



# **STRUCTURAL DESKTOP and STRUCTURAL DESKTOP LT TUTORIAL**

## **Contents**

<b>1.....</b>	<b>Introduction - Using this Tutorial</b>
<b>2.....</b>	<b>Step I - Loading an Analysis File</b>
<b>7.....</b>	<b>Step II - Adjusting the Display</b>
<b>10.....</b>	<b>Step III - Fusing Members</b>
<b>14.....</b>	<b>Step IV - Setting up Layers</b>
<b>22.....</b>	<b>Step V - Offsetting Members</b>
<b>40.....</b>	<b>Step VI - Offsetting Elements</b>
<b>47.....</b>	<b>Step VII - Grid Lines and Contract Documents</b>
<b>54.....</b>	<b>Conclusion</b>

## **Structural Desktop® Tutorial**

© 2004 Structural Desktop, Inc. All rights reserved.

Structural Desktop, Inc. reserves the right to revise and improve its products as it sees fit. This publication describes the state of this product at the time of its publication, and may not reflect the product at all times in the future. Structural Desktop makes no warranty, express or implied, as to the suitability of the software for any specific project, nor suggests substitution of software for good engineering practice and careful checking of results, drawings and documents created by this software.

### **Structural Desktop®**

Registered, U.S. Patent and Trademark Office  
Structural Desktop, Incorporated

**Autodesk, Inc., Autodesk, AutoCAD, AutoCAD Architectural Desktop** and **ObjectARX** are registered trademarks or trademarks of Autodesk, Inc.

**STAAD.Pro™** is a registered trademark or trademark of Research Engineers International, a division of netGuru, Inc.

**GT STRUDL®** is a registered trademark of the Computer-Aided Structural Engineering Center, Georgia Institute of Technology

**RISA** as applied to structural engineering software is a trademark of RISA Technologies.

**SAP2000** is a product and trademark of Computers and Structures Inc.



**Structural Desktop inputs a file from analytical software and converts this information into a Structural Desktop Model. Structural Desktop gives you, the engineer, the tools to control this process from the design phase to the production of final structural contract documents.**

## **Introduction**

This tutorial is created from the files of a structural project. The tutorial begins by directing you through the steps to read a file created by an analysis/design program into ***Structural Desktop***. You will learn to use the ***Structural Desktop*** commands to create, improve and finalize a ***Structural Desktop*** Model of the structure. This forms the basis to produce structural contract documents. After the completion of this tutorial, you will be able to:

- Transfer files from an analysis program to ***Structural Desktop***.
- **OR**
- Read a pre-prepared HOLD file into ***Structural Desktop LT***
- Create a ***Structural Desktop*** model of the structure.
- Fuse members together.
- Set up layers.
- Use the commands in ***Structural Desktop***.
- Create 2D plans, elevations and sections for Contract Documents.
- Create a 3D ***Structural Desktop*** solid model of the building.
- Create a Bill of Materials for this structure.

When you load the analysis file into ***Structural Desktop***, you will see a graphic line-drawing identical to **Figure T1.4**. This is a 3D model of the structure read from an input file. The model contains columns, beams, joists, joist girders, footings, foundation walls and double tee wall panels. ***Structural Desktop*** has the ability to read and utilize the data created for the analysis program to create structural contract documents.

In structural analysis programs, the engineer uses centroid lines and faces (elements) with design properties rather than structural objects to design and create the layout of a structure. When you load the input file from the analysis program into ***Structural Desktop***, you are going to see the centroids and faces. Other display modes provide envelopes or 3D solid objects to assist you in visualizing the model.

## **Using this Tutorial**

To use this tutorial, you should have knowledge of working with programs written for the Windows environment: opening and saving files and navigating on your computer. You should be familiar with structural analysis programs utilized in structural engineering design. You must have **AutoCAD 2000** (or later) and the ***Structural Desktop*** program or demo properly installed. The file **TUTORIAL.STD** should be copied to your **AutoCAD 2000** folder. (**TUTORIAL.HLD for users of SDT LT**) You will get maximum benefit from following along on the computer while reading this Tutorial, and taking the steps with us.

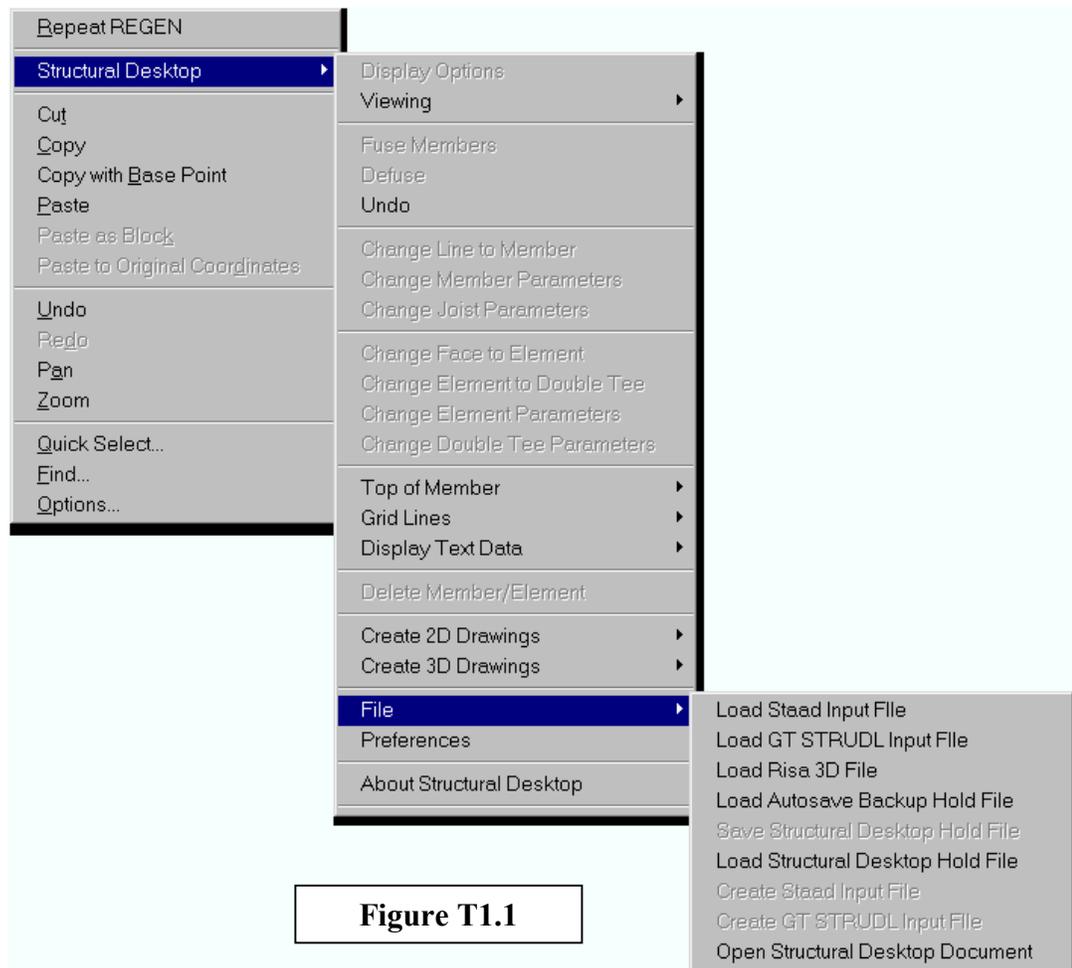
## Step I - Loading an Analysis File

*Structural Desktop* has two methods of accessing commands. The **Command Line Interface** permits you to type commands, just as with AutoCAD commands, from the command line. A chart with these commands is provided at the end of the Tutorial.

The method we will use in the Tutorial will be referred to as the "Right-Click Menu". This is referred to as the "Context-Sensitive Menu" in some Microsoft literature. To see the menu, click the **right-most** button on your two-or-three button mouse. To select from the menu, click the **left** button on the command you wish to enter.

Open your AutoCAD program, and make certain that you loaded *Structural Desktop* with the load-option of your choice...see **Installation Instructions: Getting Started** that was included with your program for details as to this process.

With the programs loaded, click the right button of your mouse. You should see the "Right-Click Menu". Move the mouse over *Structural Desktop* and down the command list until you come to the "File" section. You should see what is shown in **Figure T1.1** below.



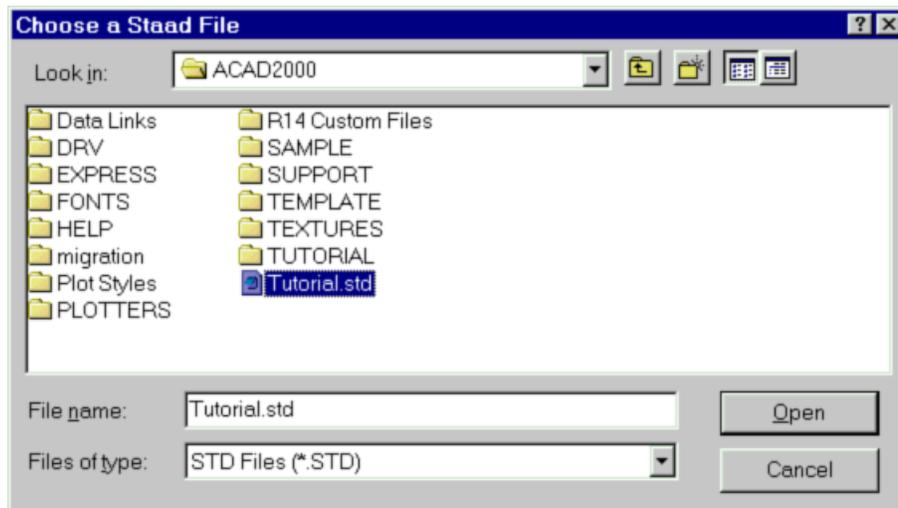
*Structural Desktop LT* Users: Please skip to page 4

**Structural Desktop Users:**

For reference, we will usually refer to a command with the entire "tree" of that command. This should make finding each command easier. In this case, we wish to click on **Load Staad Input File** in the rightmost menu box, so that command is:

**Structural Desktop** → **File** → **Load Staad Input File**

Click on that command and you will see the Dialog Box displayed in **Figure T1.2** below.



**Figure T1.2**

Select the file *Tutorial.std* for this exercise, either by double-clicking on the file in the window or selecting it, as shown above, and clicking on "Open". You should then see the image appear as shown in **Figure T1.4** on page 5.

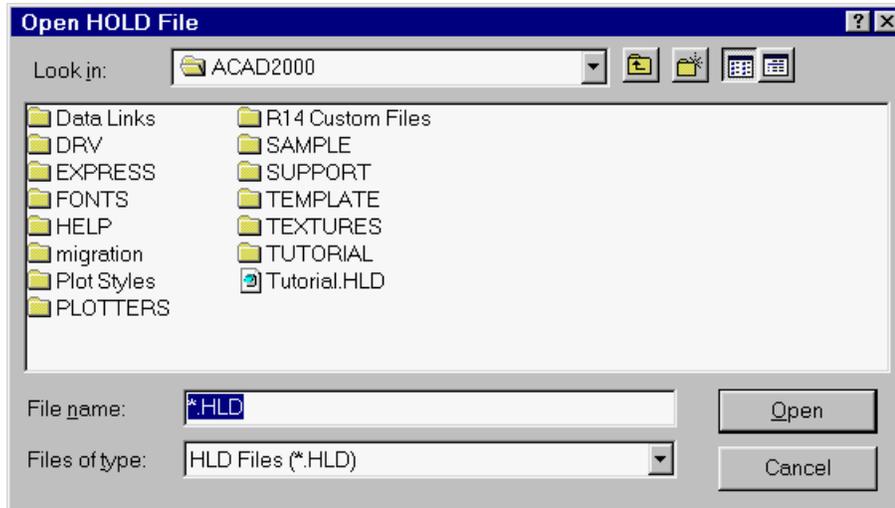
*Structural Desktop* uses Hold files to save your work between sessions. These hold files may be exchanged with other users of *Structural Desktop*, or *Structural Desktop LT*. Page 4 deals with the loading of a hold file. While it is addressed to users of *Structural Desktop LT*, the process is also available to users of *Structural Desktop*. Since hold files are covered again further in the text, feel free to skip to Page 5.

**Structural Desktop LT Users:**

For reference, we will usually refer to a command with the entire "tree" of that command. This should make finding each command easier. In this case, we wish to click on **Load Structural Desktop Hold File** in the rightmost menu box, so that command is:

**Structural Desktop** → **File** → **Load Structural Desktop Hold File**

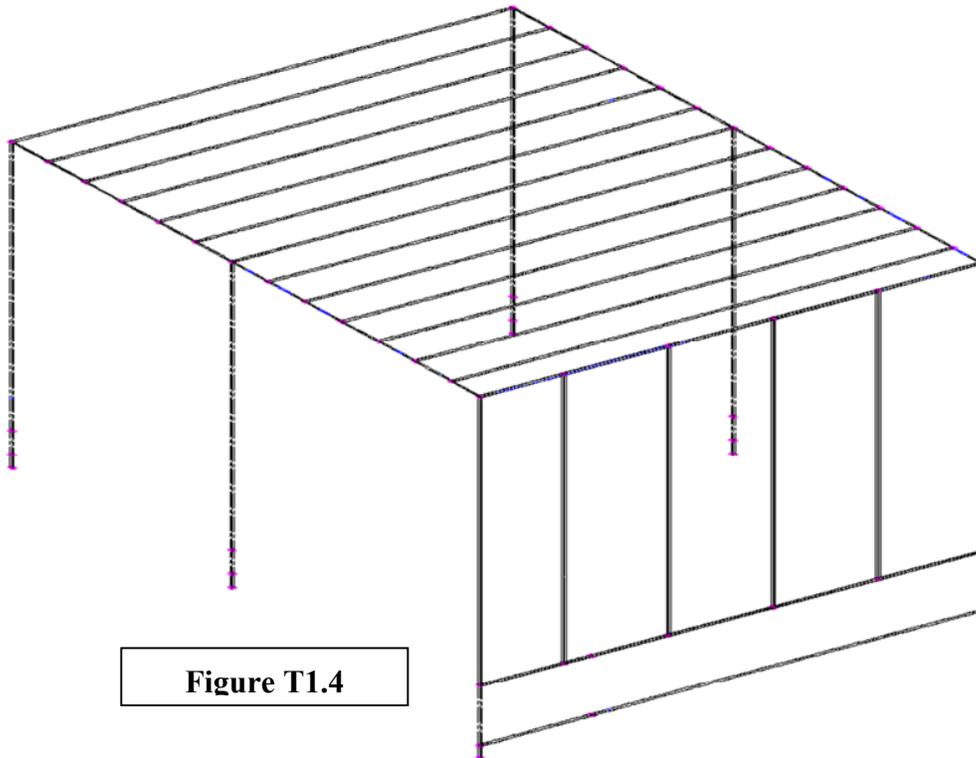
Click on that command and you will see the Dialog Box displayed in **Figure T1.3** below.



**Figure T1.3**

Select the file Tutorial.HLD for this exercise, either by double-clicking on the file in the window or selecting it, as shown above, and clicking on "Open". You should then see the image appear as shown in **Figure T1.4** on page 5.

**Structural Desktop LT** uses Hold files to save your work between sessions. These hold files may be exchanged with other users of **Structural Desktop**, or **Structural Desktop LT**. We provide you with a hold file so that you can proceed directly with the Tutorial without having to build a model *exactly* like the one we use, with the same numbering scheme. Refer to the **Structural Desktop** manual section **Change Line to Member or Change Face to Element** for more information on how to build a Model from AutoCAD lines and faces.



**Figure T1.4**

You are now ready to start the process of turning the analysis graphic drawings into Contract Documents.

In this tutorial, we are taking you through a step-by-step process to learn the commands and their functions to complete the structural drawings. Your Analytical Model is accurate for the purposes of performing the analysis, but it models the members as simple lines. It does not contain the position information that you need to have the members drawn accurately.

There are seven steps you must complete to prepare the model for creation of Contract Documents. They are:

- Loading the analysis file  
( or drawing lines representing a model in *Structural Desktop LT*)
- Adjusting the Display
- Fusing Members
- Setting up Layers
- Offsetting the Members and Elements
- Add additional members, elements, and information required to complete the *Structural Desktop* Model.
- Extracting standard AutoCAD drawings for your Contract Documents

## **Summary of Step I**

In Step I, you learned that all ***Structural Desktop*** and ***Structural Desktop LT*** commands can be accessed from the right- click menu. You learned how to load a Staad file or a Hold file from disk into the program, and that this is Step I of a seven-step sequence that will produce drawings from the file provided.

If you are confident that you can start ***Structural Desktop*** and load an analysis file, you are ready to go on to Step II.

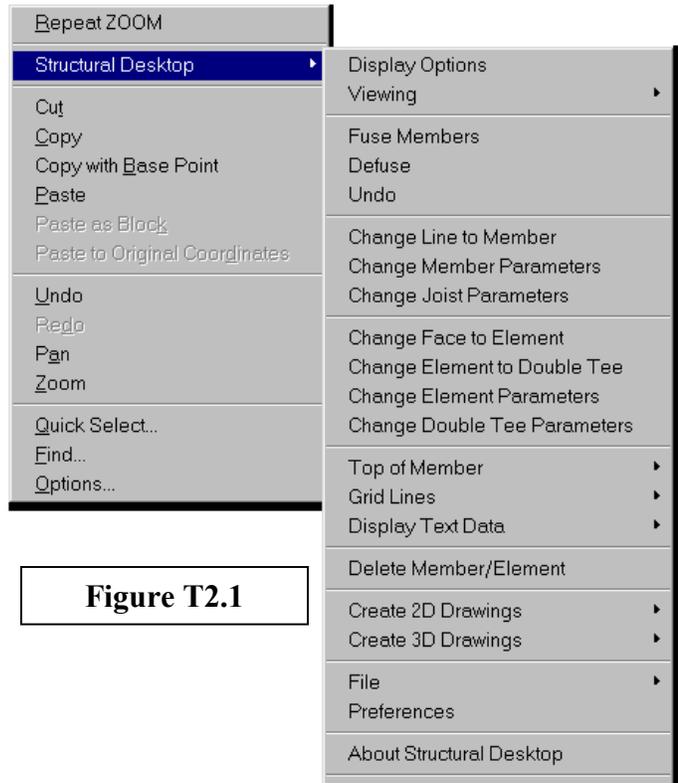
## STEP II – Adjusting the Display

*Structural Desktop* relies heavily on the high quality of graphics available in AutoCAD. Using this graphics engine, *Structural Desktop* displays your model in different ways to permit you to get the maximum performance from your hardware. **Figure T1.1**, above, is the default display. **Centroids** are displayed to represent **Members** in a dashed, yellow color. **Faces** drawn in magenta represent plate **Elements**. **Joints**, at the coordinate locations referenced in your analysis file, are represented by small green circles.

