TowerSoft
ENGINEERING SOFTWARE

TSTOWER FOR LATTICED GUYED MASTS

STRUCTURAL ANALYSIS SOFTWARE FOR COMMUNICATION TOWERS

USER'S MANUAL

BY

JOHN WAHBA, PH.D., P.ENG., P.E.

MATTHEW MALINOWSKI, P.ENG.

WWW.TOWERSFT.COM

TABLE OF CONTENTS

Chapter 1 INTRODUCTION

TSTower is a general computer program for the analysis and design of latticed guyed masts. Towers can either have triangular or square cross sections. The program will solve using the American codes TIA/EIA 222-F, ANSI-TIA/222-G, ANSI-TIA/222-H and the Canadian equivalent CSA S37-94, CSA S37-01, CSA S37-13 and CSA S37-18. Load generation, capacity assessments follow the applicable codes, and analysis is based on a three-dimensional beam model.

The program analyzes the tower for all specified wind, ice loads and Earthquake loads and determines the capacity of the members. The results are displayed graphically on the screen. This allows the designer to visually check the adequacy of the design and quickly make adjustments to achieve the optimum solution. The program allows for a full printout or a summary printout of the results. A graphical printout of the profile is also available.

HARDWARE REQUIRMENTS

The following minimum system requirements to run TSTower:

- An IBM compatible PC equipped with a Pentium processor running under Windows 2000, or XP
- 50 MB free disk space
- 128 MB RAM
- 14 inch SVGA monitor with (800 x 600 min resolution)
- A pointing device, a mouse or graphical tablet configured to work under windows.
- An optional printer that is set from windows

INSTALLING TSTOWER

The TSTower installation is initiated from windows. From start menu choose run, and choose file setup.exe

The setup program will create a TSTower directory on the hard drive at a location designated by the user. The user may change the name of the folder in which the program will copy all necessary files required to run TSTower.

Choose the folder name and click OK, the installation program will copy all necessary files into the different directories of your system. Follow the program instructions and place the subsequent disks into the disk drive.

At the end of the installation the program will notify you that the installation was completed successfully. Refer to the purchasers agreement regarding the number of authorized users allowed to run the program.

DISCLAIMER

Extensive care has been taken during the development and testing of TSTower program to ensure that both the source code and the underlying engineering principles comply with standard engineering practice. Should any discrepancies or possible program errors occur, please notify TowerSoft immediately.

> TOWERSOFT DISCLAIMS ALL WARRANTIES IMPLIED OR OTHERWISE WITH REGARDS TO THE SOFTWARE. BY USING THE SOFTWARE, THE USER AGREES THAT NEITHER TOWERSOFT NOR ITS EMPLOYEES SHALL BE LIABLE FOR ANY LOSS, DAMAGE, OR EXPENSE OF ANY KIND WHICH IS CAUSED DIRECTLY OR INDIRECTLY BY THE USE, PERFORMANCE, MAINTAINANCE, SERVICE OR CONDITION OF THE SOFTWARE. IN NO EVENT WILL TOWERSOFT BE LIABLE FOR SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGE RESULTING FROM USE OF THIS SOFTWARE.

User's comments and suggestions are welcomed. Please forward all your comments to support@towersft.com.

Chapter 2 INPUT

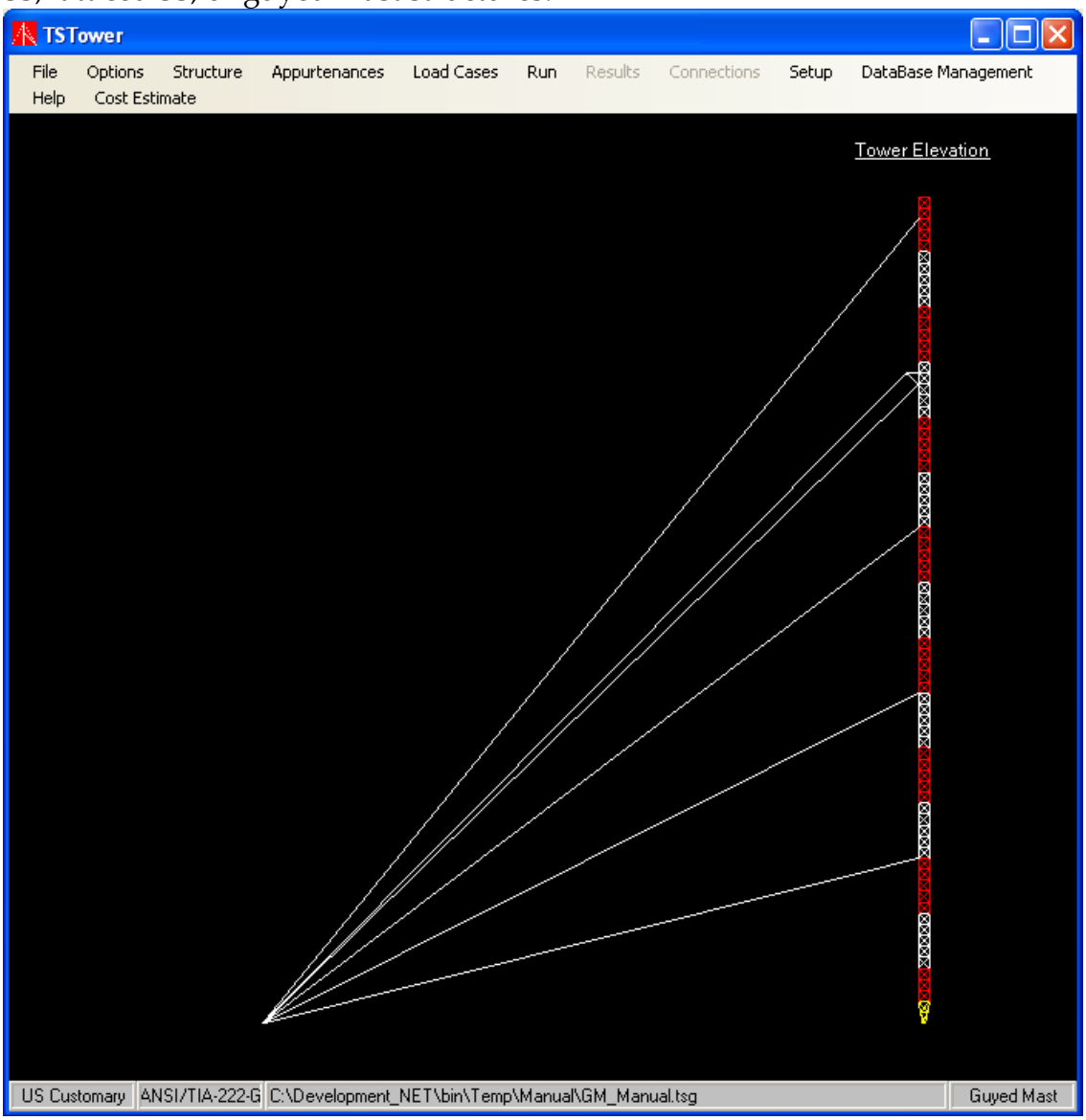
TSTower is an analysis software program. The user inputs all the necessary information required to perform the analysis, the finite element program is run, and the results are shown in a graphical format on the screen.

For design purpose, the user decides on a tentative geometry and performs an analysis. The user examines the results of the preliminary selection and modifies any of the design parameters before reanalyzing the structure. Using successive runs, the user can quickly arrive at an optimum solution. The designer makes the choices, interprets the output and has full control on the design process.

This Chapter illustrates how ***TSTower*** works by explaining the input screens. The input values and screens are shown on the figures in this chapter utilizing an example. This example may not use all the features of the software, however it gives the user a quick introduction to the main features and the use of the program. Features available in the program that are not used for this example are explained in the context.

START THE PROGRAM

When TSTower starts, the main screen shows a blank page. The user selects a structure type from the Structure menu. The three options are either tubular SS, latticed SS, or guyed mast structures.

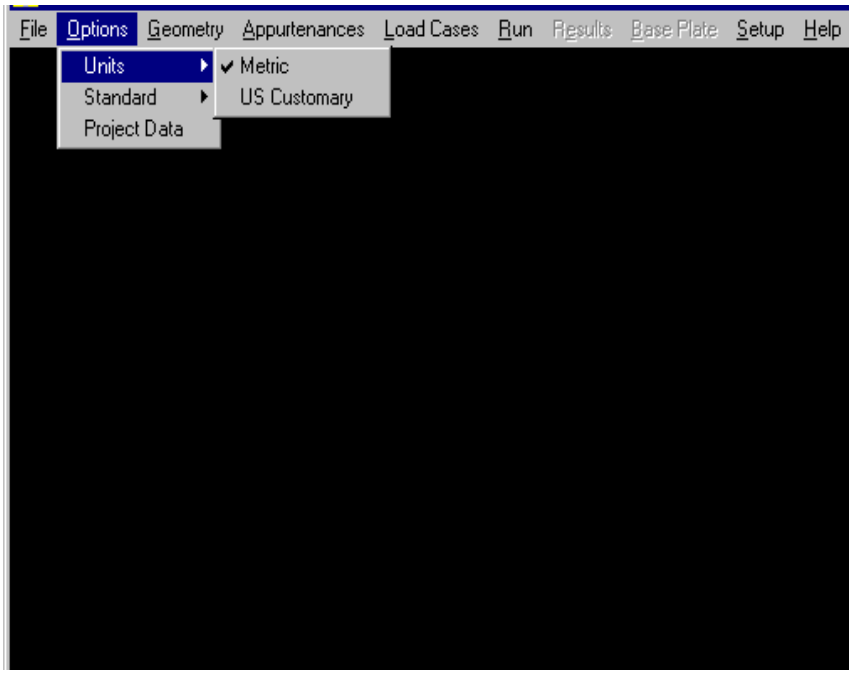


Note the status bar at the bottom of the windows showing the following data in order:

1- The current mode of units (metric or US Customary)
2- The applicable code of design (CSA S37-01, CSA S37-13, CSA S37-94, CSA S37-18, EIA 222-F, ANSI/TIA 222-G, ANSI/TIA 222-H)

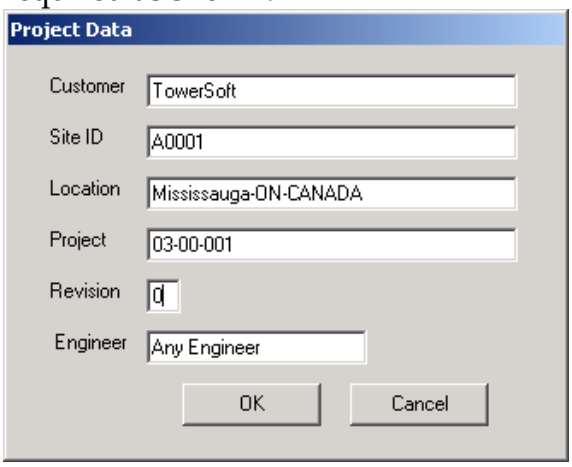
3- The problem file name, and path
4- Structure type (Tubular , Latticed or Guyed Mast)

Select **Options** from the main menu and from the list choose the required code and the system of units.



