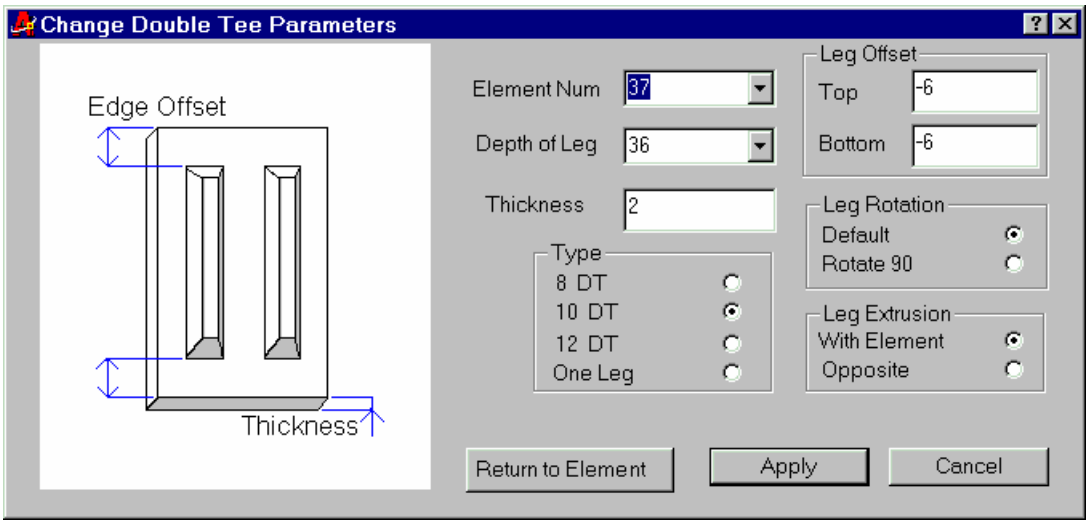


Figure 7 - 2

Functions of the Double Tee Dialog Box

- **Element Num** - Identifies the selected elements. This is a drop-down box that will display all the elements in a current selection set. The other text boxes on the dialog box will display the information about the currently viewed number. Changing the number in the **Element Num** box can change the display.

- **Depth of Leg** - This item indicates the distance from the back face of the element to the front of the leg or legs. This is a drop-down box that will permit depths from 12 inches to 48 inches for Double Tees and Single Leg Tees.

- **Thickness** - This is the depth of the double tee flange, based on the thickness of the original element. This can be any value in decimal inches.

- **Type** - This selects the standard double tee width that establishes the spacing of the legs. When the element is transformed to a double tee, the width is measured and the distance between the legs is set. As an example, if the width of the double tee element is eight feet or less, the program sets the leg spacing at four feet between center and sets the type as an eight-foot double tee. Greater than eight foot and the type is set as a ten-foot double tee with five feet between centers on the legs. Greater than ten feet is a twelve-foot with six feet between the legs. Any double tee can be set to have a single leg, which will be centered on the face.

- **Leg Offset** - The top or bottom of the double tee legs can be made shorter or longer than the edges of the element by changing this value. The legs will take their position from the edge directly over the center of the leg, even if the edge is sloping or not square with the sides of the double tee.

- **Leg Rotation** - In the event that the double tee legs are horizontal (or, in a roof double tee, simply run the wrong way) this enables you to rotate the legs 90 degrees.

- **Leg Extrusion -** This enables the user to cause the legs to be on either face of the element. If you have created an element and it is where it should be when the envelope mode is on, but the legs are inside when they should be outside, this will reverse the legs without making any other changes to the position of the double tee.

CHAPTER 8 - Creating Grid Lines and a Material Report

The objective of this chapter is to help you use the following functions:

- Create Grid Lines
- Modify Grid Lines
- Create Material Report

Create Grid Lines

In ***Structural Desktop,*** you can create a grid line that is tied to a node from your analytical file, or to any other point within the model. Each relevant grid line draws itself in every drawing extracted from the model, and dimensions are provided automatically across the grid lines and between them. Our program permits engineers to place grid lines where they want them, and the associated dimensions are generated on all 2D drawings automatically.

On the menu, select Add a Grid Line. ***Structural Desktop*** will prompt you to **Select start point of grid line** on the command line. You may enter a point by coordinates, or pick a point on the screen. We suggest that you use the Centroid/Faces display if you wish to pick a point graphically with snaps on, so that edge lines of an envelope or solid do not interfere with your selection.

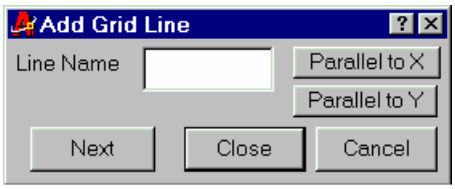


Figure 8 - 1

The dialog box in Figure 8 - 1 **will appear. You must click in the dialog box on the** Parallel to X **or** Parallel to Y **button to indicate the orientation of the grid line in terms of a plan view.**

To enter a value to be used as a label for the Grid Line, you must now click in the text entry box. Do so, and enter a number, letter, or brief label. You click on the "Next" button to enter the next Grid Line. You should have a grid line that extends generally from one side of the model area to the other, and has a label on the left-hand side or top side. The text will be 1 1/2 times the size of other text on the model, set on the Preferences **dialog box.**

The program will prompt you to select another start point. You may do so, and continue to enter labels, as long as they are of the same orientation. Click the "other" orientation button to change the direction of the Grid Lines. When you are through, click the Close **button.**

If you accidentally click Next and you are through, the program is still waiting for a point to be picked by you. Press **[Esc]** to exit the function, or click anywhere on the graphics screen and then click **Cancel** to exit.

The grid lines that you have entered will be printed on the **!SD_Gridlines** layer on every drawing you create from this model. Also, when you began to edit gridlines, the Display Options box for gridlines was automatically checked for you. You can turn off the grid line display from this dialog box at any time, but they will still print on your final drawings.

Modify Grid Lines

This menu item brings up the grid edit dialog box in **Figure 8 - 2**. To change a coordinate of a grid line, click on that coordinate number to edit it.

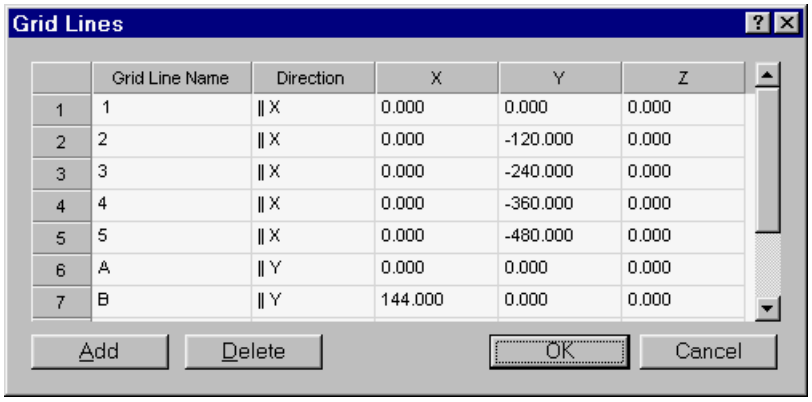


Figure 8 - 2

You may delete a grid line by clicking on the number value to the left of the grid line label or name and clicking the Delete button. When you have made changes you are satisfied with, click the OK button, or click the Cancel button to exit without your changes taking effect.

Pressing the **Add** button on this dialog box returns you to the function for creating a new grid line, as outlined above.

Create Material Report

Periodically through the adjustment of the members and elements from analytical lengths to true-cut lengths, you may wish to activate the Material Report function. This function will create a complete Material Report file and open a window over your AutoCAD session of the notepad editor to let you see the file's contents. The first part of the file at the top is intended to be a brief report of the status of members, called a Preliminary Identification Report. It details how many of each type and such members or elements that may be missing data, and can be accessed as often as you wish. An example of this part of the report follows on the next page.

```
PRELIMINARY IDENTIFICATION REPORT
-------------------------------------------------------------------
3 of the Rectangular Prism type.        Total length =    15' 0"
3 of the Cylindrical Prism type.        Total length =    15' 0"
3 of the Wide Flange type.              Total length =    15' 0"
3 of the Angle type.                    Total length =    15' 0"
6 of the Channel type.          Total length =    30' 0"
3 of the Tube type.                     Total length =    15' 0"
3 of the Double Angle type.             Total length =    15' 0"
3 of the Reverse Angle type.            Total length =    15' 0"
6 of the Pipe type.                     Total length =    30' 0"
3 of the Double Angle (Short) type.     Total length =    15' 0"
3 of the Steel Tee type.                Total length =    15' 0"
3 of the Cee Section type.              Total length =    15' 0"
3 of the Zee Section type.              Total length =    15' 0"
-------------------------------------------------------------------
All members have been identified
-------------------------------------------------------------------
-------------------------------------------------------------------
8 Valid Elements found
                    Total Volume                72.917 cubic-ft
                    Total Weight                16603.604 pounds
-------------------------------------------------------------------
-------------------------------------------------------------------
```

As you can see, this is a rough report. You may edit it, or discard it, when you prepare a final Material Report.

The next section is denoted in simple text format by the title:

```
                        Bill of Materials
                  STAAD SPACE ALL SIMPLE MEMBERS
```

The second line is taken from the top line of your analysis file. This is to identify the project or section for your convenience.

The program proceeds from that point to report the members and elements. The members are sorted as to type, variant within that type, and length within that variant. Each report sums and separates all the members of each type of a certain length, followed by a second report that combines and totals the lengths of each type. An example from an actual project heads the next page.

```
-------------------------------------------------------------------
        Total Length of Wide Flanges =        5734' 5 1/8"
        Total Weight of Wide Flanges =                      147973.245
-------------------------------------------------------------------
        Wide Flange Summary
-------------------------------------------------------------------
        47      W6x9          234' 3 1/4"   2108.438
        5       W6x25          86' 0"       2150.000
        1       W8x10        8' 6"    85.000
        2       W8x31         36' 0 15/16"  1118.422
        107     W10x12        1481' 4 1/8"  17776.125
        10      W10x33         149' 5"      4930.750
        100     W12x14        1643' 2 5/8"  23005.063
        1       W12x16         25' 0"       400.000
        9       W14x22         221' 2 3/4"  4867.042
        13      W16x26         284' 0 7/16" 7384.948
        1       W16x31         30' 0"       930.000
        1       W16x36         33' 0"       1188.000
        14      W18x35         372' 5 3/4"  13036.771
        67      W18x40         289' 8 15/16"11589.792
        1       W18x50         25' 0"       1250.000
        7       W21x44         227' 5 3/16" 10007.021
        3       W24x55         111' 0"      6105.000
        15      W27x84         476' 8 1/8"  40040.875
-------------------------------------------------------------------
```

The detailed report that preceded this is too long to include here, but if each of the 5 W6x25's listed above had a different length, that length would rate that member a separate entry. Only identical lengths are grouped in the first report.

The material report is saved in a file called MatRep.TXT before you ever see it. From our editing program (SDEdit.exe) you may save it as any text-file name you wish for later import into the Word Processor of your choice (Notepad, Word, WordPerfect, etc.) At that time you may delete or retain whatever part of the Material Report you deem useful.

It is useful to note that the display of the data in the Material Report looks quite different when viewed using different font styles. A non-proportionally spaced font, such as FIXEDSYS will display the columns in a more readable alignment. FIXEDSYS is available on most platforms, and the SDEdit program that is linked to ***Structural Desktop*** will permit change of font specifically for this reason. You should also note that text files such as this material report can be accessed through spreadsheet programs.

CHAPTER 9 - Creating Drawings

The objective of this chapter is to help you use the following functions:

- Create 2D Drawings
  - Create Plan View
  - Create Elevation X-Z
  - Create Elevation Y-Z
- Create 3d Drawings

Create 2D Drawings

The **Create 2D Drawings** menu item in ***Structural Desktop*** will display three choices as shown in **Figure 9 - 1**. The three choices can be thought of as coming from the basis of elementary drafting.

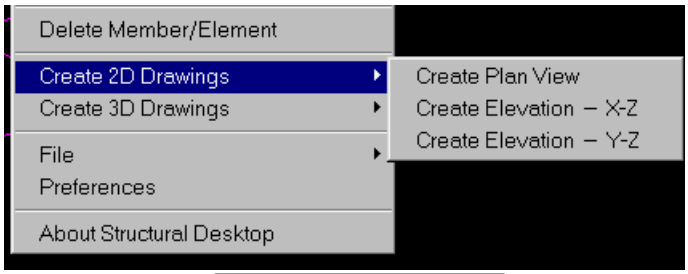


Figure 9 - 1

They enable you to draw the top, front and side views of your structure, or of any part of your structure, that you want to have drawn in a regular, 2 Dimensional drawing. These commands open a new window in your AutoCAD session and draw two-dimensional representations of your structure in that window. The more accurate your model, the less clean-up and modifications have to be made to these extracted drawings before they are checked and released.

The drawings that are produced are designed so that you may make them available to anyone who uses AutoCAD or a program compatible with AutoCAD. Other parties do not have to have a copy of ***Structural Desktop*** to take full advantage of an electronic copy of your drawings. If another group or engineer does have a copy of ***Structural Desktop***, you can send them a hold file and they will be able to take full advantage of your work. The output drawings must be saved as drawings, but may be saved in any drawing format your version of AutoCAD permits.

The drawings are drawn quickly enough that, if you check them and find that something was left out, you can simply close that drawing without saving it, make the necessary changes, and create the drawing again. Each member drawn in a 2D drawing has the name of the member...as you have either modified it or not...as it appears on the screen when you display **Member Names** and in the **DESC** field of the **Change Member Parameters** dialog box. Each name will be on the same layer as the member to which it belongs. You can change all of the text layers in AutoCAD, or leave them as they are.

The intent of ***Structural Desktop*** is to convert design information into as complete a drawing as possible. ***Structural Desktop*** does enable you to deal with beams instead of lines showing pictorially correct relationships of members.

Create 2D Drawings - Create Plan View

When you click on the **Create Plan View** menu item in the ***Structural Desktop*** Shortcut Menu, you are prompted to **Select Objects:** at the command line. **Create Plan View** will draw all, or any part, of your model with a Z value of zero and in a flat representation of each of the members. When you have indicated to AutoCAD that you are through with the selection process, the program will create a new drawing window in AutoCAD. The program draws each member selected on the layer that you have placed it on, and will create a text entity for that member on the same layer. These simple text entities can be moved, re-sized, or modified through standard AutoCAD text management tools.

You may review this drawing, and save it using the AutoCAD menu **SAVE AS**. You may keep this window open to refer to as you create other drawings, but we recommend that you close the window in order to conserve resources and optimize the speed of your computer.

This drawing can serve as an XREF drawing to be incorporated into a final drawing. If you proceed in that fashion, then later changes to the model can be propagated to a new plan drawing, and that new plan drawing can replace the former in the final document the next time it is opened. For further discussion on this subject, refer to the AutoCAD manual and the excellent texts that are available for improved efficiency with AutoCAD.

Additionally, use of paperspace to isolate sections from the drawing at more detailed scale factors will enable you to create details that are, in fact, part of the overall plan and drawn from members that are accurately sized. You do have to complete the details by hand, but the relationships of the members are correctly drawn for you.

One advantage you will find that you have using ***Structural Desktop*** is that the coordinate system is preserved from the analytical model to the ***Structural Desktop*** Model to the final drawing with just a little forethought on your part.

Create 2D Drawings - Create Elevation X-Z

When you click on the **Create Elevation X-Z** menu item in the ***Structural Desktop*** Shortcut Menu, you are prompted to **Select Objects:** at the command line. **Create Elevation X-Z** will draw all, or any part, of your model with a Z value of zero and in a flat representation of each of the members. When you have indicated to AutoCAD that you are through with the selection process, the program will create a new drawing window in AutoCAD. The program draws each member selected on the layer that you have placed it on, and will create a text entity for that member on the same layer. These simple text entities can be moved, re-sized, or modified through standard AutoCAD text management tools.

The difference in this drawing from the same drawing that would be created from **Create Plan View** is that the objects selected are drawn so that the X-axis in the drawing is the same as the X-axis in the model and the Y-axis in the drawing is the Z-axis in the model. Joists seen from an end-on view will draw only the top and bottom chords and the bearing. Seen from the side, they will be drawn in 2D as a pictorial representation of their actual side view. Grid Lines parallel to the Y-axis will show up at the X-coordinate value for that Grid Line and will be vertical, dimensioned (using the current default dimension style in your template for new AutoCAD drawings), and labeled. Grid Lines parallel to the X-axis will be ignored in an X-Z elevation, just as you would expect.

You may review this drawing, and save it using the AutoCAD menu **SAVE AS**. You may keep this window open to refer to as you create other drawings, but we recommend that you close the window as soon as it is convenient in order to conserve resources and optimize the speed of your computer.

Create 2D Drawings - Create Elevation Y-Z

When you click on the **Create Elevation Y-Z** menu item in the ***Structural Desktop*** Shortcut Menu, you are prompted to **Select Objects:** at the command line. **Create Elevation Y-Z** will draw all, or any part, of your model with a Z value of zero and in a flat representation of each of the members. When you have indicated to AutoCAD that you are through with the selection process, the program will create a new drawing window in AutoCAD. The program draws each member you have selected on the layer that you have placed it on, and will create a text entity for that member on the same layer. These simple text entities can be moved, re-sized, or modified through standard AutoCAD text management tools.

The difference in this drawing from the same drawing that would be created from the other two 2D drawing modes is that the objects selected are drawn so that the X-axis in the drawing is the same as the Y-axis in the model and the Y-axis in the drawing is the Z-axis in the model.

Again, Grid Lines parallel to the X-axis in the model will show up at the Y-coordinate value for that Grid Line (but projected to the X-axis) and will be vertical, dimensioned (using the current default dimension style in your template for new AutoCAD drawings), and labeled. Grid Lines parallel to the Y-axis will be ignored in an Y-Z elevation.

You may review this drawing, and save it using the AutoCAD menu **SAVE AS**. You may keep this window open to refer to as you create other drawings, but we recommend that you close the window as soon as it is convenient in order to conserve resources and optimize the speed of your computer.

Create 3d Drawings

Figure 9 - 2 at right displays the menu for 3D drawings. The first two drawings contain 3D solids and are created for two basic purposes. You can use them to create hidden line or rendered drawings, and you can use advanced AutoCAD solids tools to slice cross sections of the solids at angles off of the orthogonal.

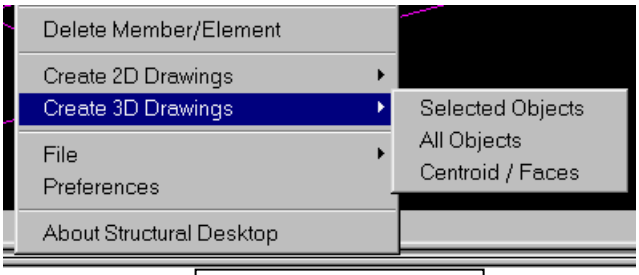


Figure 9 - 2

For formations or details that are not in line with the X or Y axis of your model, these sections provide a measure of utility to create plans, elevations, and true-length views that are beyond the scope of ***Structural Desktop*** itself. In this way, ***Structural Desktop*** also, enables you to extend its usefulness in areas of unique or unusual designs.

Once you have extracted a 3D model from AutoCAD and saved it as a drawing, it is no longer a ***Structural Desktop*** model, and is subject to all AutoCAD commands and functions. You can delete, edit, add and share this drawing as a pure AutoCAD drawing.

- **Selected Objects -** This menu item will prompt you to **Select Objects:** and will create a 3D model in a new drawing window. This model will contain only those objects you have selected, the Grid Lines for the model as reference lines, and layers as you have denoted them.

- **All Objects -** Every object that has not been expressly deleted from the database or that is not fused into another member will be expressed in a 3D model with Grid Lines and layers as you have set them. You do not have to select members or elements for this command.

- **Centroid/Faces -** The objects created with the two commands above are created without centroid lines or 3D Faces. If you have any need of these objects as reference points for your structure, this command will extract them, stripped of the additional data required to interface with ***Structural Desktop,*** and will provide them in a separate 3D model.

CHAPTER 10 - Other Functions

The objective of this chapter is to help you use the following functions:

- AutoCAD Layers with ***Structural Desktop***
- AutoCAD Stretch and Trim Commands
- AutoCAD Move and Rotate Commands
- AutoCAD Copy and Array Commands

AutoCAD Layers with *Structural Desktop*

AutoCAD permits you to place any AutoCAD entity that can be drawn or seen on the screen on a **Layer**. Layers are useful for manipulating what you do or do not see on the screen at any given time, and can be selectively manipulated in different paperspace viewports to hide or reveal different objects in different drawings when you plot.

Structural Desktop enables you to use the AutoCAD layering system with the ***Structural Desktop*** model. When you first begin an initial session of ***Structural Desktop***, all of the annotation and objects are displayed on default layers created by ***Structural Desktop***. It is useful in working with the model for you to create new layers for the members and elements in ***Structural Desktop***. These layers will permit you to turn on and off different groups of members, thus making the model easier to work with.