Once you have loaded a file, the commands that were "grayed-out" as unavailable become active. The right click menu now looks as you see it to the right in **Figure T2.1**.

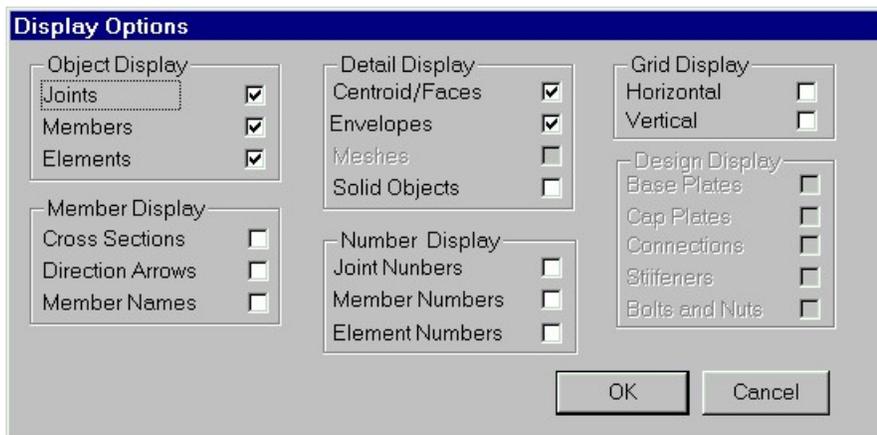
The commands we will be using in this step are in the top section of the *Structural Desktop* command menu. The Display Options command will bring up a Dialog Box that will permit you to adjust what information is presented on the screen at any given time.

Click on this command now and You will see the Dialog Box in **Figure T2.2** below.



**Figure T2.1**

**Structural Desktop** → **Display Options**



**Figure T2.2**

The Display Options dialog box permits you to display different aspects of the model at any given time. If you turn off the Joints in this dialog box, they not only disappear from the screen, they are not in your drawing, anywhere. In fact, none of the "objects" you view in the **Structural Desktop** Model Drawing are anything but temporary until you save your work in a "hold" file (\*.HLD) or export a drawing file and save it. Likewise, Members, Elements, their cross-sections, arrows, names, numbers, envelopes and solids, gridlines...all of these are temporary and can be turned on or off with this control at will. When turned back on, the representation will return to your screen.

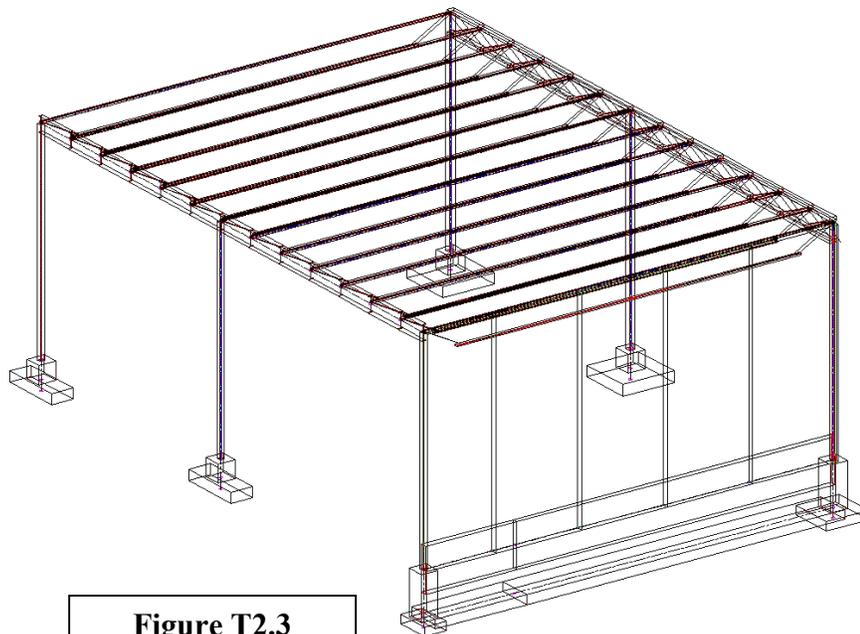
Saving the MODEL as a DRAWING will not permit you to use that drawing with **Structural Desktop** in future sessions. All you will have is the image you see before you at that time.

DRAWING FILES do not hold **Structural Desktop** Database Data. HOLD files store joints, coordinates, offsets, members, elements, properties, etc. HOLD files also store new layers that you create, new Members and Elements that you create, enough to pick up where you left off at any point in the process of creating Contract Documents.

For now, we will concentrate on working with the Envelopes checked on the Display Options dialog box. Other options will be covered as they are useful.

When the **Envelopes** box is checked, an outline of the object is created in three dimensions. This outline requires far less computer power to work with than having the full, solid model function turned on. As much as possible, the **Envelopes** mode provides a real-time method of locating the ends and edges of any of your structural sections or plates.

Make certain that **Envelopes** is checked, and click OK. Your display should be as shown in **Figure T2.3** below.



Further viewing changes will refer frequently to the Display Options box. We will be identifying Members by their Member Numbers and by the Member Names that you can display with this dialog box.

Members in the *Structural Desktop* model have a direction, like a vector, just as they do in analysis programs. The direction follows the sequence of joint numbers in the member definition. Determining the direction of a member is important when you are working with the Member parametrically...i.e. changing values in a dialog box to affect a number of Members. **Direction Arrows** indicate graphically the direction of the Member, permitting you to determine the **start** (for which we use both the nomenclature "**J**" or the term **Start**) and the **end** (for which we use both the nomenclature "**K**" or the term **End**) of the member.

## **Summary of Step II**

In Step II you were shown how to find and use the Display Options dialog box. With this dialog box you may display simple lines and faces only, cross-sections, simple envelopes, and 3d solids. You may also turn off or on grid lines (when you have created them). Individual control is available for viewing Joints, Members, Elements, all of the number labels from the analysis file, and Member Names (e.g. W14x99).

With this information, we are ready to begin fusing members in Step III.

## Step III - Fusing Members

### Reasons for Fusing

An analysis program uses centroid lines, faces and nodes (or joints) to create the design of a structure and structural loading requirements are applied to these entities. The analysis file is read into *Structural Desktop*. *Structural Desktop* reads the data analysis input file, creating centroid lines, 3d faces, and circles which are pictorial representation of structural objects such as beams, columns, plates, and nodes that create the 3D model that appears in **Figure T1-4**.

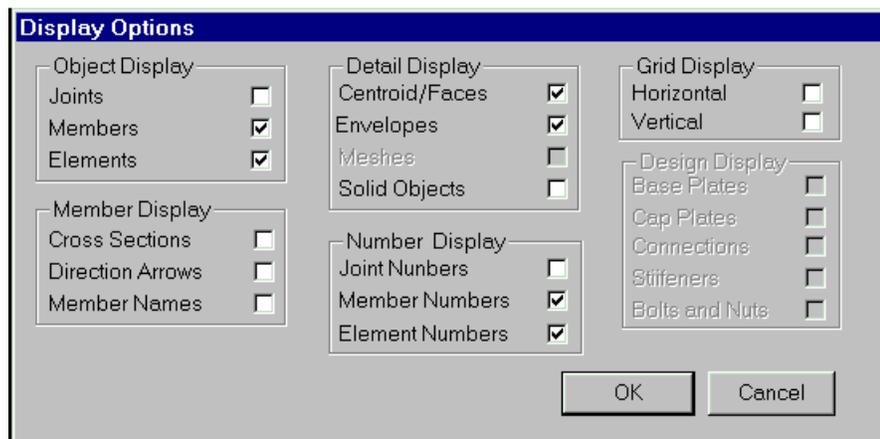
In the analysis program, many members are subdivided into segments by nodes. This "breaking" of a physical member into logical sub-members permits intersections or connections between that member and another for analysis purposes.

The members created in the analysis program found in **Figure T1.4** must be "fused" back together to represent the actual member that must be drawn. Looking at the envelope representation in **Figure T2.3**, you can see a row of small Wide Flange sections along the left edge, and a row of joist girders along the right edge of the Tutorial model. We will now fuse these members, and they will then be drawn correctly in the model and the output drawings.

### Set the Display

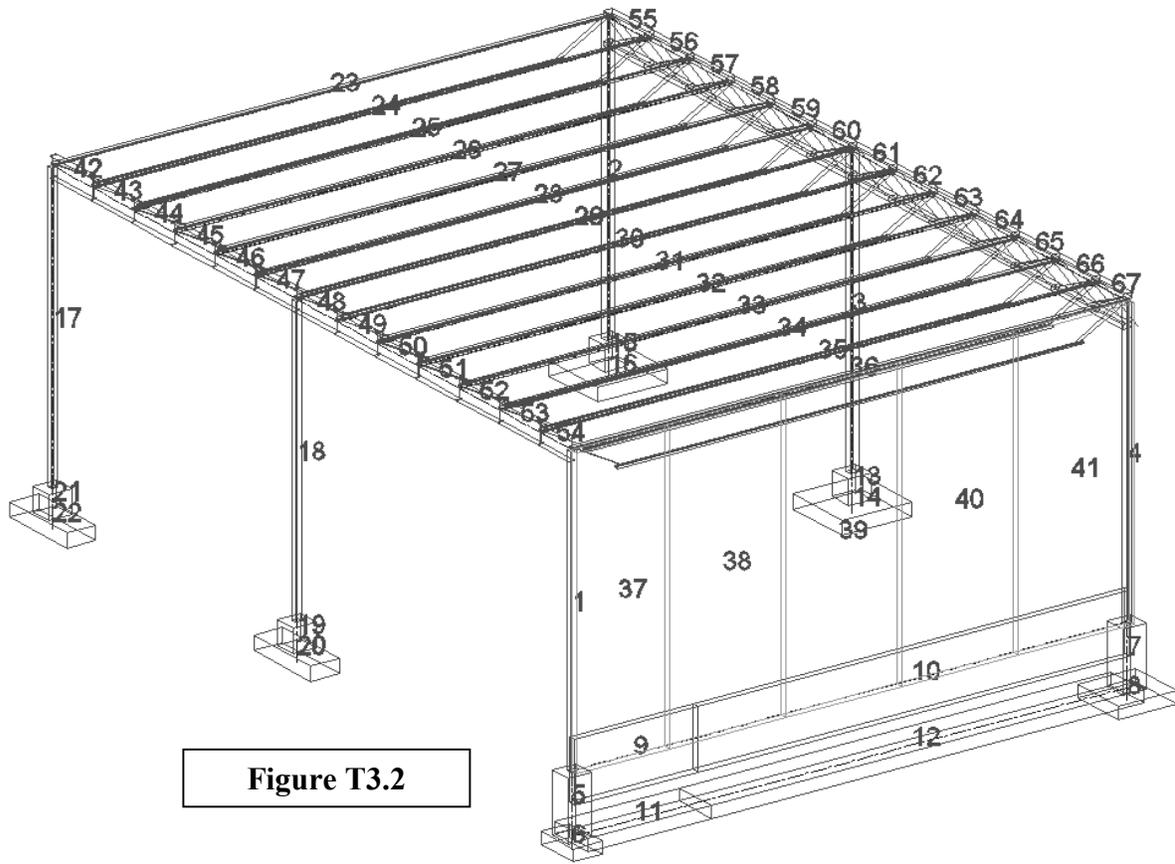
Your first objective is to prepare the graphic drawing for fusing the members. Using the right-click menu, issue the command: **Structural Desktop** → **Display Options**

In the Display Option Dialogue box select the following items as found in **Figure T3.1**:



**Figure T3.1**

In the **Object Display** section, you will turn off **Joints**, leaving on **Members** and **Elements**. In the **Detail Display** section, you want to have clicked on **Centroid/Faces** and **Envelopes**. Finally, in the **Number Display** section, you should be seeing **Member Numbers** and **Element Numbers** checked. Click "OK" and your display should look as shown in **Figure T3.2**.



**Figure T3.2**

This displays all the members and elements with bounding envelopes and identification numbers.

Your display may not exactly match this one; *Structural Desktop* places all text on the screen in the current font that is selected and stored in your template file, or selected by you while in the editing session. This image uses the True-Type Font Arial, found on virtually all Windows-based platforms. You may use whatever shape or font file you wish.

The **SIZE** of the text on the display screen is determined by the dialog box found under **Structural Desktop** —> **Preferences**. The measurement is given in inches; with practice, you can change this as required. Also, the letters and numbers change when you "zoom" into or out of the drawing, and you can move them with the move command to separate clumps and see a particular number more clearly. The **Preferences** dialog box has several options that we will ignore for now, but the two you will need to know are under the **Text Height** section: **Display Text Size** and **Drawing Text Size**. **Display Text Size** controls the height of text on the screen, while **Drawing Text Size** controls the size of text objects created by *Structural Desktop* on your final output drawings. In this model, the height is set at 16 inches for display.

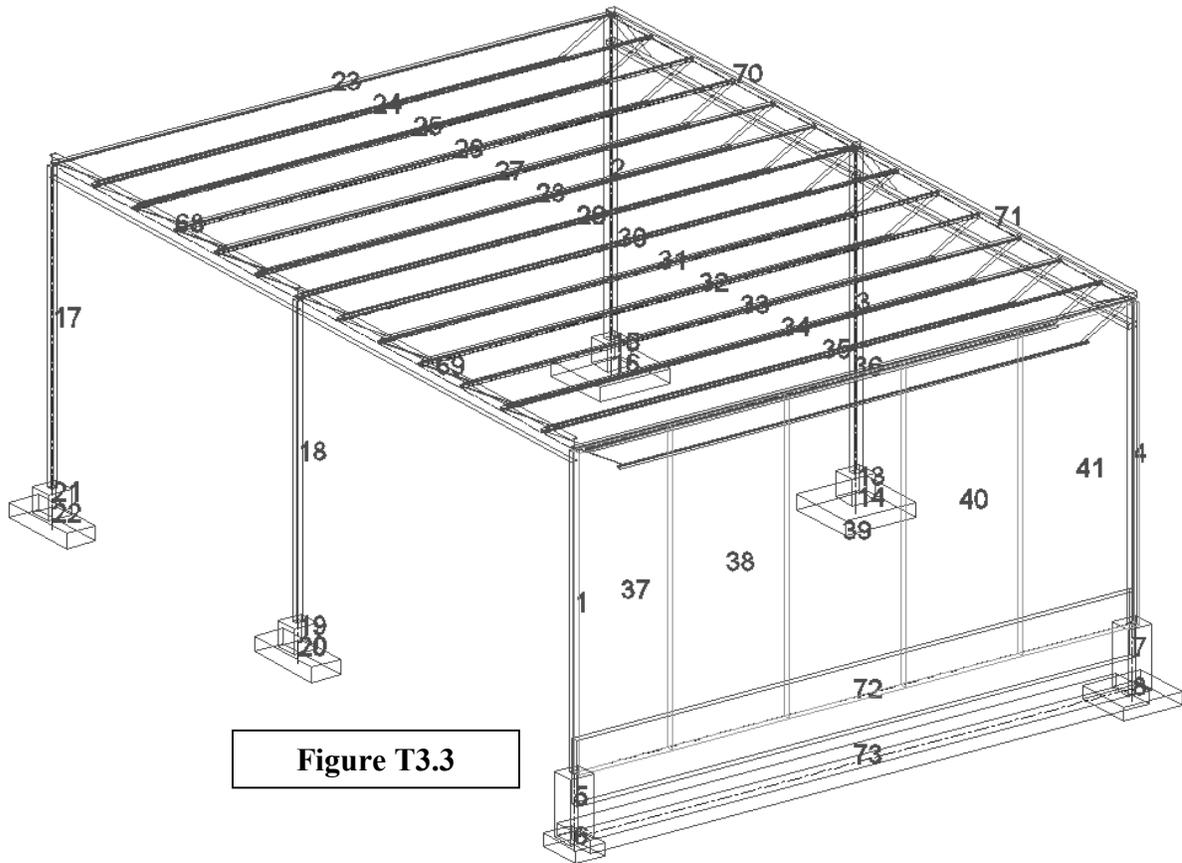
You are now ready to fuse the members together. Right-click the mouse and pick **Structural Desktop**. Select the Fuse command with the left button of your mouse.

**Structural Desktop** —————> **Fuse Members.**

- 1) The program requests that you Select Objects. With the left mouse button, use the pick box on the screen to pick members 42, 43, 44, 45, 46, 47. You can pick any part of a member envelope or pick the centroid of the member. When you are satisfied with your selection, press [Enter] and the six members disappear, into one new member with a new number, 68. The old member numbers will remain until you change the screen display, either by changing your **Display Options** or by bringing up the **Display Options** dialog and clicking "OK" without any changes.
- 2) Repeat the procedure from item one for beams 48, 49, 50, 51, 52, 53, and 54. This fuses these seven members into one member numbered 69.
- 3) Repeat again for beams 55, 56, 57, 58, 59, 60, and hit return. This fuses these members into one member numbered 70. You can see that the joist outline of the envelope for the full size member is much more obvious than for the sub-members that preceded it. In your own models, joists that are not fused will stand out clearly like this, because the program is trying to force a complete joist into a single span-spacing.
- 4) Repeat the fusing sequence for member beams 61, 62, 63, 64, 65, 66, 67, and hit return. This fuses these members into one member numbered 71.
- 5) Fuse members 9 and 10. This fuses the foundation wall into a single wall numbered 72.
- 6) Fuse member beams 11 and 12. This fuses the footings into a single footing numbered 73.

You have now completed the fusing of the members to form the beams, joists, footings, and foundation wall. Force the program to refresh your current view by clicking on **Display Options: Structural Desktop** —————> **Display Options.**

Click "OK" and your screen should appear the same as **Figure T3.3**.



**Figure T3.3**

### Summary of Step III

In Step III we explained that you fuse members in order to begin to make subdivided members into actual members that will plot correctly. You were shown how to use the **Display Options** dialog box to hide **Joints** and display **Member Numbers** and **Element Numbers**. You then stepped through six separate fusing operations to make the Members more closely approximate the actual beams, joists, foundation wall and footing that will be in the actual building. Finally, we forced the *Structural Desktop* database to completely refresh our display using the **Display Options** dialog box, showing us what is currently there as in **Figure T3.3** above.

We are now ready to place different Members and Elements on different AutoCAD layers in Step IV.

## STEP IV – Setting up Layers

### Layer Overview

When the files of an analysis program are first imported into *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. Also, these layers will also 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.

Each company has its own standards and methods for setting up layers, choosing layer names, colors, and lines types. The layers, names, colors, and line types suggested in this section are for the purpose of learning the process of managing layers using *Structural Desktop*.