PROJECT DEFINITION

Select **Project Data** from the **Options** menu. Type in any identification data required as shown.

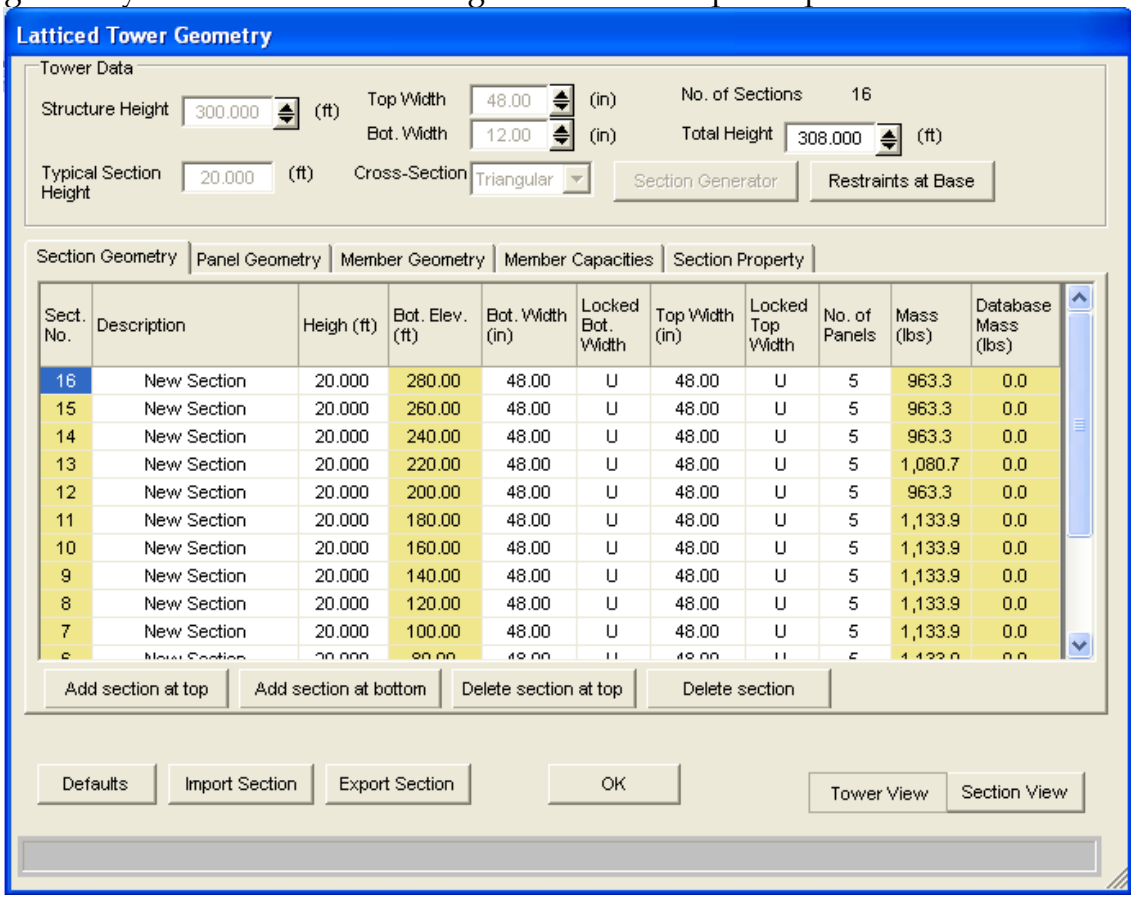


To add data to any box, move the cursor into the box, click inside the box and type in the data using the keyboard. To change any existing data in an edit

box, delete the entry using the <Del> key or the <Backspace> key. Alternatively, highlight the existing characters by pressing and dragging the left mouse button, then type over the existing entry. This is a standard Windows feature.

GEOMETRY DEFINITION

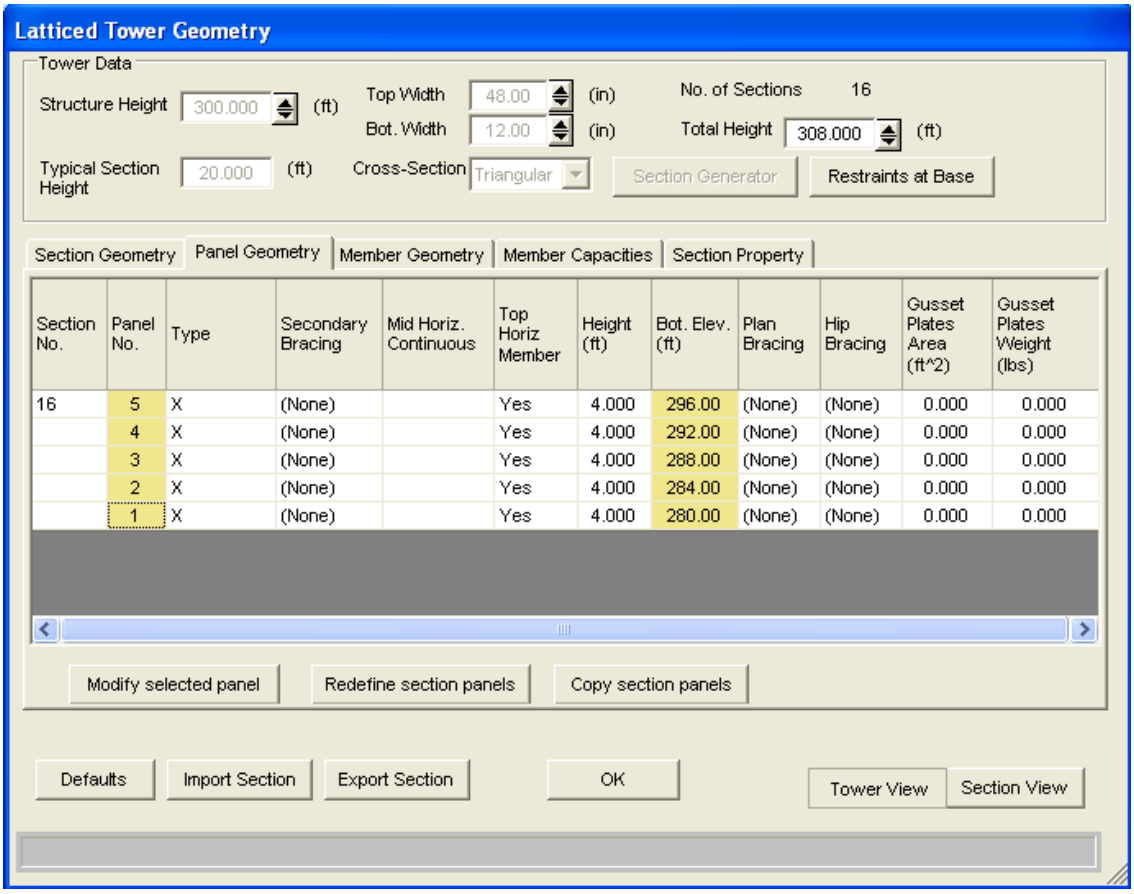
Select **Structure/Guyed Mast/Mast** from the main menu, this will show the Geometry Definition Window. Initially the window will show default geometry data and the user changes that to the required parameters.



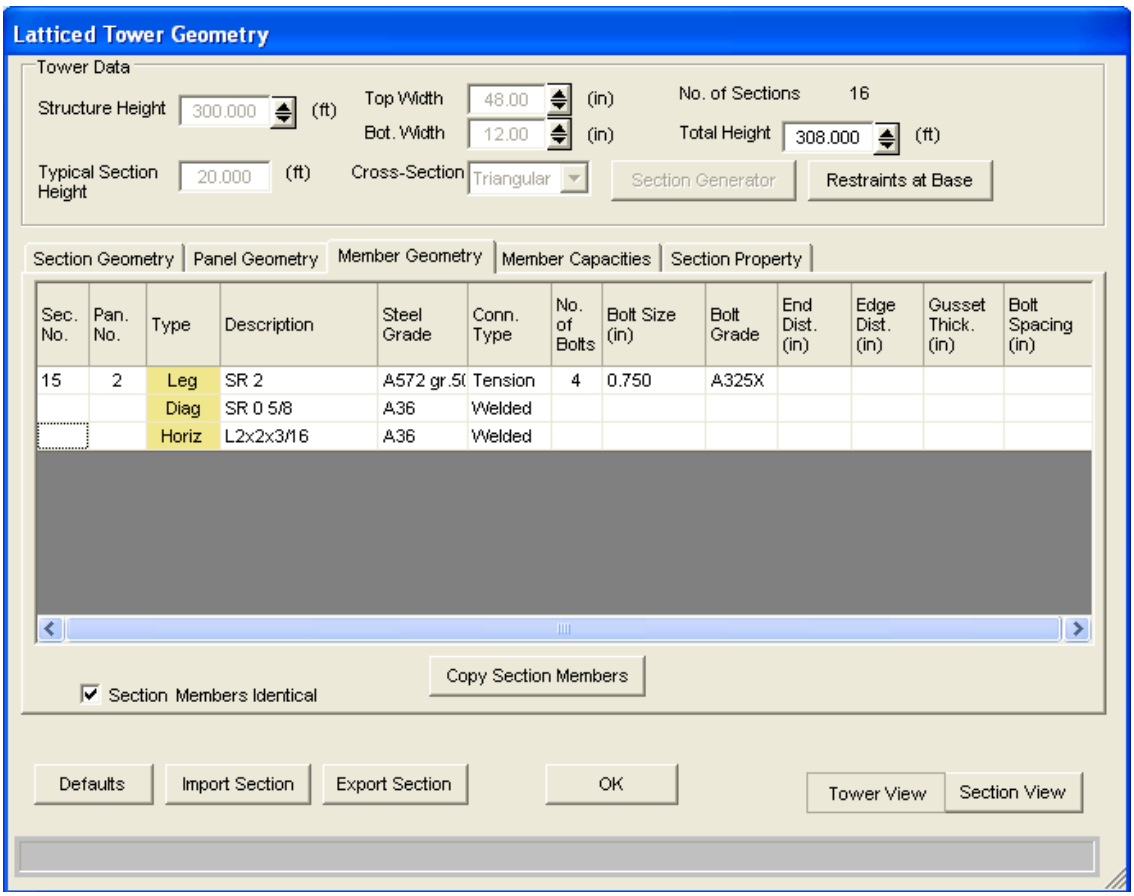
In this window the user inputs the total height, top and bottom widths of the tower, typical section height and clicks on the Section Generator button. This will create the general outline of the tower sections based on the selected typical section height. The user can model multiple slopes on the tower or straight sections for example on the top of the tower by locking top or bottom width of a selected section. From this screen, the user can add sections to the top or bottom of the tower, delete sections at the top of the tower or delete a selected number of sections.