The layers you choose and implement will be saved in your HOLD file whenever you wish to interrupt your work for a while, and will be replicated precisely (by name) in every drawing extracted from the model through the ***Structural Desktop*** process.

Creating New Layers

AutoCAD provides a number of toolbars of which ***Structural Desktop*** takes advantage. **Figure 10 - 1** below shows the **Object Properties** toolbar, which deals with Layer information. Two controls we can use to manipulate layers are the **Layer Button** and the **Layer Control Drop-Down List**.

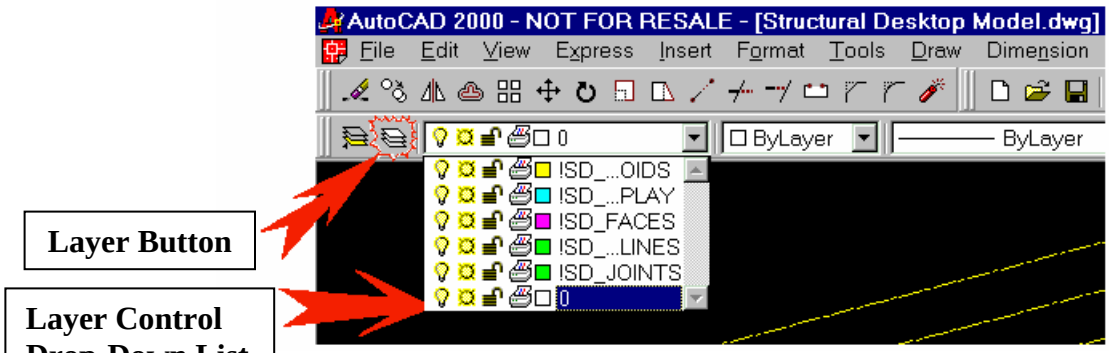


Figure 10 - 1

If you do not have the **Object Properties** toolbar active in your ***Structural Desktop*** session, you should consider adding this toolbar.

The AutoCAD **Layer Property Manager** dialog box is shown in **Figure 10 - 2** below. This is accessed by clicking on the Layer Button, typing "**Layer**" or "**LA**" at the command line, or by using the AutoCAD menu item "**Format**" and selecting "**Layer**".

The layers shown in the dialog box are those that ***Structural Desktop*** requires to operate. You should never remove or modify the names of these layers.

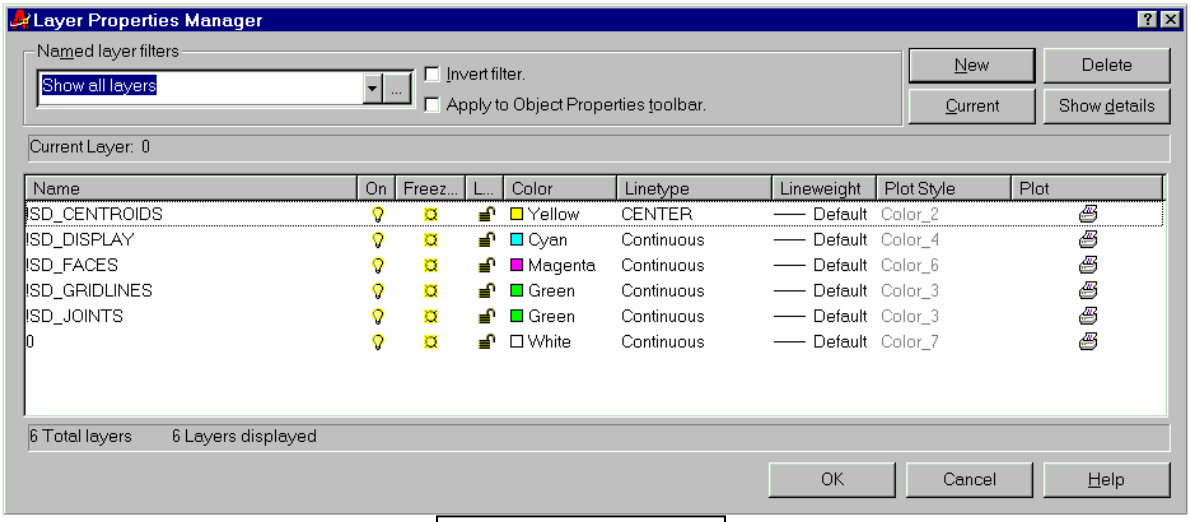


Figure 10 - 2

Clicking the **New** button in the dialog box above enables you to type a new layer name. Any layer name that begins with a number or a letter will be displayed below the AutoCAD layer 0 (zero). The ***Structural Desktop*** layers should therefore be separate from and not interfere with easy reading of your own layer names.

Double clicking on any of the properties of any layer permits you to edit those properties, such as choosing a new color for the layer. If you are operating AutoCAD with a white screen background, for example, you may prefer a color other than yellow for the !SD_Centroids layer.

Changing the Layer of an Object

Note: We suggest that you create your new object layers from the top of the structure to the bottom. This arrangement allows you to turn off each layer after it is completed, making it much easier to locate and select the objects in the next layer.

Using the "grip" method within AutoCAD is a very quick way of changing the layer of an object. As you click on each member with the left mouse button, or otherwise select members, please note that members must be selected by clicking on either the centroid

line or some line of the envelope of that member. You will know a member is selected because it will acquire "grip points" and the display of all lines in that member will display as dashed lines.

In the **Layer Control Drop-Down List (see Figure 10 - 1)**, select the layer by clicking it with the left button on the mouse. The grip-selected Members will transfer to that layer. When they do, they change color to match the color of the layer upon which you have placed them. You can check the color to make certain you have put them on the correct layer.

To freeze an AutoCAD layer, find the second icon from the left in the **Layer Control Drop-Down List**, which is a small "sun". By clicking on this "sun" icon, it changes to a "snowflake". AutoCAD will present a dialog box telling you that selected objects are being frozen; click "**OK**", and the objects will no longer display that are on that layer.

Other methods of changing the layer of an object include the command line command Change or the command Chprop. Details about these commands are available in the AutoCAD "Help" files, and in the manuals. For the purpose of this manual, any command that changes the layer of a centroid line in a ***Structural Desktop*** model will reach into the database and effect a permanent change of the member the line represents. This change will be retained from one ***Structural Desktop*** session to another through the vehicle of the Hold file.

The AutoCAD Stretch and Trim Commands – Members and Elements

The Stretch Command in AutoCAD permits you to encircle one end of a line with a bounding box and drag that end to a new location. For the sake of precision, you can stretch using numeric input or by using snap points within your drawing to locate the start and end relative locations of the stretch. The Trim Command uses the crossing of another entity to indicate a point at which the Member should be truncated.

Using the stretch or trim command with ***Structural Desktop*** will cause the member or element to update the offsets related to it and will cause the member to change to match the stretched centroid. Stretching anything other than the centroid of a member or the 3d Face of an element will NOT cause a permanent change, unless the centroid or face is also stretched. 3D Faces cannot be trimmed.

If you perform a stretch or trim command and the member or element looks questionable, you may force Structural Desktop to perform a full update at any time by clicking on the Display Options menu item and clicking OK on the Display Options dialog box.

The effects of using the Stretch command on Members and Joints with the [MOVE ALL] menu selection active are covered in the next section.

The AutoCAD Move and Rotate Commands

With the [Other Options] menu item set to [Adjust Members/Elements]
The Move Command or Rotate Command in AutoCAD will change the offsets of a member or element to reflect whatever changes you apply. It should be noted that large changes in these values will invalidate the possibility of extracting a meaningful analysis file from the ***Structural Desktop*** database should you wish to do so later. For large moves, consider creating new members or elements and deleting the current ones.

For small moves, however, the move command is a quick way to position an entire member or element, especially if you have reference points from that object or other objects to use with the snap option.

It should also be noted that, due to the high degree of precision in AutoCAD, any rotation of an object in 3 dimensional space would almost certainly result in very long decimal fractions for the offset values. These long decimals may be inconvenient to work with.

For clarification, ROTATING a member around the centroid will NOT update the Beta Angle value. This must be done through the Change Member Parameters dialog box, or to a 2D or 3D object in an output drawing in plain AutoCAD.

With the [Other Options] menu item set to [Move All]
The Move Command or Rotate Command in AutoCAD will move any Joints, Members and/or Elements that are selected. As long as Joints and Members are moved or rotated together, the SDT database will update the coordinates of the Joints as they will be written to an analytical file, and maintain relative offsets between the Joints and Members /Elements. Selecting a group of members and joints with this command option active will also permit you to Stretch a group of members and joints together. In order to reposition a joint using stretch, the bounding box must completely enclose the circle that represents the joint. Only Move, Stretch and Rotate commands using the World Coordinate system are permitted.

It cannot be stressed strongly enough that using the Viewing Commands from SDT to hide Members/Elements, the Display settings to turn off Joints or Members/ Elements, or the Layer options within AutoCAD to freeze layers with Joints/Members/ Elements make it impossible for these unseen model components to be selected. This can result in movements that will separate your structure in unanticipated ways.

Until you are familiar with the consequences of setting this preference to [Move All] you should proceed with caution and back up your Hold file frequently.

The AutoCAD Copy, Mirror and Array Commands

The Copy Command in AutoCAD can be used with ***Structural Desktop*** members and elements, but the results are unproductive in returning to or producing an analytical file. The members and elements copied will still have the same joints and properties as the member copied. This can be convenient for quick operations that are solely concerned with creating a drawing, but can be confusing if you are tracking member and joint numbers or intend to export the model to an analysis program.

Using the Copy Command in AutoCAD to produce multiple copies of a single member is also unproductive. The SDT database is only signaled to produce a new member at the final position of the multiple copy command. There is no limit, however, to how many lines you can create using the multiple copy command and then turn these lines into members afterwards.

For large changes and/or numbers of new members or elements, we suggest that you create lines and copy or array these lines **BEFORE** they are transformed into Structural Desktop members, or create Faces and copy or array them **BEFORE** changing them to elements.

Due to the nature of the array command, it will not update the database properly. An attempt to array a member will only succeed in creating the **last** member in the array specified. You may, however, array as many lines that you intend to represent members as you wish, and **then** change them to members using the SDT command.

For information purposes, the mirror command will behave as the copy command if you do NOT elect to erase originals. It will create a new member that is the mirror image of the original, but with the same joints and offset from those joints to the new position. The mirror command will perform as the move command if you do elect to delete the original member, offsetting the member from its original position.

Summary of Use of AutoCAD Commands

- Changing the layer of a Centroid Line or 3D Face will place the member or element, and all manifestations of it, on that layer.

- Stretch, Trim and Move, used with ***Structural Desktop***, work correctly when used on centroid lines or faces that represent elements. Trim will not trim 3D Faces in AutoCAD.

- When [Move All] is selected from the [Other Options] menu item, you can move or rotate part or all of a structural model and the Joints will move and update accordingly. Care must be taken to actually select all of the Joints or Members/ Elements that you wish to move. Using the Stretch Command and completely selecting the circles representing the Joints will permit you to slide an entire wall of a bay with one action.

- Rotate, Copy, and Mirror have limited direct use within the ***Structural Desktop*** session, and are more useful applied to lines or faces. Work with your lines and faces to create the stick form of your model; then they can be used to create members and elements.

- Array works only with lines or faces to be later turned into members or elements.

- If you have any question about a function, **save a hold file** before you try it. This will prevent the loss of data in the event that the function causes AutoCAD to malfunction.

STRUCTURAL DESKTOP
Appendices

Appendix

Appendix A - Command Line Equivalents

All commands have the prefix SD to associate them with ***Structural Desktop***. Commands may be entered in upper or lower case, or combinations. Each command is followed by a brief expansion of the command as a memory aid (in parenthesis) and an explanation of the command usage. Except where noted, these commands are available on the Shortcut Menu (right-click) under the ***Structural Desktop*** menu heading.

File Oriented Commands:

This block of commands allows you to open design files and check or edit those files. After you have made changes in ***Structural Desktop*** Object Model data, you save your changes, retrieve your model for later sessions, and derive a text file **Material Report** with a summation of the major structural components of your model.

SDLOAD (LOAD Staad file)
Provides a standard file dialog box to load a STAAD input file (*.std) either from a local drive or mapped network drive.

SDEDSTAAD (EDit **STAAD)**
Displays the original STAAD file that was used to create the current model in a popup session of Notepad. This command is only available during the first session with a STAAD file, and is not available when GT STRUDL files or HOLD files are loaded. The function assists the user in distinguishing specific project files and permits quick editing. Changes made to the STAAD file will only apply if the STAAD file is saved and loaded into ***Structural Desktop*** in a new session.

SDLDBX (Load **DBX)**
Loads the GT STRUDL input file set from a directory. The command expects the directory to contain the files listed below, which are the default names for the files created by GT STRUDL for interface to programs such as ***Structural Desktop***:

STDBX02
STDBX03
STDBX20
STDBX22
STDBX23 (if ELEMENTS are present)

SDLOADRISA (LOAD RISA 3D file)
Provides a standard file dialog box to load a RISA input file (*.r3d) either from a local drive or mapped network drive.

SDLOADCSI (LOAD CSI SAP 2000 file)
Provides a standard file dialog box to load a SAP 2000 input file (*.s2k) either from a local drive or mapped network drive.

SDSAVE (SAVE)
Saves a ***Structural Desktop*** HOLD file. ***Structural Desktop*** does not save the database data with any drawing file. Opening a drawing file in ***Structural Desktop*** manually is not recommended. ***Structural Desktop*** saves the coordinates, joint, member, and element incidences, offsets, parameters, newly added members or elements, and the AutoCAD Layer information for these objects in a HOLD file.

Hold files are complete, and can be shared with other ***Structural Desktop*** or ***Structural Desktop*** **LT** users. Only a hold file can retain changes that you make to the original imported database information. These files can be named any valid name that AutoCAD would recognize for a drawing name, but with the HLD extension.

SDLHOLD (Load **HOLD** file**)**
Used to Load a Hold file. Displays a dialog box for browsing and selecting from the files on your hard drive or mapped network drive.

SDLOADAUTOSAVE (LOAD AUTOmatically **SAVE**d Hold file**)**
Structural Desktop automatically saves a hold file every five minutes on your hard drive in the Backup directory under the Structural Desktop directory. This file is named StructuralDesktopBackup.Hld. Entering this command, or clicking on the menu item **Load Autosave Backup Hold File** will return you to the state you were in when the last Autosave was completed. You should load this file into an existing model, not as a blank document. This is meant as a last-ditch "undo" for emergency use, and is not meant to replace periodic saving of your work at various stages of completion.

SDMAT (MATerial report**)**
Creates a material report for the current database and opens a NOTEPAD session for you to view the results. Data can be cut-and-pasted from this file, or the file can be saved-as any text filename of your choice and can be imported into virtually any word processor. By default, the file is created in the current directory under the name MatRep.tmp.

SDCTSTAAD (Crea**T**e **STAAD)**
Structural Desktop will create a STAAD input file from the current database. This file contains all joint coordinates, member incidences, element incidences, member properties, element thicknesses, and constant values. If members are FUSED in your Structural Desktop Model, the fused member will be ignored, but the sub-members that it is composed of will be written to the STAAD file with the current Member Properties, Constants, and Beta Angles. ***Structural Desktop*** does not write **Offsets** to a STAAD file because these are used primarily to adjust members to real-world relative positions; it does, however, read **Offsets** when they are included in a STAAD file.

Members created in ***Structural Desktop*** that rely on the **USER PROVIDED TABLES** are defined in terms of those table names. A section of a STAAD file created using these members is provided below. In order for these members to be used within STAAD III/STAAD.Pro, you must copy the relevant USER files from the directory on your hard disk or on the supplied installation disk to the same directory from which you intend to run the STAAD file created. The files are located in:

C:\Program Files\Structural Desktop\STAAD User Files

SDCTSTRUDL (Crea**T**e **STRUDL)**
Creates a GT STRUDL file from the current database. This file contains joint coordinates, member incidences, element incidences, member properties, element thicknesses, and constant values. Members created in Structural Desktop that rely on the USER PROVIDED TABLES are defined in terms of those table names. When you load the input file, you must specify the data file StructDT.ds which is provided on your hard drive in the directory

C:\Program Files\Structural Desktop\GT STRUDL User Files

SDCTCSI (Crea**T**e **CSI** SAP 2000 file**)**
Creates a CSI SAP 2000 file from the current database. This file contains joint coordinates, member incidences, element incidences, member properties, element thicknesses, and constant values. Members created in Structural Desktop that rely on the USER PROVIDED TABLES are not supported in SAP 2000.

SDABOUTSD (ABOUT Structural **D**esktop**)**
Version Information, Internet Link to ***Structural Desktop*** Web Site for updates, information and technical support. Phone and fax numbers for Structural Desktop Inc. as well as physical address are provided. The Security Code that you must have to perform the Unlock Code request when you first install ***Structural Desktop*** is available at any time on this dialog box.

View Oriented Commands:

This block of commands permits you to isolate members or elements into groups quickly. Rather than relying on Layering, you may select members or elements to view or hide temporarily, regardless of what layer they are on, and continue to remove members from view until you see only what you wish to see, the way you wish to see it.

SDVCOM (View **COM**ponents**)**
Displays the **COMPONENT TABLES** of all MEMBERS for which ***Structural Desktop*** contains data. Quick access to Steel Tables and other data.

SDVDB (View **D**ata**B**ase**)**
Creates a table of Joints, Members, and Elements with critical information for quick review. Members or Elements you have added to the database can be confirmed as to their status through this box. A quick, convenient summary.

SDMEM (**MEM**ber information)
You are prompted to select a single member and the information for that member will be displayed

SDVOPT (Viewing **OPT**ions**)**
Displays the **Display Options** dialog box to toggle different ways of viewing the objects. You can choose whether to display or not from the Joint, Member and Element classes, and to display the numbers associated with these objects. Members can display arrows to indicate direction, cross sections to show shape and orientation, and their section name. Grid Lines you create can be toggled on or off. The Centroid/Face display mode will always be active...envelopes are available to display the extents of a member or element, with special display of Joist or Double Tee data. Finally, solid modeling is available to check final parameters for a member or element. Editing is not advised in the Solid View Mode of ***Structural Desktop***; in fact, this mode turns itself off (returning to envelopes) before many of the ***Structural Desktop*** commands begin, in order to provide the faster processing of the data. NOTE: This does NOT refer to the 3D Solid Model available for final output and use in a regular AutoCAD session, only to the VIEWABLE model available while actually running ***Structural Desktop***.

SDVBS (View **B**y **S**election**)**
You are prompted to make a selection; selected members and elements are the only ones that will be visible until you select another **View** function such as **View All** or **View by Plane**.

SDHBS (**H**ide **B**y **S**election**)**
Each object you select within this selection will be selectively removed from visibility. This can be applied to previous selections to remove members that obscure what you desire to see and work with. Sequential selections will remove members without making any visible; to recover visibility of an object you must use the View All function.

SDVBR (View **B**y **R**ange)
A dialog box will appear that has two ranges, one for Members and one for Elements. Using the Member and Element numbers assigned from your design file and, subsequently, to fused or new members or elements, you may specify a single range of numbers to be visible. If you do not put a range into one of these boxes, the relevant objects remain visible...to turn off ALL of a type, use the checkbox on the **Display Options.**

SDVBP (View **B**y **P**lane**)**
You are prompted to select a point. This could be any point in the structure that you can snap to, a point entered by coordinates, or any other method used in AutoCAD. A dialog box appears with the coordinates displayed. You then select the X, Y, or Z coordinate to govern the selection set and a value for an envelope width. Finally, you select the criteria desired and click **[OK]**.

If you select the Y coordinate to govern, you will select everything at that Y coordinate, plus or minus the envelope width, along all X and Z values. If you select the criteria Completely Within Envelope, all the points of a member or element must be within the envelope for that object to be selected. The criteria At Least 1 Joint in Envelope requires that a single point (or more) of the endpoints of an object lie within the envelope of selection. Finally, the criteria Passes through Envelope will select any member that interacts with the envelope in any fashion, whether it has an endpoint within the envelope, all points within the envelope, or merely passes through the envelope. Whatever is selected through this process becomes visible and all other objects become invisible.

SDVALL (View **ALL)**
Sets all objects to visible and recreates the screen from the database information. This is the only way to recover objects once they are set to "invisible" by the ***Structural Desktop*** conventions.

Parametric Editing Commands:

These commands permit you to change the descriptions and parameters of objects in dialog boxes, and the objects in the database shown on the screen and in drawings will change to match your new information.

Members Only Commands:

SDFUSE (FUSE)
Command to unify two or more Members into a single new Member. This Member will be assigned the next available Member number that is higher than any existing Member or Element number. By unifying all of the sub-members of an analysis file BEFORE creating any solid models, the final size of the solid model is minimized (as compared to UNIONING small solids in AutoCAD to create a single solid object). Output drawings will reflect the unified member status only.

SDDEFUSE (DEFUSE)
Reverses the **SDFUSE** command, even in a later ***Structural Desktop*** work session. A **FUSED MEMBER** is thereby deleted and the original members, with their original data and offsets, are returned to you.