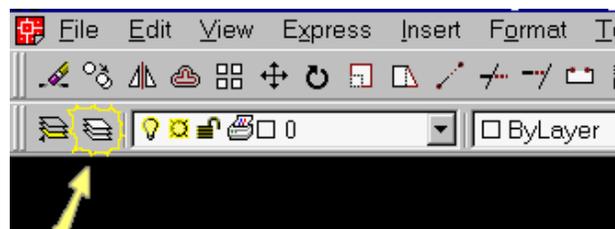
In this section we are going through a step-by-step process to show you how to:

- Use different options for opening AutoCAD's **Layer Properties Manager** dialog box.
- Changing an object to a different layer.
- Choose new colors for each layer.
- Turning layers off and on to manage the object selection process.

**Note:** Please see the AutoCAD 2000 USER'S GUIDE for more concepts and details in working with AutoCAD layer commands and the selection process.

### Creating New Layers

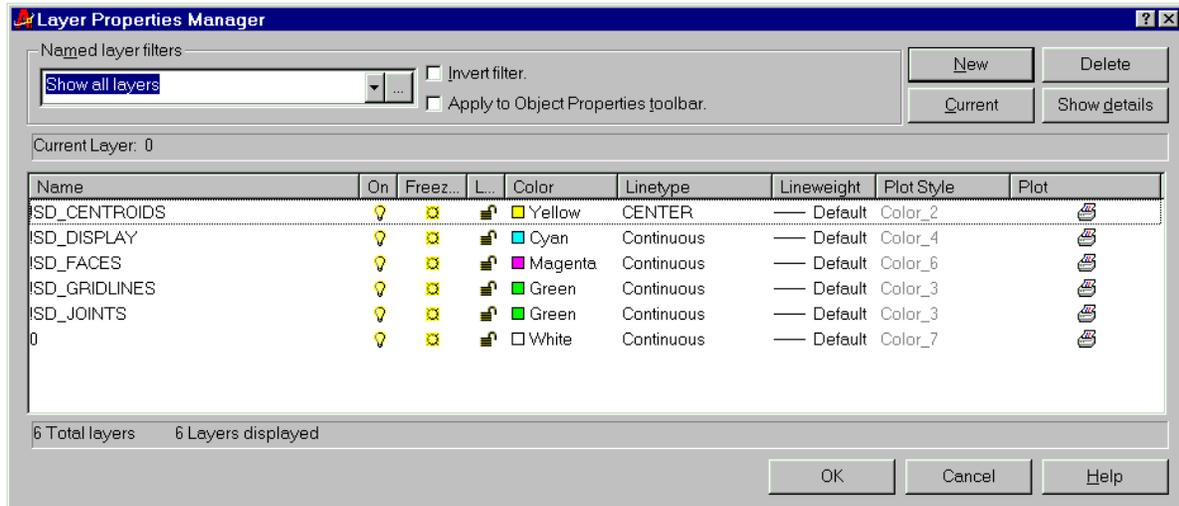
AutoCAD provides a number of toolbars of which *Structural Desktop* takes advantage. **Figure T4.1** below shows the **Object Properties** toolbar, which deals with Layer information. Two controls we will need for Step IV are the **Layer Button** and the **Layer Control Drop-Down List**. If you do not have the **Object Properties** toolbar active in your *Structural Desktop* session, please take a moment to add this toolbar.



Layer Button

Figure T4.1

We are now ready to create new layers; layers on which our objects will be changed to reside. Open the AutoCAD **Layer Property Manager** dialog box. This is done 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**". However you choose to arrive there, the box in **Figure T4.2**, below, should be what you see.



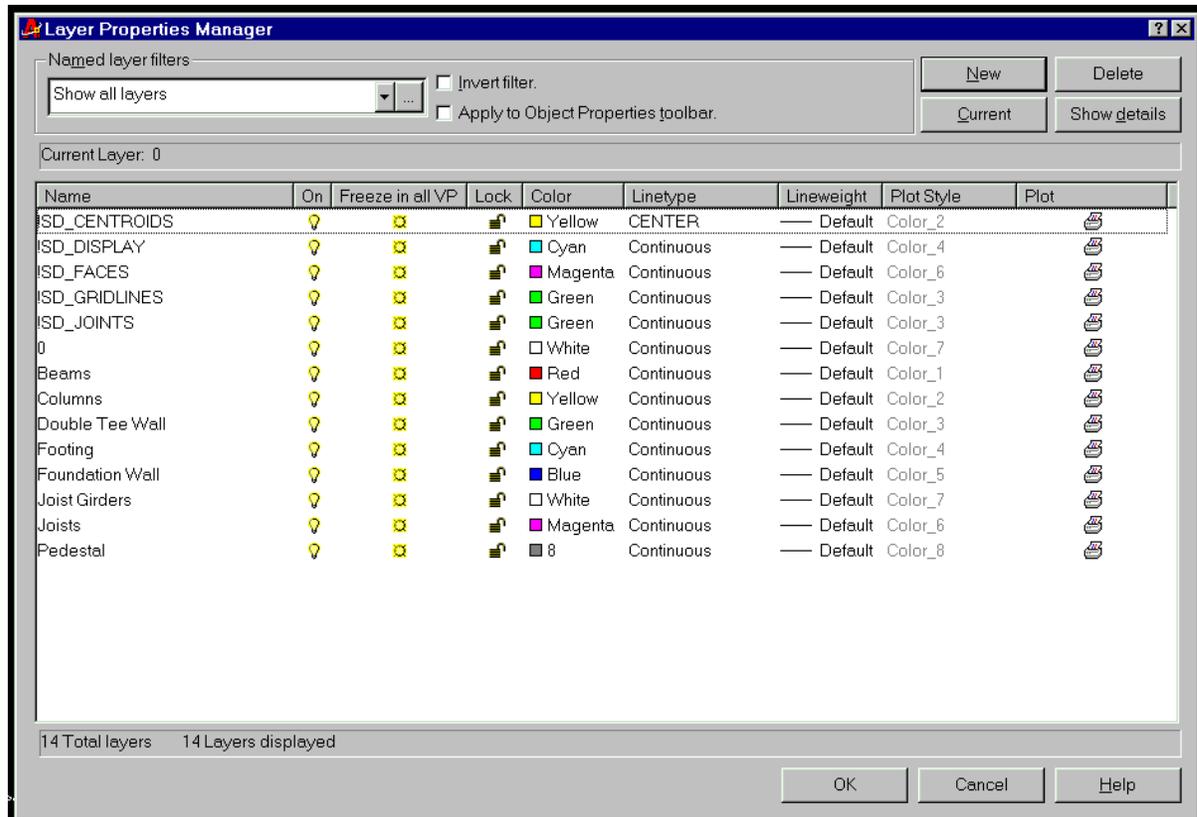
**Figure T4.2**

Select the **New** button in the dialog box. Type the new layer name and create the new layers as listed below. For each new layer created, choose a new color for the layer.

#### New Layers

- Beams
- Columns
- Double Tee Wall
- Footing
- Foundation Wall
- Joists
- Joist Girders
- Pedestal

The dialog box should end up looking a great deal like the one on the top of the next page in **Figure T4.3**.



**Figure T4.3**

Please notice that the first five layers appear above the Layer 0. These layers are required for the operation of *Structural Desktop*, and should never be removed from a Model. They may be purged from an Output Drawing.

When you have completed creating the new layers, select **OK** in the dialog box, which will return you to the *Structural Desktop* Model Drawing.

## Changing the Layer of an Object

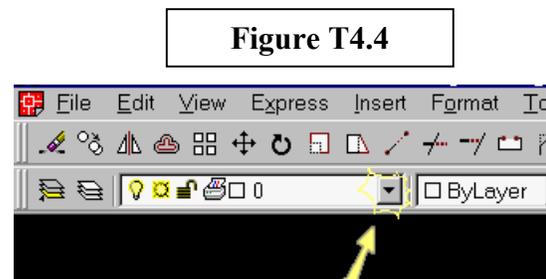
**Note:** We suggest that you create your new object layers from the top of the structure (joists) to the bottom (footings). This arrangement allows you to turn off each layer after it is completed, making it much easier to locate and select the objects in the next layer. We will follow this sequence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

### **Joist Layer**

Objective: Change the bar joists numbered 23 through 36 (as shown in **Figure T3.2**) from the *Structural Desktop* default layers to your "Joists" layer.

1. With the left mouse button select each joist member (object) from number 23 through 36. We are using the "grip" method within AutoCAD 2000. 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.

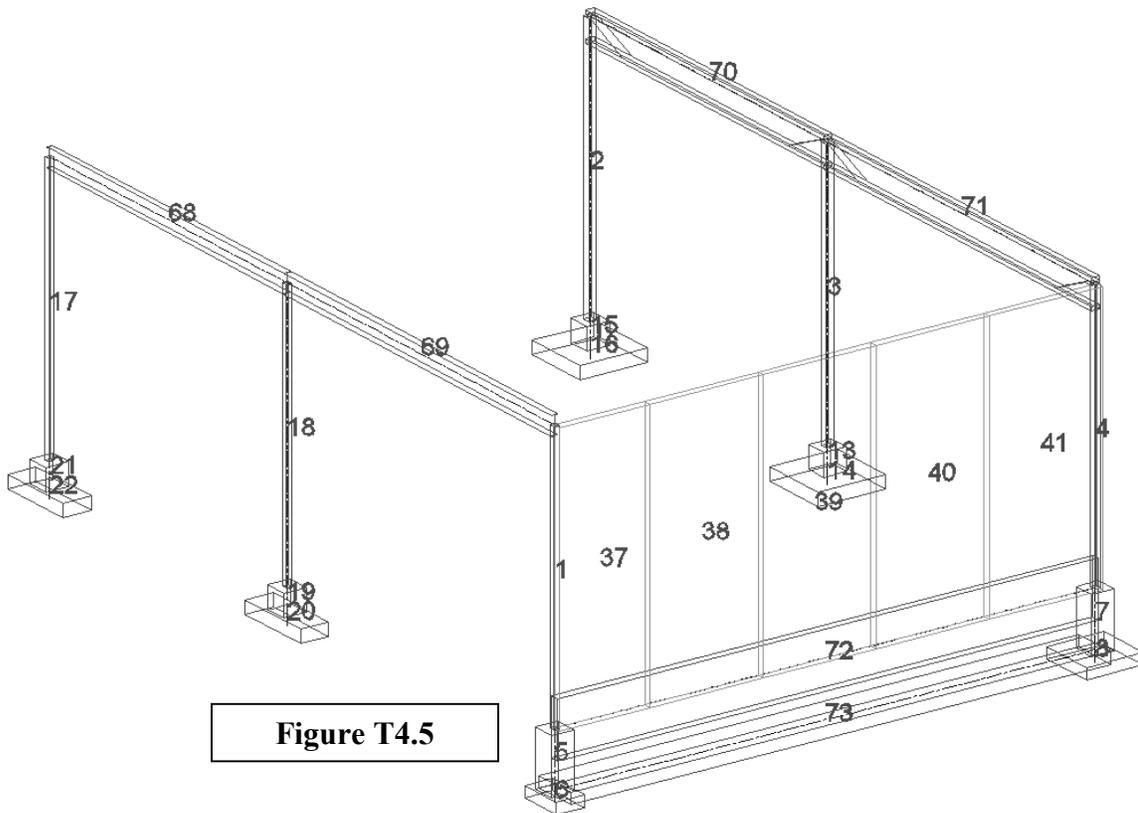
2. In the **Layer Control Drop-Down List** (see **Figure T4.4** at right), select the **Joists** layer by clicking it with the left button on the mouse. The grip-selected Members will transfer to that layer. When they do, they change color to match the color of the layer you have placed them upon. Check the color to make certain you have put them on the correct layer.



**Layer Control Drop-Down List**

3. In the **Layer Control Drop-Down List**, we will freeze the joist layer. On the same line as the joist layer, the second icon from the left 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**.

After turning off the joist layer, force an update of the screen from the database by clicking on **Display Options** and clicking **OK**. The model figure on your screen will appear the same as **Figure T4.5** on the next page.



### Joist Girder Layer

Objective: Change the joist girders numbered 70 and 71 as shown in **Figure T4.5** from the *Structural Desktop* default layer to the your Joist Girder layer.

1. With the left mouse button select the joist girders numbered 70 and 71. (This is the grip command within AutoCAD 2000.)
2. In the **Layer Control Drop-Down List**, select the **Joist Girder** layer by selecting it with the left button on the mouse. The selected joist girders are now on the new joist girder layer.
3. In the **Layer Control Drop-Down List** freeze the joist girder layer.

## Beam Layer

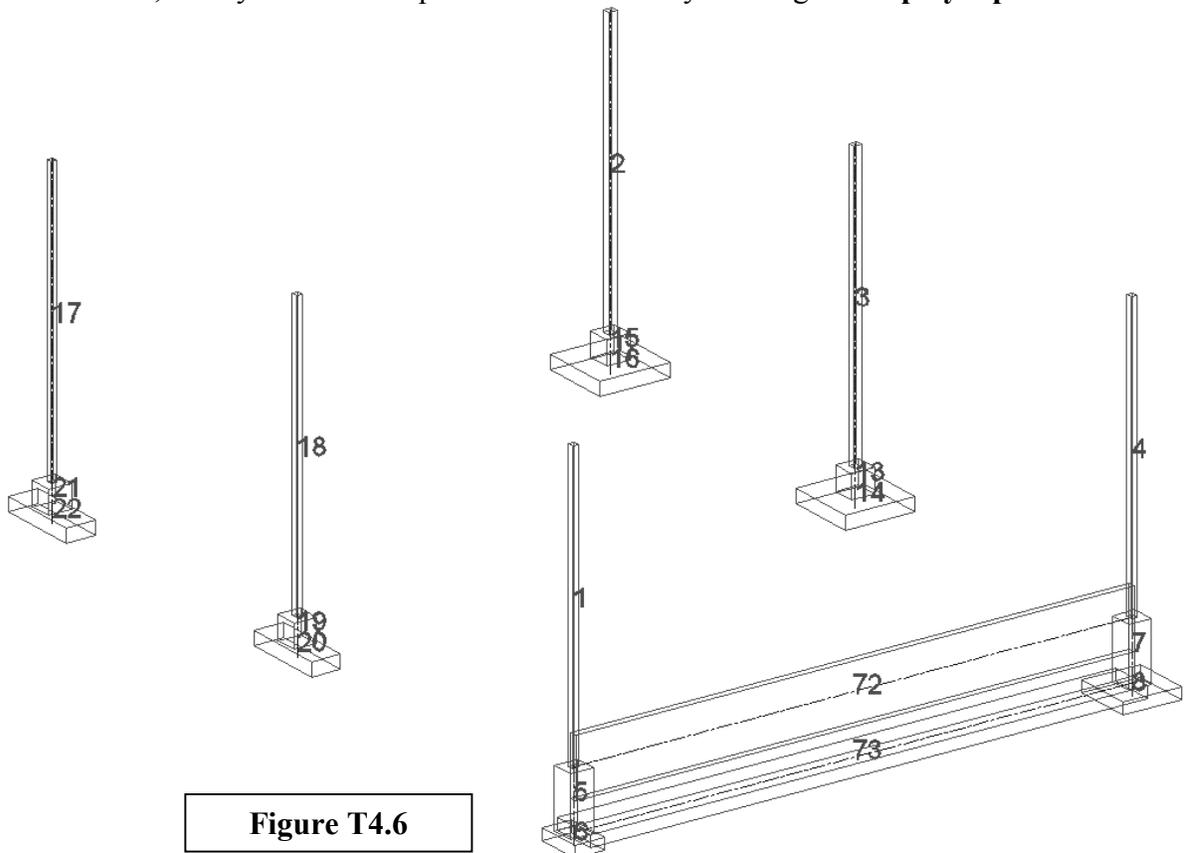
Objective: Change the wide flange beams numbered 68 and 69 as shown in **Figure T4.5** from the *Structural Desktop* default layer to the new Beam Layer.

1. With the left mouse button select the wide flanges numbered 68 and 69. (This is the grip command within AutoCAD 2000.)
2. In the **Layer Control Drop-Down List**, select the **Beam** layer by selecting it with the left button on the mouse. The selected beams are now on the new beam layer.
3. In the **Layer Control Drop-Down List**, freeze the **Beam** layer by clicking on the small "sun" icon on the line of icons for that layer.

## Double Tee Wall Layer

Using the same procedure as in the three detailed examples above, select the wall elements across the front of the structure, numbered 37 through 41 and place them on the **Double Tee Wall** layer. Then freeze the **Double Tee Wall** layer.

When you have completed these steps, your model should look as the one in **Figure T4.6** does below. Remember that the extra member numbers are left, for your convenience and information, until you force an update of the screen by clicking on **Display Options** and **OK**.



## Column Layer

Objective: Change the columns numbered 1, 2, 3, 4, 17 and 18, as shown in **Figure T4.5** from the *Structural Desktop* default layer to the new Column Layer.

1. With the left mouse button select all columns. (This is the grip command within AutoCAD 2000.)
2. In the **Layer Control Drop-Down List**, select the **Column** layer by selecting it with the left button on the mouse.
3. In the **Layer Control Drop-Down List**, freeze the **Column** layer by clicking on the small "sun" icon for that layer.

## Pedestal Layer

Repeat the procedure for the Pedestals numbered 5, 19, 21, 15, 13, and 7 as shown in **Figure T4.6** from the *Structural Desktop* default layer to your **Pedestal** layer, and freeze that layer.

## Foundation Wall Layer

Change the Foundation Wall numbered 72 as shown in **Figure T4.6** from the *Structural Desktop* default layer to the **Foundation Wall** layer and freeze that layer,

## Footing Layer

Change the Footings numbered 6, 20, 22, 16, 14, 8, and 73 as shown in **Figure T4.6** from the *Structural Desktop* default layer to the new **Footing** layer.

## Layering Complete

All of the objects in the Model are now on your layers.

You have now completed setting up the layers. Open the Layer Dialog box and unfreeze all the layers by clicking on the snowflake and turning it into a sun. Click **Okay** in the layer dialog box. All the layers in the 3D model will be visible on the display.

## **Summary of Step IV**

In Step IV you were shown that objects in *Structural Desktop* could be placed on layers that you create. You were taken through a process to create a series of layers, place objects on those layers, and use layering to make objects visible and invisible.