Note that the defaults of the sections generation can be edited from the Defaults button. The user can also switch from Tower View or zoom to Section View to see an enlarged view of the selected section.

The user can also utilize the import and export section functionality to save or retrieve a section from the user-defined database of standard sections.



On the next Tab "Panel Geometry" the user can define the number, type, height of panels for the selected section. From the section column, other sections can be selected for Panel definition. Also on the same window the user can modify selected panels, change heights for different panels, redefine section panels and or copy section panels to other section(s).



From the Member Geometry Screen, the user defines the member sizes, steel grades, connection type, number of bolts, bolt size, bolt grade, end distance, edge distance and gusset thickness. The following is a definition of the different fields:

Member Description: By double clicking the user can select another member from the same type of members (Angles, Tubes, Solid Rounds, etc.). By left clicking the Member Data form will open and allow the user to change the member type or size.

Steel Grade or Bolt Grade: By double clicking the user selects from a drop down list of available grades. The user can add or edit different grades from the Database Management Menu. By left clicking the yield and ultimate values of the selected grade are displayed. Note that for bolt grades, the threads included or excluded from the shear plane are available.

Connection Type: By double clicking the user can select connection type. For legs the user selects from tension, single shear or double shear. For other members the selections are either welded to bolted.

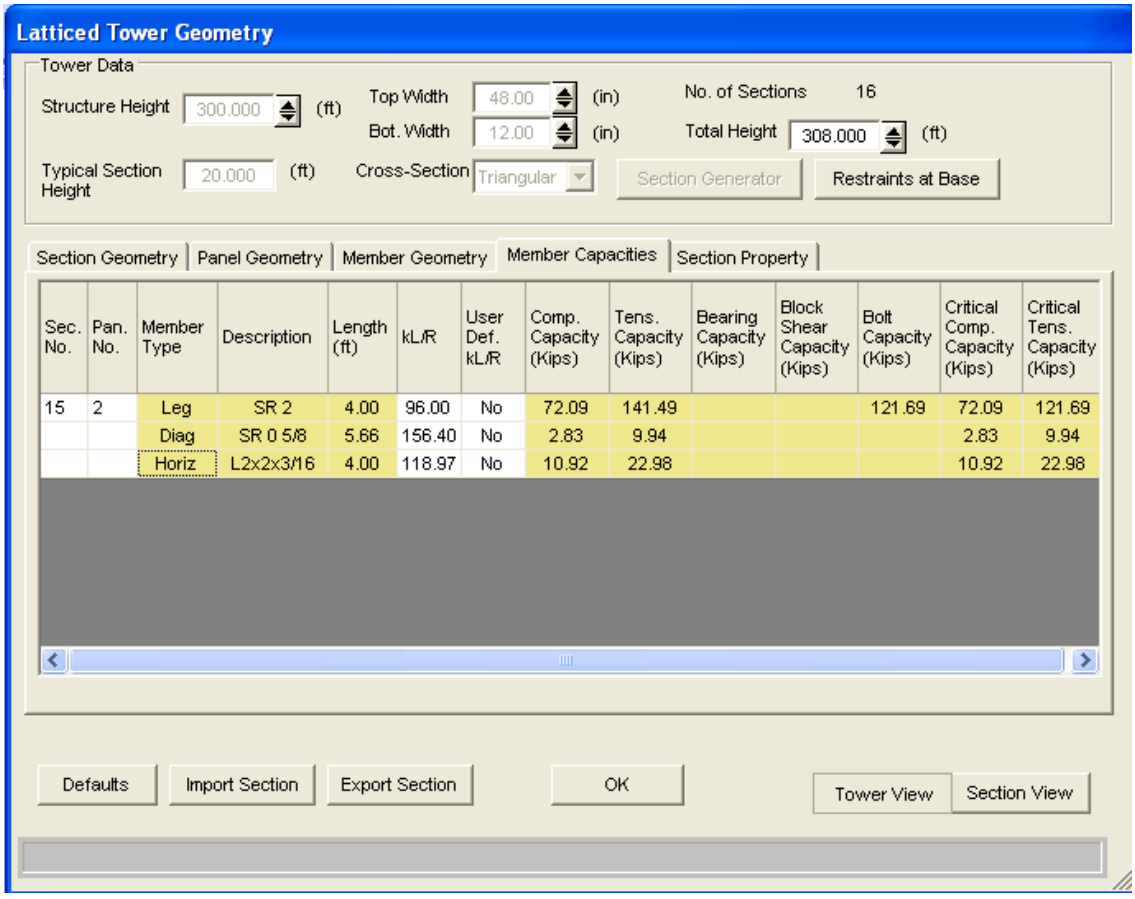
Number of Bolts: the user selects or types in the number of bolts for the connection.

End distance: defined as distance from the center of bolt to the end of the member along the axis of the member (force).

Edge Distance: defined as the distance from the center of bolt to the edge of the member (normal to the line of force).

Gusset Thickness: the gusset thickness for connection of legs to other members. For leg angles, the thickness of the leg is assumed to be the gusset thickness but can be over written by the user.

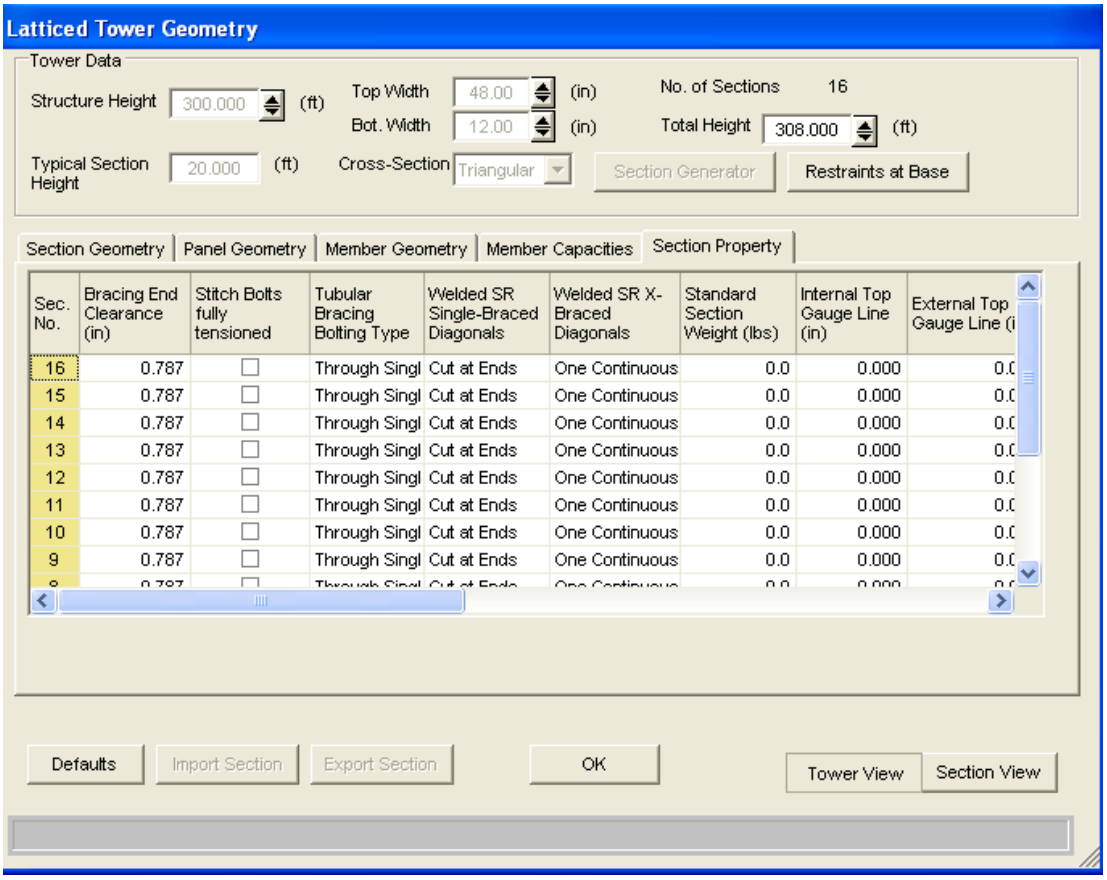
Note that the click button at the bottom allows the user to select either identical or different members for each of the panels of the section.



The Member Capacities tab displays the following data:

- Section number (from bottom to top)

- Panel number (From bottom to top)
- Member Description
- Length of member (m or ft)
- KL/R (program calculated effective slenderness ratio – can be overwritten by User)
- User Def. kL/R (No) means kL/R calculated by program, (Yes) means that the User has overwritten the kL/r and the Users entry will be used for capacities
- Compression Capacity (kN or Kips)
- Tension Capacity (kN or Kips)
- Bearing Capacity (kN or Kips)
- Block Shear Capacity (kN or Kips)
- Bolt Capacity (kN or Kips)
- Critical (Governing) Compression Capacity (kN or Kips)
- Critical (Governing) Tension Capacity (kN or Kips)



On the next Tab "Section Property" the user can define following section properties:

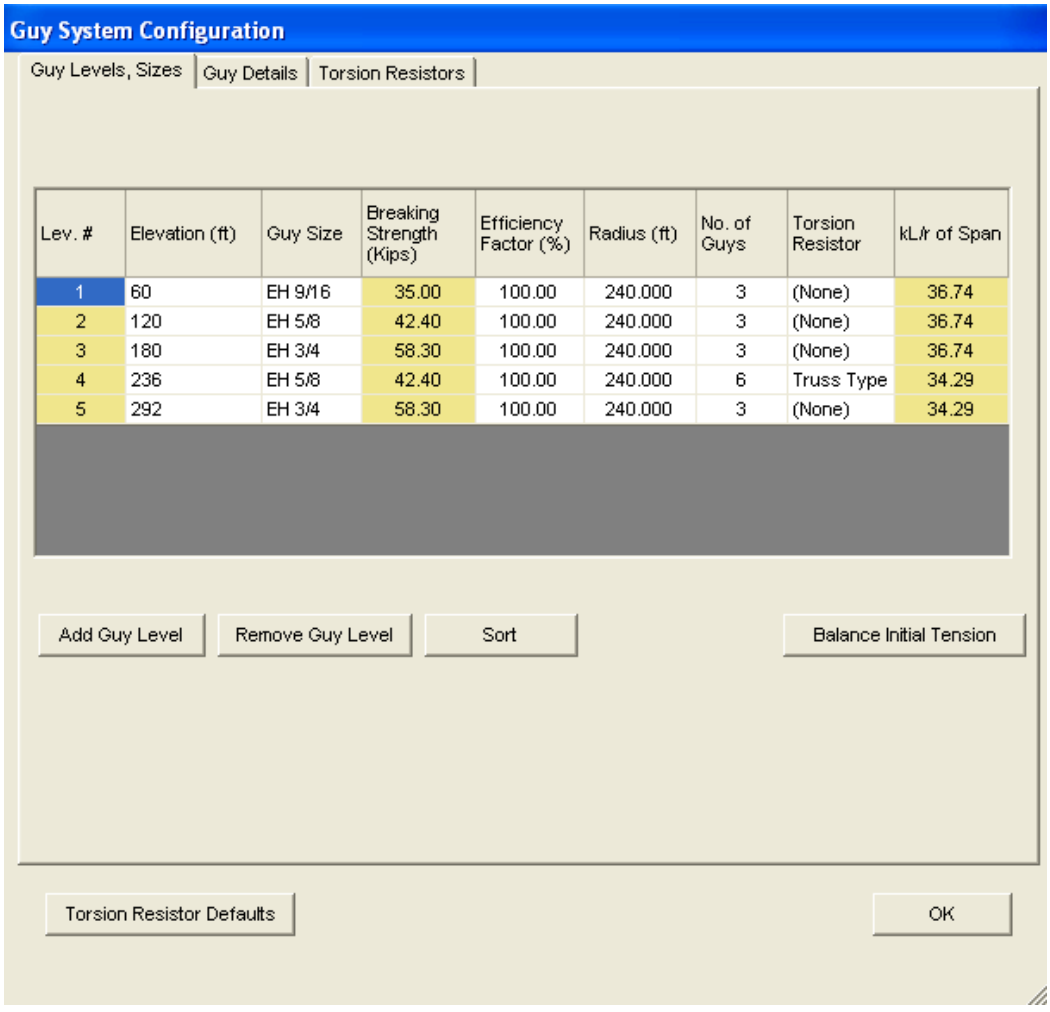
Bracing End Clearance: Applies to bracing members (angles) attached to solid round or tubular legs using gusset plate. It is the distance from the corner of the bracing angle to the surface of the leg measured perpendicular to the leg.
Stitch Bolts fully tensioned: Applies to back-to-back angles. By default the stitch bolts are considered not fully tensioned. By clicking on the tick box this will be changed to fully tensioned (tick in the box indicates fully tensioned).
Tubular Bracing Bolting Type: Applies to tubular bracing members attached by bolting. Following options are available: Through Double Wall (default), Through Single Wall and Through Gusset Wall.
Welded SR Single Braced Diagonals: Applies to single braced panels with solid round bracing members welded to legs. Following options are available: Cut at Ends and Bent Continuous. Not applicable when using EIA-222-F standard.
Welded SR X-Braced Diagonals: Applies to X-Braced panels with solid round bracing members welded to legs. Following options are available: One Continuous, One Broken, Welded; Both Continuous, Welded; Both Continuous and Bent, Tack Welded. Not applicable when using EIA-222-F standard.

Standard Section Weight (lbs or kg): Applies only to design/analysis of standard towers, where structure data comes from databases.

Remaining section properties like Internal Top Gauge Line, External Top Gauge Line, etc., are applicable only for Users performing design/analysis of standard tower for the sake of matching sections to each other.

GUY SYSTEM CONFIGURATION

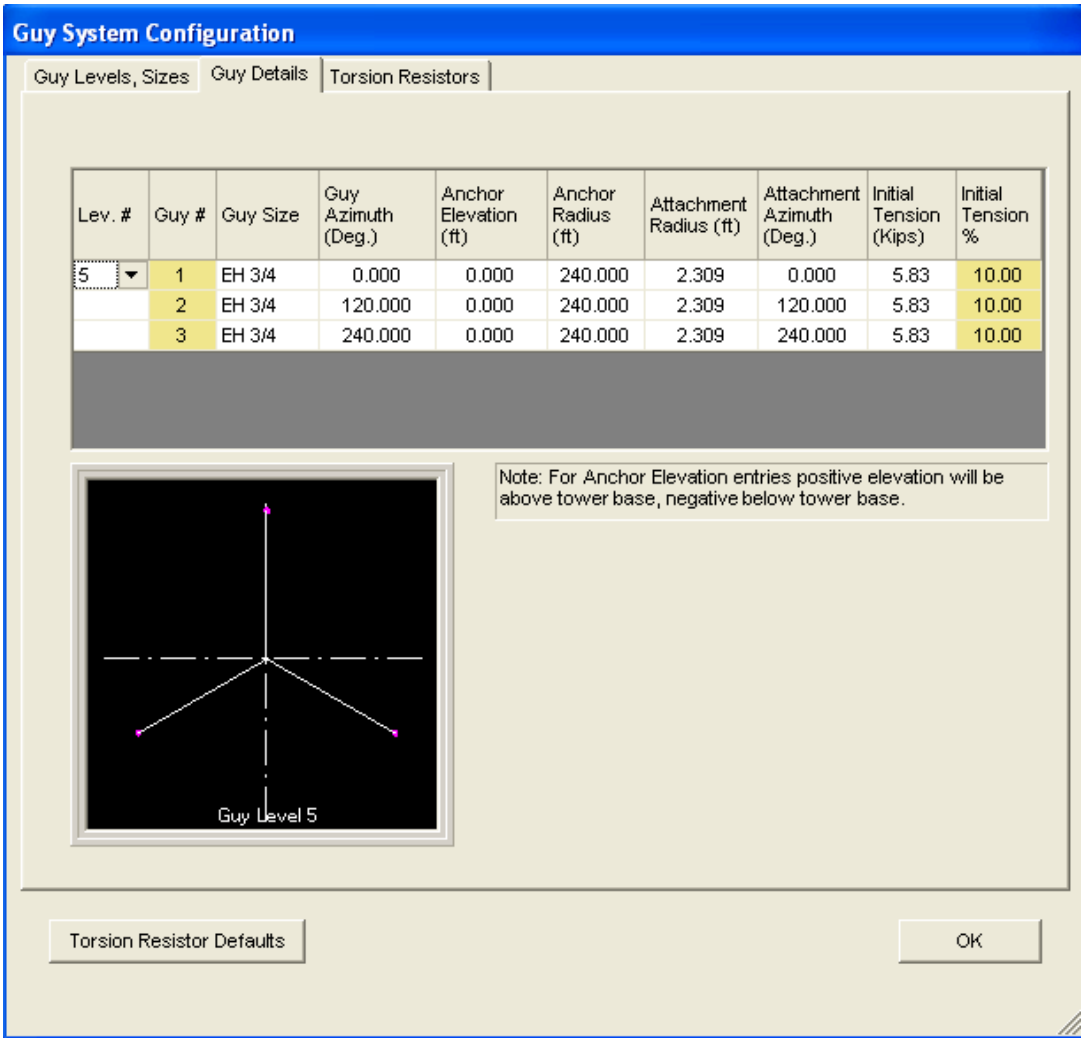
Select **Structure/Guyed Mast/Guy System** from the main menu, this will show the Guy System Configuration Window.



The following is a definition of the different fields:
Elevation: Distance from base of the tower to guy attachment at mast
Guy Size: Selected from Database table
Breaking Strength: Read-only field – matching selected guy size with Breaking Strength from Database table
Efficiency Factor: By default 100%. Users should determine if lesser efficiency factor should be used depending on guy hardware
Radius: Distance from the center of the mast to guy attachment at anchor
Number of Guys: Default (minimum) 3 for triangular towers and 4 for square towers at given guy level. For Torsion Resisting (Torque Arm) systems the number of guys is twice the default (6 for triangular and 8 for square towers).
KL/R of Span: Span slenderness calculated by the program.

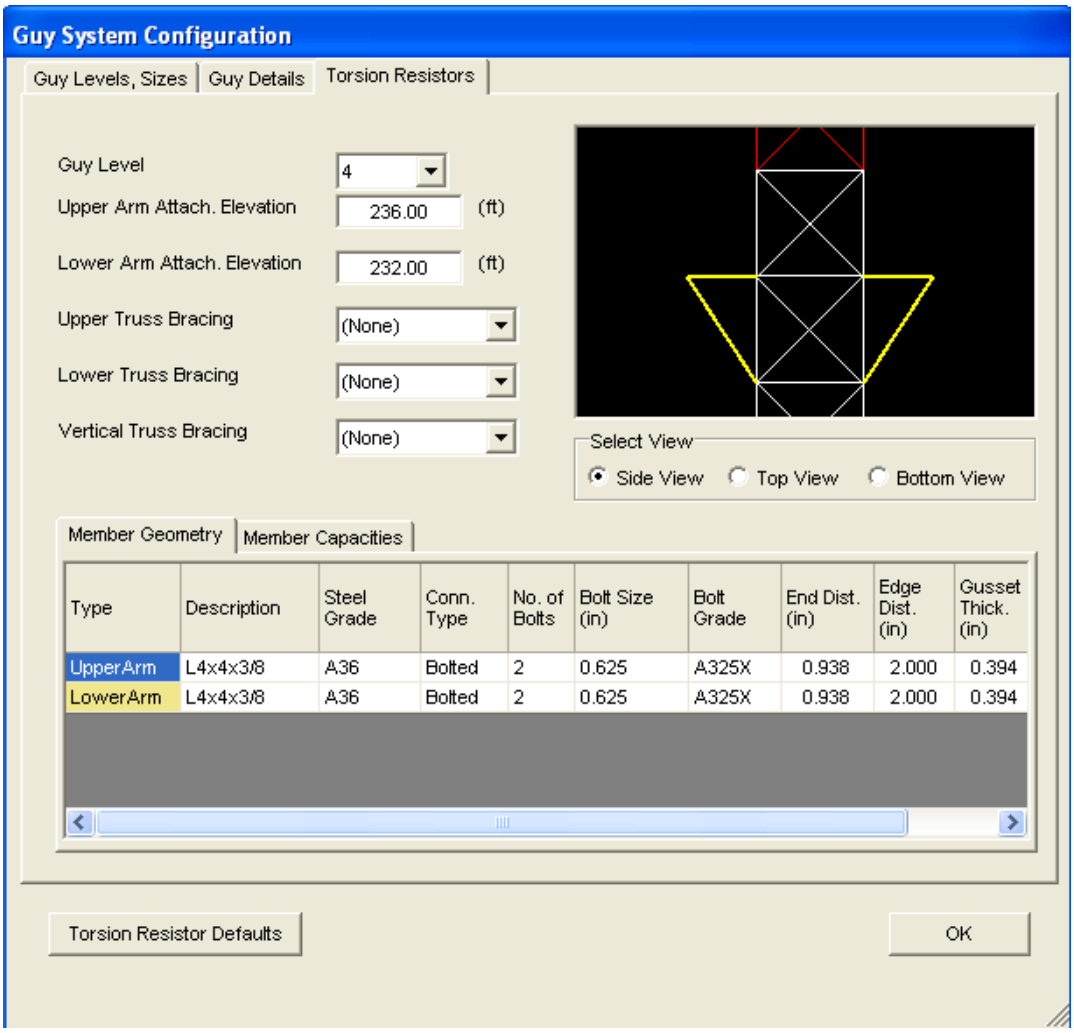
From the Guy System Configuration Screen, the user defines the guy level' elevation, guy size, guy efficiency factor, guy radius, number of guys and torsion resistor type (if applicable).
Clicking on "Add Guy Level" button creates a new guy level. The choice of elevations in form of "drop-down" box will appear in cell of "Elevation" column. Please, note that the program chooses the available elevations as nodes (panel points). You may, however, type desired elevation directly in the cell instead of selecting it from the drop-down box.
A guy level can be deleted by selecting (setting focus) at given row and by clicking on "Remove Guy Level" button.
Guy size and type can be determined by clicking in the cell of column "Guy Size". A drop-down box allowing selection of any guy from available Guys database appears. Breaking Strength (read only) appears in the column adjacent to Guy Size column.
Torsion Resistor type (applicable only when the number of guys is two times more than the number of tower legs – 6 guys for triangular tower and 8 guys for square tower) can be selected from drop-down box as "Truss Type" or "Beam Type". Please note that the selection of 6 guys for triangular and 8 guys for square towers with guys attached symmetrically to torque arms, even without selecting Torsion Resistor, will still be treated by the analysis as a torsion resisting system. The difference of choosing explicitly a torsion resistor will make the program to consider the Torsion Resistor members for loading (both wind load and their weights) and will show in the output the assessment of the Torsion Resistor members.