SDLTM (Line **T**o **M**ember**)**
This function permits you to select any number of regular AutoCAD LINES and will change them into a neutral MEMBER entity. The lines selected must be on AutoCAD Layer 0, any lines on other layers, or any other entities (including existing MEMBERS or ELEMENTS) will be ignored by this command. If the START or END of the line is within an inch of an existing JOINT, the MEMBER created will reference that joint, otherwise a new JOINT is created. As of Version 1.6, also changes selected faces into elements.

SDCMEM (Change **MEM**ber parameters**)**
Displays a dialog box for editing parameters of the MEMBER objects. In this box you may adjust offsets, change the length of the member, change the type, density, or beta angle of the MEMBER, and apply a description to be added to the MEMBER output drawings.

SDUNDO (UNDO)
Reverses last changes made with Change Member Parameters by one step backwards.

SDJP (Joist **P**arameters**)**
The joist type of members is a special case. They include additional information about the geometry of the simulated Joist to assist in more detailed drawings as output. The Joist Parameter dialog box permits viewing and changing data specific to these members.

Elements Only Commands:

SDFTE (Face **T**o **E**lement)
In order to create a new element from scratch, you must first create a 3dFace object on Layer 0. These objects are either triangles or four points describing a planar four-sided polygon. The points describing these objects are in a sequence, and the order of that sequence will determine the direction in which the thickness of the element will be expressed from the face. A RIGHT-HAND RULE applies, in that a counter-clockwise enumeration of these points in the order they are created (curling in the direction of the fingers of the right hand) will generate a direction determined by the thumb as being "UP". If the order is not around the outside edges of the 3dFace, but has edges that cross (referred to as a "Bow-Tie" due to the distinctive appearance) the ELEMENT created from this face will not be predictable. For viewing purposes, the user-created element is generated with a thickness of one inch until the user edits the object. Since Version 1.6, will also change any lines on layer zero that are not already members into members.

SDCELM (Change **EL**e**M**ent parameters**)**
Displays a dialog box for editing parameters of the ELEMENT objects to include thickness, offsets, density, and extrusion direction. You are limited to selecting up to 100 elements at a time for editing with this parameter box.

SDCT (Change to double **T**ee**)**
The user selects ELEMENT objects in ***Structural Desktop*** and transforms them to DOUBLE TEE objects. These can then be edited with the DOUBLE TEE tab of the CHANGE ELEMENT PARAMETERS dialog box. A 3dFace must first be converted into an ELEMENT by the user before he can change that ELEMENT into a DOUBLE TEE.

SDCHGDBLT (CHan**G**e **D**ou**BL**e **T**ee**)**
Displays the Change Double Tee Parameters dialog box to permit the user to edit internal structure of a Double Tee's description and display characteristics.

Other Editing Commands:

SDTOM (Top **O**f **M**ember - flat**)**
This command will manipulate the offsets of a large number of members for you. The user selects horizontal members of differing depths, and inputs an elevation (in inches). All members and elements selected will be adjusted so that the top of the member is set to that elevation. Horizontal coordinate values for the member endpoints are not changed. (Note - this function will ignore a perfectly vertical column in the event it is accidentally selected).

SDTOMS (Top **O**f **M**ember - **S**lope**)**
This command invokes a function to align the depths of members through offsets, similar to the TOS command above. However, TOSS requires a slope in the X or Y-axis and a 3-D point that the plane of the sloped roof passes through. Selected members will be adjusted to fit into this slope. This works best if the original file has taken slope into account in the location of the original member joints, and they are placed in the plane desired. Please see the description of this function for more information on this involved topic.

SDAGL (Add**G**rid**L**ine)
Begins the process for inserting grid lines.

SDGL (edit **G**rid**L**ines**)**
Permits viewing of all gridline data and editing of that data to change grid lines.

SDDELETE (DELETE)
Totally and permanently erases a Member or Element. If you want it back, you must create a new one, because it is gone.

Output Drawing Commands:

The intent of ***Structural Desktop*** is to take your design file and give you complete drawings of all the objects that file supports. The following commands will open a new window in your AutoCAD session and draw the requested objects in that window. It is then up to the user to check the drawing, save it as an AutoCAD drawing file under the name of his choosing, and close the drawing window before proceeding with further ***Structural Desktop*** functions.

3D Drawing Files

SD3DALL (3D ALL)
Creates a 3D solid model of the entire file, all Members and Elements. Similar to the display you see when you choose **Display Options**: Solid Mode, but there are no Joints, Centroids, Faces, or data associated with these objects as in ***Structural Desktop***. The file is strictly 3D solids for you or your user/client to do with as you wish.

SD3DSEL (3D SELected objects**)**
Prompts you to select MEMBERS and ELEMENTS, then creates a 3D solid model of the selection set.

SD3DCENT (3D CENTroids/faces**)**
Creates a 3d Model for export of Centroids, Faces, and Joints only. Centroids can be useful for the user to compare the original data file against subsequent outputs in ***Structural Desktop***, therefore we provide the user this access to the skeletal drawing.

2D Drawing Files

SDPLOTPLAN (PLOT PLAN)
Prompts the user to select MEMBERS and ELEMENTS to be included, then creates a 2-D Plan view of the selected objects with names and gridlines on appropriate layers.

SDPLOTXZ (PLOT Elevation **X-Z)**
Prompts the user to select MEMBERS and ELEMENTS to be included, then creates a 2-D Plan view of the selected objects with names and gridlines on appropriate layers.

SDPLOTYZ (PLOT Elevation **Y-Z)**
Prompts the user to select MEMBERS and ELEMENTS to be included, then creates a 2-D Plan view of the selected objects with names and gridlines on appropriate layers.

Appendix B - Member Information

This appendix contains information about each of the members supported in ***Structural Desktop***.

Prismatic Members - Rectangular

Rectangular Prismatic members are described in terms of, and can be modified by, depth and width.

The centroid of the member passes through the center of the rectangles or squares formed by the Start and End faces as shown to the right in **Figure B - 1**.

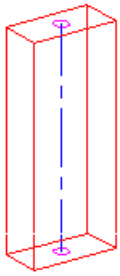


Figure B - 1

Prismatic Members - Cylinder

Round Prismatic members are described in terms of, and can be modified by, diameter.

The centroid of the member passes through the center of the circles formed by the Start and End faces as shown to the right in **Figure B - 2**.

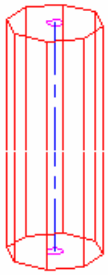


Figure B - 2

Wide Flanges, M, HP, and S Sections

W, M, HP, and S sections are described by a name in terms of nominal depth and weight per unit foot. They are defined by depth, width, thickness of flange, and thickness of web. Representations are exact in these dimensions but do not include fillets at corners.

The centroids of members with these shapes pass exactly through the center of the web as measured in both of the member axis as shown in **Figure B - 3**.

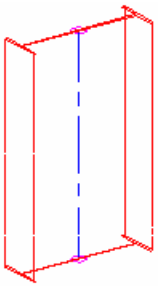


Figure B - 3

Tapered I-Shaped Sections

Version 2.0 of Structural Desktop introduces Tapered I-Shaped Members. These members are read from (or written to) a STAAD.Pro file with the relevant Member Properties in the following format:

MEMBER PROPERTY AMERICAN
1 TAPERED 18 0.375 6 12 0.5 8 0.475

This indicates:

Start Depth	**18**
Web Thickness	**0.375**
End Depth	**6**
Top Flange Width	**12**
Top Flange Thickness	**0.5**
Bottom Flange Width	**8**
Bottom Flange Thickness	**0.475**

This member would be displayed as shown in **Figure B - 4** through the relevant portions of the Member Parameters dialog box.

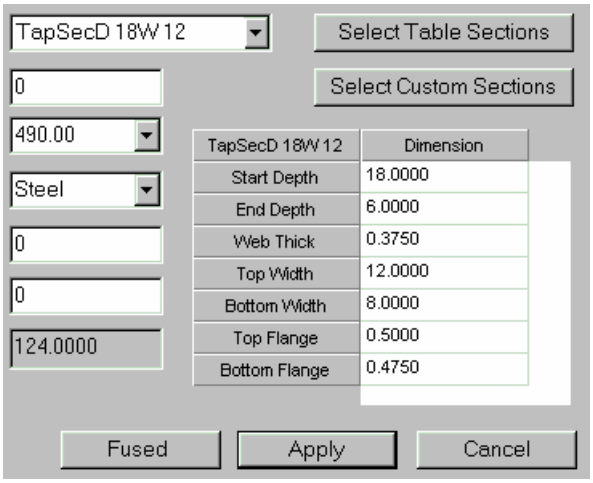


Figure B - 4

Note the "Select Custom Sections" button that is now displayed on the Change Member Parameters Dialog Box.

Selecting this button would display a new dialog, which is shown in **Figure B - 5,** at right. This dialog box comes up with all values empty. Inputting the values as shown with all the relevant data will create the same section, from scratch, in Structural Desktop or permit you to change the values prior to writing the file to STAAD.Pro.

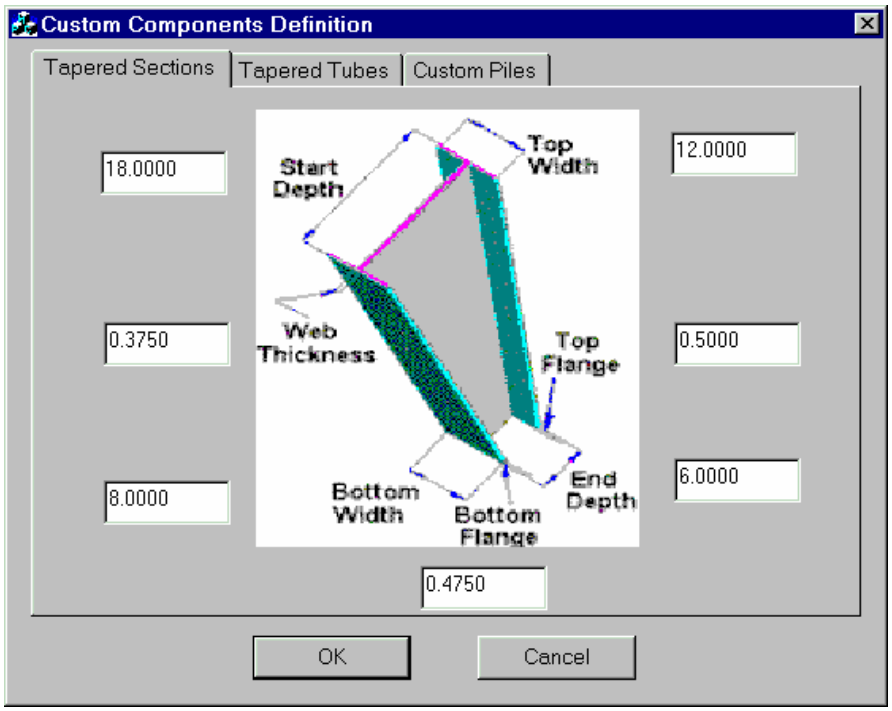


Figure B - 5

Finally, there is the representation of the Tapered I section in Structural Desktop graphically. This section appears as shown in **Figure B - 6** at right in the model. Note that the member Centroid is depicted as being at the center of the TOP FLANGE. For drawing purposes, the member is designed with the larger depth at the START of the member, and the member will rotate around the Top Center of the Top Flange when a Beta angle is applied.

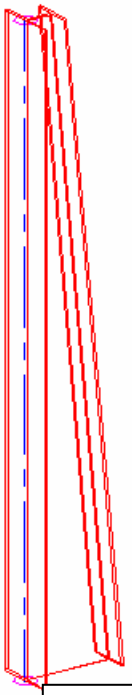


Figure B - 6

This representation presents the top flange flush with the analytical centroid line. Basing your positions of girts and purlins on the centroid of a tapered member means that there exist some configurations where the drawing will be ready to be converted into a model or drawn as two-dimensional output drawings without any offsets being applied.

These members will display in your Material Report with a name derived from their dimensions. Members will not be considered identical unless the lengths are identical in the Material Report. The weights of each member are calculated based on a calculated sum of the flanges and web.

Tapered Tube Sections

Selecting the second tab of the Custom Components Definition dialog box will permit you to enter values to create a Tapered Tube section, as shown in **Figure B - 7** at right.

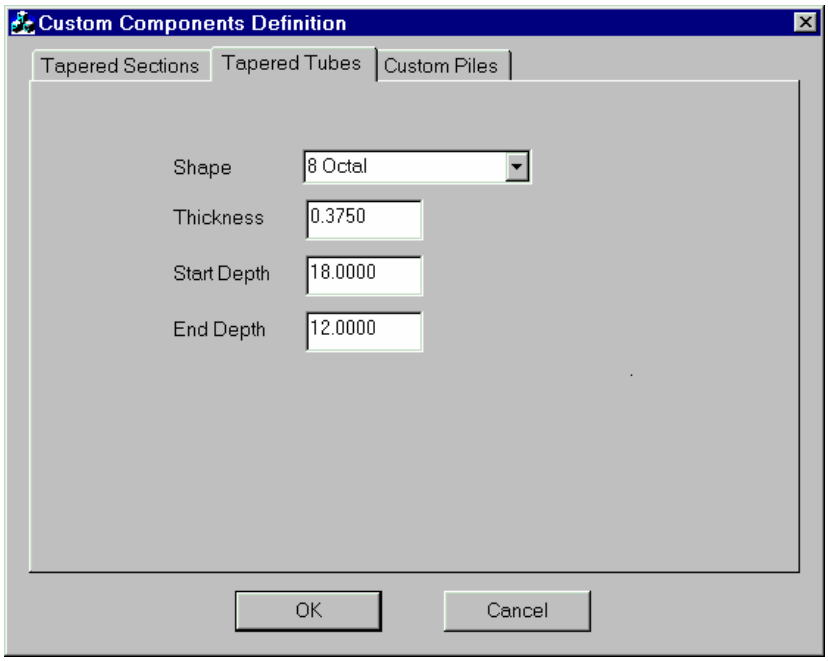


Figure B - 7

Custom Tapered Tubes are both read and written to a STAAD.Pro file. You have the choices of 4, 6, 8, 10, 12, and 16-sided tubes, as well as a round section and you specify the outside diameter at start and end, and the thickness of the tube. The centroid is through the center of the tube.

Tapered Tubes have an estimated weight which is calculated and provided in the Material Report.

Custom Pile Sections

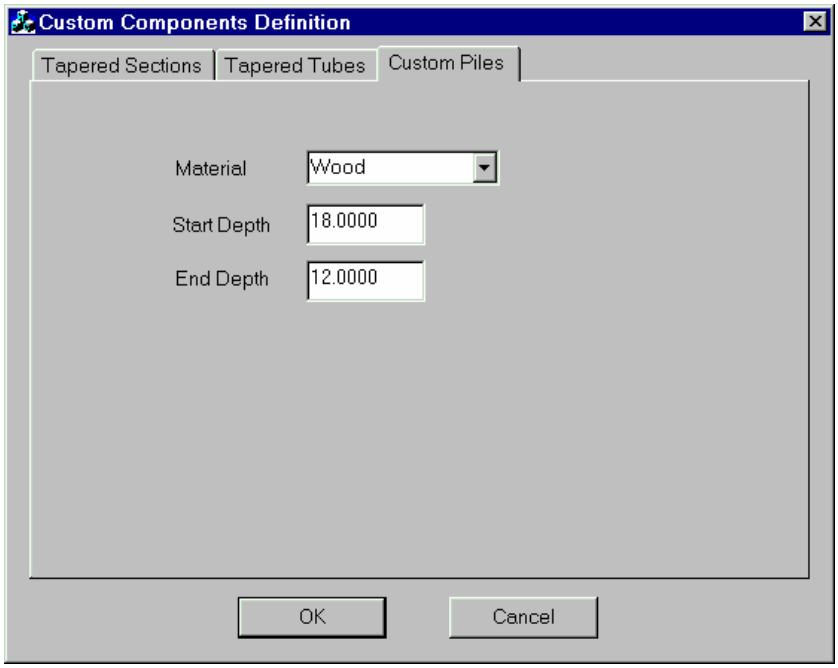


Figure B - 8

Structural Desktop now has a member that is not available for analytical use. Custom Piles are defined as a frustum from a cone. These members can be selected from Wood, Steel, or Concrete materials, and are specified by the Start and End Depth as shown in **Figure B - 8.** The members can be used in conjunction with cylindrical prisms to draw bell-bearing piers, by themselves, or in any place where a frustum of a conical section is required. The section will be drawn in 3D models, plans, and elevations and will be included in calculations for weight and volume of steel, wood or concrete in the Material Report.

Since there is no corresponding section, this member will be written to analytical files without a section being assigned.

Steel T Sections

T sections' names in STAAD are derived from the W or other section they are cut from, but in GT STRUDL the name is that from the Steel Manual. Structural Desktop converts T names to the Steel Manual convention.

T sections are defined by depth, width, thickness of flange, and thickness of web. Representations are exact in these dimensions but do not include fillets at corners. The centroid of members with these shapes pass exactly through the center of the web as measured in both of the member axis and as shown in **Figure B - 9**.

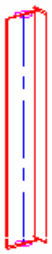


Figure B - 9

Angles & Reverse Angles
Angles and Reverse Angles are handled separately in Structural Desktop to permit alignment of angles with unequal legs to display properly, due to the "handedness" of such members. These members are lumped together for purposes of Material Reports. They are defined by depth, width and thickness.

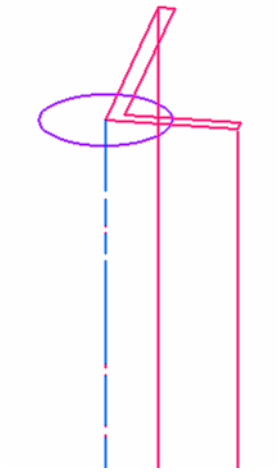


Figure B - 10

To assist in positioning, the centroid line creates an angle member as if the centroid represented the back corner of the angle, and is not in the position of the physical centroid (center of gravity) as shown in the steel tables. **Figure B - 10** shows the centroid in relation to its angle.

Channels and Miscellaneous Channels
Channels and Miscellaneous Channels are recognized and incorporated into ***Structural Desktop*** with the names given in the steel manual. These members are represented by their depth, width, thickness of flange, and thickness of web. Representations are exact in these dimensions but do not include fillets at corners.

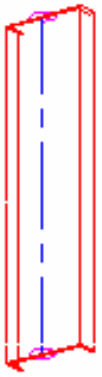


Figure B - 11

The centroids of members with these shapes have been set for ***Structural Desktop*** to pass exactly down the center of the back face as shown in **Figure B - 11**. This is to permit their analysis position to represent a location flush with another member on the back plane.

Hollow Steel Shapes - Square and Rectangular
Tube sections are defined by depth, width, and thickness. Representations are exact in these dimensions but do not include fillets at corners. Tubes are drawn with a single line indicating the outside surface. The centroids of tubes pass exactly through the center of the faces, just as with rectangular prismatic members (**Figure B - 1)**.

Double Angles
Double angles are represented by depth, width and thickness. In addition, a spacing factor is given to represent the clearance between the back legs. Double angles with short legs back-to-back and long legs back-to-back are handled separately within ***Structural Desktop*** to give you the flexibility to have these members drawn properly for you.

In both cases, the centroid passes through the space between the angles that make up the double angle, centered between the angles and at the center of the depth of the construct.

Pipe

Pipe can be represented either by inside/outside diameter or from the standard pipe tables. The centroid passes through the center of the pipe, just as with cylindrical prismatics (see **Figure B - 2**).

Light Gage (LGSI) C and Z Sections

The LGSI (Light Gage Steel Institute) C and Z sections are represented by a naming convention that is derived from the measurements and thickness of the gage of the material. The naming convention was designed to fit the requirements of analysis programs.

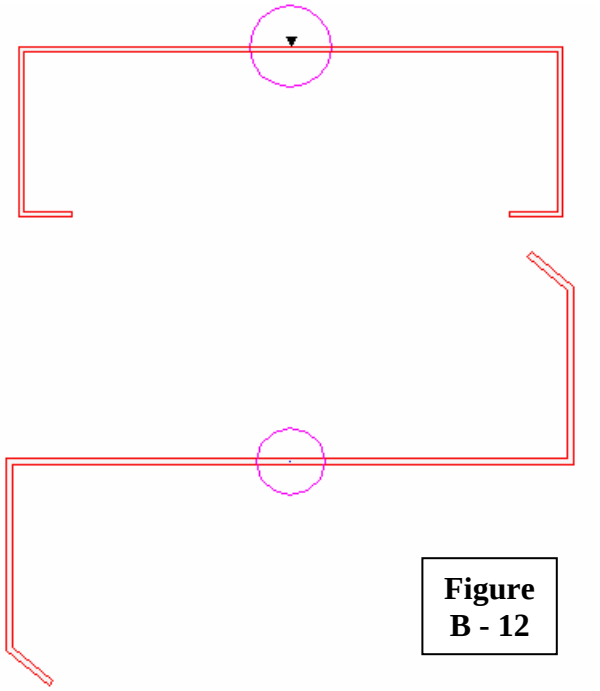


Figure B - 12

C sections are named by the letter **C** followed by the depth in inches, the letter **X**, the width and the gage of the material. If the width is a whole number the integer is shown; if a fraction the fraction is given without a decimal.
(e.g. a C section that is 4 by 2 and 16 gage is **C4X216** while a **C4X2516** has a depth of 2.5 inches.