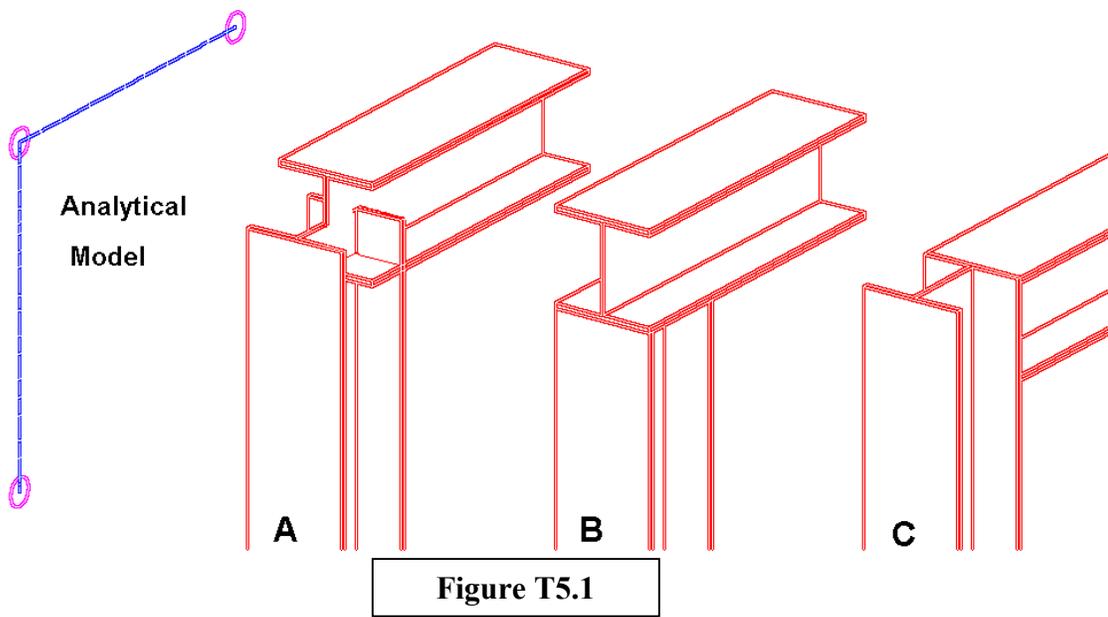
In Step V you will be directed to save your work, start a new session of *Structural Desktop*, retrieve the file you saved. You will then be shown methods and reasons to apply offsets to the Members.

## STEP V – Offsetting Members

### Offsets Overview

Centroid lines, faces, and nodes represent the analytical model. The arrangements of these components follow the rules of analysis programs. Within *Structural Desktop* the centroid lines are converted into objects having structural definition and solid outline such as a beam or joist. Faces are converted to elements, such as a wall panel or floor slab.

The mathematical abstraction that is an analysis model functions by simplifying reality. Centroid lines and faces meet at mathematical points, rather than through complex connections. Consider **Figure T5.1**, below.



From the pure model, an indication cannot be derived as to which of the three outcomes at the right the engineer desired. The first, "A", is what would be displayed if a solid image were simply created around the centroids as they are in the analysis file. The other physical configurations could be modeled in this file. It is likely that the engineer desires "B" or "C". Connection requirements (cap plates, clip angles, etc) would require more adjustment, depending on the level of detail desired.

The result of *Structural Desktop* converting centroid lines and faces to structural objects creates a physical definition of size, shape, and density. This creates the need for you to adjust the physical relationship of the members or elements. *Structural Desktop* accomplishes this by using the concept of "offsets", already present in one form or another in popular analysis packages. In "B" above, the beam was raised and the end extended to rest on the column. In "C", the beam was shortened and the column increased in length.

## Centroid Lines and Structural Members

In the analytical files you created the members and elements using centroid lines, faces connected at nodes (joints). You also applied the designation, material and properties of members and thickness, material and properties of elements. *Structural Desktop* took this information and turned the members and elements into structural objects used to construct the structure such as steel beams (member) or concrete panels (element).

As noted in the examples in the previous section of this chapter, *Structural Desktop* created a relationship between the location of the centroid line and the structural object. Before you can “offset” the member from its analytical model location to the building model location, it is necessary to know the exact placement of the centroid line relative to the position of the member.

Figure T5.2 below shows examples of the centroid line location for each member found in this model. The centroid runs from the point indicated at the start end of the member to the corresponding point at the opposite end.

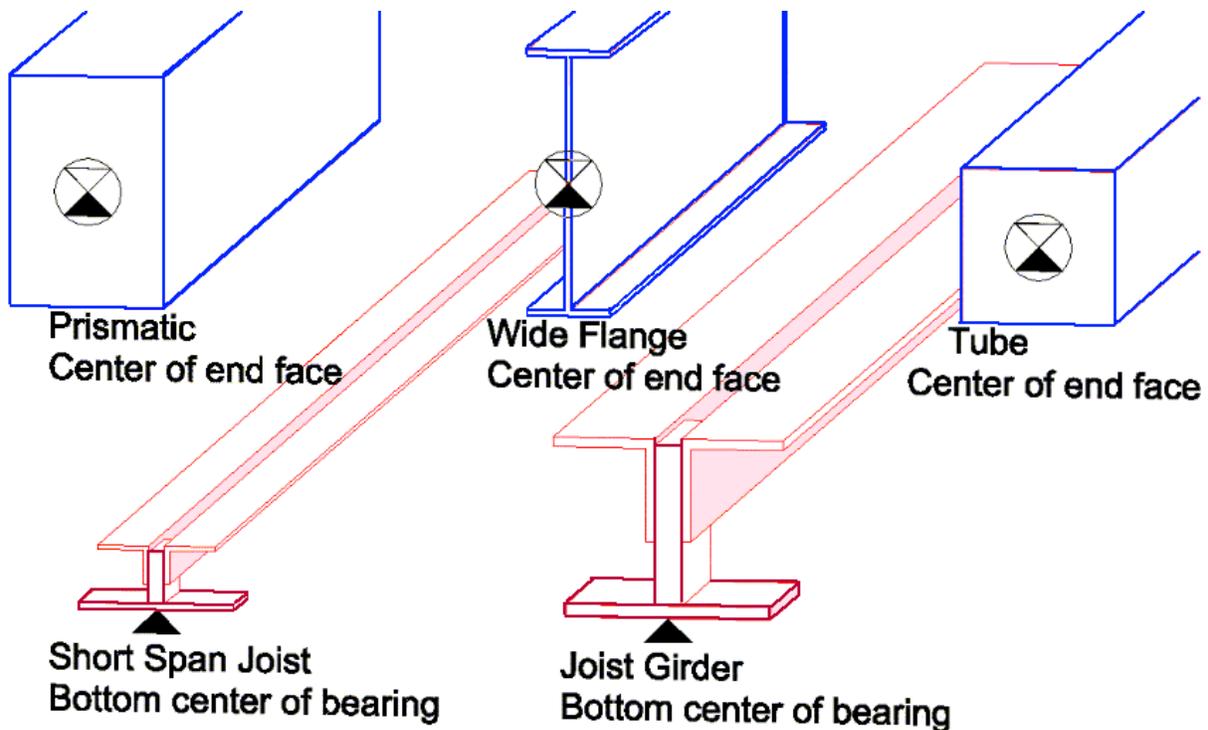


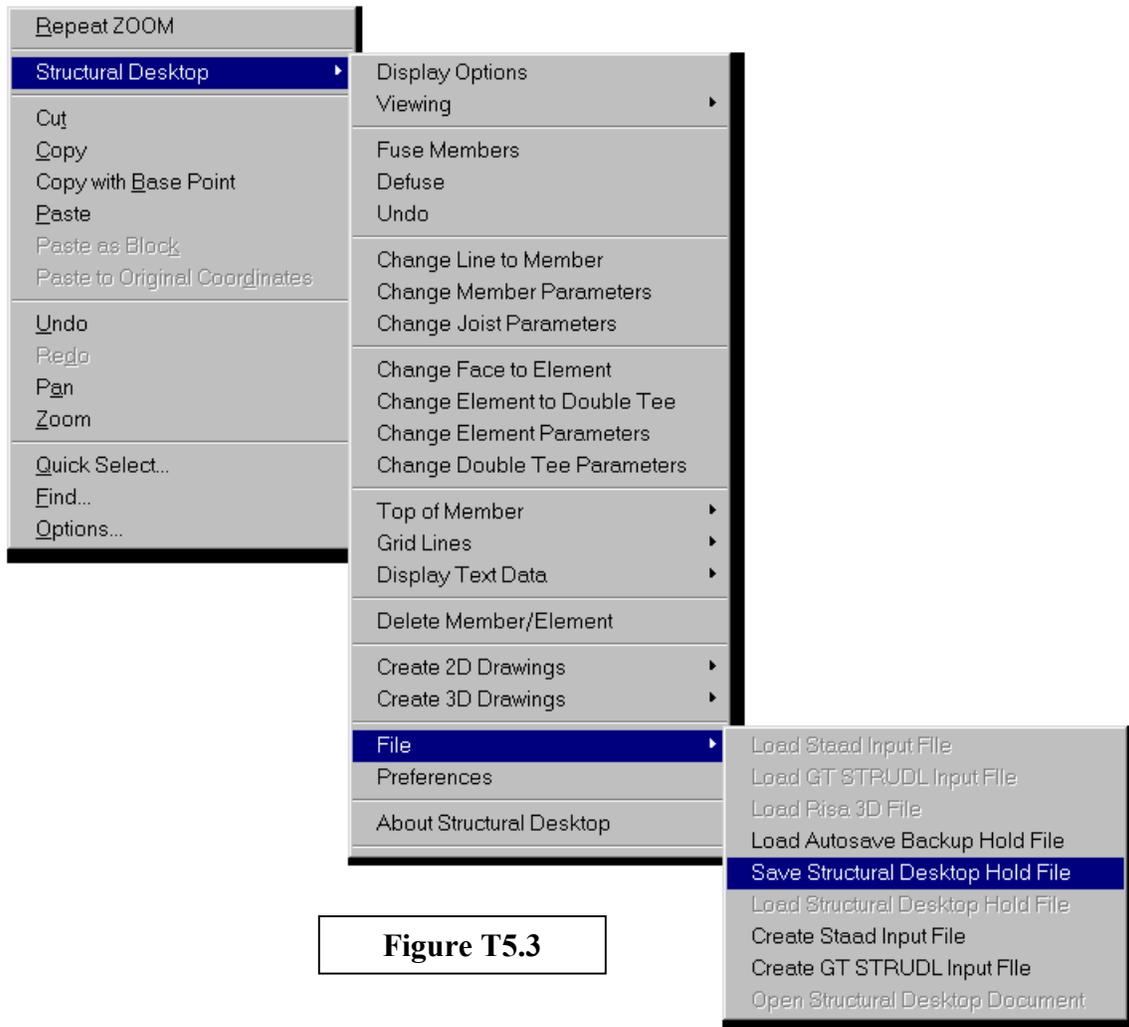
Figure T5.2

## A Pause for Safety

If there are two phrases that are never said enough in computer use, they are "**Backup often**" and "**Save Your Work**". **Structural Desktop** maintains a database outside of the AutoCAD objects on the screen, but linked to them. In order to save your work so that you may come back to it at a later time, you will want to use the option under:

**Structural Desktop** → **File** → **Save Structural Desktop Hold File**

This is important enough to show below in **Figure T5.3**.



**Figure T5.3**

This is a very important concept to learn and remember. Once you have it down, it will give you no trouble. Simply stated, the only time you SAVE A DRAWING (using the AutoCAD menu command "Save") is when you are FINISHED creating that drawing in **Structural Desktop**. To save the MODEL you are working with, save a Hold file. We will touch on this point one more time when we actually extract drawings from this model.

For now, let us save the work done so far. Right-Click to the "Save Structural Desktop Hold File" menu-item. You will be presented with a standard "save a file" dialog box, with a default ending on the file of **"\*.HLD"**. Make certain that you enter a name that has meaning to you and that follows the standard Windows file-naming requirements...i.e. no question marks or asterisks in the name, etc. If you leave off the ending, the program will supply it for you (**HLD**) appended to the file name written to disk.

You can save Hold files as often as you wish during an AutoCAD session. Most certainly you should save often. The file will "hold" your work for you, preserving fusing, layers, layer colors, and other **Structural Desktop** information for an hour or a year.

Before you **LOAD** a hold file, exit out of AutoCAD completely. Future versions of **Structural Desktop** will offer multiple models being worked on simultaneously, and other benefits that were unrealizable in AutoCAD until AutoCAD 2000 was available. Currently, however, when you are finished with **Structural Desktop** exit AutoCAD completely, and you should begin **Structural Desktop** from a "clean slate" before loading a new analysis file or a hold file you have already worked on.

So, save your work. Exit AutoCAD. When you wish to begin on this Tutorial again, load **Structural Desktop**, then invoke:

**Structural Desktop** → **File** → **Load Structural Desktop Hold File**

Select the file that you have saved, and you are ready to continue on to work with offsets.

### **The Offset Process**

To begin the process of offsetting the members and elements load the hold file you saved in the last section of this step. We are going to start by moving the foundation wall to the building model position.

First, select the **Display Options** dialog box. Turn **OFF Joints** and **Elements** in the Object Display Frame, leaving **Members** visible. Add **Envelopes** in the Detail Display Frame and **Member Numbers** in the Number Display Frame.

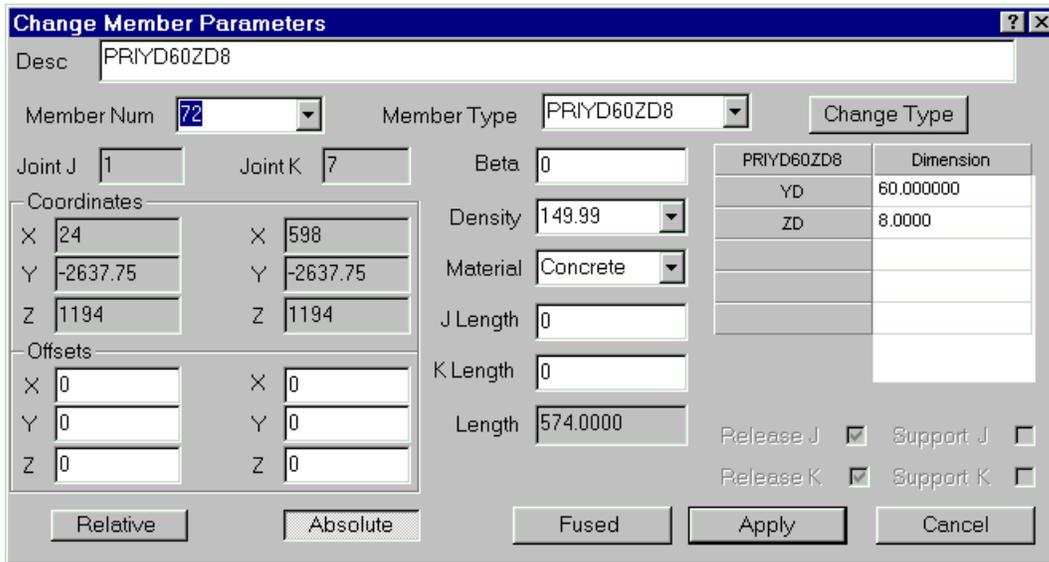
Click **OK** and the display will be set to show members you need while not displaying the element wall that could obscure your view.

We will use one other tool now. Select:

**Structural Desktop** → **Viewing** → **View by Selection**

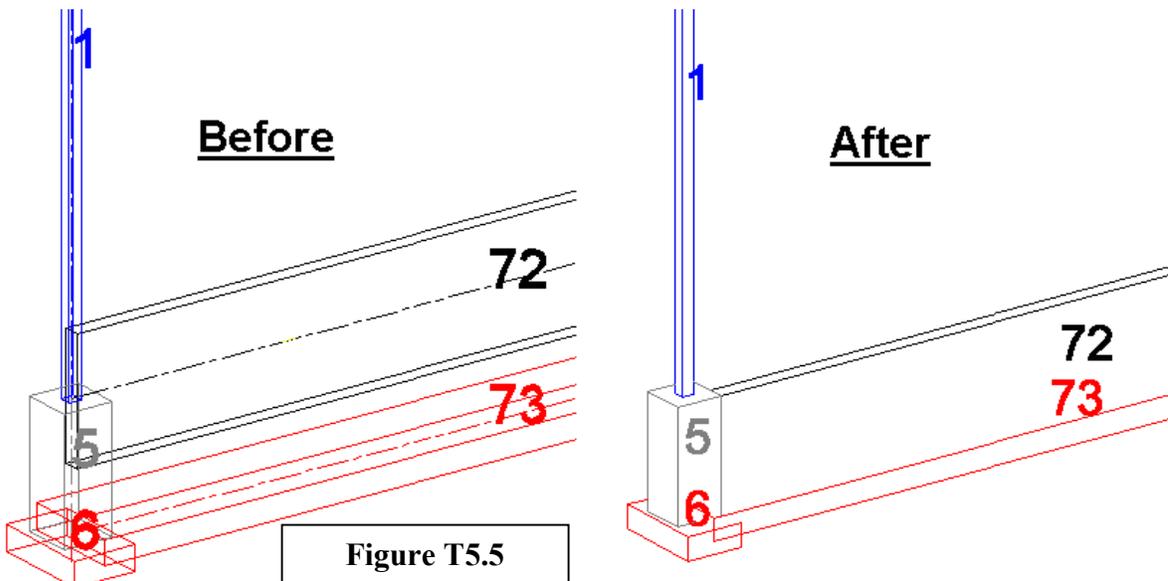
When prompted, select those members across the front of the structure; columns 1 and 4, pedestals 5 and 7, footings 6, 8 and 73, and the foundation wall, member 72. The screen will refresh with only those members visible. The left corner will appear as in the "Before" image in **Figure T5.5** below.

To accomplish our first objective, we will use the most powerful command in *Structural Desktop*, the **Change Member Parameters** dialog box, **Figure T5.4**.



**Figure T5.4**

The first objective is to move the foundation wall to align with the pedestal and foundation footing as shown in **Figure T5.5**.



**Figure T5.5**

Note that the foundation wall was design-modeled as being connected at the top of the pedestal. By moving it forward, down half its length, and shortening each end, it becomes flush with the front of the pedestals and at the proper elevation and length to be the prototype for a correct drawing.

Select:

**Structural Desktop**  **Change Member Parameters**

Select the foundation wall (Member 72).

The **Change Member Parameters** dialog box, as shown in **Figure T5.4**, is used for most operations you will want to do to members. This dialog box gives all current information regarding the members and permits you to change the rotation (Beta), density, type, length, and location of either or both ends. It allows you to offset the members in all three axes. The current data on the selected member should match the dialog box above. This includes a description of the member (PRIYD60ZD8, meaning prismatic, Y depth 60", Z width 8") and the member number (72). We use the convention in **Structural Desktop** of referring to the start of a member as the "J" end, and the end of the member as the "K" end. The numbers of joints J and K from the original input file are displayed. (1, 17)

The X, Y, and Z coordinates of the **Joints** associated with the member will be displayed. Any offsets that have been applied to either end of the member will be displayed in the boxes below the joint coordinates as **Offsets**.