Please note that the values contained in the tab "Guy Levels, Sizes" of the form are general and some of them (such as Guy Sizes and Radiuses) may differ for each guy at a given level. More detailed definitions of the guys' parameters for a given level are maintained in tab "Guy Details" of the form.



The following is a definition of the different fields:

Guy Azimuth: Azimuth of guy anchor measured from tower's North (triangular towers have North going through the apex leg and for square towers the North line is perpendicular to the face).
Anchor Elevation: Vertical Distance of the anchor from the tower's base. Positive elevations above the base, negative below the base.
Attachment Radius: Distance from the center of the tower to the attachment of the guy at mast (for Torsion Resistors to the attachment of the guy to the arm).
Attachment Azimuth: Angle between the tower's North and line going through the guy's attachment.
Initial Tension: By default 10% of the Breaking strength but may be overwritten by User.



Torsion Resistor Geometry and members:

The following is a definition of the different fields:

For Torsion Resistor Geometry and Bracing type (for Truss Type):

Upper Arm Attach. Elevation: Elevation of the attachment of the upper arm of the Torsion Resistor to the mast
Lower Arm Attach. Elevation: Elevation of the attachment of the lower arm of the Torsion Resistor to the mast
Upper Truss Bracing: Available choices are None, Single Braced, X-Braced
Lower Truss Bracing: Available choices are None, Single Braced, X-Braced
Vertical Truss Bracing: Available choices are None, Diagonal, Diagonal + Horizontal

Each of the above-mentioned bracings, if selected, will be shown in Selected View (Side, Top and Bottom Views) and the default bracing members will be added to the grid below (Member Geometry tab).

For Member Geometry:

Member Size: the cell in column "Description" serves to select the member size and type from Steel Database tables. Double clicking on selected cell allows to re-select member from the same member type table (e.g. equal angles). Right hand clicking on the selected cell allows to choose any table of steel shapes and to select a member from such table.

Steel Grade or Bolt Grade: By double clicking the user selects from a drop down list of available grades. The user can add or edit different grades from the Database Management Menu. By left clicking the yield and ultimate values of the selected grade are displayed. Note that for bolt grades, the threads included or excluded from the shear plane are available.

Connection Type: By double clicking the user can select connection type. For arms the user selects from tension or bolted. For other members the selections are either welded to bolted.

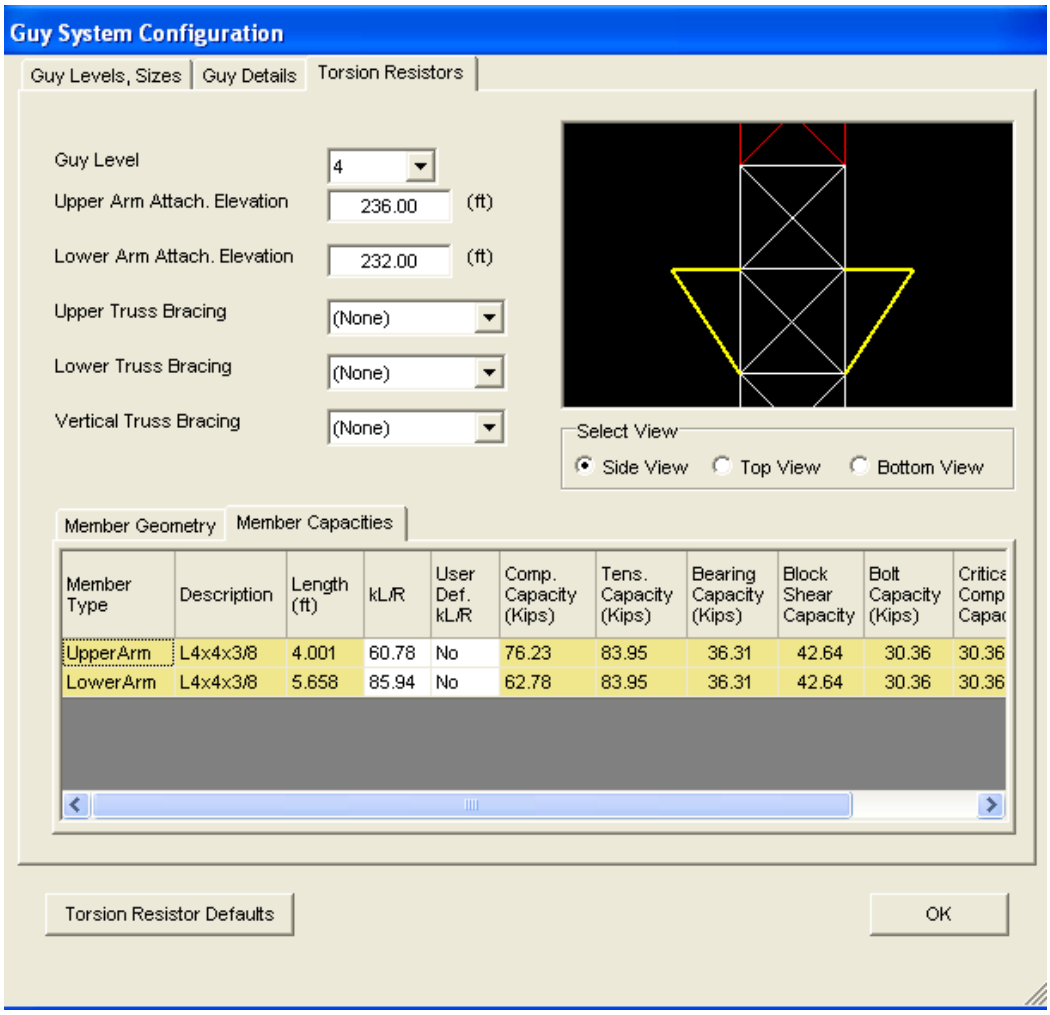
Number of Bolts: the user selects or types in the number of bolts for the connection.

End distance: defined as distance from the center of bolt to the end of the member along the axis of the member (force).

Edge Distance: defined as the distance from the center of bolt to the edge of the member (normal to the line of force).

Gusset Thickness: the gusset thickness for connection of arms to other members. For leg angles, the thickness of the leg is assumed to be the gusset thickness but can be over written by the user.

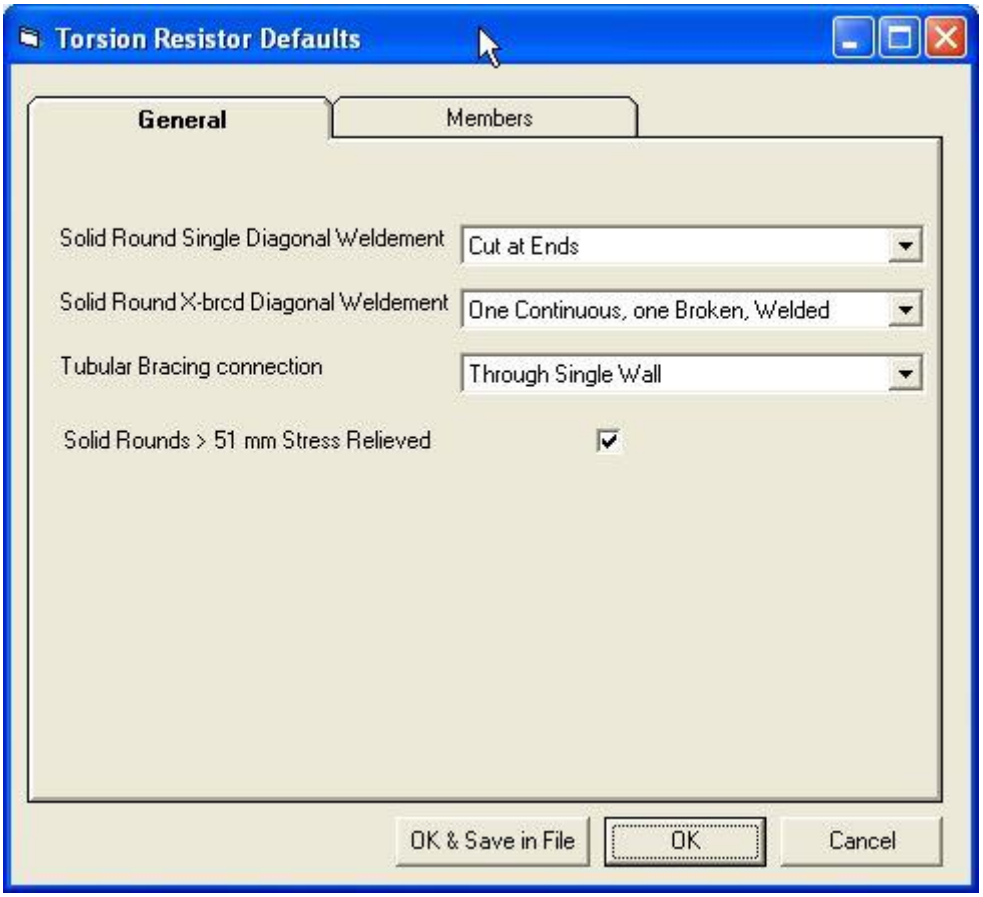
Bolt Spacing: defined as the distance between the centers of adjacent bolts (normal to the line of force).