They have a centroid that, just as in the case of the Channel members, passes flush with the back face of the member and is centered on the back face. This is to permit precise alignment of the member when arranging it for drawing purposes.

The centroid of the Z section is centered as to the depth of the member but is placed to be in the center of the thickness of the material.

Plan views of the two members are shown **Figure B - 12**.

Table B - 1 and **Figure B - 13** (next page) illustrate the measurements and designation nomenclature for the C Sections that are available for your use in the User Files.

Table B - 2 and **Figure B - 14** (on the following page) illustrate the measurements and designation nomenclature for the Z Sections that are available for your use in the User Files.

In the User file for RISA 3D, each of these is prefaced by the letters "**SD**" and an Underline. [**SD_C4X216**]

Light Gage C Members					
Designation	Depth	Width	H1	Gage	WT/FT
C4X216	4.0	2.000	0.84	16	1.89
C4X214	4.0	2.000	0.87	14	2.36
C4X213	4.0	2.000	0.90	13	2.89
C4X212	4.0	2.000	0.93	12	3.32
C4X2516	4.0	2.500	0.75	16	2.06
C4X2514	4.0	2.500	0.78	14	2.57
C4X2513	4.0	2.500	0.81	13	3.14
C4X2512	4.0	2.500	0.89	12	3.62
C4X416	4.0	4.000	0.75	16	2.68
C4X414	4.0	4.000	0.78	14	3.34
C4X413	4.0	4.000	0.81	13	4.09
C4X412	4.0	4.000	0.89	12	4.71
C6X2516	6.0	2.500	0.75	16	2.48
C6X2514	6.0	2.500	0.78	14	3.08
C6X2513	6.0	2.500	0.81	13	3.77
C6X2512	6.0	2.500	0.89	12	4.34
C6X416	6.0	4.000	0.75	16	3.10
C6X414	6.0	4.000	0.78	14	3.86
C6X413	6.0	4.000	0.81	13	4.72
C6X412	6.0	4.000	0.89	12	5.43
C7X2516	7.0	2.500	0.75	16	2.68
C7X2514	7.0	2.500	0.78	14	3.34
C7X2513	7.0	2.500	0.81	13	4.09
C7X2512	7.0	2.500	0.89	12	4.71
C7X416	7.0	4.000	0.75	16	3.31
C7X414	7.0	4.000	0.78	14	4.12
C7X413	7.0	4.000	0.81	13	5.04
C7X412	7.0	4.000	0.89	12	5.80
C8X216	8.0	2.000	0.75	16	2.68
C8X214	8.0	2.000	0.78	14	3.34
C8X213	8.0	2.000	0.81	13	4.09
C8X212	8.0	2.000	0.89	12	4.71
C8X2516	8.0	2.500	0.75	16	2.89
C8X2514	8.0	2.500	0.78	14	3.60
C8X2513	8.0	2.500	0.81	13	4.41
C8X2512	8.0	2.500	0.89	12	5.07
C8X3516	8.0	3.500	0.75	16	3.31
C8X3514	8.0	3.500	0.78	14	4.12
C8X3513	8.0	3.500	0.81	13	5.04
C8X3512	8.0	3.500	0.89	12	5.80
C8X416	8.0	4.000	0.75	16	3.51
C8X414	8.0	4.000	0.78	14	4.38
C8X413	8.0	4.000	0.81	13	5.36
C8X412	8.0	4.000	0.89	12	6.16

Table B - 1

Light Gage C Members					
Designation	Depth	Width	H1	Gage	WT/FT
C9X2516	9.0	2.500	0.75	16	3.10
C9X2514	9.0	2.500	0.78	14	3.86
C9X2513	9.0	2.500	0.81	13	4.72
C9X2512	9.0	2.500	0.89	12	5.43
C10X216	10.0	2.000	0.75	16	3.10
C10X214	10.0	2.000	0.78	14	3.86
C10X213	10.0	2.000	0.81	13	4.72
C10X212	10.0	2.000	0.89	12	5.43
C10X2516	10.0	2.500	0.75	16	3.31
C10X2514	10.0	2.500	0.78	14	4.12
C10X2513	10.0	2.500	0.81	13	5.04
C10X2512	10.0	2.500	0.89	12	5.80
C10X3514	10.0	3.500	0.78	14	4.64
C10X3513	10.0	3.500	0.81	13	5.67
C10X3512	10.0	3.500	0.89	12	6.53
C10X414	10.0	4.000	0.78	14	4.89
C10X413	10.0	4.000	0.81	13	5.99
C10X412	10.0	4.000	0.89	12	6.89
C12X2514	12.0	2.500	0.78	14	4.64
C12X2513	12.0	2.500	0.81	13	5.67
C12X2512	12.0	2.500	0.89	12	6.53
C12X3514	12.0	3.500	0.78	14	5.15
C12X3513	12.0	3.500	0.81	13	6.30
C12X3512	12.0	3.500	0.89	12	7.25
C12X414	12.0	4.000	0.78	14	5.41
C12X413	12.0	4.000	0.81	13	6.62
C12X412	12.0	4.000	0.89	12	7.62

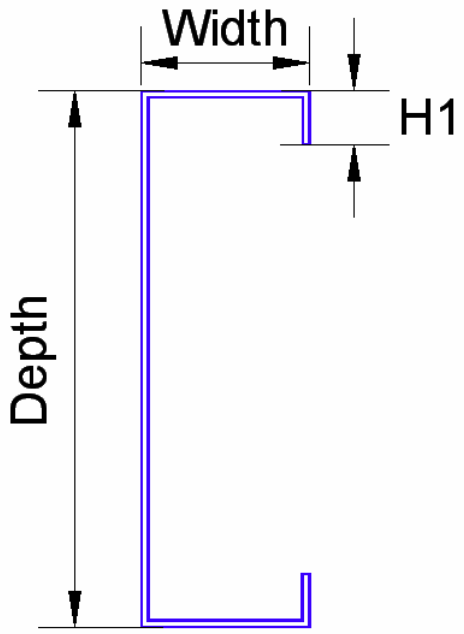


Figure B - 13

Light Gage Z Members						
Designation	Depth	H1	H2	H3	Gage	WT/FT
Z35X1516	3.5	1.500	1.500	0.74	16	1.58
Z35X1514	3.5	1.500	1.500	0.77	14	1.97
Z4X2516	4.0	2.375	2.125	0.90	16	2.06
Z4X2514	4.0	2.375	2.125	0.92	14	2.57
Z6X2516	6.0	2.375	2.125	0.90	16	2.48
Z6X2514	6.0	2.375	2.125	0.92	14	3.08
Z6X2513	6.0	2.375	2.125	0.95	13	3.77
Z6X2512	6.0	2.375	2.125	0.97	12	4.34
Z7X2516	7.0	2.375	2.125	0.90	16	2.68
Z7X2514	7.0	2.375	2.125	0.92	14	3.34
Z7X2513	7.0	2.375	2.125	0.95	13	4.09
Z7X2512	7.0	2.375	2.125	0.97	12	4.71
Z8X2516	8.0	2.375	2.125	0.90	16	2.89
Z8X2514	8.0	2.375	2.125	0.92	14	3.60
Z8X2513	8.0	2.375	2.125	0.95	13	4.41
Z8X2512	8.0	2.375	2.125	0.97	12	5.07
Z8X316	8.0	2.875	2.625	0.90	16	3.10
Z8X314	8.0	2.875	2.625	0.92	14	3.86
Z8X313	8.0	2.875	2.625	0.95	13	4.72
Z8X312	8.0	2.875	2.625	0.97	12	5.43
Z8X3516	8.0	3.375	3.125	0.90	16	3.31
Z8X3514	8.0	3.375	3.125	0.92	14	4.12
Z8X3513	8.0	3.375	3.125	0.95	13	5.04
Z8X3512	8.0	3.375	3.125	0.97	12	5.80
Z9X2516	9.0	2.375	2.125	0.90	16	3.10
Z9X2514	9.0	2.375	2.125	0.92	14	3.86
Z9X2513	9.0	2.375	2.125	0.95	13	4.72
Z9X2512	9.0	2.375	2.125	0.97	12	5.43
Z10X2516	10.0	2.375	2.125	0.90	16	3.31
Z10X2514	10.0	2.375	2.125	0.92	14	4.12
Z10X2513	10.0	2.375	2.125	0.95	13	5.04
Z10X2512	10.0	2.375	2.125	0.97	12	5.80
Z10X316	10.0	2.875	2.625	0.90	16	3.51
Z10X314	10.0	2.875	2.625	0.92	14	4.38
Z10X313	10.0	2.875	2.625	0.95	13	5.36
Z10X312	10.0	2.875	2.625	0.97	12	6.16
Z10X3516	10.0	3.375	3.125	0.90	16	3.72
Z10X3514	10.0	3.375	3.125	0.92	14	4.64
Z10X3513	10.0	3.375	3.125	0.95	13	5.67
Z10X3512	10.0	3.375	3.125	0.97	12	6.53
Z12X2514	12.0	2.375	2.125	0.92	14	4.64
Z12X2513	12.0	2.375	2.125	0.95	13	5.67
Z12X2512	12.0	2.375	2.125	0.97	12	6.53
Z12X314	12.0	2.875	2.625	0.92	14	4.89
Z12X313	12.0	2.875	2.625	0.95	13	5.99
Z12X312	12.0	2.875	2.625	0.97	12	6.89
Z12X3514	12.0	3.375	3.125	0.92	14	5.15
Z12X3513	12.0	3.375	3.125	0.95	13	6.30
Z12X3512	12.0	3.375	3.125	0.97	12	7.25

Table B - 2

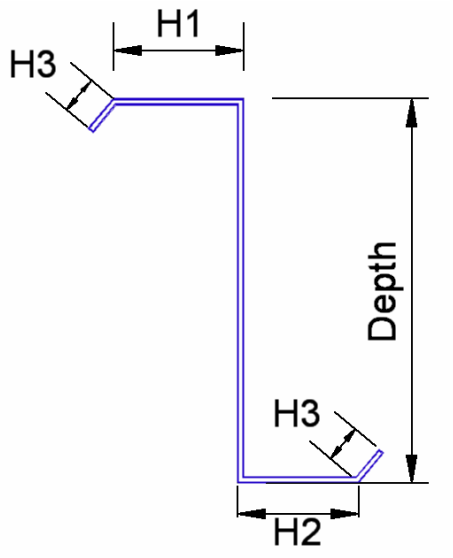


Figure B - 14

AISI SECTIONS - CS, CU, ZS, ZU and HU

In addition to the LGSI sections, ***Structural Desktop*** interfaces to the AISI (American Iron and Steel Institute). These members were added with the release of Version 1.6.

The CS members are similar to the C members of the LGSI table. The CU members are without the distinctive lips of the C or CS, as shown in **Figure B - 15** at right.

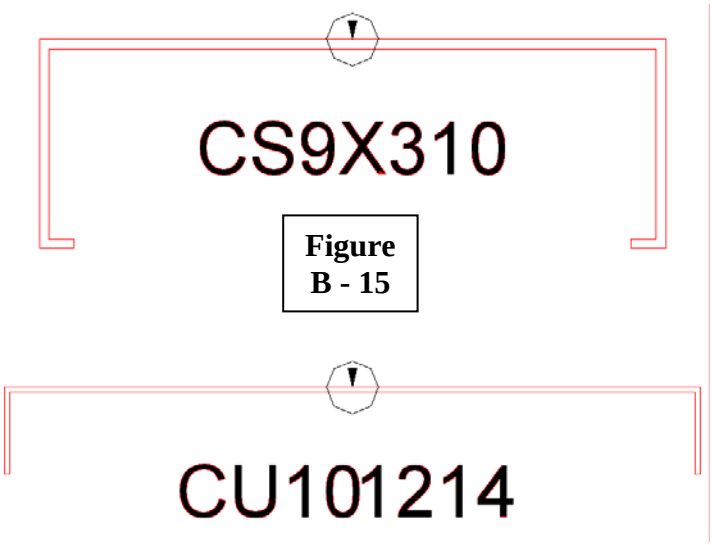


Figure B - 15

Both types of AISI member have been created to display with the centroid line centered on the midpoint of the back face.

The naming convention adopted for these members in ***Structural Desktop*** is also similar to that of the LGSI members for compatibility with some of the analytical programs.

Structural Desktop will read these members from RISA 3D, for example, in the native AISI format that RISA 3D uses. The CS9X310 would be the 9CS3X135. In this case, the "9" and "CS" are reversed, and the 135, representing 135-one-thousanths, is converted to the nearest gage, 10. A 10CU1.25X071 likewise is converted to CU10X1214, in line with other naming conventions for custom members. However, due to the fact that this conversion results in a name nine characters long, the name is shortened by dropping the "X" in the center, resulting in "CU101214" (a CU, 10 by 1.2(5) with 14 gage material). If you read this information from a RISA file, the members will be correctly read and the names will be changed. If you use our User files to access these members in another analytical/design program, the names will be in the shortened format in the User file. These names are listed in the tables that follow.

Table B - 3 and **Figure B - 16** below show the names and sizes of the **AISI CS** members as used in the User Files.

AISI CS Members						
Designation	Depth	Width	H1	Thickness	Gage	WT/FT
CS151620	1.5	1.625	0.375	0.035	20	0.616
CS151618	1.5	1.625	0.375	0.045	18	0.783
CS151616	1.5	1.625	0.375	0.057	16	0.977
CS151614	1.5	1.625	0.375	0.071	14	1.188
CS251620	2.5	1.625	0.375	0.035	20	0.735
CS251618	2.5	1.625	0.375	0.045	18	0.936
CS251616	2.5	1.625	0.375	0.057	16	1.171
CS251614	2.5	1.625	0.375	0.071	14	1.429
CS3X1620	3.0	1.625	0.375	0.035	20	0.793
CS3X1618	3.0	1.625	0.375	0.045	18	1.014
CS3X1616	3.0	1.625	0.375	0.057	16	1.269
CS3X1614	3.0	1.625	0.375	0.071	14	1.552
CS3X316	3.0	3.000	0.500	0.060	16	1.895
CS3X314	3.0	3.000	0.500	0.075	14	2.348
CS3X313	3.0	3.000	0.500	0.090	13	2.797
CS3X312	3.0	3.000	0.500	0.105	12	3.236
CS3X310	3.0	3.000	0.500	0.135	10	4.097
CS351620	3.5	1.625	0.500	0.035	20	0.885
CS351618	3.5	1.625	0.500	0.045	18	1.13
CS351616	3.5	1.625	0.500	0.057	16	1.412
CS351614	3.5	1.625	0.500	0.071	14	1.732
CS4X1620	4.0	1.625	0.375	0.035	20	0.895
CS4X1618	4.0	1.625	0.375	0.045	18	1.143
CS4X1616	4.0	1.625	0.375	0.057	16	1.436
CS4X1614	4.0	1.625	0.375	0.071	14	1.773
CS4X416	4.0	4.000	0.500	0.060	16	2.508
CS4X414	4.0	4.000	0.500	0.075	14	3.114
CS4X413	4.0	4.000	0.500	0.090	13	3.716
CS4X412	4.0	4.000	0.500	0.105	12	4.308
CS4X410	4.0	4.000	0.500	0.135	10	5.475
CS551620	5.5	1.625	0.500	0.035	20	1.123
CS551618	5.5	1.625	0.500	0.045	18	1.436
CS551616	5.5	1.625	0.500	0.057	16	1.8
CS551614	5.5	1.625	0.500	0.071	14	2.215
CS551612	5.5	1.625	0.500	0.102	12	3.097
CS8X1618	8.0	1.625	0.500	0.045	18	1.817
CS8X1616	8.0	1.625	0.500	0.057	16	2.287
CS8X1614	8.0	1.625	0.500	0.071	14	2.821
CS8X1612	8.0	1.625	0.500	0.102	12	3.964
CS9X316	9.0	3.000	0.500	0.060	16	3.12
CS9X314	9.0	3.000	0.500	0.075	14	3.879
CS9X313	9.0	3.000	0.500	0.090	13	4.635
CS9X312	9.0	3.000	0.500	0.105	12	5.38
CS9X310	9.0	3.000	0.500	0.135	10	6.853
CS101616	10.0	1.625	0.500	0.057	16	2.675
CS101614	10.0	1.625	0.500	0.071	14	3.304
CS101612	10.0	1.625	0.500	0.102	12	4.658
CS10X316	10.0	3.000	0.500	0.060	16	3.325
CS10X314	10.0	3.000	0.500	0.075	14	4.134
CS10X313	10.0	3.000	0.500	0.090	13	4.941
CS10X312	10.0	3.000	0.500	0.105	12	5.737
CS10X310	10.0	3.000	0.500	0.135	10	7.313
CS113716	11.0	3.750	0.500	0.060	16	3.835
CS113714	11.0	3.750	0.500	0.075	14	4.774
CS113713	11.0	3.750	0.500	0.090	13	5.706
CS113712	11.0	3.750	0.500	0.105	12	6.632
CS113710	11.0	3.750	0.500	0.135	10	8.459
CS121614	12.0	1.625	0.500	0.071	14	3.787
CS121612	12.0	1.625	0.500	0.102	12	5.353
CS123716	12.0	3.750	0.500	0.060	16	4.039
CS123714	12.0	3.750	0.500	0.075	14	5.029
CS123713	12.0	3.750	0.500	0.090	13	6.013
CS123712	12.0	3.750	0.500	0.105	12	6.989
CS123710	12.0	3.750	0.500	0.135	10	8.919
CS143716	14.0	3.750	0.500	0.060	16	4.447
CS143714	14.0	3.750	0.500	0.075	14	5.54
CS143713	14.0	3.750	0.500	0.090	13	6.625
CS143712	14.0	3.750	0.500	0.105	12	7.704
CS143710	14.0	3.750	0.500	0.135	10	9.837
CS163714	16.0	3.750	0.500	0.075	14	6.05
CS163713	16.0	3.750	0.500	0.090	13	7.238
CS163712	16.0	3.750	0.500	0.105	12	8.418
CS163710	16.0	3.750	0.500	0.135	10	10.76

Table B - 3

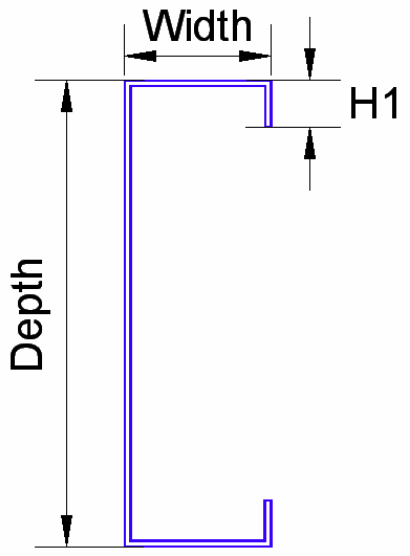


Figure B - 16

The **Table B - 4** at the left and the **Figure B - 17** below show the names and sizes of the **AISI CU** members as used in the User Files.