You are now ready to offset the foundation wall by providing the new position in the Offset section of the **Change Member Parameters** dialog box. The steps are as follows:

1. Click the **Absolute** button with the left mouse key.
2. In the "Offset" sections under the Joint J and K labels, enter the following values:

<u>Joint J</u>	<u>Joint K</u>
X: 12	X: -12
Y: -8	Y: -8
Z: -30	Z: -30

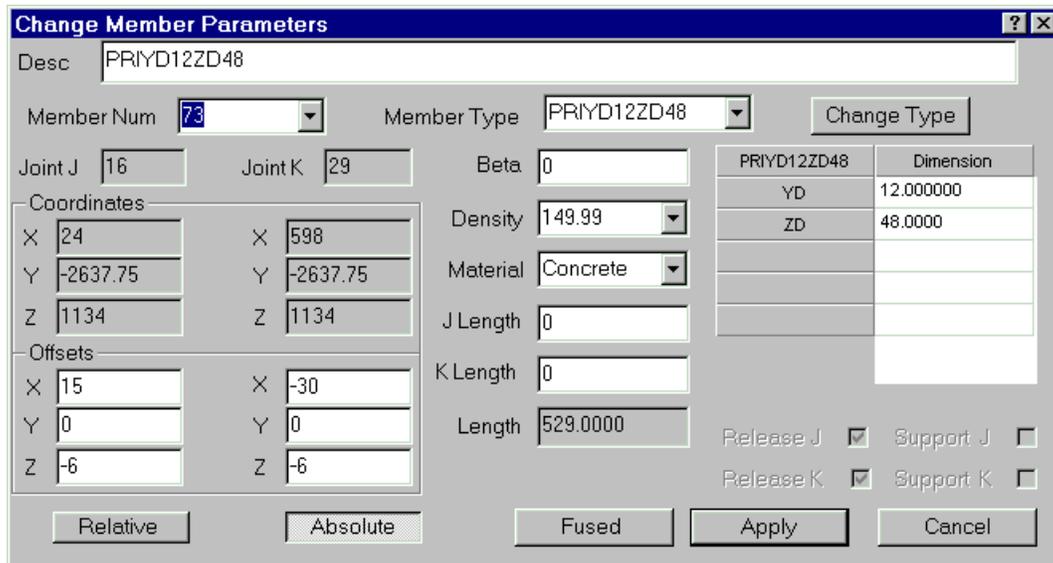
3. Click the **Apply** button.

The foundation wall will move to new coordinate position. The foundation wall appears in the drawing in the same position as found in the "After" side of **Figure T5.5**. What you have done is offset the J end 12 inches to the right, 8 inches toward you, and half of its depth downward. You have done the same to the K end, except you have moved it 12 inches to the left.

Following the same procedure, we will offset the foundation footing (Member 73) as we did the foundation wall. Select "**Change Member Parameters**" from the *Structural Desktop* menu and select the footing.

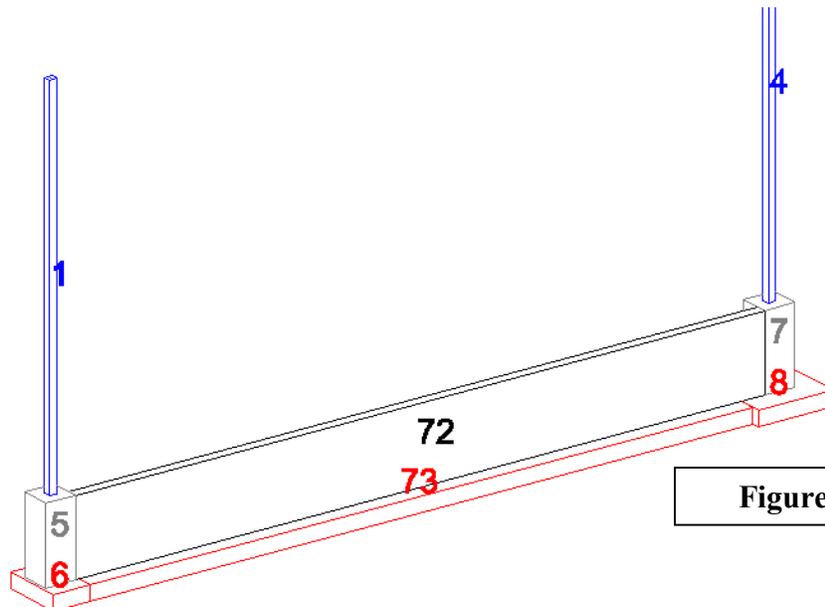
Footing, Member 73, needs to drop 6 inches to be flush with the (current) bottom of the foundation wall and the tops of adjacent footings. The footing to the left is 30 inches wide and the footing on the right is sixty inches wide, so the offsets need to be 15 inches and 30 inches respectively.

The **Change Member Parameters** should appear as in **Figure T5.6** below:



**Figure T5.6**

This offset, when applied, will make the front of the model look like the final assembly you want, as in Figure T5.7 below. (For clarity, we show this as a solid model with **Hide** enabled).



**Figure T5.7**

Our next objective is to place the columns. The column locations are correct with respect to plan, but the bottom of each column needs to be adjusted upwards by 1 ½” for grout and 1” for the base plate, reducing the total length by 2 ½”. These columns were modeled with the starting point at the bottom. Grout and the baseplate will have to be added to the final drawing after **Structural Desktop** produces it, but this will be easier with the column already placed with the bottom at the precise location desired.

We are able to adjust all the columns with a single pass, using a similar procedure to that used for the foundation wall and the foundation wall footing.

Select:

**Structural Desktop** → **Viewing** → **View all**

All members on unfrozen layers will appear. Freeze the layers for beams, joists, and joist girders to simplify the display.

The offset is 2 ½”, which is to be removed from the length of the member at the start of the member.

Select:

**Structural Desktop** → **Change Member Parameters**

Select column members 1, 2, 3, 4, 17, and 18.

We will use an alternative method of applying an offset. Near the center of the dialog box are two data-entry boxes labeled "J Length" and "K Length" with the total length, in inches, displayed below. Enter **-2.5** in the “J” Length box and click the apply button.

The columns will update, shortening by 2 1/2 inches. The "Length" modification function will add or subtract a length to any member, regardless of the orientation of that member. This saves you the trouble of calculating the vectors if you want to shorten a member by a set length, even if the member is not orthogonal. A positive value adds length to a member, a negative value, to either end, subtracts length.

If you choose **Change Member Parameters** again and select one of the columns, you will see that the offset reports a 2.5-inch positive offset, on the J end, in the Z-axis. For some offsets, this is easier than zooming and panning and changing viewing direction to try to see a result, especially on a large structure.

Our next objective is to complete the process of offsetting the bar joists, beams and joist girders. This can be achieved with two different commands:

- **Change Member Parameters**
- **Top of Member**

We will move the members using the top of steel as the key reference point. The reference point for the elevation of the building model was set by the engineer during the process of designing the structure in the analysis program at an elevation of 1488 inches. We will move the members to the proper elevations in this order:

- Bar joists (Member 23 through 36)
- Beams (Members 68 & 69)
- Joist girders (Members 70 & 71)
- Columns (Members 1 through 4, 17, & 18)

To offset the bar joists, beams, joist girders, and the “K” joint of the columns you must have the following information:

- The top of steel with a reference elevation of 1488”.
- The size and dimensions of each structural object to be offset. This can be found in the **Change Member Parameters** dialog box.
- The current location of each structural object to be offset.
- The distance and direction the structural objects must be moved to meet the requirements of the building model.

The bottom of a bar joist bearing plate is located on the centroid line of the bar joist. Currently, all of these are at the reference elevation of 1488 inches. The top of the top chord of the bar joist is the designated or designed location of the top of steel. You are going to offset bar joist 23 through 36 in the negative z-axis by 2 ½”. (The depth from bearing plate to the top of the top chord of K joists, as they are represented in *Structural Desktop*.)

1. Turn on the **Joists** layer and select:

**Structural Desktop** → **Change Member Parameters**

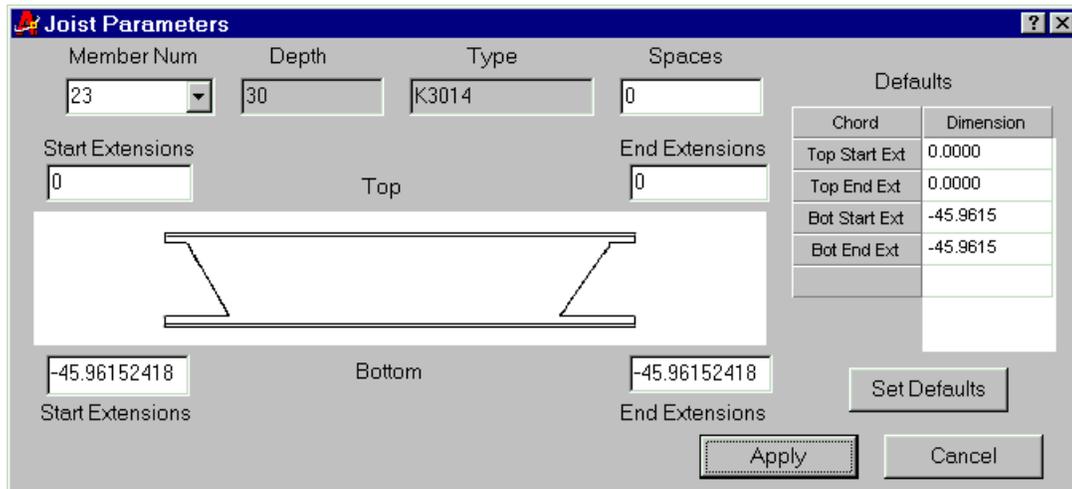
2. Select bar joists 23 through 36 with the left mouse key and hit return.

- This brings up the **Change Member Parameters** dialog box.
- In the Offset section, enter a -2.5 in the Z-axis at **both** ends, and **Apply**.
- The top chords of the bar joists are now positioned to the reference elevation of 1488”. This determines elevation for the tops of beams 68 and 69, and joist girders 70 and 71. The bearing plates of bar joists 23 through 36 have an elevation of 1485 ½”. ( $1488'' - 2\ 1/2'' = 1485\ 1/2''$ )

We can check the elevation of the top of the bar joists by using the AutoCAD "ID" command. First, enter "UNITS" on the command line. The **Unit** dialog box will become visible. If it is not already set for "Decimal", enter "d" and press **Enter**. This will cause AutoCAD to report all number values for coordinates and distances in decimal inches.

Next, enter **ID** on the command line. Select an end point of the top of an envelope of a bar joist, and identify the exact elevation of the top of steel in the z-axis. (1488") The "ID" command displays the coordinate values of a point on the command line in the current units.

We will now perform one more operation on the bar joists. Sometimes, it is necessary to extend the top chord at one, the other, or both ends of a joist or joist girder beyond the end of the bearing plate. *Structural Desktop* permits this operation through a special adjustment dialog box specifically for joists: the **Change Joist Parameters** dialog box, located on the menu directly below the **Change Member Parameters** dialog box and shown in **Figure T5.8**.



**Figure T5.8**

Note that this dialog box displays default extension values at the right, and in the boxes where you may change them. At any time you can return to default values by clicking on "Set Defaults" and then clicking "Apply".

Select:

**Structural Desktop** —————> **Change Joist Parameters**

- Select bar joists 23 through 36 and press enter. This brings up the **Joist Parameters** dialog box, as seen above.
- In the top left-side extension box ("Start"), enter a positive value of 15 and click **Apply**. The top chord of the "J" end of bar joists 23 through 36 will extend past the bearing plate by 15 inches.

- You may check the extensions by zooming in on the bar joists and use the AutoCAD distance command. If you measure the distance from the end of the extension of the top chord to the outside edge of bearing plate on the “J” end, you will find it shows the 15-inch extension in the envelope.

Notes: To add length to the top chord of a bar joist, enter a positive number. You cannot reduce the length of a top chord except by changing the total length of the member, but a bottom chord can be modified to either a positive or a negative value. To summarize, the **length** of a Joist determines the distance between the outside edge of the bearing plates, the **extension** of the top chord extends past that value positively in either direction, and the bottom chord can be extended (or retracted), with a default value (based on depth) pre-determined by the program for display purposes.

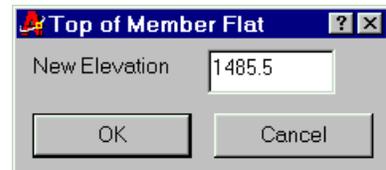
We have completed adjusting the top of the bearing plate of the bar joists to place the top chords at an elevation of 1488 inches. As noted, from the bar joist bearing plates to the top chords is 2 ½ inches, leaving the bottom edge of the bearing plate at an elevation of 1485 ½”. The bar joists are placed on top of beams 68 and 69; therefore, the elevation of the top of beams 68 and 69 must be offset to an elevation of 1485 ½”.

We will use the **Top of Member** command to place the beams at this elevation. First, use the **Layer Control Drop-Down List** to Freeze the **Joists** and Thaw the **Beams**.

Select:

**Structural Desktop** → **Top of Member** → **Top of Member - Flat**

Select beams 68 and 69. The **Top of Member - Flat** dialog box will pop up in the upper left corner of the display. Enter 1485.5, as shown in **Figure T5.9** at right. Click "OK" and the beams will be automatically offset to the amount required to set the top of the beam at an elevation of 1485.5 inches.



**Figure T5.9**

This function is very useful in setting a large number of beams to the same elevation simultaneously, especially for its simplicity. No matter how many you select, each beam is internally evaluated separately by the program, so that any number of primary and secondary beams in your model will all assume exactly the same "top of steel" with one entry on your part. We refer to this as "**Top of Member**" because it will also set precast concrete members at the elevation selected.

You are now ready to change the length of beams 68 and 69 at both ends.

First, we will decrease the length of beam 68 at both ends. Select:

**Structural Desktop** → **Change Member Parameters**

- Select beam 68 and enter - 3.5 inches in the “J” and -.25 at the “K” end of the Length section of the dialog box, and **Apply**.

Next, we will decrease the length of beam 69 at the “J” joint by - .25 and increase the length of the “K” joint by 3.00 inches.

- Follow the same procedure as above, except you are to select beam member 69.
- Within the **Change Member Parameter** dialog box enter in the Length section -0.25 in the “J” joint and a positive 3 inches in the “K” joint and **Apply**.

There is now a ½” gap between beams 68 and 69 at column 18. The "J" end of beam 68 is retracted to the face of column 17, and the “K” end of beam 69 has been extended to the front edge of column 1.

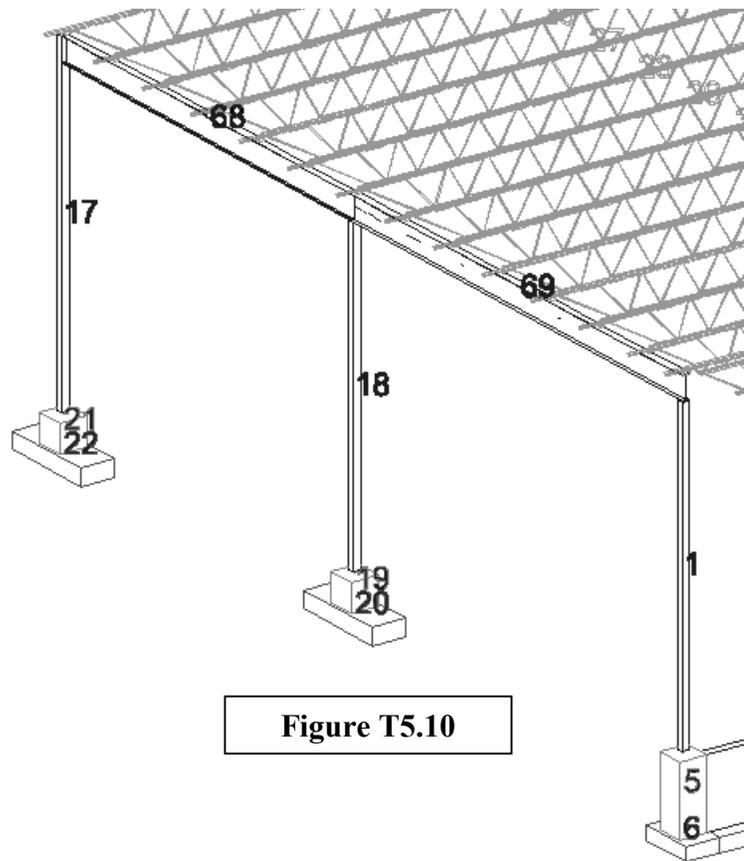
We will now set the columns to be consistent with this development. Use **Change Member Parameters** with column 17 to change its length by subtracting 3 inches from the "K" end, or top. By reducing its height by 2 1/2 inches, we shorten it to the bottom of the bearing plate of the joist that rests upon it; an additional 1/2 inch makes room for a cap plate to be added to final drawings, hence the 3 inches. Go ahead and do this now.

Next, we will use the **Distance** command in AutoCAD to determine how much to shorten column 18. Set your snap functions to snap to endpoints. **Zoom** (window) until you can see the top of the column and the bottom of beams 68 and 69 clearly. You need to be able to see the bottom flange, so that you can be certain that you pick the bottom of the flange and not the top of the flange. Type "**dist**" at the command line and **Enter**. Pick the bottom of the flange (for either 68 or 69) and the top of the column. AutoCAD should report a Z distance of 23.1600 inches. We want another 1/2" cap plate here, so we will shorten column 18 by  $23.16 + 0.5$  or 23.66. (If this was new to you, try measuring some distances in AutoCAD until you can get the proper value.)

Now, use **Change Member Parameters** with column 18 to change its length by subtracting 23.66 inches from the "K" end, or top. Refer back to previous length changes if this is unclear, but by now you should be getting the hang of it. Remember to use a negative value to subtract length, as a positive value will add length to either end. The column should drop down neatly to 1/2 inch below the bottoms of the flanges of 68 and 69.

Finally, apply a reduction in length to the "K" end of column number 1. As column 1 also needs to drop below a beam with the same depth as the beam above column 18, you need to shorten the top (end) of column 1 by 23.66 inches.

When you have completed all of the above adjustments, the model for this side of the structure will appear just as it should be in reality when constructed. From the model so far, a solid model for this end of the building with a hidden line view is shown in **Figure T5.10**. Note the relative positions of the columns and beams, and the extensions on the bar joists.



**Figure T5.10**

Thaw all of the frozen layers in the layer control, and you will see this same image in the envelope form.

Next, we will adjust the joist girders numbered 70 and 71, at the other end of the structure. These members are denoted as JG3210's, which can be determined by looking at one through the **Change Member Parameters** dialog. The centroid line on the joist girder is located at the bottom of the bearing plate. The top chord tops out 7 1/2 inches above the centroid line and the centroid line is currently located at the elevation of 1488 inches.