The Member Capacities tab displays the following data:

- Member Type
- Member Description
- Length of member (m or ft)
- kL/R (calculated effective slenderness ratio) but User can directly overwrite this value
- User Def. kL/R (No) means kL/R calculated by program, (Yes) means that the User has overwritten the kL/r and the Users entry will be used for capacities
- Compression Capacity (kN or Kips)
- Tension Capacity (kN or Kips)
- Bearing Capacity (kN or Kips)
- Block Shear Capacity (kN or Kips)
- Bolt Capacity (kN or Kips)
- Critical (Governing) Compression Capacity (kN or Kips)

- Critical (Governing) Tension Capacity (kN or Kips)



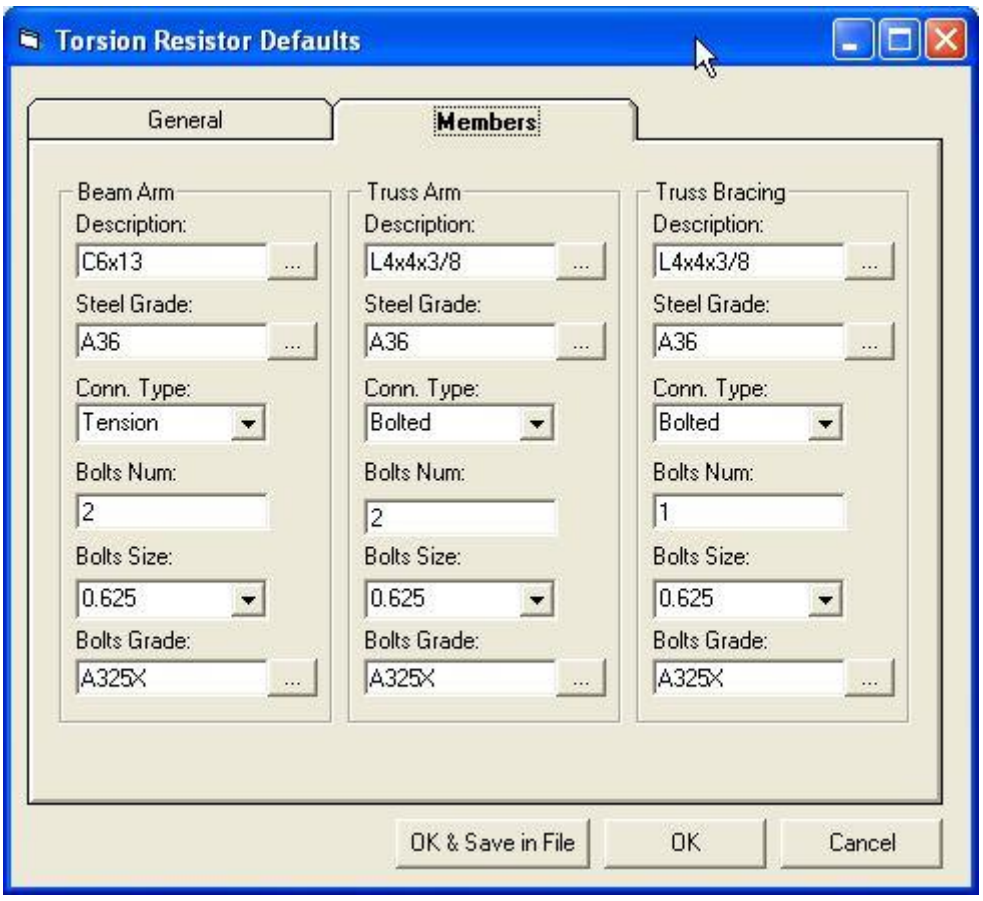
It is recommended that the User sets up Torsion Resistor Defaults before working with the details of the members.

The form above illustrates the General properties of the members.

The next tab, "Members", allows declaring default member types, steel grades, connections, number of bolts, bolt sizes and grades.
Separate selections can be made for beam arm, truss arm and for truss bracing members.

When defaults are defined the User may choose to:

Save in File – that will keep the current selection until the next change and saving – Button "OK & Save in File",
Save for the current tower only – Button "OK",
Cancel – to ignore all selections – Button "Cancel".

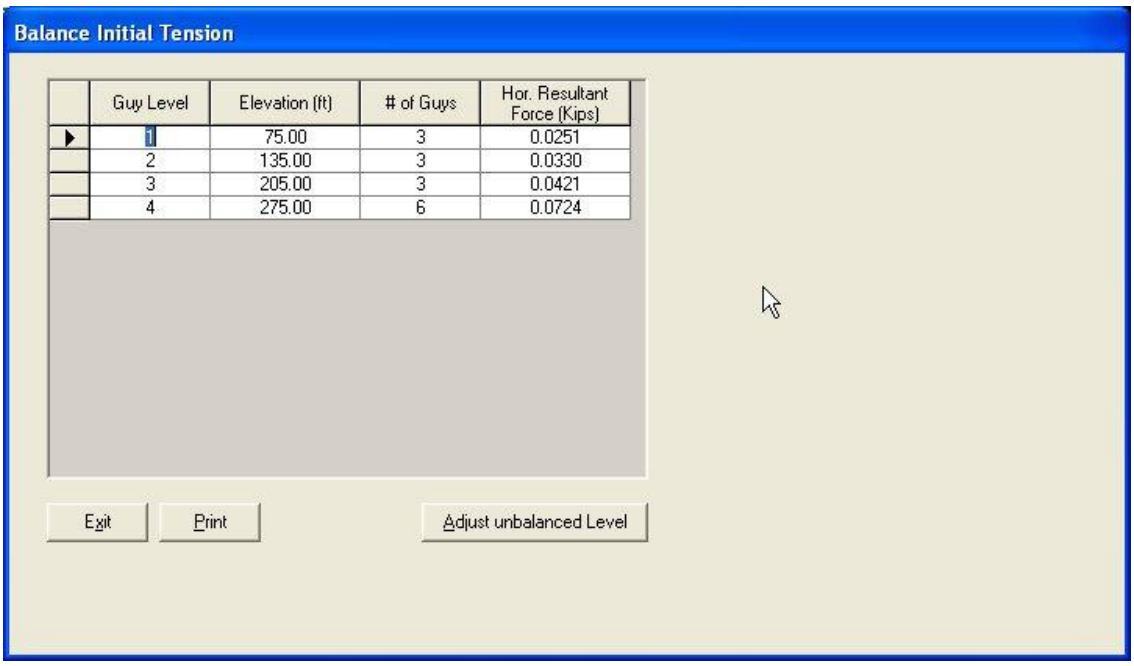


Balancing Initial Tension

The variations in anchor radiuses and elevations at particular guy level may cause significant lack of balance of the resultant horizontal force acting at this guy level. For newly designed towers and for proposed modifications, reinforcing and maintenance of existing towers it is recommended to bring the guy system to the balanced state, where the resultant horizontal is equal or close to zero.

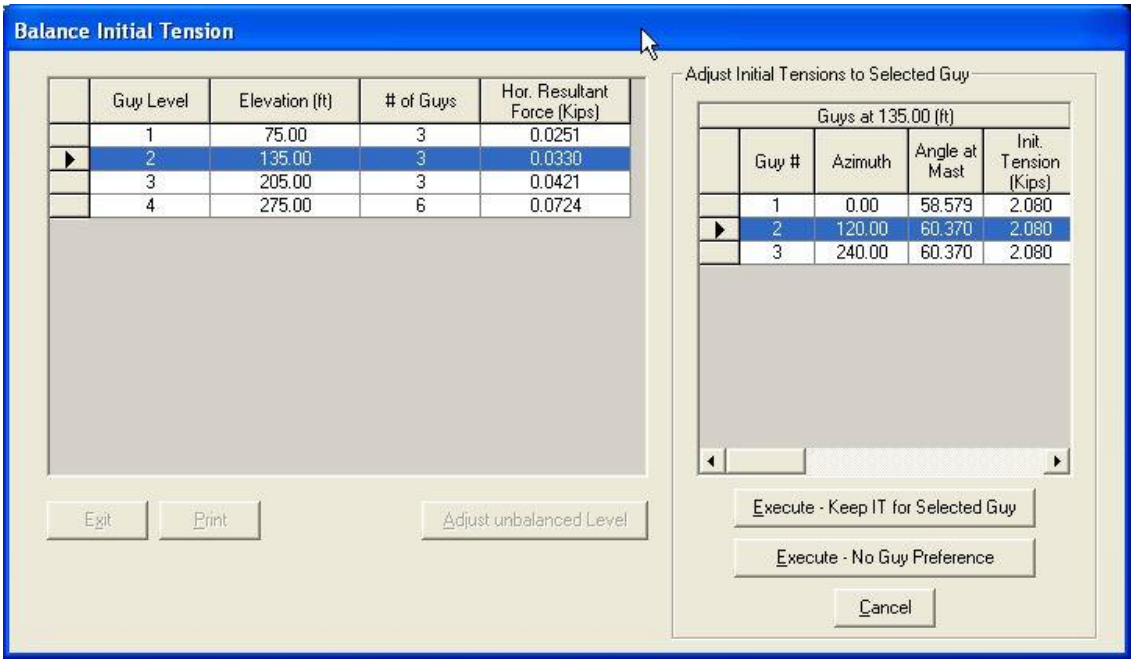
Click on "Balance Initial Tension" button on the form "Guy System Configuration", tab "Guy Levels, Sizes".

Following screen "Balance Initial Tension" will show for each guy level the value of Horizontal Resultant Force.