AISI CU Members					
Designation	Depth	Width	Thickness	Gage	WT/FT
CU151220	1.5	1.240	0.035	20	0.446
CU151218	1.5	1.250	0.045	18	0.572
CU151216	1.5	1.250	0.057	16	0.718
CU151214	1.5	1.250	0.071	14	0.885
CU251220	2.5	1.250	0.035	20	0.565
CU251218	2.5	1.250	0.045	18	0.725
CU251216	2.5	1.250	0.057	16	0.912
CU251214	2.5	1.250	0.071	14	1.126
CU3X1220	3.0	1.250	0.035	20	0.626
CU3X1218	3.0	1.250	0.045	18	0.800
CU3X1216	3.0	1.250	0.057	16	1.007
CU3X1214	3.0	1.250	0.071	14	1.249
CU351220	3.5	1.250	0.035	20	0.684
CU351218	3.5	1.250	0.045	18	0.878
CU351216	3.5	1.250	0.057	16	1.106
CU351214	3.5	1.250	0.071	14	1.368
CU4X1220	4.0	1.250	0.035	20	0.745
CU4X1218	4.0	1.250	0.045	18	0.953
CU4X1216	4.0	1.250	0.057	16	1.201
CU4X1214	4.0	1.250	0.071	14	1.490
CU551220	5.5	1.250	0.035	20	0.922
CU551218	5.5	1.250	0.045	18	1.184
CU551216	5.5	1.250	0.057	16	1.494
CU551214	5.5	1.250	0.071	14	1.851
CU8X1220	8.0	1.250	0.035	20	1.222
CU8X1218	8.0	1.250	0.045	18	1.565
CU8X1216	8.0	1.250	0.057	16	1.977
CU8X1214	8.0	1.250	0.071	14	2.457
CU101218	10.0	1.250	0.045	18	1.872
CU101216	10.0	1.250	0.057	16	2.365
CU101214	10.0	1.250	0.071	14	2.940
CU121216	12.0	1.250	0.057	16	2.753
CU121214	12.0	1.250	0.071	14	3.423

Table B - 4

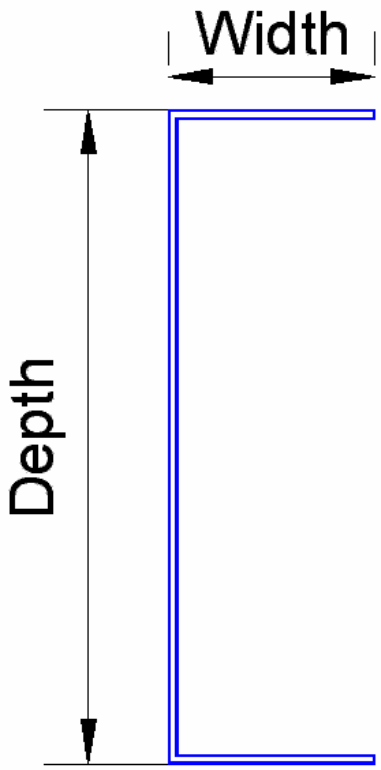


Figure B - 17

AISI ZS Members						
Designation	Depth	Width	H1	Thickness	Gage	WT/FT
ZS3X1720	3.0	1.75	0.75	0.036	20	0.943
ZS3X1718	3.0	1.75	0.75	0.048	18	1.249
ZS3X1716	3.0	1.75	0.75	0.060	16	1.555
ZS3X1714	3.0	1.75	0.75	0.075	14	1.929
ZS3X1713	3.0	1.75	0.75	0.090	13	2.300
ZS4X220	4.0	2.00	0.75	0.036	20	1.126
ZS4X218	4.0	2.00	0.75	0.048	18	1.494
ZS4X216	4.0	2.00	0.75	0.060	16	1.861
ZS4X214	4.0	2.00	0.75	0.075	14	2.310
ZS4X213	4.0	2.00	0.75	0.090	13	2.760
ZS5X220	5.0	2.00	0.75	0.036	20	1.249
ZS5X218	5.0	2.00	0.75	0.048	18	1.657
ZS5X216	5.0	2.00	0.75	0.060	16	2.065
ZS5X214	5.0	2.00	0.75	0.075	14	2.566
ZS5X213	5.0	2.00	0.75	0.090	13	3.066
ZS6X218	6.0	2.00	0.75	0.048	18	1.820
ZS6X216	6.0	2.00	0.75	0.060	16	2.270
ZS6X214	6.0	2.00	0.75	0.075	14	2.821
ZS6X213	6.0	2.00	0.75	0.090	13	3.372
ZS6X212	6.0	2.00	0.75	0.105	12	3.917
ZS7X218	7.0	2.00	0.75	0.048	18	1.984
ZS7X216	7.0	2.00	0.75	0.060	16	2.474
ZS7X214	7.0	2.00	0.75	0.075	14	3.076
ZS7X213	7.0	2.00	0.75	0.090	13	3.678
ZS7X212	7.0	2.00	0.75	0.105	12	4.274

AISI ZS Members						
Designation	Depth	Width	H1	Thickness	Gage	WT/FT
ZS8X218	8.0	2.00	0.75	0.048	18	2.147
ZS8X216	8.0	2.00	0.75	0.060	16	2.678
ZS8X214	8.0	2.00	0.75	0.075	14	3.331
ZS8X213	8.0	2.00	0.75	0.090	13	3.985
ZS8X212	8.0	2.00	0.75	0.105	12	4.631
ZS8X2518	8.0	2.50	0.75	0.480	18	2.310
ZS8X2516	8.0	2.50	0.75	0.060	16	2.882
ZS8X2514	8.0	2.50	0.75	0.075	14	3.587
ZS8X2513	8.0	2.50	0.75	0.090	13	4.291
ZS8X2512	8.0	2.50	0.75	0.105	12	4.988
ZS9X316	9.0	3.00	0.75	0.060	16	3.290
ZS9X314	9.0	3.00	0.75	0.075	14	4.097
ZS9X313	9.0	3.00	0.75	0.090	13	4.903
ZS9X312	9.0	3.00	0.75	0.105	12	5.703
ZS9X310	9.0	3.00	0.75	0.135	10	7.282
ZS10X316	10.0	3.00	0.75	0.060	16	3.495
ZS10X314	10.0	3.00	0.75	0.075	14	4.352
ZS10X313	10.0	3.00	0.75	0.090	13	5.210
ZS10X312	10.0	3.00	0.75	0.105	12	6.060
ZS10X310	10.0	3.00	0.75	0.135	10	7.741
ZS123216	12.0	3.25	0.75	0.060	16	4.005
ZS123214	12.0	3.25	0.75	0.075	14	4.992
ZS123213	12.0	3.25	0.75	0.090	13	5.975
ZS123212	12.0	3.25	0.75	0.105	12	6.952
ZS123210	12.0	3.25	0.75	0.135	10	8.891

Table B - 5

Table B - 5 above and **Figure B - 18** demonstrate the names, sizes and values used by ***Structural Desktop*** for the **AISI ZS** members as they are provided in the User files.

The top and bottom flanges **(Width)** and the lip lengths **(H1)** are identical for these members.

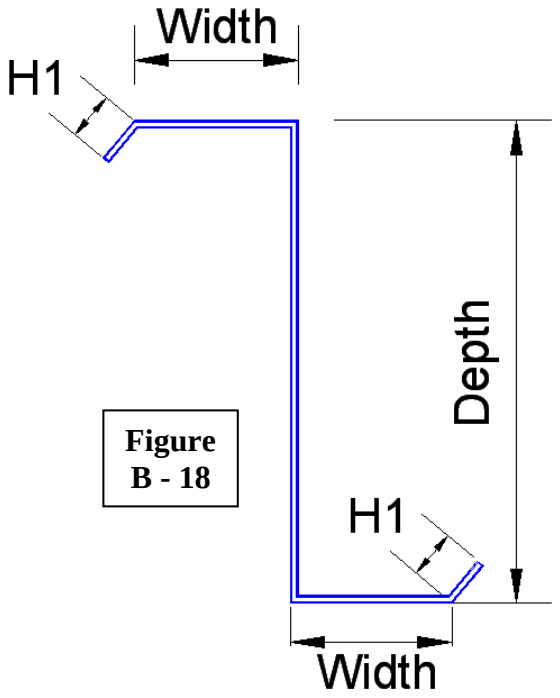


Figure B - 18

The **AISI ZU** members that are provided in the User files are listed in **Table B - 6** below and refer to the values in **Figure B - 19.**

AISI ZU Members					
Designation	Depth	Width	Thickness	Gage	WT/FT
ZU151220	1.500	1.25	0.036	20	0.459
ZU151218	1.500	1.25	0.048	18	0.609
ZU151216	1.500	1.25	0.060	16	0.755
ZU151214	1.500	1.25	0.075	14	0.932
ZU151213	1.500	1.25	0.090	13	1.109
ZU251220	2.500	1.25	0.036	20	0.582
ZU251218	2.500	1.25	0.048	18	0.772
ZU251216	2.500	1.25	0.060	16	0.960
ZU251214	2.500	1.25	0.075	14	1.188
ZU251213	2.500	1.25	0.090	13	1.416
ZU361220	3.625	1.25	0.036	20	0.721
ZU361218	3.625	1.25	0.048	18	0.956
ZU361216	3.625	1.25	0.060	16	1.188
ZU361214	3.625	1.25	0.075	14	1.477
ZU361213	3.625	1.25	0.090	13	1.759
ZU4X1220	4.000	1.25	0.036	20	0.766
ZU4X1218	4.000	1.25	0.048	18	1.017
ZU4X1216	4.000	1.25	0.060	16	1.266
ZU4X1214	4.000	1.25	0.075	14	1.572
ZU4X1213	4.000	1.25	0.090	13	1.875
ZU6X1218	6.000	1.25	0.048	18	1.344
ZU6X1216	6.000	1.25	0.060	16	1.674
ZU6X1214	6.000	1.25	0.075	14	2.082
ZU6X1213	6.000	1.25	0.090	13	2.487
ZU6X1212	6.000	1.25	0.105	12	2.889
ZU8X1218	8.000	1.25	0.048	18	1.671
ZU8X1216	8.000	1.25	0.060	16	2.082
ZU8X1214	8.000	1.25	0.075	14	2.593
ZU8X1213	8.000	1.25	0.090	13	3.100
ZU8X1212	8.000	1.25	0.105	12	3.604

Table B - 6

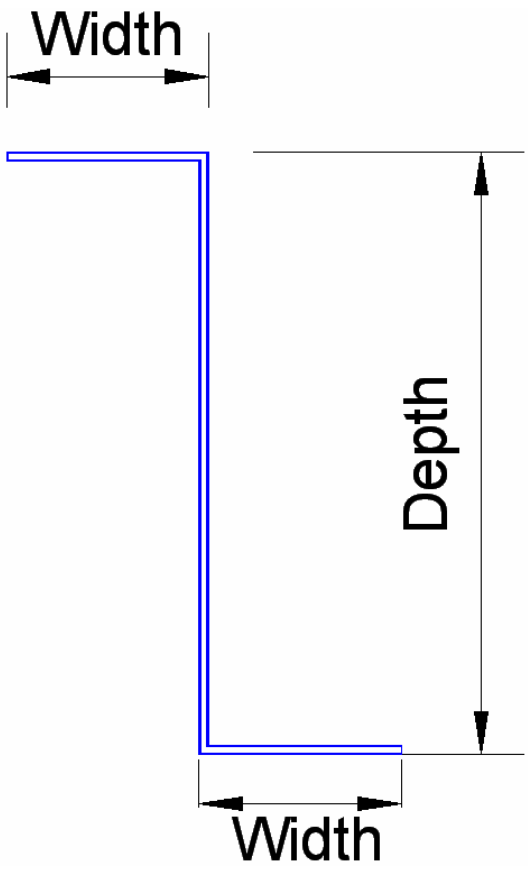


Figure B - 19

The **AISI HU** members that are provided in the User files are listed in **Table B - 7** below and refer to the values illustrated in **Figure B - 20.**

AISI HU Members						
Designation	Depth	Width	H1	Thickness	Gage	WT/FT
HU3X314	3.0	3.0	0.915	0.075	14	2.590
HU3X312	3.0	3.0	1.340	0.105	12	3.876
HU3X4512	3.0	4.5	1.340	0.105	12	4.413
HU3X4510	3.0	4.5	1.670	0.135	10	5.907
HU4X218	4.0	2.0	0.618	0.048	18	1.746
HU4X216	4.0	2.0	0.750	0.060	16	2.222
HU4X214	4.0	2.0	0.915	0.075	14	2.845
HU4X414	4.0	4.0	0.915	0.075	14	3.355
HU4X412	4.0	4.0	1.340	0.105	12	4.948
HU4X612	4.0	6.0	1.340	0.105	12	5.662
HU4X610	4.0	6.0	1.670	0.135	10	7.517
HU6X318	6.0	3.0	0.660	0.048	18	2.576
HU6X316	6.0	3.0	0.760	0.060	16	3.246
HU6X314	6.0	3.0	0.915	0.075	14	4.121
HU6X614	6.0	6.0	0.915	0.075	14	4.886
HU6X612	6.0	6.0	1.340	0.105	12	7.091
HU6X912	6.0	9.0	1.340	0.105	12	8.163
HU6X910	6.0	9.0	1.670	0.135	10	10.730
HU8X416	8.0	4.0	0.840	0.060	16	4.301
HU8X414	8.0	4.0	0.980	0.075	14	5.431
HU8X812	8.0	8.0	1.340	0.105	12	9.235
HU8X1212	8.0	12.0	1.340	0.105	12	10.660
HU8X1210	8.0	12.0	1.670	0.135	10	13.950
HU10X514	10.0	5.0	1.050	0.075	14	6.741

Table B - 7

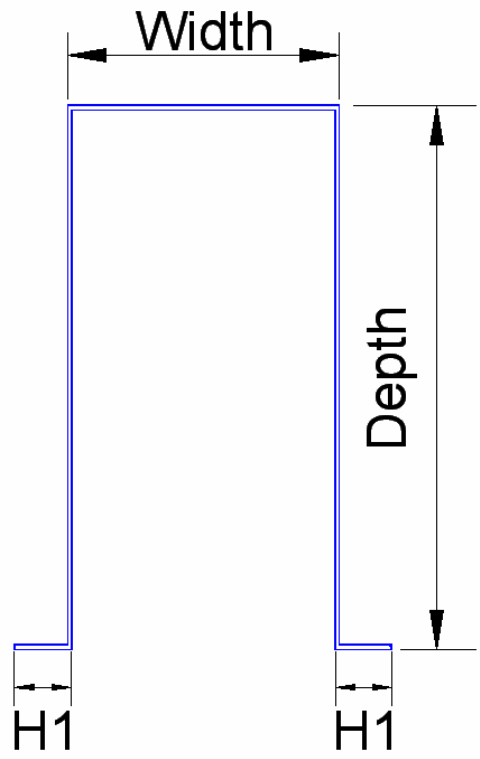


Figure B - 20

The centroid of the HU member is treated as if the member were a C or Channel member, i.e. the center of the measurement above marked "Width" is treated as a back face, and the center of that back face is the location of the centroid.

Concrete L's and Inverted T's

Concrete L's have a naming convention derived from their depth and width. The members have either an "R" (for right handed) or "L" (for left handed) prefix. This is followed by the Width at the base of the member, the letters "LB" (for L Beam) and the Depth.

These members are available in the User files for your use. The centroids of the L members are set vertically through the center of the depth. Horizontally, the centroid runs through the center of the width, less the 6-inch cutaway that gives the member its characteristic L shape, as shown in **Figure B-21** at right.

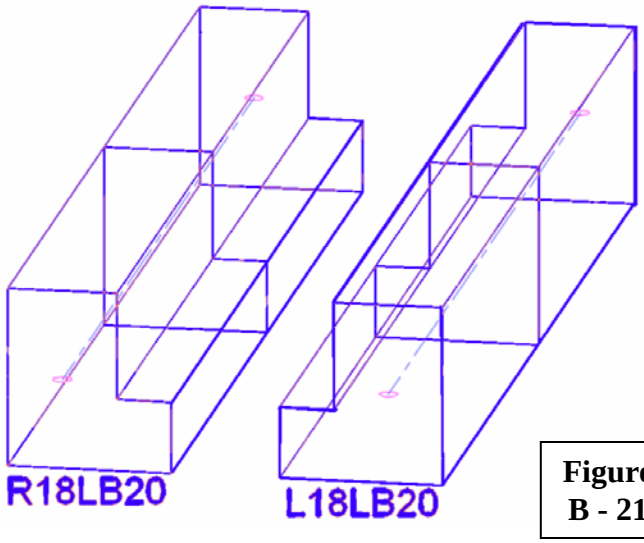


Figure B - 21

Measurements and definitions for these members are further detailed in **Table B - 8** and **Figure B - 22** below.

L - SHAPED BEAMS				
Designation	Width	Depth	H1	H2
18LB20	18"	20"	12"	8"
18LB24	18"	24"	12"	12"
18LB28	18"	28"	16"	12"
18LB32	18"	32"	20"	12"
18LB36	18"	36"	24"	12"
18LB40	18"	40"	24"	16"
18LB44	18"	44"	28"	16"
18LB48	18"	48"	32"	16"
18LB52	18"	52"	36"	16"
18LB56	18"	56"	40"	16"
18LB60	18"	60"	44"	16"
24LB20	24"	20"	12"	8"
24LB24	24"	24"	12"	12"
24LB28	24"	28"	16"	12"
24LB32	24"	32"	20"	12"
24LB36	24"	36"	24"	12"
24LB40	24"	40"	24"	16"
24LB44	24"	44"	28"	16"
24LB48	24"	48"	32"	16"
24LB52	24"	52"	36"	16"
24LB56	24"	56"	40"	16"
24LB60	24"	60"	44"	16"

Table B - 8

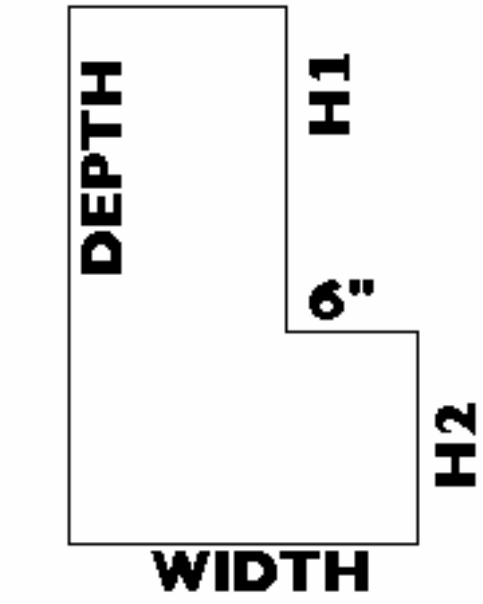


Figure B - 22

Concrete T's have a naming convention derived from their depth and width. The members have a "T" prefix. This is followed by the Width at the base of the member, the letters "IT" (for Inverted Tee beam) and the Depth.

INVERTED TEE BEAMS				
Designation	Width	Depth	H1	H2
24IT20	24"	20"	12"	8"
24IT24	24"	24"	12"	12"
24IT28	24"	28"	16"	12"
24IT32	24"	32"	20"	12"
24IT36	24"	36"	24"	12"
24IT40	24"	40"	24"	16"
24IT44	24"	44"	28"	16"
24IT48	24"	48"	32"	16"
24IT52	24"	52"	36"	16"
24IT56	24"	56"	40"	16"
24IT60	24"	60"	44"	16"
30IT20	30"	20"	12"	8"
30IT24	30"	24"	12"	12"
30IT28	30"	28"	16"	12"
30IT32	30"	32"	20"	12"
30IT36	30"	36"	24"	12"
30IT40	30"	40"	24"	16"
30IT44	30"	44"	28"	16"
30IT48	30"	48"	32"	16"
30IT52	30"	52"	36"	16"
30IT56	30"	56"	40"	16"
30IT60	30"	60"	44"	16"
36IT20	36"	20"	12"	8"
36IT24	36"	24"	12"	12"
36IT28	36"	28"	16"	12"
36IT32	36"	32"	20"	12"
36IT36	36"	36"	24"	12"
36IT40	36"	40"	24"	16"
36IT44	36"	44"	28"	16"
36IT48	36"	48"	32"	16"
36IT52	36"	52"	36"	16"

Table B - 9

These members are available in the User files for your use. The centroids are centered on the T both horizontally and vertically.

Measurements and definitions for these members are further detailed in **Table B - 4** and **Figure B - 23** below.