From the previous calculations for beams 68 and 69, we know that the placement of the top chord of the joist girders would have to be set to an elevation of 1485 1/2 inches, so that the bar joists will rest upon them. The joist girders are seven and one-half inches from the bearing plate to the top of chord, so their centroids need to be set 7 1/2 inches lower, or to an elevation of 1478 inches. This requires an offset in the Z-axis of 10 inches.

Select:

**Structural Desktop** → **Change Member Parameters**

- Select joist girders 70 and 71. Click **Absolute** and enter - 10 inches in the “J” and “K” offset boxes, and **Apply**.

In order to get a better view of the joist girders, we will use the AutoCAD viewpoint command. Currently, the view is set three dimensionally as if you were viewing the structure from an angle of 235 degrees with an elevation of 23 degrees from the XY plane. To change that from a "southwest" to a "southeast" view, consider the following interchange at the command line of AutoCAD:

**Command:** vpoint

**Current view direction: VIEWDIR=-0'-0 1/2",-0'-0 3/4",0'-0 3/8"**

**Specify a view point or [Rotate] <display compass and tripod>: r**

**Enter angle in XY plane from X axis <235>: 315**

**Enter angle from XY plane <23>:**

**Regenerating model.**

Therefore, type:       vpoint[Enter]  
                          r[Enter]  
                          315[Enter]  
                          [Enter]

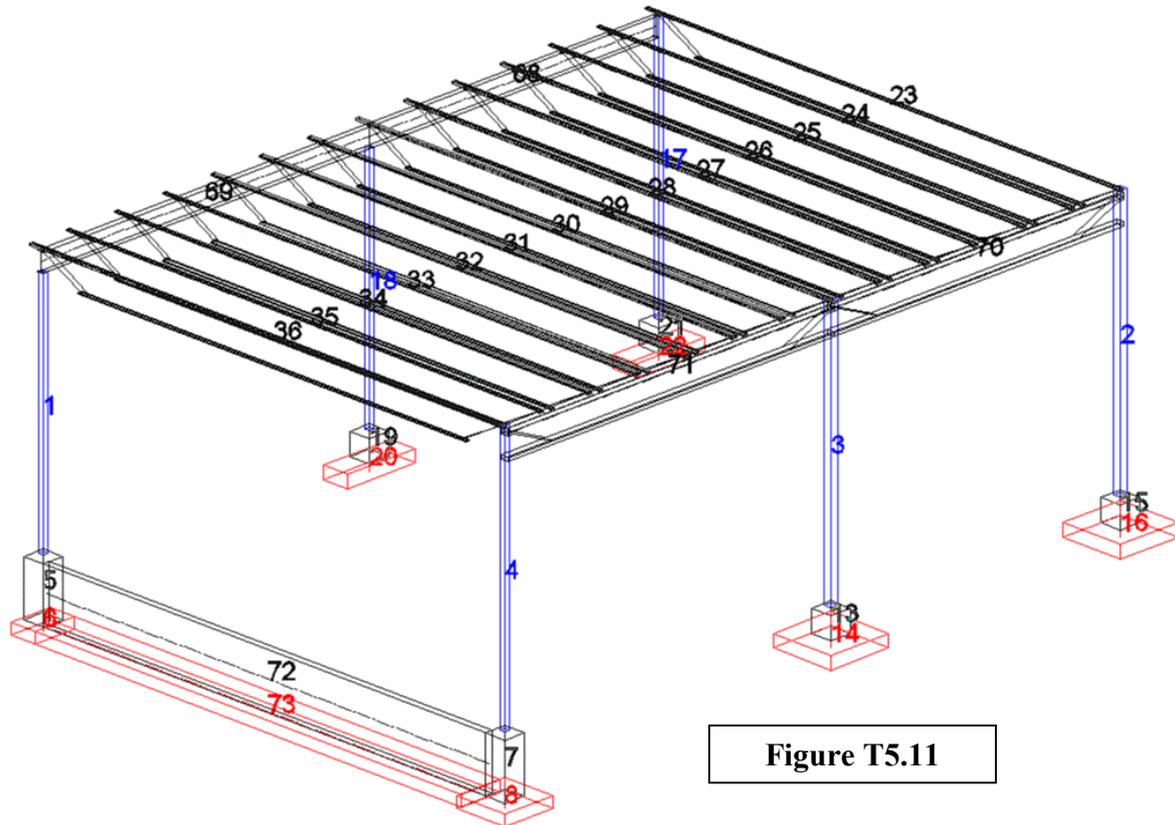
You will be viewing the model from a better vantage-point for the joist girders. Notice that the member numbers are still facing in the direction you "were looking from"...to update them, use:

**Structural Desktop —————> Display Options**

and **click OK**. This, again, is what you should do in the event that a display looks odd or has not refreshed properly. Once you have clicked **OK** on the **Display Options** dialog box, what you see is what is in the database. As a point of information, remember:

- That you may have to use the **Structural Desktop** menu item **View All** if you have used a **Viewing** command to simplify your view. Only this command returns objects to view.
- That items will be invisible if they are on layers that are frozen.
- That Members and Elements can be turned on and off separately on the **Display Options** dialog box.

With the elements still turned off at the **Display Options** dialog box, everything else visible by layer and viewing style, and viewing in the envelope mode, your display should look quite similar to the one pictured in **Figure T5.11** at the top of the next page if you have followed along thus far.



When you examine to **Figure T5.11**, you can readily determine that the bottom chords of the joist girders meet inside column 3. We will use the function:

**Structural Desktop** → **Change Joist Parameters**

We will decrease the length of the lower chords to clear all columns by 1/2 inch. First we must determine the size of the columns, and we must make any desired changes to the joist girder length. The extensions of a joist or joist girder are measured from the end of the member...if you lengthen or shorten the member, the extension will move with it. We will therefore first change the length of the member, and then change the joist parameters to position the bottom chord with respect to the final positions of the ends of the member.

- Use the **Change Member Parameter** function and select columns 2 through 4. When the dialog box comes up, notice the drop-down box that displays the member number. If you click on a particular member, the dialog box will display the information for that member. Doing so, you will find column 4 is a TS6x6x1/4 and columns two and three are TS8x8x1/4. Click **CANCEL** to close the box.
- Use the **Change Member Parameters** box (**Not the Change Joist Parameters!**) to shorten the length of member 70 by 1/4-inch at each end. This will create a 1/2-inch gap when we do the same to member 71.

- Use the **Change Member Parameters** box to shorten member 71 by 1/4-inch at the "J" end, and lengthen it in the "K" direction by three inches. This completes the 1/2-inch gap between 70 and 71, and places the bearing plate of member 71 forward and flush with the front face of the Tube member upon which it rests.
- The bottom chord of joist girder 70 needs to be reduced by 4 1/2 inch to clear the side walls of columns 2 and 3 by 1/2-inch. However, since you have already shortened member 70 by 1/4 inch, the amount of reduction is now only 4 1/4 inches.
- Select **Change Joist Parameters** and choose joist girder 70.
- Enter a negative 4.25 for the bottom extensions as shown in **Figure T5.12** and **Apply**.

Defaults	
Chord	Dimension
Top Start Ext	0.0000
Top End Ext	0.0000
Bot Start Ext	0.0000
Bot End Ext	0.0000

**Figure T5.12**

Joist girder 71 is attached to column number 3 (that is a TS 8x8x1/4) at the "J" joint. This requires a negative 4 1/2" extension of the bottom chord. However, we shortened the length of 71 from the centroid of the column by 1/4 inch, just as in 70 above, so the value at the "Start" of the member must be 4 1/4 inch. The "K" joint of joist girder 71 is attached to a TS6x6x1/4" which would require 3 1/2" to provide 1/2" clearance from the face of the column, except we added 3 inches in length at the "K" end of 71, so we must subtract 6 1/2 inches from the "End Extension".

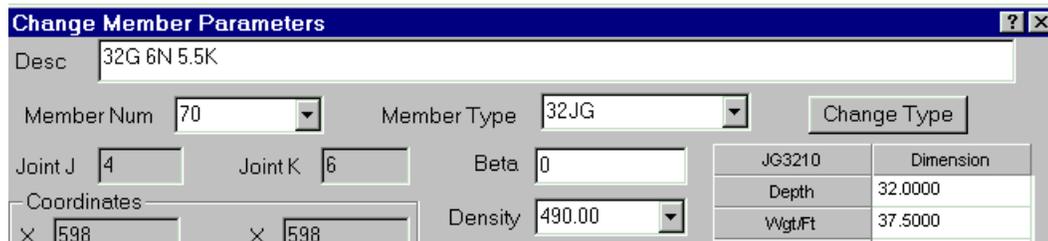
Use the **Change Joist Parameters** dialog box to make the necessary modifications to the bottom chord of member 71, as outlined above and following the example of member 70. This example illustrates how changing one value (length) affects other values (extensions) of the same member.

At this point, we will make one more small change to joist girders 70 and 71, to the nomenclature they carry. First, take another look at the **Change Joist Parameters** dialog box for joist girder number 70 in **Figure T5.12** above.

Notice the text entry box labeled **Spaces**. This refers to the number of spaces or **Panels** that make up the joist girder, and is determined automatically from the number of members you have fused to create the girder...assuming that each load (connecting member) applied to the girder indicates a panel. For joist girder 70, we fused six members to create one joist girder.

You may require a label quite different from the one supplied for this member - 32JG - in your final drawings. We are going to change the given label to one standard designation based on a 5.5 kip load: **32G 6N 5.5K**

To accomplish this, click **Change Member Parameters**. In the **Desc** (description) text box, type **32G 6N 5.5K** (as shown in **Figure T5.13** below) and **Apply**.



**Figure T5.13**

This is the description that will print on the output drawings. You will be able to change the nomenclature for any member in *Structural Desktop*, and it will remain as you have changed it unless you change the member type, at which time the description will revert to the member name. You can also change any number of members through this function, at once, no matter what types are represented, to have a single identical label if that is what you want.

Member 71 would not have an identical label in this example. We fused seven members to create joist girder 71, so assuming the same load, the label should be: **32G 7N 5.5K**

Use the **Change Member Parameters** box to change the description of joist girder 71 to **32G 7N 5.5K**.

Columns number 2, 3, and 4 support joist girders numbered 70 and 71.

- The top elevation for columns 2, 3, and 4 prior to offset is 1488 inches. We need to reduce the lengths of the columns at the top by the combined depths of the bar joist from top to bearing plate (2.5 inches) and the joist girders (7.5 inches) plus the thickness of the top-of-column cap-plate that we need to leave room for, which we will assume requires 1/2 inch.

- The length reduction for the columns can be determined by adding 2.5, 7.5, and 0.5 inches. A 10 1/2-inch reduction will place the top of the column tube at 1477.5 inches.
- Select the **Change Member Parameters** Dialog box, selecting columns 2, 3, and 4. Enter a negative 10.5 in the “K” Length text field and **Apply**.

This completes the manipulation of the members. All of the steel members and concrete footing and foundation members are now exactly where, in relation to the model, they would be in an actual erected structure.

### **Summary of Step V**

In Step V you were shown how to save your work, start a new session of *Structural Desktop*, and retrieve the file you saved. You were introduced to the **Change Member Parameter** dialog box that permits parametric editing of a member. You were given opportunities to change the length and position of different kinds of members, from the foundation to the Top of Steel, and you were introduced to the **Top of Member** command to set members of the same or different depths at a common top elevation. You were also introduced to the **Change Joist Parameter** dialog box. With this tool we extended the top chords and reduced the lengths of bottom chords of different types of joist members.

We moved to another view of the building (for anyone not familiar with changing point of view) and adjusted lengths, extensions, and column heights on that side. We finally changed the description of the joist girders to reflect a more meaningful label that will print on every view of these members drawn for the contract documents.

In Step VI, we will change the Element plates that we banished earlier in the session into Double Tee members. We will interact with these objects and show you how to change their properties.

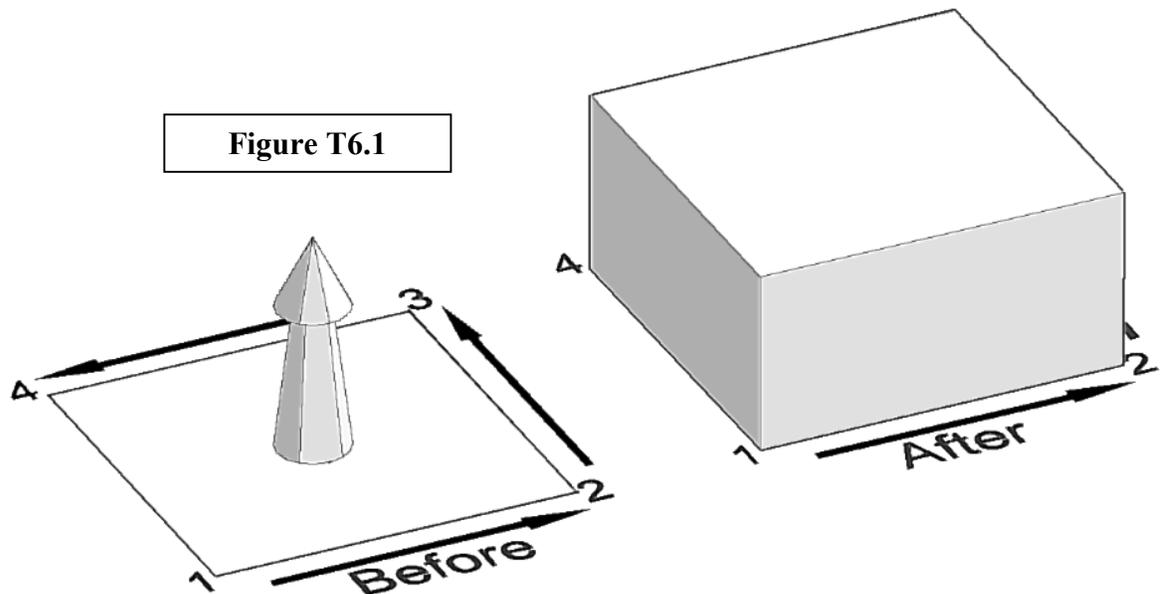
## STEP VI – Offsetting Elements

### Elements Overview

Centroid lines, faces, and nodes represent the analytical model. The arrangement of these components follows the rules of analysis programs. Within *Structural Desktop* we match the properties of the AutoCAD entities called 3d Faces to the finite element mesh or slab entities of analysis programs. We refer to analytical objects of two nodes as Members; those with three or four we refer to as Elements within this text and the program interface.

Faces represent elements, such as a wall panel or floor slab, and can be any three or four points in space. Any three points, no matter how they are arranged, will represent a triangle in a plane; the program will do its best to create a coplanar object from four points, even if they are not coplanar.

The pure model works with the nodes and, for most models, uses a value of thickness as one of the properties tying those nodes together. *Structural Desktop*, attempting to convert this abstraction to a reality, applies the thickness as an extrusion. The envelope or solid representation of an element follows a **Right Hand Rule**: If you face an element so that the fingers of the right hand curl in the direction of the sequence of nodes (counter-clockwise) then the thumb points in the direction of extrusion of the solid thickness...the "unseen face" of the element will be towards you. See **Figure T6.1**.



The result of *Structural Desktop* converting centroid lines and faces to structural objects creates a physical definition of size, shape, and density. This creates the need for you to adjust the physical relationship of the members or elements. *Structural Desktop*

accomplishes this by using the concept of "offsets", already present in one form or another in popular analysis packages. In the case of flat plate elements, the offsets are adjusted in the **Change Element Parameters** dialog box.

This would be a good time to save a hold file.

**Structural Desktop** —————> **File** —————> **Save Structural Desktop Hold File**

Save your work. Exit AutoCAD. When you wish to begin on this Tutorial again, load **Structural Desktop**, and then invoke:

**Structural Desktop** —————> **File** —————> **Load Structural Desktop Hold File**

Select the file that you have saved, and you are ready to continue on to work with offsets on Elements.

### **The Offset Process**

To begin the process of offsetting the elements, load the hold file you saved in the last part of this step. We are going to start by turning on the elements, moving the elements to their proper position, and changing the thickness of the elements.

First, select the **Display Options** dialog box. Turn **ON Elements** in the Object Display Frame, leaving **Members** visible. Add **Element Numbers** in the Number Display Frame.

Click "OK" and the display will be set.

We will use one other tool now. Select:

**Structural Desktop** —————> **Viewing** —————> **View by Selection**

When prompted, select those members across the front of the structure; columns 1 and 4, pedestals 5 and 7, footings 6, 8 and 73, the foundation wall, 72, and the elements across the front. The screen will refresh with only those members and elements visible, and will appear as in **Figure T6.2** below.

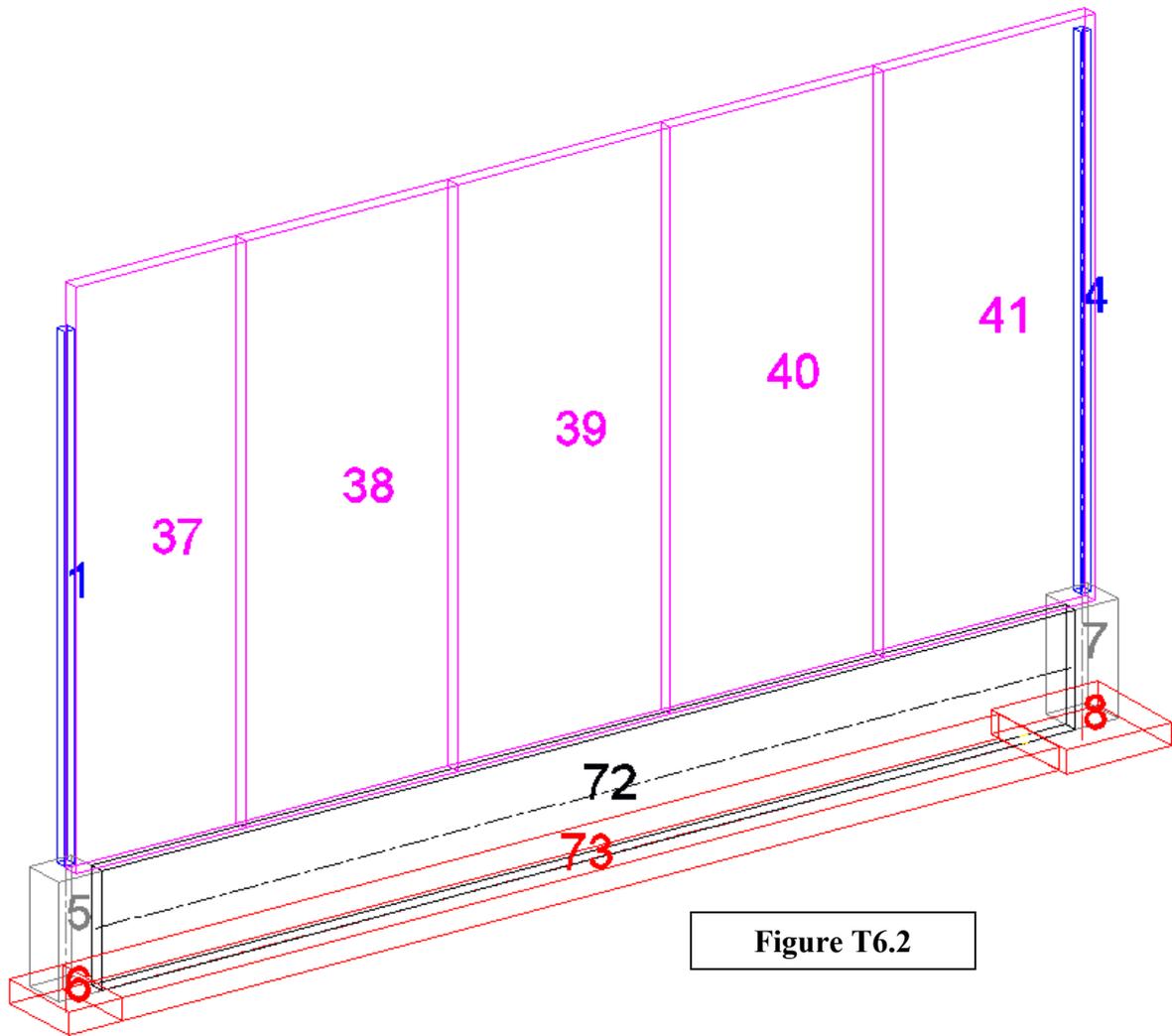


Figure T6.2

Just to restate, the image above is the default viewing angle (after saving a hold file, exiting AutoCAD, and reloading) with the View by Selection function used to eliminate all but the members and elements shown.