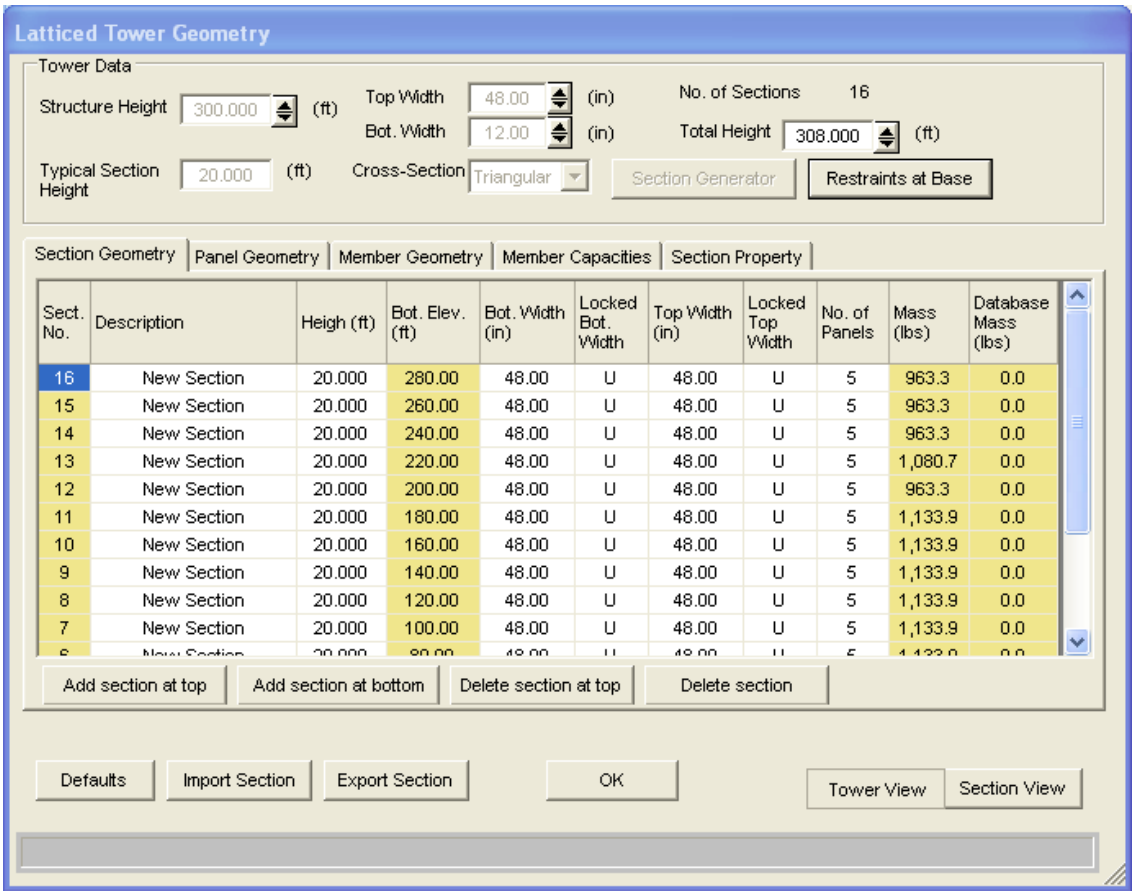
In case the horizontal resultants are considered too large (need for balancing) please click on "Adjust unbalanced Level" button for a selected guy level.

Following screen appears. You may choose to execute the balancing with Initial Tension kept "as is" for selected guy, to execute the balancing without any preference or to cancel the balancing.

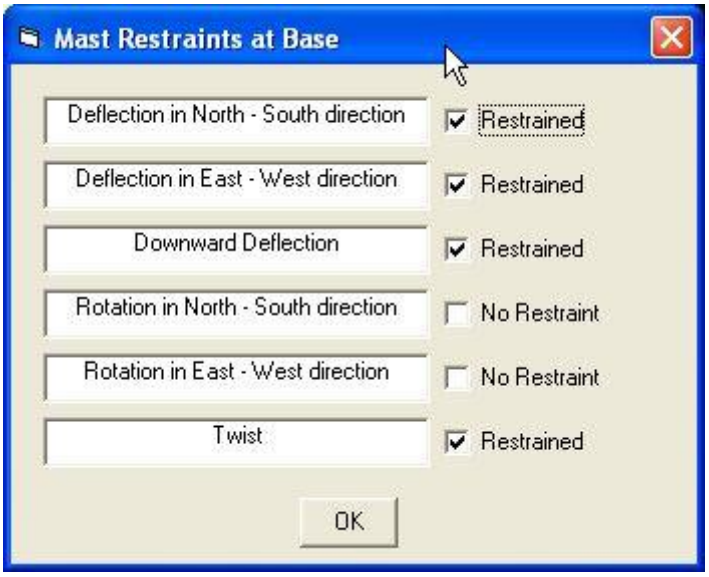


Setting the Restraints of the mast at base

Click on "Restraints at Base" button

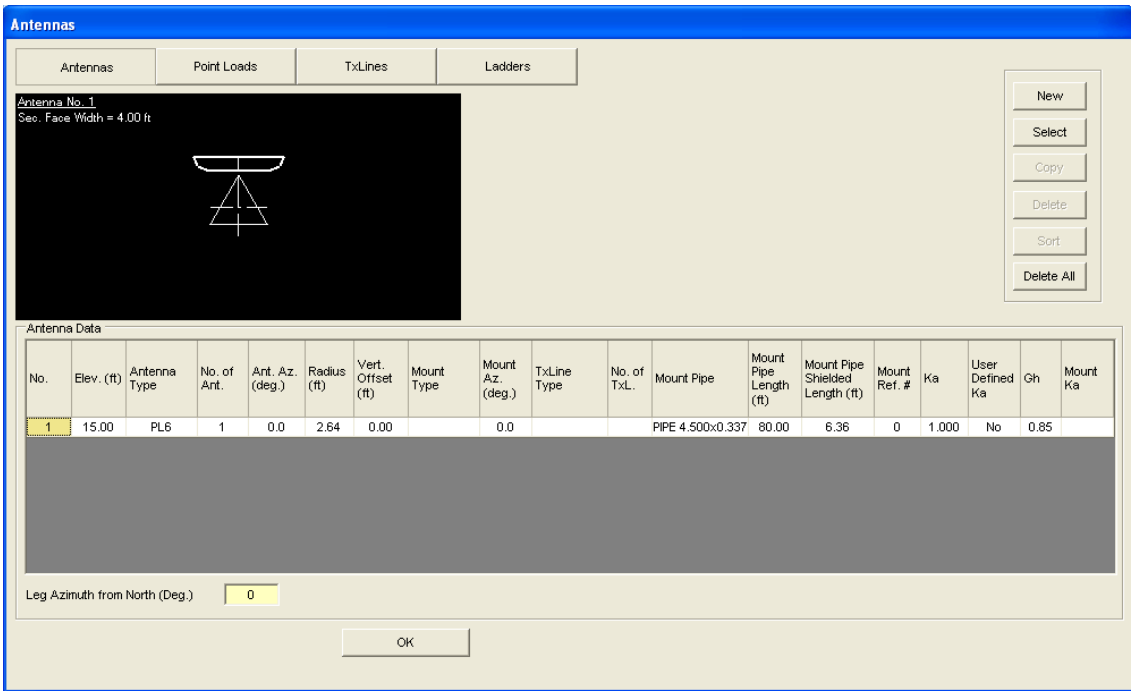


And select the restraints conditions:



ANTENNAS DEFINITION

From the main menu choose **Appurtenances** then sub-menu **General Appurtenances** and then select **Antennas,** the following window is displayed.



In this window the tower elevation is shown along with the plan cross-section at the marked antenna. To add a new antenna select **New** and a blank line with an antenna type (none) is shown. The user inputs the following data as defined below:

- ***Elevation:*** Elevation of the center of the antenna marked from the bottom of the pole and shown in meters or ft.
- ***Antenna Type:*** Type of antenna and can be chosen from the antennas database available. To choose an antenna click on this field and an antenna type window will be displayed and the required type and size is specified.
- ***No of Ant.:*** Number of Antennas. For M/W antennas the number cannot be more than one.
- ***Ant. Az. (deg.):*** Antenna beam azimuth measured from the zero azimuth of the pole and may be referred to as pole's north (specified in degrees).
- ***Radius:*** Radius is measured from the pole center to the mounting point of the antenna (m or ft.). Also, note that the pole radius at that elevation is shown for guidance on the section drawing.

- ***Vert. Offset:*** Vertical distance from antenna mount point to attachment point on tower (m or ft.). This distance may be positive (meaning that the attachment point is above the antenna) or negative (meaning that the attachment point is below the antenna). It should be applied only for cases of stand-off mount supporting the antenna and having only one point of attachment to tower (cantilever mount).
- ***Mount Type:*** Type of antenna mount and can be chosen from the database available. To choose a mount click on this field to select from the database.
- ***Mount Az. (deg.):*** Angle between the tower's north and the antenna's mount point measured in the clockwise direction.
- ***TxLine Type:*** Type of transmission lines associated with that antenna and can be chosen from a database available.
- ***No. of TxL:*** Quantity of TX lines associated with that antenna.
- ***Mount Pipe:*** Size of mounting pipe selected from database.
- ***Mount Pipe Length:*** Full length of the mounting pipe (m or ft.)
- ***Mount Pipe Shielded Length:*** Length of the mounting pipe shielded by antenna (m or ft.)
- ***Mount Ref. #:*** Reference number (antenna number) of the mount for an antenna, where mount type was defined.
- ***Ka:*** Shielding factor Ka (default Ka=1) can be overwritten by User. (Does not apply to EIA-222-F standard)
- ***User Defined Ka (No or Yes):*** Indicates if Ka was defined by User (Yes) or left as default (No). If the indicator is "Yes" then by clicking on this cell the Ka will be changed back to default and the indictor will be back to "No". (Does not apply to EIA-222-F standard)
- ***Gh:*** Gust factor for the antenna. Default is equal to gust factor for the tower but can be overwritten by User. (Does not apply to EIA-222-F standard)
- ***Mount Ka:*** Shielding factor for mount, if applicable. Default Ka = 1. (Does not apply to EIA-222-F standard)
- ***Leg Azimuth from North (Deg.):*** Angle between true geographical north and tower north measured clockwise. The tower north is the line going through tower centre and tower apex (for triangular towers) or through tower centre perpendicular to face (for square towers). This azimuth allows Users to specify antenna azimuths and mount azimuths referring to true north.

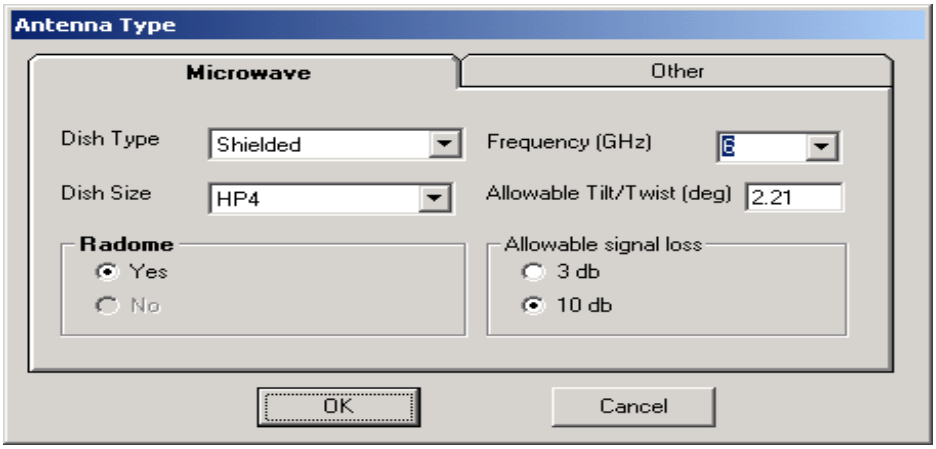
Following functions are available:

- Select: Marks an antenna for copying or deleting

- Copy: Allows to copy selected antenna
- Delete: Allows to delete selected antenna
- Sort: Sorts antennas by elevation
- Delete All: Allows to delete all antennas

Antenna Type:

Click on type in the antenna table and the following window is displayed and from which the antenna type and size is specified.