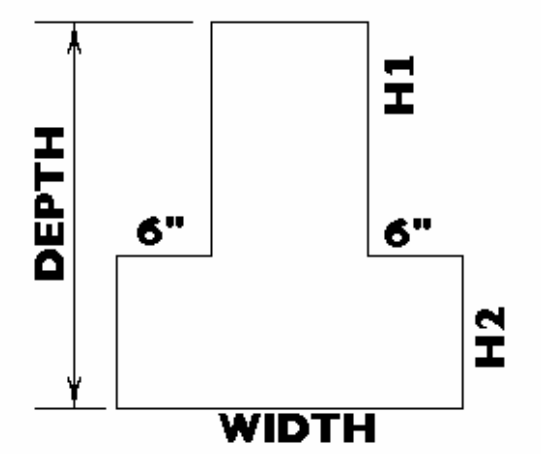


Figure B - 23

Joist Girders, LongSpan and ShortSpan Joists

The Joist members in the tables that follow are available to you in the User files. The centroid of a joist member is located at the bottom of the bearing plate. Please see **Tables B - 10 . 1** through **B - 10 . 7** and **Figures B - 24** through **B - 27.**

Joist Girders Available in User Files												
NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT
JG201	20	16.0	JG281	28	17.5	JG361	36	18.0	JG441	44	19.5
JG202	20	19.0	JG282	28	20.0	JG362	36	20.0	JG442	44	21.5
JG203	20	23.5	JG283	28	23.0	JG363	36	21.5	JG443	44	23.0
JG204	20	21.0	JG284	28	22.0	JG364	36	23.0	JG444	44	24.5
JG205	20	22.0	JG285	28	23.0	JG365	36	24.5	JG445	44	25.5
JG206	20	25.0	JG286	28	26.0	JG366	36	26.5	JG446	44	28.5
JG207	20	27.5	JG287	28	29.0	JG367	36	29.5	JG447	44	30.5
JG208	20	30.5	JG288	28	31.5	JG368	36	32.0	JG448	44	33.5
JG209	20	34.5	JG289	28	36.5	JG369	36	35.5	JG449	44	37.0
JG2010	20	35.5	JG2810	28	38.0	JG3610	36	38.5	JG4410	44	40.0
JG2011	20	40.5	JG2811	28	43.0	JG3611	36	41.0	JG4411	44	43.5
JG2012	20	44.5	JG2812	28	46.5	JG3612	36	46.0	JG4412	44	46.5
JG2013	20	47.5	JG2813	28	49.0	JG3613	36	49.5	JG4413	44	50.5
JG2014	20	51.0	JG2814	28	57.0	JG3614	36	53.5	JG4414	44	55.5
JG2015	20	58.0	JG2815	28	63.0	JG3615	36	61.5	JG4415	44	63.5
JG2016	20	73.5	JG2816	28	73.5	JG3616	36	71.0	JG4416	44	72.0
JG2017	20	78.5	JG2817	28	79.5	JG3617	36	76.0	JG4417	44	78.0
JG2018	20	84.0	JG2818	28	91.0	JG3618	36	86.0	JG4418	44	89.0
JG2019	20	99.5	JG2819	28	101.5	JG3619	36	101.5	JG4419	44	105.0
JG2020	20	136.0	JG2820	28	135.0	JG3620	36	130.0	JG4420	44	131.5
JG241	24	17.5	JG321	32	18.0	JG401	40	18.5	JG481	48	19.0
JG242	24	19.5	JG322	32	19.5	JG402	40	20.5	JG482	48	20.0
JG243	24	23.5	JG323	32	21.0	JG403	40	22.0	JG483	48	21.0
JG244	24	21.5	JG324	32	22.5	JG404	40	23.5	JG484	48	22.0
JG245	24	22.5	JG325	32	23.5	JG405	40	25.0	JG485	48	24.0
JG246	24	26.0	JG326	32	26.5	JG406	40	27.0	JG486	48	26.0
JG247	24	28.0	JG327	32	28.5	JG407	40	30.0	JG487	48	29.5
JG248	24	32.5	JG328	32	32.0	JG408	40	32.5	JG488	48	32.0
JG249	24	35.5	JG329	32	34.5	JG409	40	35.5	JG489	48	35.0
JG2410	24	38.5	JG3210	32	37.5	JG4010	40	39.0	JG4810	48	37.0
JG2411	24	43.0	JG3211	32	41.0	JG4011	40	42.0	JG4811	48	41.0
JG2412	24	46.0	JG3212	32	45.0	JG4012	40	45.0	JG4812	48	44.0
JG2413	24	49.0	JG3213	32	47.5	JG4013	40	49.0	JG4813	48	46.5
JG2414	24	57.5	JG3214	32	52.5	JG4014	40	53.5	JG4814	48	54.0
JG2415	24	63.5	JG3215	32	61.0	JG4015	40	61.5	JG4815	48	59.5
JG2416	24	75.5	JG3216	32	72.0	JG4016	40	69.5	JG4816	48	67.5
JG2417	24	79.5	JG3217	32	77.0	JG4017	40	75.5	JG4817	48	75.5
JG2418	24	93.5	JG3218	32	85.5	JG4018	40	85.5	JG4818	48	83.5
JG2419	24	101.0	JG3219	32	100.5	JG4019	40	101.0	JG4819	48	99.5
JG2420	24	138.5	JG3220	32	130.5	JG4020	40	129.0	JG4820	48	124.0

Table B - 10 . 1

Joist Girders Available in User Files												
NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT
JG521	52	19.0	JG601	60	23.0	JG681	68	24.5	JG841	84	40.0
JG522	52	20.5	JG602	60	23.5	JG682	68	24.5	JG842	84	40.5
JG523	52	21.5	JG603	60	23.5	JG683	68	25.0	JG843	84	40.5
JG524	52	22.5	JG604	60	23.5	JG684	68	25.5	JG844	84	41.0
JG525	52	24.0	JG605	60	24.5	JG685	68	26.0	JG845	84	42.0
JG526	52	26.5	JG606	60	27.0	JG686	68	29.0	JG846	84	42.0
JG527	52	29.5	JG607	60	30.5	JG687	68	32.0	JG847	84	42.5
JG528	52	32.0	JG608	60	33.0	JG688	68	35.0	JG848	84	43.0
JG529	52	35.0	JG609	60	36.5	JG689	68	38.5	JG849	84	46.0
JG5210	52	37.5	JG6010	60	38.0	JG6810	68	40.5	JG8410	84	48.0
JG5211	52	41.0	JG6011	60	42.0	JG6811	68	44.0	JG8411	84	49.5
JG5212	52	43.5	JG6012	60	44.5	JG6812	68	47.0	JG8412	84	50.0
JG5213	52	46.5	JG6013	60	47.0	JG6813	68	49.5	JG8413	84	51.0
JG5214	52	53.0	JG6014	60	53.0	JG6814	68	55.0	JG8414	84	54.0
JG5215	52	58.5	JG6015	60	60.0	JG6815	68	62.0	JG8415	84	61.0
JG5216	52	67.0	JG6016	60	67.5	JG6816	68	70.0	JG8416	84	68.0
JG5217	52	75.5	JG6017	60	75.5	JG6817	68	79.5	JG8417	84	81.0
JG5218	52	83.5	JG6018	60	84.0	JG6818	68	88.5	JG8418	84	88.0
JG5219	52	98.5	JG6019	60	100.5	JG6819	68	103.5	JG8419	84	101.5
JG5220	52	123.5	JG6020	60	124.0	JG6820	68	128.0	JG8420	84	130.5
JG561	56	20.0	JG641	64	23.5	JG721	72	29.5	JG961	96	56.0
JG562	56	21.0	JG642	64	24.0	JG722	72	29.5	JG962	96	56.5
JG563	56	22.0	JG643	64	24.5	JG723	72	29.5	JG963	96	57.0
JG564	56	23.0	JG644	64	24.5	JG724	72	30.5	JG964	96	57.0
JG565	56	25.0	JG645	64	25.5	JG725	72	31.0	JG965	96	58.0
JG566	56	27.0	JG646	64	28.5	JG726	72	32.5	JG966	96	59.0
JG567	56	30.5	JG647	64	31.0	JG727	72	35.0	JG967	96	59.5
JG568	56	32.5	JG648	64	33.5	JG728	72	35.5	JG968	96	60.5
JG569	56	36.5	JG649	64	37.5	JG729	72	38.4	JG969	96	61.0
JG5610	56	38.5	JG6410	64	39.5	JG7210	72	40.5	JG9610	96	61.5
JG5611	56	42.0	JG6411	64	42.5	JG7211	72	44.5	JG9611	96	62.0
JG5612	56	44.5	JG6412	64	46.0	JG7212	72	47.5	JG9612	96	62.5
JG5613	56	47.5	JG6413	64	48.0	JG7213	72	50.5	JG9613	96	63.0
JG5614	56	54.5	JG6414	64	54.0	JG7214	72	55.5	JG9614	96	67.5
JG5615	56	60.0	JG6415	64	60.5	JG7215	72	61.5	JG9615	96	71.5
JG5616	56	67.5	JG6416	64	68.5	JG7216	72	70.0	JG9616	96	75.0
JG5617	56	76.5	JG6417	64	77.5	JG7217	72	78.5	JG9617	96	82.5
JG5618	56	85.0	JG6418	64	86.0	JG7218	72	88.5	JG9618	96	92.0
JG5619	56	100.5	JG6419	64	102.0	JG7219	72	103.5	JG9619	96	108.0
JG5620	56	125.0	JG6420	64	126.0	JG7220	72	127.0	JG9620	96	134.0

Table
B - 10 . 2

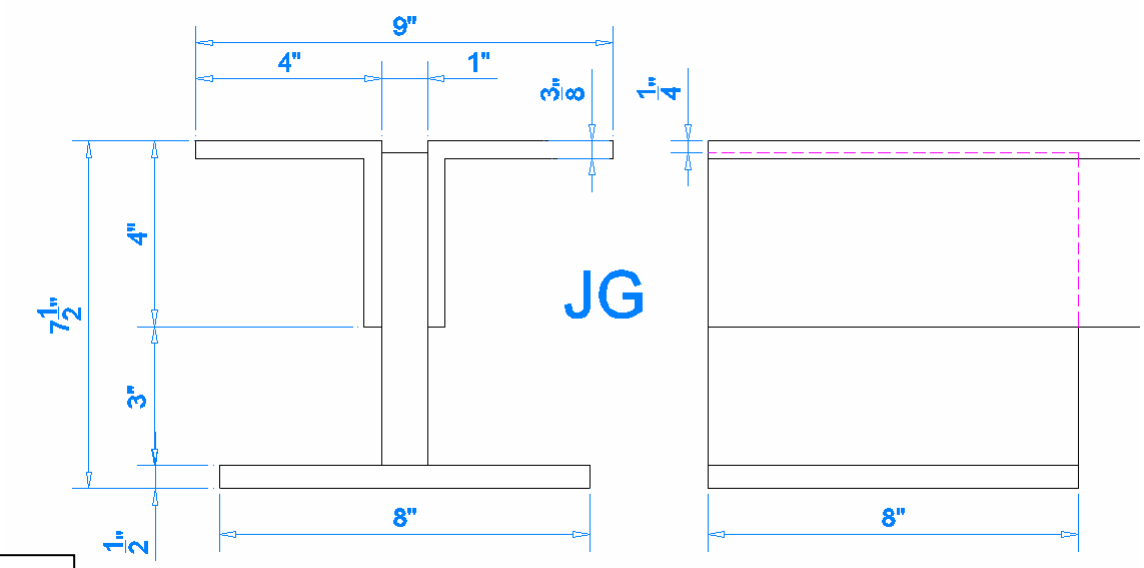


Figure B - 24

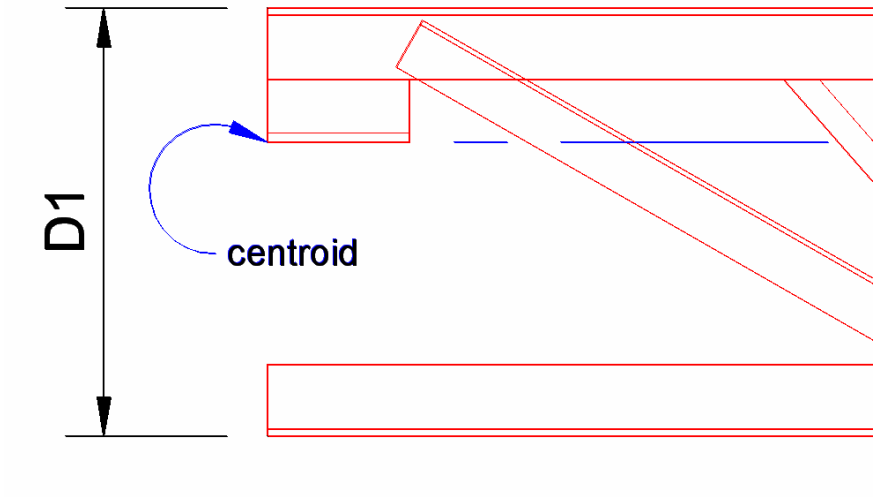


Figure B - 25

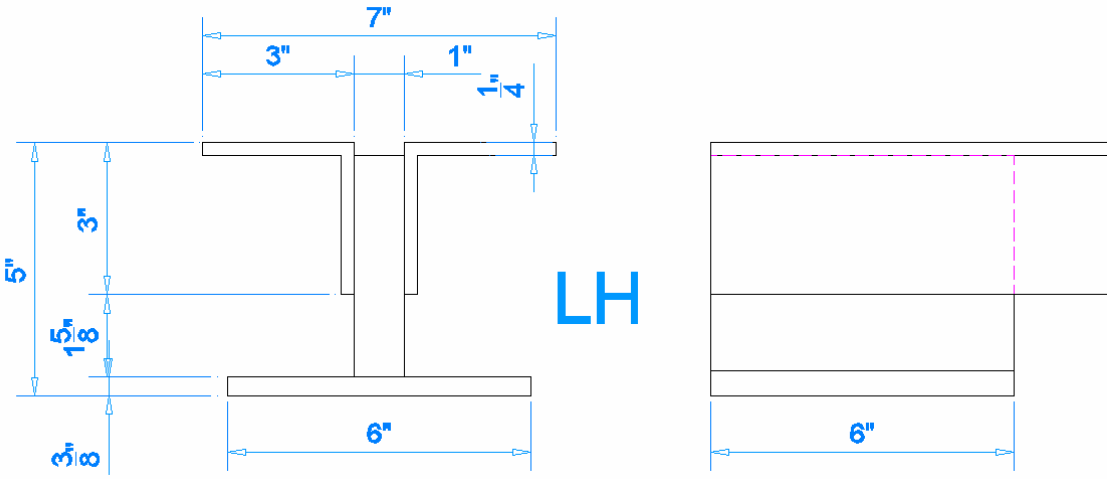


Figure B - 26

Long Span Joists Available in User Files												
NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT
LH181	18	10.0	LH241	24	9.5	LH321	32	10.5	LH401	40	11.0
LH182	18	11.0	LH242	24	10.5	LH322	32	10.5	LH402	40	11.5
LH183	18	11.0	LH243	24	10.5	LH323	32	11.5	LH403	40	11.5
LH184	18	11.5	LH244	24	11.0	LH324	32	11.5	LH404	40	12.0
LH185	18	11.5	LH245	24	11.0	LH325	32	12.0	LH405	40	12.5
LH186	18	12.5	LH246	24	12.0	LH326	32	12.5	LH406	40	13.0
LH187	18	13.0	LH247	24	13.0	LH327	32	13.0	LH407	40	14.5
LH188	18	13.5	LH248	24	13.5	LH328	32	14.0	LH408	40	14.5
LH189	18	14.5	LH249	24	14.5	LH329	32	14.5	LH409	40	15.0
LH1810	18	14.5	LH2410	24	14.5	LH3210	32	15.5	LH4010	40	16.0
LH1811	18	16.0	LH2411	24	16.5	LH3211	32	17.0	LH4011	40	17.5
LH1812	18	17.5	LH2412	24	17.0	LH3212	32	19.0	LH4012	40	18.5
LH1813	18	19.0	LH2413	24	18.0	LH3213	32	19.5	LH4013	40	19.5
LH1814	18	20.0	LH2414	24	21.0	LH3214	32	20.5	LH4014	40	21.0
LH1815	18	21.0	LH2415	24	21.0	LH3215	32	21.0	LH4015	40	21.5
LH1816	18	22.0	LH2416	24	21.5	LH3216	32	23.0	LH4016	40	23.0
LH1817	18	24.5	LH2417	24	23.0	LH3217	32	24.5	LH4017	40	25.5
LH1818	18	25.0	LH2418	24	25.0	LH3218	32	27.5	LH4018	40	27.5
LH1819	18	28.5	LH2419	24	27.5	LH3219	32	28.5	LH4019	40	29.5
LH1820	18	31.0	LH2420	24	31.0	LH3220	32	31.0	LH4020	40	31.5
LH1821	18	34.5	LH2421	24	33.5	LH3221	32	35.5	LH4021	40	35.5
LH1822	18	38.0	LH2422	24	37.5	LH3222	32	39.0	LH4022	40	40.5
LH1823	18	42.0	LH2423	24	41.0	LH3223	32	42.0	LH4023	40	42.5
LH1824	18	48.0	LH2424	24	44.0	LH3224	32	44.5	LH4024	40	45.5
LH1825	18	54.0	LH2425	24	53.0	LH3225	32	52.0	LH4025	40	54.5
LH201	20	10.0	LH281	28	9.5	LH361	36	10.5	LH441	44	11.0
LH202	20	10.5	LH282	28	10.5	LH362	36	10.5	LH442	44	11.5
LH203	20	11.0	LH283	28	10.5	LH363	36	11.0	LH443	44	11.5
LH204	20	11.5	LH284	28	11.0	LH364	36	11.0	LH444	44	11.5
LH205	20	12.0	LH285	28	11.0	LH365	36	11.5	LH445	44	12.0
LH206	20	12.0	LH286	28	12.0	LH366	36	12.0	LH446	44	13.0
LH207	20	13.0	LH287	28	12.5	LH367	36	13.0	LH447	44	13.5
LH208	20	14.0	LH288	28	13.0	LH368	36	14.0	LH448	44	14.5
LH209	20	14.5	LH289	28	14.5	LH369	36	15.0	LH449	44	15.0
LH2010	20	14.5	LH2810	28	14.5	LH3610	36	15.0	LH4410	44	15.5
LH2011	20	16.0	LH2811	28	16.0	LH3611	36	16.5	LH4411	44	17.0
LH2012	20	17.5	LH2812	28	17.5	LH3612	36	18.5	LH4412	44	18.5
LH2013	20	19.0	LH2813	28	18.5	LH3613	36	19.0	LH4413	44	19.0
LH2014	20	20.0	LH2814	28	20.0	LH3614	36	20.5	LH4414	44	21.0
LH2015	20	21.0	LH2815	28	21.5	LH3615	36	21.0	LH4415	44	22.0
LH2016	20	22.0	LH2816	28	21.5	LH3616	36	22.0	LH4416	44	22.5
LH2017	20	23.0	LH2817	28	23.5	LH3617	36	23.5	LH4417	44	24.0
LH2018	20	25.5	LH2818	28	25.5	LH3618	36	26.5	LH4418	44	26.5
LH2019	20	28.5	LH2819	28	26.5	LH3619	36	28.5	LH4419	44	28.5
LH2020	20	31.0	LH2820	28	30.0	LH3620	36	31.0	LH4420	44	32.0
LH2021	20	34.0	LH2821	28	32.0	LH3621	36	34.0	LH4421	44	34.5
LH2022	20	38.0	LH2822	28	39.0	LH3622	36	37.5	LH4422	44	38.0
LH2023	20	42.0	LH2823	28	41.5	LH3623	36	40.0	LH4423	44	42.0
LH2024	20	46.0	LH2824	28	45.0	LH3624	36	45.0	LH4424	44	45.5
LH2025	20	53.5	LH2825	28	53.5	LH3625	36	53.0	LH4425	44	54.0

Table
B - 10 . 3

Long Span Joists Available in User Files												
NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT
LH481	48	12.0	LH561	56	13.5	LH641	64	18.5	LH721	72	20.0
LH482	48	12.0	LH562	56	13.5	LH642	64	18.5	LH722	72	20.0
LH483	48	12.5	LH563	56	13.5	LH643	64	18.5	LH723	72	20.0
LH484	48	12.5	LH564	56	14.5	LH644	64	18.5	LH724	72	20.0
LH485	48	13.0	LH565	56	14.5	LH645	64	18.5	LH725	72	20.0
LH486	48	13.5	LH566	56	15.0	LH646	64	18.5	LH726	72	20.5
LH487	48	14.5	LH567	56	15.5	LH647	64	18.5	LH727	72	21.5
LH488	48	14.5	LH568	56	15.5	LH648	64	20.0	LH728	72	22.0
LH489	48	15.5	LH569	56	17.5	LH649	64	19.5	LH729	72	23.0
LH4810	48	16.5	LH5610	56	17.5	LH6410	64	20.5	LH7210	72	23.0
LH4811	48	18.0	LH5611	56	18.5	LH6411	64	21.5	LH7211	72	23.0
LH4812	48	20.0	LH5612	56	20.0	LH6412	64	22.5	LH7212	72	23.5
LH4813	48	20.5	LH5613	56	21.0	LH6413	64	22.0	LH7213	72	24.5
LH4814	48	21.5	LH5614	56	21.5	LH6414	64	23.0	LH7214	72	25.5
LH4815	48	22.0	LH5615	56	23.0	LH6415	64	24.5	LH7215	72	26.5
LH4816	48	23.5	LH5616	56	24.0	LH6416	64	26.0	LH7216	72	28.5
LH4817	48	25.5	LH5617	56	26.0	LH6417	64	27.0	LH7217	72	30.5
LH4818	48	28.0	LH5618	56	29.5	LH6418	64	29.5	LH7218	72	31.5
LH4819	48	30.0	LH5619	56	30.5	LH6419	64	32.0	LH7219	72	34.5
LH4820	48	31.5	LH5620	56	32.0	LH6420	64	34.5	LH7220	72	37.0
LH4821	48	36.5	LH5621	56	36.5	LH6421	64	37.5	LH7221	72	40.5
LH4822	48	40.0	LH5622	56	40.0	LH6422	64	42.5	LH7222	72	45.5
LH4823	48	42.0	LH5623	56	42.0	LH6423	64	45.5	LH7223	72	46.5
LH4824	48	46.0	LH5624	56	45.5	LH6424	64	47.0	LH7224	72	51.5
LH4825	48	54.0	LH5625	56	55.0	LH6425	64	59.5	LH7225	72	59.5
LH521	52	13.5	LH601	60	14.5	LH681	68	18.5	LH781	78	20.0
LH522	52	13.5	LH602	60	14.5	LH682	68	18.5	LH782	78	20.0
LH523	52	14.0	LH603	60	15.0	LH683	68	18.5	LH783	78	20.5
LH524	52	14.5	LH604	60	15.0	LH684	68	18.5	LH784	78	20.5
LH525	52	14.5	LH605	60	15.0	LH685	68	18.5	LH785	78	21.0
LH526	52	15.0	LH606	60	16.0	LH686	68	19.0	LH786	78	21.0
LH527	52	15.5	LH607	60	16.0	LH687	68	19.5	LH787	78	21.0
LH528	52	15.5	LH608	60	16.0	LH688	68	20.5	LH788	78	22.0
LH529	52	16.0	LH609	60	17.5	LH689	68	21.0	LH789	78	23.5
LH5210	52	17.0	LH6010	60	17.5	LH6810	68	22.0	LH7810	78	23.5
LH5211	52	18.0	LH6011	60	19.0	LH6811	68	22.5	LH7811	78	24.5
LH5212	52	20.0	LH6012	60	20.5	LH6812	68	23.0	LH7812	78	24.5
LH5213	52	20.5	LH6013	60	21.5	LH6813	68	23.5	LH7813	78	25.0
LH5214	52	21.5	LH6014	60	22.0	LH6814	68	25.5	LH7814	78	25.5
LH5215	52	22.5	LH6015	60	23.5	LH6815	68	26.0	LH7815	78	26.5
LH5216	52	23.5	LH6016	60	24.5	LH6816	68	28.0	LH7816	78	28.5
LH5217	52	26.0	LH6017	60	27.0	LH6817	68	30.0	LH7817	78	30.5
LH5218	52	28.5	LH6018	60	29.0	LH6818	68	30.5	LH7818	78	33.0
LH5219	52	30.5	LH6019	60	31.0	LH6819	68	33.5	LH7819	78	34.0
LH5220	52	32.0	LH6020	60	32.5	LH6820	68	37.0	LH7820	78	36.5
LH5221	52	36.5	LH6021	60	37.0	LH6821	68	40.0	LH7821	78	41.5
LH5222	52	40.5	LH6022	60	40.5	LH6822	68	43.5	LH7822	78	44.5
LH5223	52	43.5	LH6023	60	42.5	LH6823	68	45.5	LH7823	78	46.5
LH5224	52	46.0	LH6024	60	46.5	LH6824	68	50.5	LH7824	78	52.5
LH5225	52	54.0	LH6025	60	54.0	LH6825	68	58.5	LH7825	78	59.5