Our objective with the elements will be to bring them forward, flush with the front edge of the pedestals 5 and 7 and the foundation wall, 72. We will also move the tops of the elements to form a parapet, by offsetting them two feet upward. Our final goal is to change these elements into precast double-tees, so we shall also modify the element thickness.

In AutoCAD, we can determine that the elements are 8 inches thick and, as shown in **Figure T6.3** at the right, the front face of the element is 4 inches from the foundation wall face.

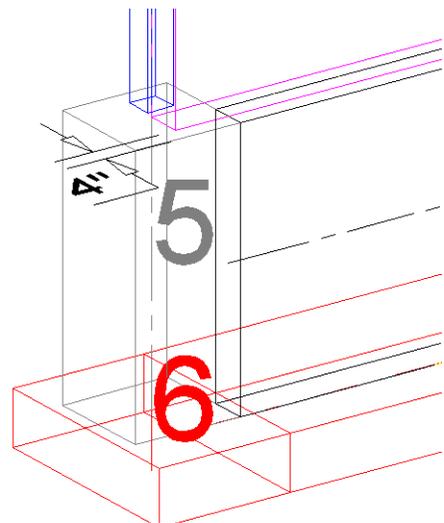


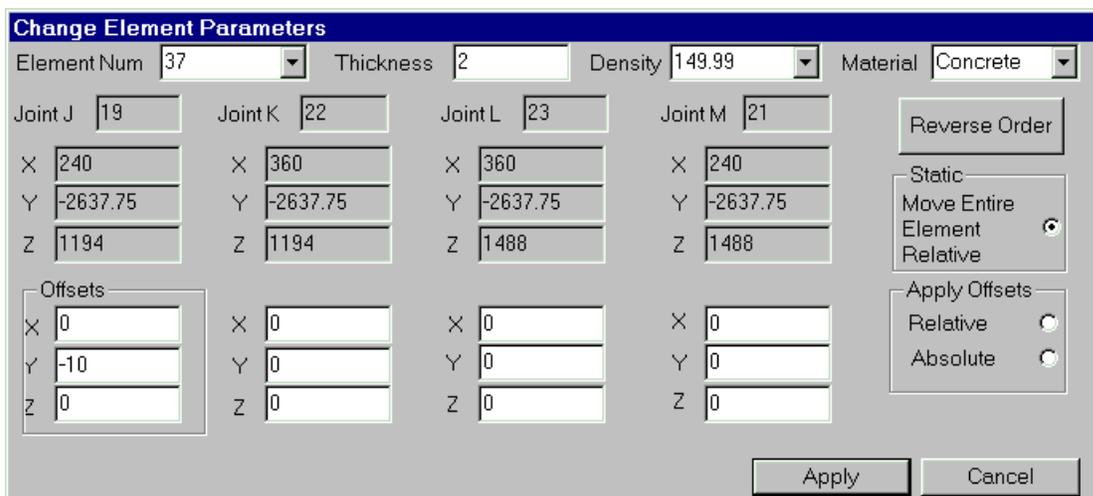
Figure T6.3

The double-tees will be two inches thick, although elements with a thickness of eight inches were used here for design purposes. Since the element extrudes from the face toward us in this instance, if you change the thickness of the element to two inches from eight inches, the front face will be ten inches from the face of the foundation wall member. (The distance is four inches now; reducing the thickness will pull back towards the 3d face object that was established by the original nodes, and we are reducing the thickness by six inches; adding the six to four requires an offset toward us of ten inches.)

Select:

**Structural Desktop** → **Change Element Parameters**

Select all of the elements, and the dialog box will appear.



**Figure T6.4**

Observe the illustration in Figure T6.4 above. The elements you have selected are viewed in the top left drop-down box, followed by thickness, density, and material. The joints are listed in four columns, with the coordinates of the joints listed below each. Below that set of boxes are boxes to display and change the offsets applied to the elements from the original joint coordinates. When the dialog box pops up, the thickness will read "8" and the offsets are zero. Note, also, that if you have followed along, the offsets on your dialog box look different from the framed box to the bottom left above; we will explain that in a moment.

The two frames set to the right of the dialog box deserve consideration. The "Static" box will move an entire element, relative to it's current set of offsets, with just one set of inputs. In other words, if you click the button (as shown above) the offsets will set a frame around the first set of input boxes (again, as shown above) and those values will be added or subtracted from every joint in the elements...the element will move as a whole.

On the other hand, if you click "Relative" or "Absolute" in the "Apply Offsets" frame, all four sets of input boxes will become active. You can move any joint any direction for any element or group of elements, individually. We will use this function soon, but for now,

click in the button in the Static frame and see the frame appear around the input boxes, then input a negative ten into the "Y" axis value of the framed set of boxes. Since the positive "Y" axis points away from us in the view shown, to move toward us the element must be given a negative "Y" value.

Finally, change the "Thickness" value from "8" to "2" and click the "Apply" button. The screen will update and show thinner elements poised on the edge of the foundation wall. The Element Numbers will appear doubled...if this is disturbing, the image can be updated completely by manually clicking the **Display Options** function and clicking "OK".

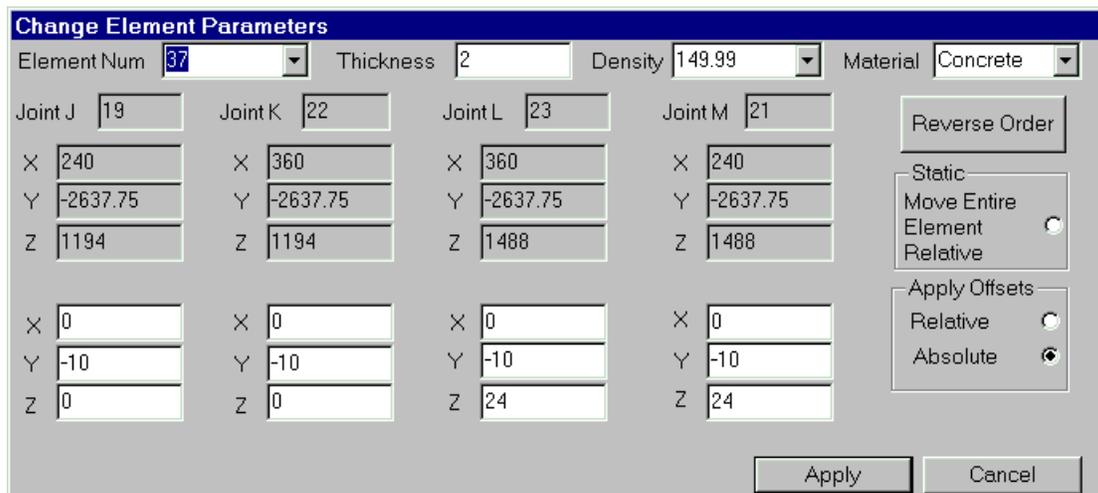
We now have two-inch thick elements flush with the front of the foundation wall. This would be a good time to save a hold file! You do not have to exit from AutoCAD when we save a hold file, so we will continue to extend the tops of the elements to form a parapet.

Another good habit to get into is useful in both analysis file work and in *Structural Desktop*. That habit is consistent numbering; if you always build your members in one direction, you do not need to check to know which node is the "Start" and which is the "End". If you apply a consistent numbering scheme to elements, that also pays off in *Structural Desktop*.

These elements were all numbered counter-clockwise, starting from the lower-left corner. As a result, the last two joint columns in the dialog box are the joints at the top of the element.

**Select:    Structural Desktop    →    Change Element Parameters**

Select all of the elements, and the dialog box will re-appear. In order to "stretch" each element upward by 24 inches, choose the "Absolute" radio button on the "Apply Offsets" frame. Then, enter 24 in the "Z" box of the last two nodes, reading from left to right. The dialog box should look like the one in **Figure T6.5** below. Notice that the earlier application of a negative 10 still shows in the dialog box, but is now shown for all four nodes.



**Figure T6.5**

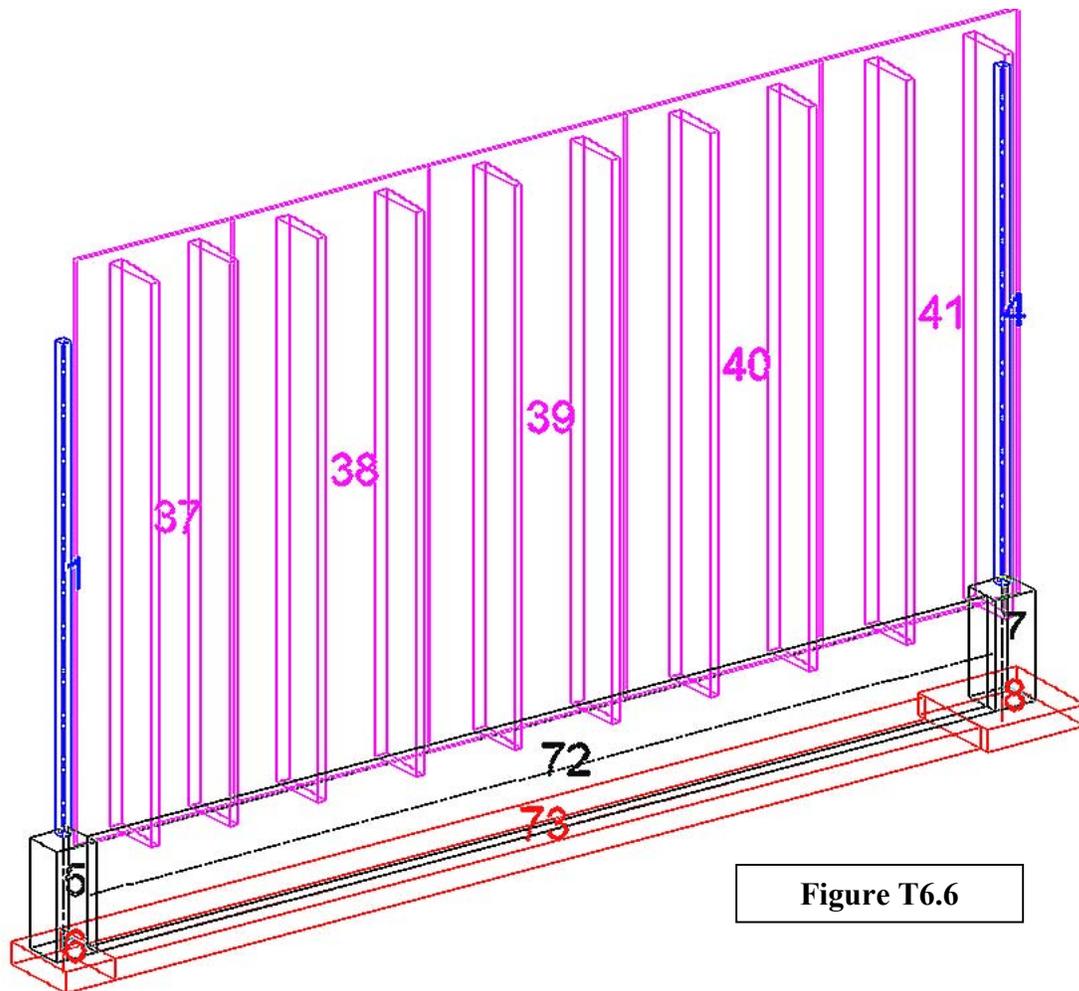
Now, with the location and height set to what we want, we will change these elements into a double-tee object.

Select:

**Structural Desktop** → **Change Element to Double Tee**

Select the elements across the front of the structure, and press enter.

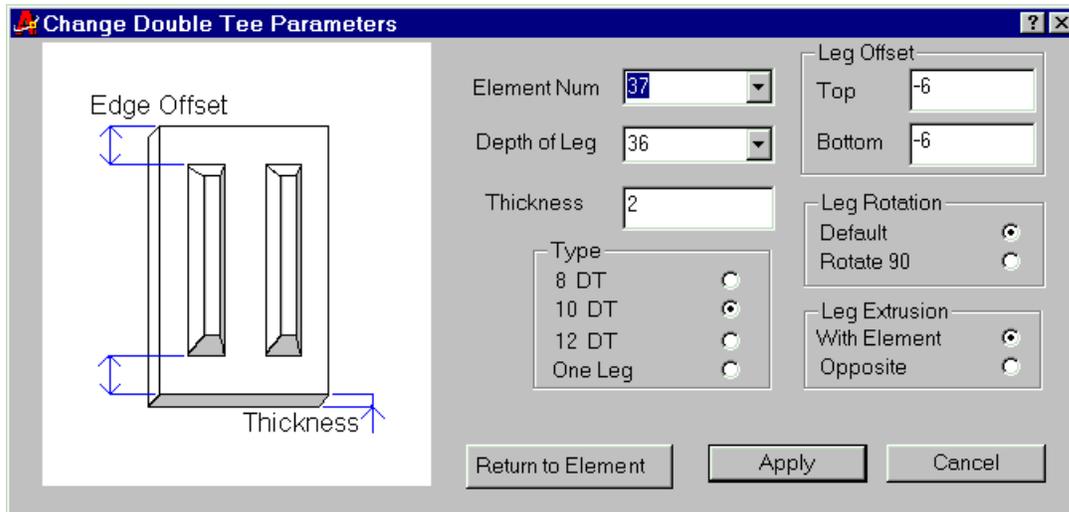
Immediately, the screen should change to appear as in Figure T6.6 below.



There is one more operation to perform on the double tees and we will be finished with the existing structure, as imported from the analysis file. When the double tees are created, they appear with defaults for depth of leg and for the cut at the top and bottom of the double tee leg. Additionally, the double tees are divided into types, based on the distance between the first two points at the base of the element. Each type of double tee (8, 10, or 12) assumes an eight foot, ten foot, or twelve foot width, and positions the legs, centered and half the nominal width apart. If you wish to change any of these defaults, it is time to select:

**Structural Desktop** → **Change Double Tee Parameters**

The dialog box that will appear after you select the double tees is as shown in **Figure T6.7**.



**Figure T6.7**

The function that creates the double tee from an element applies default characteristics. We will change the depth of leg and leg offset settings. Click on the drop-down box for **Depth of Leg** and change the value to 32. Change the **Top Leg Offset** to -24 (the additional height we added to the elements) and the **Bottom Leg Offset** to 0 (zero). Click **Apply** and note the changes in the model.

### **Summary of Step VI**

In Step VI you were introduced to the **Change Element Parameter** dialog box that permits parametric editing of an element. You were shown how to move an element as a unit and you were given instruction in changing just two nodes of an element. You were shown how to change an element into a double tee and introduced to the **Change Double Tee Parameter** dialog box. With this tool we changed the depth of legs and changed top and bottom leg offsets.

In Step VII, we will add grid lines and create plans and elevations from the completed model that will be ready to clean up, put a border around, and check for final approval.

## STEP VII – Grid Lines and Contract Documents

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 simple grid lines where they want them, and to see the dimensions on a drawing before anyone else sees them.

For this simple structure we will place grid lines only at the columns. In the last Step, we had everything turned off except for the front of the structure that had relevance to the double tee elements. Select:

**Structural Desktop** → **Viewing** → **View All**

Next, go to the **Display Options** dialog box, and turn off **Elements** and **Element Numbers**. Turn on **Joints**. When the display has updated, select:

**Structural Desktop** → **Viewing** → **View by Selection**

and select the columns. If you accidentally have selected something in addition to the columns, use the **Hide by Selection** command in the same sub-menu to remove the unwanted additional members.

You will notice that the only joints visible are those integral to the columns, even though the columns have been offset and no longer touch them.

We will perform one more customization of AutoCAD to speed up the grid line process. At the command line, type "OSNAP". A dialog box will appear with all of the snap override options. If you use "OSNAP", make a note of the settings you are currently using so that you may return to them when we are through with the gridlines.

Turn on the "center" snap, and turn any other snaps off. If the check box at the top left of the dialog box is not checked, check it at this time to make "OSNAP" active. Osnap can be toggled on or off at any time by pressing F3 or by clicking on the "OSNAP" toggle at the bottom of the screen.

With "OSNAP" on and set to pick up only the centers of arcs or circles, we are ready to proceed. Select:

**Structural Desktop** → **Grid Lines** → **Add Grid Line**

*Structural Desktop* will respond **Select start point of grid line**. Move the selection cursor over the green joint circle near the base of column 17. When the cursor indicates that it senses the center of the circle representing the joint, click on the joint.



You have completed all the Steps required to add grid lines to the model. Now, we will create drawings from the model. As you create each drawing, take time to examine it and make certain that the objects you intended to have there are there. It takes only a minute to close a drawing (without saving it) and re-create it, once you have the model complete.

## Foundation Plan

To create a foundation plan from this model, use the **View All** command under the **Viewing** group of menu commands. Use the Layer control to freeze the beams, joists, and joist girders. Now, select:

**Structural Desktop** → **Create 2d Drawings** → **Create Plan View**

The program prompts you to select objects. You may select everything on the screen, or just enter "all" at the command line. When you press the enter key again, Structural Desktop will open a NEW drawing window, copy your layer information to it, and draw a flat, 2 dimensional plan representation of everything you have currently selected.