Table
B - 10 . 4

Long Span Joists Available in User Files									
NAME	D1	WT/FT		NAME	D1	WT/FT		NAME	D1	WT/FT
LH841	84	22.5		LH901	90	23.5		LH961	96	29.5
LH842	84	22.5		LH902	90	23.5		LH962	96	29.5
LH843	84	22.5		LH903	90	24.0		LH963	96	30.0
LH844	84	22.5		LH904	90	24.0		LH964	96	30.0
LH845	84	23.0		LH905	90	24.0		LH965	96	30.0
LH846	84	23.0		LH906	90	24.5		LH966	96	30.0
LH847	84	23.0		LH907	90	25.0		LH967	96	30.5
LH848	84	24.5		LH908	90	26.0		LH968	96	30.5
LH849	84	25.5		LH909	90	27.0		LH969	96	30.5
LH8410	84	25.5		LH9010	90	27.0		LH9610	96	30.5
LH8411	84	26.0		LH9011	90	26.5		LH9611	96	31.5
LH8412	84	26.0		LH9012	90	28.0		LH9612	96	31.5
LH8413	84	27.5		LH9013	90	28.5		LH9613	96	32.0
LH8414	84	30.0		LH9014	90	30.0		LH9614	96	34.0
LH8415	84	30.0		LH9015	90	30.5		LH9615	96	34.0
LH8416	84	32.5		LH9016	90	32.0		LH9616	96	35.5
LH8417	84	33.5		LH9017	90	34.0		LH9617	96	35.5
LH8418	84	35.0		LH9018	90	34.5		LH9618	96	35.5
LH8419	84	36.5		LH9019	90	36.5		LH9619	96	37.5
LH8420	84	39.0		LH9020	90	40.0		LH9620	96	41.0
LH8421	84	43.0		LH9021	90	43.5		LH9621	96	45.5
LH8422	84	45.5		LH9022	90	49.0		LH9622	96	49.5
LH8423	84	50.0		LH9023	90	50.5		LH9623	96	51.5
LH8424	84	53.0		LH9024	90	55.5		LH9624	96	56.5
LH8425	84	63.0		LH9025	90	63.5		LH9625	96	63.5

Table
B - 10 . 5

Short Span (Bar) Joists Available in User Files												
NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT
K81	8	4.5	K141	14	5.0						
K82	8	5.0	K142	14	5.5						
K83	8	6.0	K143	14	6.0	K183	18	6.5	K223	22	7.0
K84	8	6.5	K144	14	7.0	K184	18	6.5	K224	22	7.5
K85	8	6.5	K145	14	7.5	K185	18	7.0	K225	22	8.0
K86	8	7.0	K146	14	7.5	K186	18	7.0	K226	22	8.5
			K147	14	7.5	K187	18	7.0	K227	22	8.5
K101	10	5.0	K148	14	8.5	K188	18	7.5	K228	22	8.5
K102	10	5.5	K149	14	8.5	K189	18	8.0	K229	22	9.0
K103	10	6.0	K1410	14	9.5	K1810	18	9.0	K2210	22	10.0
K104	10	6.5	K1411	14	10.0	K1811	18	9.5	K2211	22	10.5
K105	10	7.0	K1412	14	11.0	K1812	18	10.5	K2212	22	11.5
K106	10	7.5	K1413	14	11.5	K1813	18	11.0	K2213	22	12.0
K107	10	7.5	K1414	14	13.0	K1814	18	13.0	K2214	22	13.5
K108	10	8.0	K1415	14	14.5	K1815	18	14.0	K2215	22	14.5
K109	10	8.0	K1416	14	15.0	K1816	18	14.0	K2216	22	15.0
K1010	10	9.5	K1417	14	16.0	K1817	18	15.5	K2217	22	16.5
K1011	10	10.0	K1418	14	17.0	K1818	18	17.0	K2218	22	16.5
K1012	10	11.5	K1419	14	18.5	K1819	18	18.0	K2219	22	19.0
			K1420	14	20.0	K1820	18	19.5	K2220	22	21.5
K121	12	4.5									
K122	12	5.5	K162	16	5.5						
K123	12	6.0	K163	16	6.0	K203	20	6.5			
K124	12	6.5	K164	16	7.0	K204	20	7.0	K244	24	8.5
K125	12	7.0	K165	16	7.0	K205	20	7.5	K245	24	9.0
K126	12	7.5	K166	16	7.5	K206	20	8.0	K246	24	9.0
K127	12	7.5	K167	16	7.5	K207	20	8.0	K247	24	9.0
K128	12	8.5	K168	16	8.0	K208	20	8.5	K248	24	9.5
K129	12	8.5	K169	16	8.5	K209	20	8.5	K249	24	10.0
K1210	12	9.5	K1610	16	9.0	K2010	20	9.5	K2410	24	11.0
K1211	12	10.0	K1611	16	10.0	K2011	20	10.0	K2411	24	11.5
K1212	12	11.0	K1612	16	11.0	K2012	20	11.0	K2412	24	12.5
K1213	12	11.5	K1613	16	11.5	K2013	20	11.5	K2413	24	12.5
K1214	12	13.0	K1614	16	13.0	K2014	20	13.0	K2414	24	14.0
K1215	12	14.5	K1615	16	14.5	K2015	20	13.5	K2415	24	15.0
K1216	12	15.0	K1616	16	15.0	K2016	20	14.5	K2416	24	15.5
			K1617	16	16.0	K2017	20	16.0	K2417	24	17.0
			K1618	16	16.5	K2018	20	16.5	K2418	24	17.5
			K1619	16	18.0	K2019	20	18.5	K2419	24	18.5
			K1620	16	20.0	K2020	20	20.5	K2420	24	21.5

Table
B - 10 . 6

Short Span (Bar) Joists Available in User Files								
NAME	D1	WT/FT	NAME	D1	WT/FT	NAME	D1	WT/FT
K267	26	9.5	K287	28	10.0	K307	30	10.0
K268	26	10.0	K288	28	10.5	K308	30	10.5
K269	26	10.5	K289	28	10.5	K309	30	11.0
K2610	26	11.5	K2810	28	11.5	K3010	30	12.0
K2611	26	12.0	K2811	28	12.0	K3011	30	12.5
K2612	26	13.0	K2812	28	13.5	K3012	30	14.0
K2613	26	13.0	K2813	28	13.5	K3013	30	14.0
K2614	26	14.5	K2814	28	15.0	K3014	30	16.0
K2615	26	16.0	K2815	28	16.5	K3015	30	17.0
K2616	26	16.0	K2816	28	16.5	K3016	30	17.0
K2617	26	17.0	K2817	28	18.0	K3017	30	18.5
K2618	26	18.0	K2818	28	18.5	K3018	30	19.0
K2619	26	19.0	K2819	28	19.5	K3019	30	20.0
K2620	26	21.0	K2820	28	21.0	K3020	30	22.0

Table
B - 10 . 7

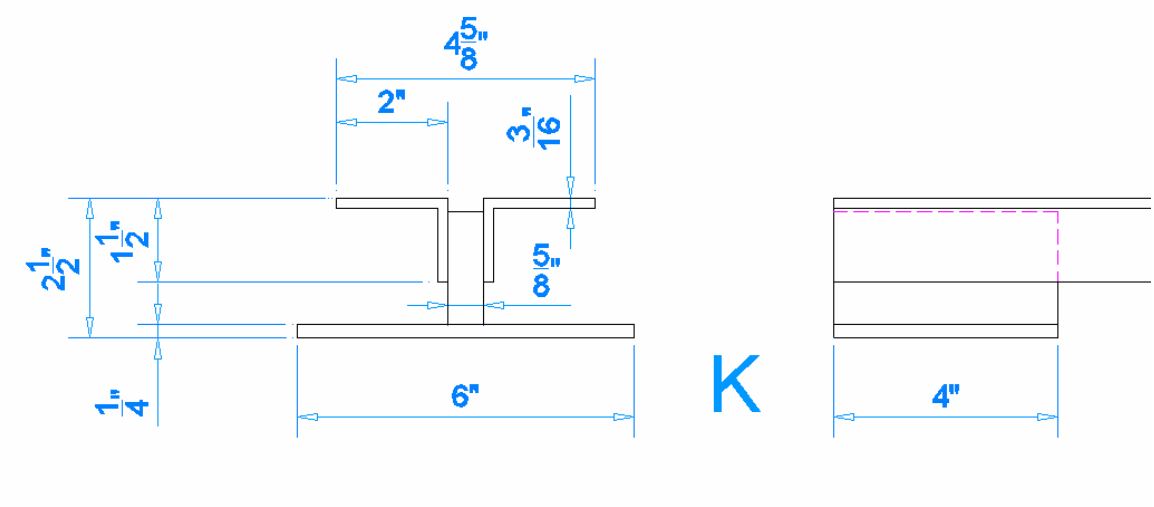


Figure
B - 27

Appendix C - STAAD User Files

The STAAD user files are installed on your hard disk at the time that you install ***Structural Desktop***. They can also be found on the CD-ROM that you used to install ***Structural Desktop***.

All of the members that are made available to you in these files require further design, once the loads and forces associated with those members have been determined. Using the members in the user files you can generate the selfweight loads, transfer loads, and provide a more complete model which includes the stiffness of the frames. The members you include from your user files will also be drawn in your drawings, bringing your drawings that much closer to completion.

To use these files in STAAD, you must include (copy) the user files in the appropriate folder (or directory). This should be the same folder that has the STAAD input file and the file must remain there during subsequent runs of the analysis file. Please refer to the STAAD or STAAD-Pro references, documentation, and help files for further information on the use of User Created Tables and to **Appendix B** for information about the members available under each type.

The files available to you for use with STAAD or ***Structural Desktop*** are as follows:

Name	Contents	Example
JG_DATA.DAT	Joist Girders	JG3218
LH_DATA. DAT	LongSpan Joists	LH2412
K_DATA. DAT	ShortSpan Joists	K1210
CEEZEE. DAT	LGSI and AISI	Z12X3512
L_T_DATA. DAT	Concrete L's and T's	T24IT48

A brief section of a STAAD file that would use some of these objects is listed as an example on the next page.

An Abridged Sample Staad File with concrete PCI, LGSI, AISI, and LongSpan Joists:

```
STAAD SPACE SOME CUSTOM MEMBERS
UNIT IN KIP
JOINT COORDINATES
1 0.000 0.000 0.000 ; 2 0.000 60.000 0.000
. . . . .

MEMBER INCIDENCES
1 1 2 ; 2 2 3 ; 3 3 4
. . . . .

START USER TABLE
TABLE 1 L_T_DATA.DAT
TABLE 2 CEEZEE.DAT
TABLE 3 LH_DATA.DAT
END
MEMBER PROPERTY
1 TO 36 PRIS YD 24.0
37 TO 49 UPT 1 R18LB20
50 TO 63 UPT 1 L18LB24
64 TO 77 UPT 1 T24IT20
78 TO 90 TABLE T W6X9
91 TO 100 UPT 2 C8X2514
101 TO 110 UPT 2 CS9X314
111 TO 120 UPT 2 CU8X1214
121 TO 130 UPT 2 Z10X3512
131 TO 140 UPT 2 HU10X514
141 TO 146 UPT 3 LH2412
CONSTANTS
E CONC MEMB 1 TO 77
E STEEL MEMB 78 TO 146
DEN CONC MEMB 1 TO 77
DEN STEEL MEMB 78 TO 146
ALPHA 6.9E-7 ALL
SUPPORT
. . . . .
```

Appendix D - The GT STRUDL User File

The GT STRUDL user file is installed on your hard disk at the time that you install ***Structural Desktop***. It can also be found on the CD-ROM that you used to install ***Structural Desktop***.

All of the members that are made available to you in these files require further design, once the loads and forces associated with those members have been determined. Using the members in the user file you can generate the selfweight loads, transfer loads, and provide a more complete model which includes the stiffness of the frames. The members you include from your user files will also be drawn in your drawings, bringing your drawings that much closer to completion.

To use this file in GT STRUDL, you must include the name of the user file (StructDT.ds) in the appropriate dialog text box when you open the program GT STRUDL and during subsequent runs of the analysis file. Please refer to the GT STRUDL references, documentation, and help files for further information on the use of this table and to **Appendix B** for information about the members available under each type.

The file contains the following types of members:

Name	Contents	Example
StructDT.ds	Joist Girders	JG3218
	LongSpan Joists	LH2412
	ShortSpan Joists	K1210
	LGSI and AISI	Z12X3512
	Concrete L's and T's	T24IT48

Appendix E - RISA 3D User Files

The RISA 3D user file is installed on your hard disk at the time that you install ***Structural Desktop***. The file is named **SDT.fil** and can be found in the folder:

C:\PROGRAM FILES\Structural Desktop\RISA User Files.

In the event that this file should be unavailable or corrupted, the **SDT.fil** file may also be found on the original Structural Desktop installation CD in the folder:

[CD-ROM]:\USER_FILES\RISA.

All of the members in these files require further design of the member itself, once the loads and forces associated with those members have been determined. Using the members in the user file will permit you to generate the selfweight loads for these members, transfer loads applied to these members to the structure, and will provide a more complete model which includes the stiffness of the frames. Any members you include from your user files will also be drawn in your drawings, bringing your drawings that much closer to completion.

To use this file in RISA 3D, you must copy the user file (**SDT.fil**) to your RISA directory on your hard disk. You must also be using RISA 3D with the **Version 4.5b** patch or later.

To determine which version of RISA 3D you are currently running, run the program and under "Help" on the drop-down menu, click "About". The dialog box that displays should reference Version 4.5b (or later). If this is not the case, go to the RISA website at:

http://www.risatech.com/

and download the latest patch. This patch was uploaded by RISA Technologies on 2/11/2002.

Structural Desktop is very grateful to the team at RISA Technologies. Due to their hard work, if the Structural Desktop file is in your RISA directory RISA will access those objects as arbitrary objects through the shape database under the group SDT.

Refer to RISA documentation, and help files for further information on the use of this table and the shapes database, and to **Appendix B** for information about the members available under each type.

The file contains the following types of members:

Name	Contents	Example
SDT.fil	Joist Girders	SD_JG3218
	LongSpan Joists	SD_LH2412
	ShortSpan Joists	SD_K1210
	LGSI	SD_Z12X3512
	Concrete L's and T's	SD_T24IT48

Note that each member has the prefix SD_ when selected from the dialog box in the RISA 3D program. This will be stripped from the member name when your RISA file is read by ***Structural Desktop.***

The file does not contain the AISI members (CS, CU, ZS, ZU, and HU) because these members were already available from the RISA shape database. You may use these members in your RISA file and they will import into ***Structural Desktop***.

Finally, you should check your numbering scheme before attempting to read a RISA file into ***Structural Desktop. Structural Desktop*** works only with numeric member and plate identifiers. ***Structural Desktop*** assumes that you will use a letter in the naming of joints, members, and plates and removes that letter. However, ***Structural Desktop*** also requires that the numbering of each joint be unique within the joints, of each member be unique within the members, and each plate be unique within the plates. Luckily, RISA has a Relabel command available either through the menu or by right-clicking on the data entry template for joints, members, and plates. We suggest you relabel Joints N1 - N???, Members M1 - M???, and Plates P1 - P???. This will permit unique labeling of each object with a number that is not repeated in objects of the same type.

Appendix F - Version History

Changes to Versions 1.7 through 3.0 are documented below.

Version 1.7 Compiled June 2002 -

Version 1.7 included the following new concepts/commands:

- The **Intersect Break** command. (SDBreak)
- The **OpenSTAAD Interface** command. (OpenSTAAD)
- The **Create 3D ADT Model** command. (SDADTScript)

Version 1.7 also included the following improvements:

- Improved interface action for Change Member Parameters dialog box
- Improved interface action for Change Joist Parameters dialog box
- Changed Fuse functions to distribute changes-to-a-member to the members that compose it. This is important because components of a member are written to input files instead of members made up by fusing other members, so changes made to a member (that was made out of fused members) are now distributed to the input file members.
- Defusing a member now defuses it totally, not incrementally.
- Completed Generation of STAAD input files to include properties and constants
- Completed Generation of GT STRUDL input files to include properties and constants
- Additional members for AISC tables, Custom Tables

Version 1.8 Compiled September 2002 -

Version 1.8 included the following new concepts/commands:

- The **Load SAP 2000** file command. (SDLoadCSI)
- The **Create SAP 2000** file command. (SDCTCSI)

Version 1.8 also included the following improvements:

- Improved DELETE command to distribute deletion to sub-members of a fused member. The members were no longer available or accessible but were locked in the database.

Version 1.9 Compiled February 2003 -

Version 1.9 included the following new concepts/commands:

- The **Create RISA 3D** file command. (SDCTRISA)
- The Structure Line Generator (SDLineWiz)

Version 1.9 also included the following improvements:

- Minimum Joint Space Selection - permitting the user to specify how close joints can be in a model created by Structural Desktop before they are considered the same model joint. Accounted for joints created that can be less than a 1,000,000th of an inch apart due to infinitesimal model discrepancies.
- Joint Display Size Selection - permitting the user to specify the display size of the individual circles representing joints.
- Introduced Tutorial and Manual available through Adobe® PDF formats from the Structural Desktop menu.
- Permitted Line-to-Member transformation of lines not on Layer 0 to preserve layering in complex, pre-drawn AutoCAD drawings.
- Created an optional upgrade for users of AutoCAD 2004 (May 2003) to account for total incompatibility between ARX modules for AutoCAD 2000 - 2002 and AutoCAD 2004.

Version 1.95 Compiled October 2003 -

Version 1.95 included the following improvements:

- Modified Component Table to accept variable Tables from multiple countries. Modified base tables for Database Format. Component Tables now Tree based rather than Spreadsheet based in display format.
- Modified Input Routines to permit input of a model without closing AutoCAD between different models.
- Button Toolbar introduced for toggle between Envelope and Line display modes.
- Significant groundwork laid for additional improvements in Versions 2.0 and 2.1 currently under development.

Version 2.0 Compiled August 2004 -

Structural Desktop Version 2.0 is based on Version 1.9 and incorporates changes that were released in overseas markets as Version 1.95. This version tested a new database format for tables of sections and a separate module for section selection. American, Canadian, British, and Euro sections were included in tables that are wholly Metric, designed for use with input and output files that are metric in measurement.
These tables currently work with STAAD.Pro and GT STRUDL files.

The regular American Tables (working only with Imperial units) were also made available in Version 1.95 for use with all supported programs.

The unique improvements between Version 1.95 and Version 2.0 are:

STAAD.Pro Interface Improvements

Tapered Sections

- Tapered I sections
  Custom Tapered I Sections are now read from the STAAD.Pro file, can be changed or created in Structural Desktop, and are written to the STD input files when Structural Desktop creates them. The weight and dimensions of these members are included in the Material Report created by Structural Desktop.

- Tapered Tube Sections
  Custom Tube Sections are now read from the STAAD.Pro file, can be changed or created in Structural Desktop, and are written to the STD input files when Structural Desktop creates them. . The weight and dimensions of these members are included in the Material Report created by Structural Desktop.

- Piles
  Structural Desktop now includes a PILE member for drawing and Material Report purposes. This member can be specified in terms of starting and ending depth of a round member and is treated as a truncated cone. Density for the Pile can specify a Wood, Concrete, or Steel pile. The parameters of this member are not written to the STAAD.Pro file beyond the endpoints and location/member number assigned.

Offsets

Structural Desktop Version 1.5 read Offsets from a STAAD file. Version 2.0 now writes the offsets back to a STAAD.Pro file to preserve your work in a back-and-forth situation. The addition of offset information can also be useful for users who wish to use the STAAD.Pro file in generation of a CIS/2 format file for further processing.

STAAD.Pro Interface Improvements (continued)

Releases

Structural Desktop now permits releases to be assigned to members quickly. When you load Structural Desktop, three new linetypes are automatically loaded into AutoCAD. These are

REL_BOTH -- < > -- Released Both Ends (MY MZ)
REL_START -- < -- Released at the Start (MY MZ)
REL_END -- > -- Released at the End (MY MZ)

Changing a Structural Desktop member to any of these mutually exclusive linetypes will update the internal database. Thereafter, the member will display the linetype on the model drawing and creation of a STAAD.Pro file will export these releases to STAAD.Pro. This functionality does not currently extend to reading the releases from the input file, although that is planned for the next release of Structural Desktop.

GT STRUDL Interface Improvements

Structural Desktop Version 2.0 has been upgraded to reflect changes in GT STRUDL table names for the changes between Version 26 and Version 27.

When generating a GT STRUDL input (GTI) file, the user is now prompted for (P)hysical or (A)nalytical members. Selecting Analytical members will produce the same GT STRUDL file as before, with fused members being broken into their sub-components and any changes to the fused member being propagated to the sub-members. Selecting Physical members will write a GT STRUDL file that contains the fused members in preference over the sub-members, and all other regular members that are neither fused nor part of a fused member.

Offsets in Structural Desktop are now propagated to the GT STRUDL input file as ECCENTRICITIES. This preserves the offsets for analytical purposes, for re-importation into Structural Desktop, and for use (with the Physical member option outlined above) in creating CIS/2 files with true-length members for export to other programs through the GT STRUDL CIS/2 export function.

RISA 3D and RISA Floor Interface Improvements

Structural Desktop Version 2.0 is now configured to read RISA 3D files for versions through RISA 3D Version 5.05, current as of August 1, 2004.

Structural Desktop Version 2.0 includes an option under the RISA Interface menu item for reading of RISA Floor files. Each layer of the RISA Floor file is imported at the elevation specified in the file for that layer.

Version 3.0 Compiled August 2005 -

General Improvements - A New Command

Structural Desktop Version 3.0 incorporated the new **Recover Data from Hold File** command in order to retrieve information lost when a file was written to an analytical model and changed, then read into Structural Desktop from the modified analytical files. Users were using Structural Desktop to create complex models with thousands of members and elements, complete with offsets, fusing, and AutoCAD layering layouts but this information was lost going from SDT to analytical and then reading the modified analytical file back to SDT. This command recovers the relevant data from a user-selected hold file written prior to the creation of the analytical file and automatically creates and assigns layers, fusing, and other SDT file information, while NOT making any changes to new, erased, or changed members.

General Improvements - Improved Stability

Version 3.0 of Structural Desktop uses a new security system to limit unauthorized copies. This system is compatible with all current versions of the Windows Operating System from Microsoft. Other changes have been made to Structural Desktop to continue to support all versions of AutoCAD to include: AutoCAD 2000, AutoCAD 2000i, AutoCAD 2002, AutoCAD 2004, AutoCAD 2005, and AutoCAD 2006. This also includes support for Architectural Desktop Version 3.3, Architectural Desktop 2004, Architectural Desktop 2005, and Architectural Desktop 2006.

Structural Desktop continues to read and write HOLD files consistently between all versions of Structural Desktop that have been offered to the public from Version 1.5 to the present. This includes all versions of Structural Desktop LT, and all variations of Structural Desktop version 3.0. You can continue to share information in the HOLD file format between users of Structural Desktop under AutoCAD 2000 with users of Structural Desktop under AutoCAD 2006 without loss of data

STAAD.Pro Interface Improvements - HSST Members are now supported.

The Hollow Steel Structural Tube objects from STAAD.Pro are now supported in **Version 3.0 of Structural Desktop**. These members can be written to a STD file, read from a STD file, changed in Structural Desktop to another type or size, and will produce 3d models, 2d drawings, and Material Report output separate from and separate from the TS members from the earlier STAAD.Pro database. (e.g. TUB80808)

RISA 3D Improvements - RISA 3D Version 5.5 is now supported

Structural Desktop Version 3.0 was recompiled to provide support for the new file format for RISA 3D version 5.5. SDT still reads prior file formats for the RISA 3D file data.

CSI SAP2000 Improvements – SAP2000 Version 9 is now supported

Structural Desktop Version 3.0 was recompiled to provide support for changes in the S2K file format for version 9. SDT still reads prior file formats for versions 7 and 8.

Version 4.0 Compiled February 2008 -

General Improvements - New Options

Structural Desktop Version 4.0 incorporated the new **Adjust Members Elements: Move All** functionality to permit models to move, rotate, or stretch groups of Joints without changing their offsets relative to their members, or return to the offset-based methods at will in order to adjust positions of members for drawing purposes without affecting the underlying analytical file.

General Improvements – Expanded Support

Structural Desktop Version 4.0 now includes three separate installation modules. One installation supports AutoCAD 2000, 2000i, and 2002. The second installation supports AutoCAD 2004, 2005, and 2006. The newest installation supports AutoCAD 2007, 2008 and AutoCAD 2009. In order to provide this support, **Structural Desktop** was split into ten separate modules, with only one dependant upon AutoCAD programming. This takes full advantage of more advanced Windows and multi-core personal computer CPU modules while remaining friendly to lower-powered, older systems.

STAAD.Pro Interface Improvements – The Full Round Turn

Structural Desktop Version 4.0 now reads and retains the information from Supports, Releases, Loads, and Load Combinations, Parameters, and even different "Print" and "Perform Analysis" statements in the original STAAD.Pro file. This information is written back to any STAAD.Pro file generated from the model.

GT STRUDL Interface Improvements – The Full Round Turn

Structural Desktop Version 4.0 is a complete refactoring of the approach to GT STRUDL used in previous versions. **Structural Desktop** no longer relies upon the user to generate the GT STRUDL dbx files in the proper order and with the proper names, but rather reads and writes the input files for GT STRUDL with the *.GTI extension. The ability to both read and write the same file formats that are comprehensible to humans also permits the extension of the format to include Eccentricities, Releases, Loads, and Load Combinations, and Parameters. This information is both read and written back to any GT STRUDL file generated from the model.

RISA 3D Improvements - RISA 3D Versions 5.5 – 7.0 are now supported
- and the Round Turn Too

Structural Desktop Version 4.0 is a complete refactoring of the approach to RISA 3D used in previous versions. **Structural Desktop** formerly read RISA 3D files from Version 4.52 to Version 5.5, and all files that were written from **Structural Desktop** were written in the Version 4.52 format. We have now moved our support base to Version 5.5. **Structural Desktop** writes this version, which can be read by all Versions of RISA 3D Software 5.5 or greater, and reads all versions from 5.5 to 7.0. **Structural Desktop** extended reading capability to include Boundary Conditions, Grid Lines, Releases, Loads, and Load Combinations, and Parameters. This information is both read and written back to any RISA 3D file generated from the model.

Appendix G - Working with Architectural Desktop

Structural Desktop (SDT) / Architectural Desktop (ADT) Interface

Installation and Setup for ADT Users

Structural Desktop (SDT) works with a proprietary set of objects created separately from and with different data and properties than ADT structural objects. The internal database for SDT retains information required for two-way interface with structural analysis packages, while this information is not necessary for ADT. In order to minimize drawing files and give the best performance possible with any computer, it is strongly recommended that SDT be run in a plain "Vanilla" AutoCAD environment, and that ADT be run in a separate environment and with the specifications set by ADT documentation and any additions you may have. Setting this up is not difficult, requires only a few minutes, and only has to be performed once for each machine upon which you wish to run SDT.

Once you have installed Structural Desktop in your Program Files directory folder on drive C, there is a simple five-step process to set up three different icons on your desktop. One icon will invoke Architectural Desktop with no changes to the program and with all of the unique capabilities it provides. The second icon will invoke a plain AutoCAD session and load Structural Desktop for you. The third icon will open Architectural Desktop with a template that holds definitions for all of the supported structural members. You will use this template file to create the final model in ADT from your Structural Desktop model.

STEP 1 - Create a Script file to run Structural Desktop

The file SDT.SCR contains the following line:

arx L "c:\Program Files\Structural Desktop\consConsEnt.arx"

Copy this file into your **Architectural Desktop** directory. SDT.SCR can be found in your Structural Desktop directory under a sub-folder named ADT, and on the installation CD-ROM that came with your Structural Desktop package. (This complete directory is also available for download from the Structural Desktop website.) The script file can be called from within a Windows shortcut (a desktop icon), and will load Structural Desktop when that shortcut is invoked.

Please note that a space is included at the end of the line of text after the closing quotation mark. This executes the command without having to press the [Enter] key each time you run Structural Desktop.

STEP 2 - Create a Profile to run Basic AutoCAD

Start your copy of Architectural Desktop.

Go to the **Options** menu item.
(This is found at the bottom of the right-click menu or the **Tools** menu item.)

Click on the **Profiles** tab.

Click the **ADD to List** button.

Enter the Profile Name "SDT".
(If you wish, you may enter a description such as "Structural Desktop")

Click the **Apply & Close** button.

Click on this newly created (SDT) profile and then click the **RESET** button

Click **Yes** to the Alert that pops up. You **DO** want to reset this profile.

Click **OK** on the **Options** Dialog Box to close it.

Exit Architectural Desktop. You do not need to save the drawing file.

Note: Creating a separate profile will guarantee that no AEC functions are demand loaded. This permits you to get maximum value from your system memory and resources when you are running Structural Desktop and helps to separate the two programs.

STEP 3 - Create Alternate Desktop Icons for Architectural Desktop

If you already have a Desktop Icon to load Architectural Desktop, right-click on the icon and select **Copy**.

If you do not, click the **Start** button for Windows, find the link to Architectural Desktop, right-click on this item and select **Copy**.

Click on a blank portion of your screen, right-click and select **Paste**. Create a total of **Three** such Icons by pasting twice on your desktop. Right-click on each copy and select **Rename**. You may rename them to whatever you find appropriate, such as **Structural Desktop** and **SDT to ADT**.

STEP 4 - Copy Template Files and Modify First Desktop Icon

In your **Program Files/Structural Desktop/ADT** directory you will find two **Template** files: **SDT.dwt** and **SDTADT.dwt**. Copy these two files to your **Architectural Desktop/Template** directory. **SDT.dwt** is a plain AutoCAD template with linetypes defined for Structural Desktop. **SDTADT.dwt** is an Architectural Desktop template based on **Aec Arch (Imperial - Intl).dwt** with all of the Structural Desktop members that ADT supports defined and given names that will work with Structural Desktop.

Right-click the Shortcut you created for **Structural Desktop** and select **Properties** from the drop-down menu. Select the **Shortcut** tab. You wish to edit the **Target** text box.

Your default will look something like this before you begin:

"C:\Program Files\Autodesk Architectural Desktop 3\acad.exe" /t "Aec Arch (Imperial - Intl).dwt" /p "C:\Program Files\Autodesk Architectural Desktop 3\adt.arg"

This shortcut loads AutoCAD, loads a template drawing, and loads the necessary ADT program modules to supply the ADT user interface and custom objects. To change this icon to load **Structural Desktop**:

Change **/t "Aec Arch (Imperial - Intl).dwt"** to **/t "SDT.dwt"**
Change **/p "C:\Program Files\Autodesk Architectural Desktop 3\adt.arg"** to **/p "SDT"**

Add **/b SDT** to the end of the target text. This invokes the script file mentioned in step 1 above to load Structural Desktop.

Note: You can copy and paste the text from the target box into a blank notepad file for editing and then paste the results back into the target box on the shortcut properties dialog.

If your ADT is on drive C in the program files area, the target box will now read:

"C:\Program Files\Autodesk Architectural Desktop 3\acad.exe" /t "SDT.dwt" /p "SDT" /b SDT

or, if you installed ADT on drive D, you may require:

"D:\Autodesk Architectural Desktop 3\acad.exe" /t "SDT.dwt" /p "SDT" /b SDT

Close the **Properties** dialog

Structural Desktop will now load when you click on this icon. This sequence causes AutoCAD to load, loads the **Template** file for **Structural Desktop**, loads a **Profile** without any calls to **Architectural Desktop** functions or menus, and uses a **Script** to load the **Structural Desktop** program.

Any changes you wish can be made to the **Template** file (**SDT.dwt**) such as adding more linetypes, changing dimension styles or text styles, etc. Removing linetypes or adding layers can affect the performance of Structural Desktop and should be tested on a case-by-case basis.

If you require assistance with this, do not hesitate to contact Structural Desktop, Inc.

STEP 5 - Modify Second Desktop Icon

The other Windows Icon is used to load **Architectural Desktop** and a drawing **Template** that permits import of **Structural Desktop** members as **Architectural Desktop** members. This icon will require ONLY that you change the **Template** file referenced by the shortcut.

The result of this change will look like:

"C:\Program Files\Autodesk Architectural Desktop 3\acad.exe" /t "SDTADT.dwt" /p "C:\Program Files\Autodesk Architectural Desktop 3\adt.arg"

Any changes you wish can be made to the **Template** file (**SDTADT.dwt**) such as adding more linetypes, changing dimension styles or text styles, etc. Do not purge any of the Structural Member styles from the Template file; you **may** purge unused styles from your drawing once the Structural Desktop members have been imported. Once this has been accomplished, you will be totally within an Architectural Desktop session, and any questions on that program or the Architectural Desktop Structural Members should be researched through Architectural Desktop documentation and help resources.

Structural Desktop (SDT) / Architectural Desktop (ADT) Interface

Transference of files from SDT to ADT

Structural Desktop is designed to work as an entity unto itself. The functions required to create a valid model in Structural Desktop from an analytical file or from scratch have been simplified and minimized with users of existing Structural Analysis Programs in mind. Once the model has been created, modified, and completed the user may extract 2D drawings or 3D models for use with any AutoCAD-based program.

More advanced users of **Architectural Desktop** will wish to have structural members in a format compatible with the **Architectural Desktop 3.3** standard. With these, they can complete their ADT Building model, rather than extracting 2D drawings to block or insert into a set of contract documents.

On the ***Structural Desktop*** menu, version 1.7 contains a new entry: **Create 3D ADT Model**. When you click on this entry, every Structural Desktop Member will be accessed by the program and processed into a script file named **SDT_to_ADT.SCR**.. This script will draw these members inside of Architectural Desktop 3.3. It is important that you complete all offsets and fusing prior to using this function, as the location and number of members will be determined at the time you create the script file. Once you have opened an Architectural Desktop session and run the script file, the resulting members are 100% Architectural Desktop members.

The structural members are dependant on the definitions in the initial drawing file. You may set up an Icon to start an Architectural Desktop session that uses the provided template. You also have the option of running a regular Architectural Desktop session and choosing to start a new drawing using this template, or you may wish to open a new drawing once with the template and save this as a drawing file. Then, every time you wish to have Structural Desktop members, you may open this empty drawing file (that has all the structural members defined) and import the script file processed by Structural Desktop. Then, use the "Save as" function in Architectural Desktop to save this drawing under its final name.

To run the Script file, start **Architectural Desktop. Make certain that you have loaded the template file, or open a drawing file, that contains the Member Styles under the naming convention provided with your Structural Desktop program.** This can be confirmed by clicking the menu item **Design**, **Structural Members**, and **Member Styles.** There should be defined styles for wide flanges, joists, angles, channels, and others.

Type "**script**" on the **Architectural Desktop** command line. When the dialog box comes up, select the script **SDT_to_ADT.SCR**. Extremely large files on older machines (such

as Pentium II processors) may take a half-hour to draw all of the members specified with beta angles and proper centroid locations for the members.

Once the program scripting has completed, you may purge the Member Styles. Only those members not in the current model will be purged automatically. This is covered in the Architectural Desktop documentation.

You may make whatever changes you wish to the drawing template, adding layers or linetypes or changing dimensional styles or grid information, but do not remove the structural members from the template or initial drawing that you intend to use. Further, automated drawing does not work well when the **OSNAP** functions have been set, so **Structural Desktop** starts each script file with a command to nullify the SNAP functions.

Once you have imported the members, purged unused styles, and saved your drawing under the name that you have chosen, you are solely in an **Architectural Desktop** session**.** For questions about **Structural Desktop**, please see your users guide or contact our technical assistance group. For questions relating to **Architectural Desktop** functions and objects, please use the avenues provided with that program by **AutoDesk**.

If you are unsure as to whether a problem with structural members relates to Structural Desktop or Architectural Desktop, please DO NOT HESITATE to call Structural Desktop, Inc. first.

Appendix H - Notes for International Users

Structural Desktop **Version 1.95** has been designed specifically with the purpose of introducing this program to engineers in countries other than the United States. This utility is designed to read and write STAAD.Pro and GT STRUDL files. Information read in one program may be written to the other with the following limitations:

- A file generated and usable in one analytical program can create an input file with members not recognized by the other analytical program.

- Structural Desktop works with either Metric or Imperial units, but is not able to jump between the two systems. The user should work consistently within the Measurement system that the final drawings will be created in. Output drawings can be scaled to Meters or mm after they are produced from Structural Desktop, using the scale function of AutoCAD.

- Two tables of American Members (with identical contents) are provided. One table is wholly Metric and the other is wholly Imperial.

- Canadian, British, and Euro sections are included in tables that are wholly Metric. Only Metric input and output files can be associated with the members in these tables.

When Structural Desktop reads an analytical file in STAAD.Pro, it will assume that measurements in inches will relate to American Tables in inches. Using the statement "Member Property" instead of "Member Property British", for example, will also default to the American Tables. Use of Metric values will prompt the program to look for specific clues in your analytical file that will determine the tables to be loaded. If you open an analytical file of any particular type, in order to properly load a file of another Country Code, you must close AutoCAD and run the program again. Each file is limited to a single Properties definition and only one Country Database can be associated with a file.

To load the DBX files created by GT STRUDL, you must select the country that your members relate to in the Preferences dialog box of Structural Desktop before indicating the directory in which the DBX files reside. Remember to use the default names when creating DBX files in GT STRUDL, and models should be constrained to those files displayed under GT STRUDL when the respective country (BRITISH, EUROPEAN) is selected.

CANADIAN Tables

The following member types have been provided in the Structural Desktop database for engineers who use Canadian tables and codes.

W types, or I - shaped members:	W, WW, M, S, HP
Channels	C, MC
Steel T's:	WT, WWT, MT, ST, HPT
Tubes, Square and Rectangular	
Angles	
Double Angles, both Short and Long Leg Back-to-Back	
Pipes	

Prismatic members, either steel or concrete, can be defined as per each program and will display. All dimensions are assumed to be in Metric and are converted to Centimeters inside Structural Desktop.

UK Tables

The following member types have been provided in the Structural Desktop database for engineers who use UK tables and codes.

W types, or I - shaped members:	UC, UB, UP, JO
Channels	CH
Steel T's:	UB and UC Tees
Tubes, Square and Rectangular	
Angles	
Double Angles, both Short and Long Leg Back-to-Back	
Pipes	

Prismatic members, either steel or concrete, can be defined as per each program and will display. All dimensions are assumed to be in Metric and are converted to Centimeters inside Structural Desktop

EURO Tables

The following member types have been provided in the Structural Desktop database for engineers who use European tables and codes.

W types, or I - shaped members:	DIL, HE, IPE, IEPA, IPEO, IPER, IPEV, IPN
Channels	U, UPN
Tubes, Square and Rectangular	
Angles	
Double Angles, both Short and Long Leg Back-to-Back	
Pipes	

Prismatic members, either steel or concrete, can be defined as per each program and will display. All dimensions are assumed to be in Metric and are converted to Centimeters inside Structural Desktop

American Metric Tables

All member types have been provided in the Structural Desktop database for engineers who use the American tables and codes but wish to use metric values. The Nominal nomenclature remains as in the A.I.S.C. tables and reflects the Imperial units; for example, the W14x99 is still called out by that designation. However, if you wish to use metric measurements, the alternate American Metric Table will be invoked and the units for the members therein are designated internally for display on your drawing in a CM per drawing-unit scheme.

Prismatic members, either steel or concrete, can be defined as per each program and will display. All dimensions are assumed to be in Metric and are converted to Centimeters inside Structural Desktop With the American Metric units, all Structural Desktop User Provided Tables (Joist and Joist Girders, Concrete L's and T's, and Light-Gage sections) can be used in STAAD.Pro or GT STRUDL, since the units will be converted by these programs for you. When the members are read into Structural Desktop, they will be matched and drawn from Metric tables of measure included in the American Metric Database that we provide.

Any questions or suggestions about the Member Tables provided can be addressed through our in house support email, support@figure5.com or through your local dealer.

Index

E **Page**

F

G

H

I

J

K

L