Your drawing should look like **Figure T7.3** below:

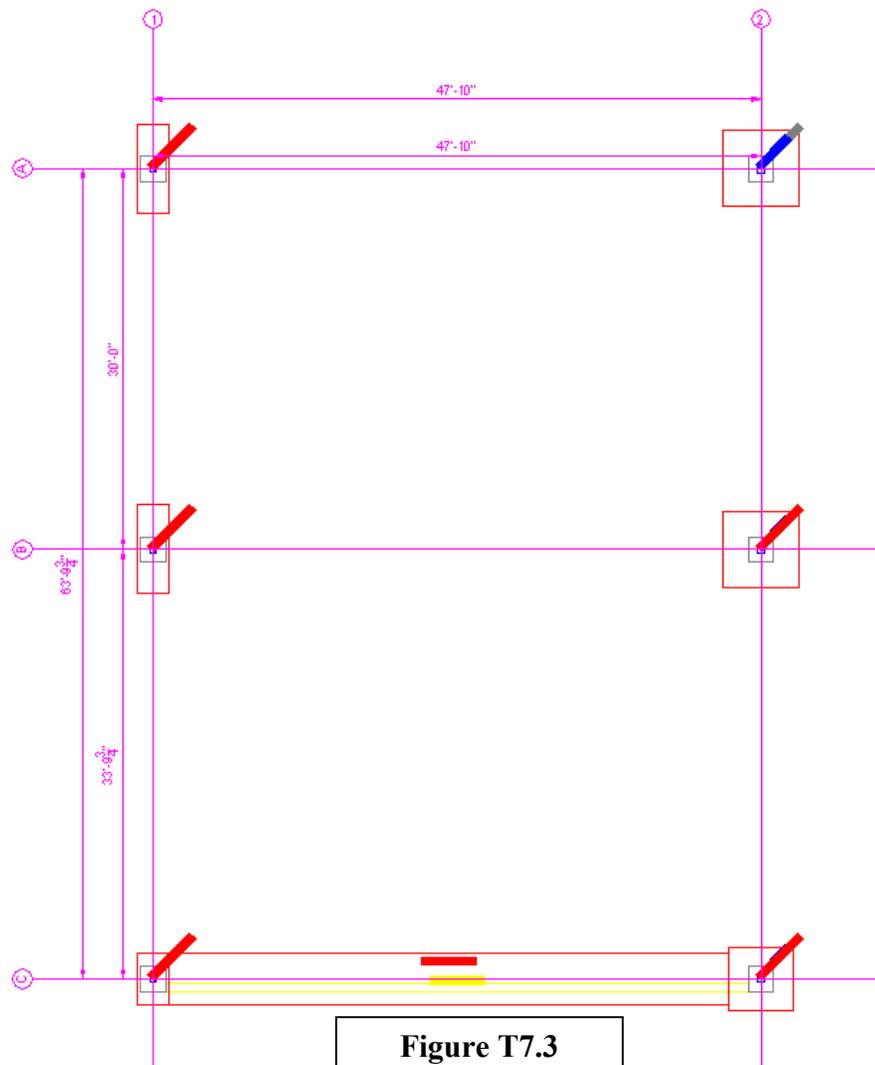


Figure T7.3

Note that the text associated with the different objects is on the same layer as the object, and the objects are on the same layers as they were in your model. Check the drawing to make certain that it contains all of the objects that you want from the model for your foundation plan drawing. Then use the standard AutoCAD "Save" command to save this drawing in the folder of your choice, and close the drawing.

When the drawing has been closed, you should be looking at the model again. Now that you have saved the foundation drawing, it is a pure, AutoCAD drawing. No special preparation or programs are required to open this drawing in any session of AutoCAD (without Structural Desktop). You are completely free to work with this drawing in AutoCAD, share this drawing with others who only have AutoCAD, and merge the drawing with other, AutoCAD compatible, drawing outputs.

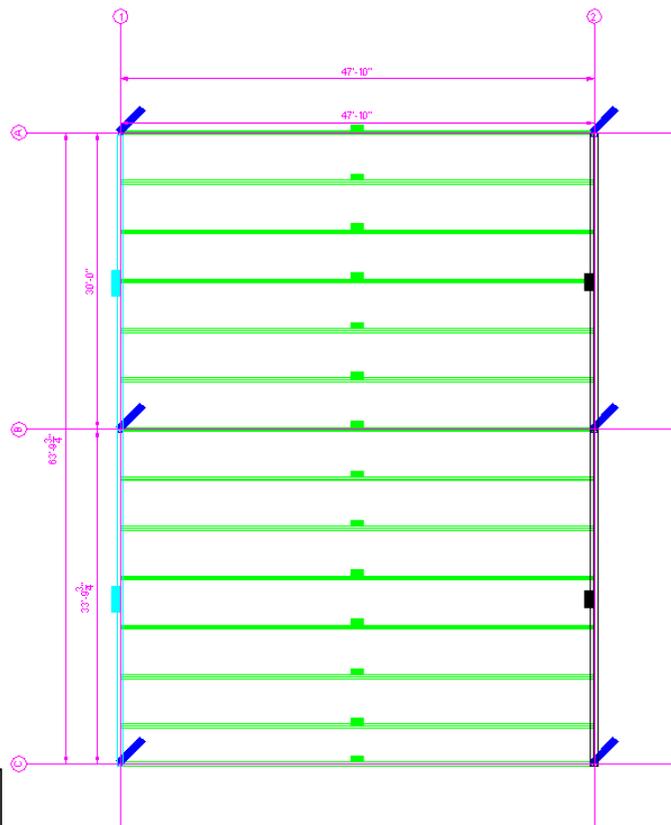
We would suggest that this drawing be attached to a final, output drawing using XREF (see the AutoCAD documentation if this is unclear). That way, if you make changes to the model, this drawing can be re-extracted and notes, borders, paperspace settings, and details will remain largely unchanged. In short, any changes you make to this drawing in AutoCAD will be lost if you recreate the drawing from the model without sufficient forethought.

### Roof framing plan

Next, we will extract a roof-framing plan from the model. In the layer control, turn on the beams, joists, and joist girders and turn off the footing, foundation wall, and pedestal layers. Select:

**Structural Desktop** → **Create 2d Drawings** → **Create Plan View**

and select all the members that remain. Your drawing should look like the one in **Figure T7.4**. Inspect your drawing, save the drawing, close the drawing and return to the model. You will want to move the names around to suit yourself and finish the drawing but, for the most part, the drawing is done. Further work should be done in AutoCAD with **Structural Desktop** not loaded in order to permit your system to run at peak efficiency and to avoid accidentally executing a **Structural Desktop** command on a drawing that does not contain a **Model**.



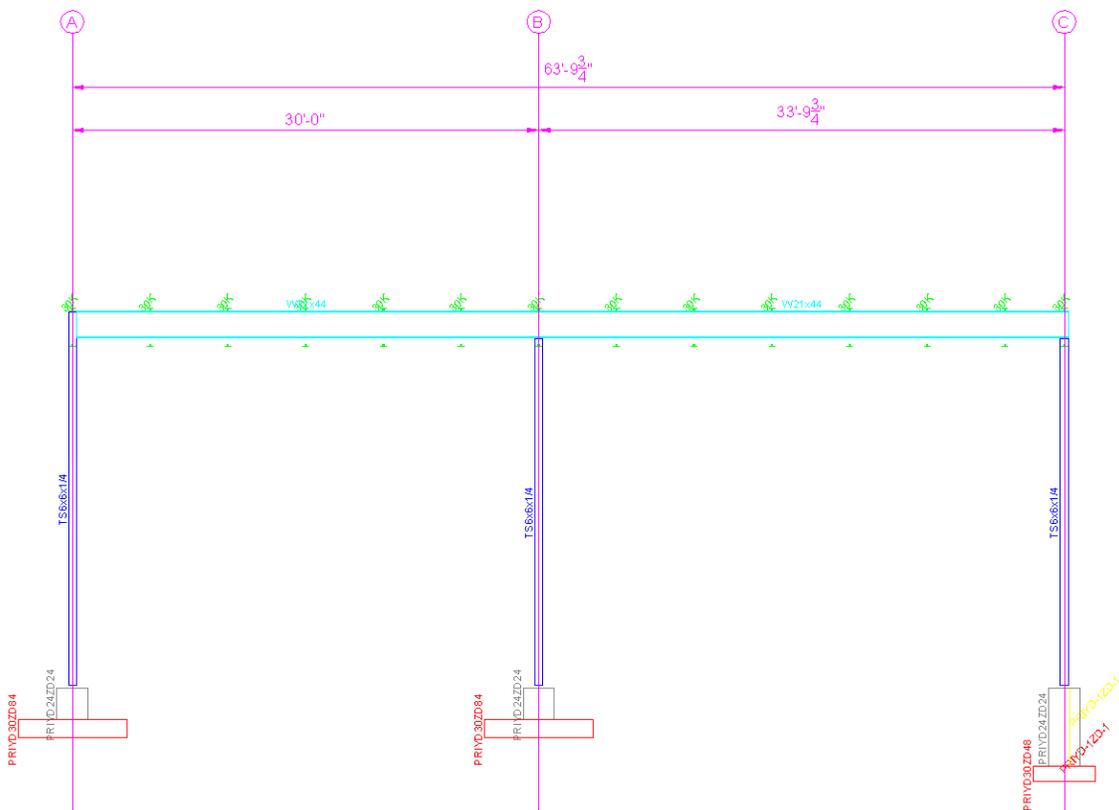
**Figure T7.4**

## Elevations

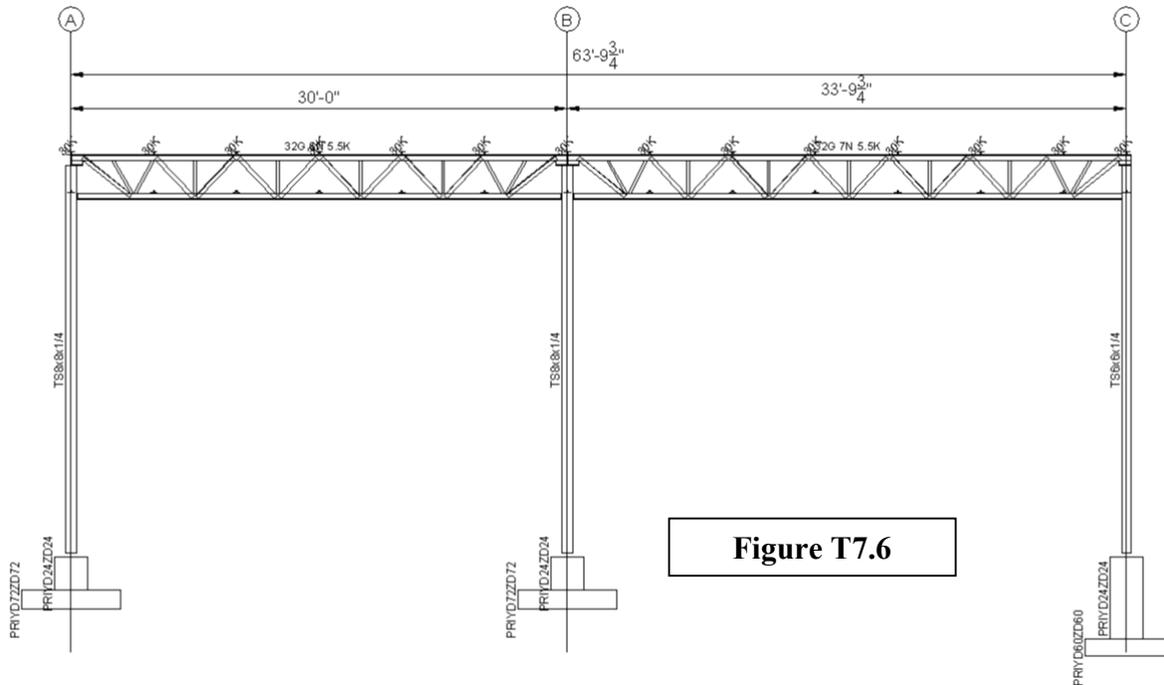
We will finish the Tutorial by creating four elevations. Turn on all layers, and place the model in a plan view. (This can be done simply by typing **plan** on the command line and answering each prompt with **Enter**. See your AutoCAD reference.) Select:

**Structural Desktop** —————> **Create 2d Drawings** —————> **Create Elevation Y-Z**

Select a box bounding all of the objects that cross or interact with Grid Line 1. An elevation will be created in a new drawing that should look like **Figure T7.5**. Save this drawing using the standard AutoCAD "save" function and close the drawing. This should bring you back to the model.



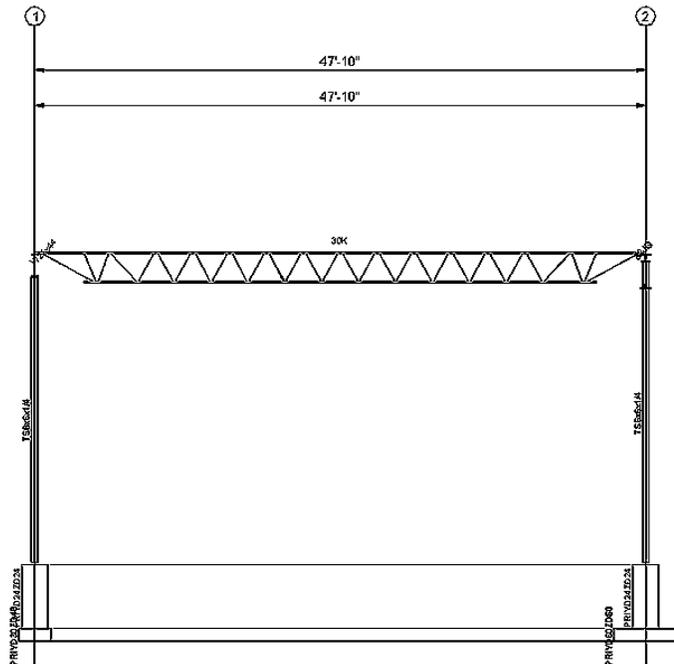
**Figure T7.5**



Invoke the **Create Elevation Y-Z** command again and select the objects crossing Grid Line 2, which should bring you to **Figure T7.6**. Save this drawing and close it. Again, remember: this is an AutoCAD drawing file. Save it using the AutoCAD menu across the top of your screen, or by typing **SAVE** at the command line.

Invoke the **Create Elevation -- X-Z** command and select the objects passing around and through Grid Line C, at the front of the structure.

The drawing that will appear should look like the one at the right in **Figure T7.7**. Save this drawing and close it.

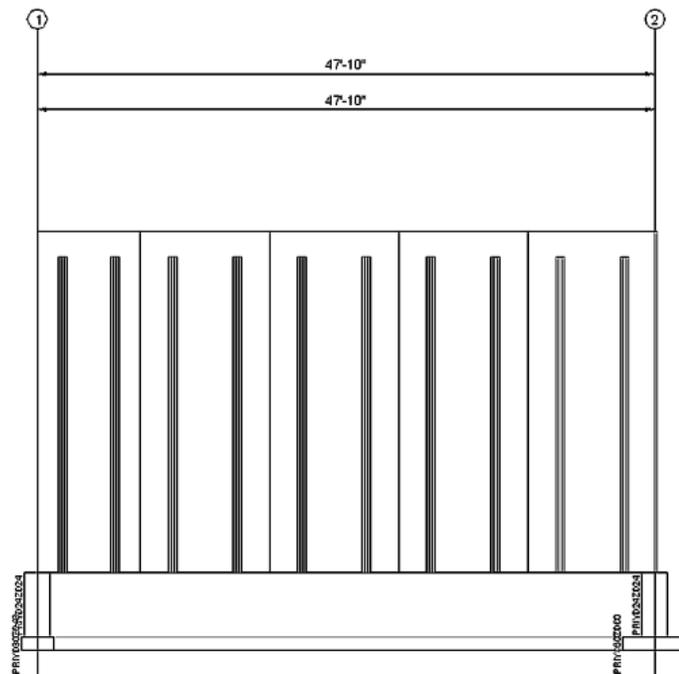


**Figure T7.7**

For our final drawing in the Tutorial, return to the Display Options dialog box and turn on the Elements. Using layers, turn off everything but the Double Tee Wall, Foundation Wall, Footings and Pedestals.

In plan view, select the objects that cross Grid Line C once again, using the **Create Elevation -- X-Z** command. The drawing that is created should look like **Figure T7.8** to the right.

**Figure T7.8**



## **Summary of Step VII**

In Step VII you were shown how to set snap overrides and create Grid Lines linked to the joints at the base of the columns. With grid lines in place, we produced two plan drawings in two dimensions that you could finish, or pass along to a draftsman to be incorporated into a set of contract documents. All of the objects drawn in the drawings were on the layers created within the *Structural Desktop* model.

We produced two elevations along Grid Lines 1 & 2 using the **Create Elevation -- Y-Z** command that depicted the beams and joists, columns and foundation. We created two more elevations across the front line, Grid Line C, using the X-Z variant of the command. One showed the steel and foundation, the other showed the double tee wall and concrete support.

## Conclusion

This concludes the Tutorial for Structural Desktop. Further activities that we would recommend you try on your own with the Tutorial Model are:

- Create 3d Drawings from the model using the **Create 3d Drawings** functions. Try the hide command with them, then save the 3d drawing and return to the model.
- Use the Grid Line Edit functions to change the labels on the Grid Lines.
- Draw a line on layer 0 in the model to represent a brace. Change it to a member using the Change Line to Member command. Use the Change Member Parameters box to give a member type (e.g. a Channel or Angle), give it a material type, and use offsets to place it where it should be to be drawn as correctly as it can be.
- Create a 3d Face on layer 0 to represent a rectangular slab for the structure, making certain to place the nodes of the face in a counterclockwise direction and at the bottom of the slab. Change the slab into an element using the Change Face to Element command. Change the thickness of the slab to six inches, and create a plan drawing showing the slab and pedestals.
- Look at the options under **Display Text Data**. Create a basic **Bill of Materials** for the structure, use save-as with the text file (created, then opened and displayed for you in notepad) to a folder of your choosing, and open the file in your word-processor-of-choice. Delete any sections you do not want in your document.

As your knowledge of *Structural Desktop* increases, you may wish to use AutoCAD commands such as **MOVE** or **STRETCH** to adjust lengths and positions of members. *Structural Desktop* will react to a member being moved or stretched by adjusting the database automatically, and the next time you observe the offsets in the **Change Member Properties** dialog box, they will reflect this movement. We do not use these functions in the Tutorial in order to emphasize that a minimum of proficiency with AutoCAD is all that is necessary to manipulate a model and produce drawings in *Structural Desktop*.

We hope that you find *Structural Desktop* to be a very useful, and profitable, program.