For microwave dishes available in the database the following dish types and sizes are available:

Shielded	(2, 4, 6, 8, 10, 12, 15 ft.)
Focal plane	(4, 6, 8, 10, 12 ft. – with or without radome)
Standard	(2, 4, 6, 8, 10, 12 ft. – with or without radome)
Grid	(4, 6, 8, 10, 12, 15 ft.)
GRIDPAK	(4, 6, 8, 10, 12, 13 ft.)

Also, for microwave antennas the allowable Tilt/Twist is calculated by the program as a function of frequency, dish diameter and allowable signal degradation (3 db or 10 db) based on the following formula:

a) For a parabolic reflector with an allowable 10dB signal degradation:

$$\theta = \frac{C_{10}}{D \quad \alpha}$$

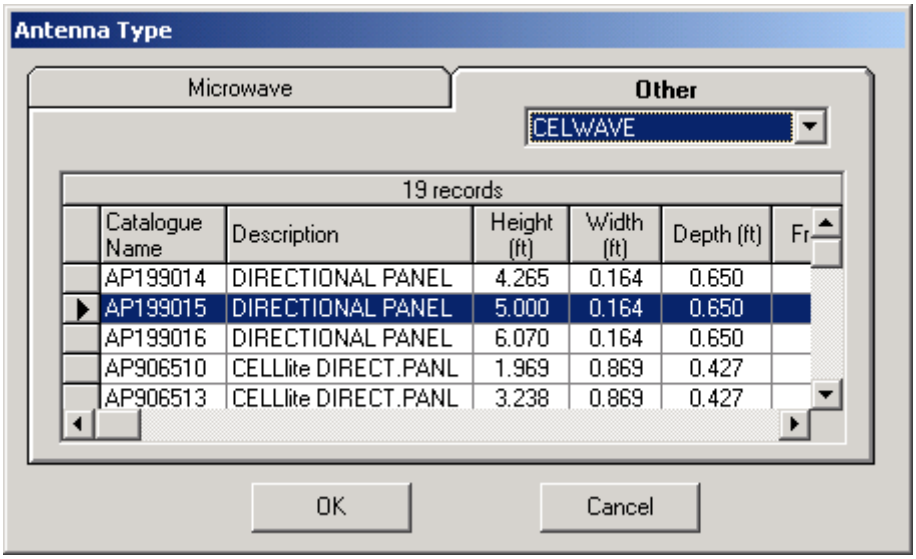
b) For a parabolic reflector with an allowable 3dB signal degradation:

$$\theta = \frac{C_3}{D \quad \alpha}$$

where:

θ = twist or sway limit, degrees
C_{10} = 53.1 GHz.ft.deg [16.2 GHz.m.deg]
C_3 = 31.0 GHz.ft.deg [9.45 GHz.m.deg]
D = Diameter of dish, ft [m]
α = Dish Frequency, GHz.

For "Other" (Non-Dish Antennas) a variety of Antennas, organized by manufacturers is available from the database.
Following screen is displayed when the Tab "Other" is clicked.



The data shown includes all parameters of these antennas necessary for the program to calculate wind loads and weight effects.

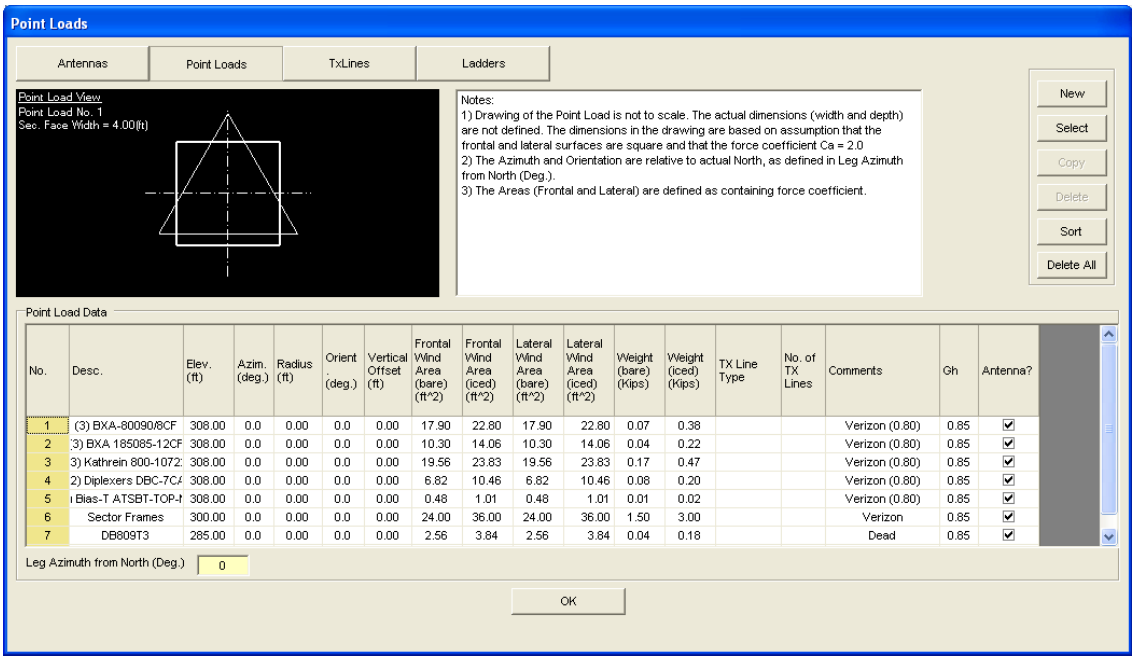
The User may add any type of Non-Dish Antenna to the database –using the "Database Management" option from the main menu. Refer to Database Management Chapter for details.

After specifying an antenna, the user may wish to copy it by highlighting the specific line and click on **select** and then on **Copy**. A new line of antenna will be displayed and the user may edit that line. A similar procedure is used to delete an antenna.

Loads are calculated for microwave dishes as per Andrew's catalogue number 36. Also load calculations for wind loads under different directions are based on ANSI/TIA-222-G tables (ANNEX C: DESIGN WIND FORCE ON TYPICAL ANTENNAS (Normative).

POINT LOADS DEFINITION

From the main menu choose **Appurtenance** and then sub-menu **General Appurtenances** and then select **Point Loads,** the following window is displayed.



In this window the mast elevation is shown along with the plan cross-section at the point load elevation. To add a new point load select **New** and input line with zero values is shown. The user inputs the following data as defined below:

- ***Description:*** Text description of the point load. This description will be displayed on the design profile. If the default description is not over written no description is displayed on the profile.
- ***Elevation:*** Elevation from the bottom of the tower to the center of the applied load (m or ft.)
- ***Azimuth:*** angle between the north and the point load radius measured in the clockwise direction (specified in degrees).
- ***Radius:*** Radius is measured from the tower center to the point load (m or ft.). Also, note that the tower radius at that elevation is shown for guidance on the section drawing.
- ***Orient.:*** Angle between the tower's north and the point load mount point measured in the clockwise direction.

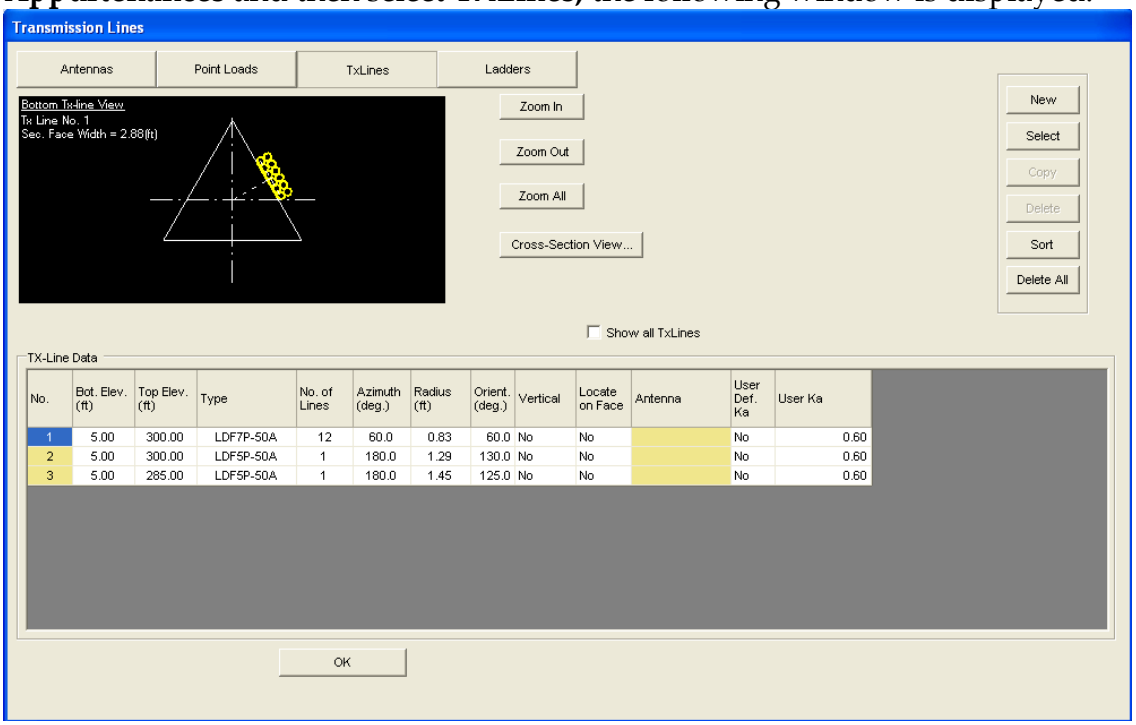
- ***Vertical Offset:*** Vertical distance from antenna mount point to attachment point on tower (m or ft.). This distance may be positive (meaning that the attachment point is above the antenna) or negative (meaning that the attachment point is below the antenna). It should be applied only for cases of stand-off mount supporting the antenna and having only one point of attachment to tower (cantilever mount).
- ***Frontal Wind Area (Bare):*** Bare wind area perpendicular to the point load azimuth of the point load multiplied by the appropriate force coefficient or drag factor depending on the shape of the appurtenance (m^2 or ft^2).
- ***Frontal Wind Area (Iced):*** Iced wind area perpendicular to the point load azimuth of the point load multiplied by the appropriate force coefficient or drag factor depending on the shape of the appurtenance and the ice accretion (m^2 or ft^2).
- ***Lateral Wind Area (Bare):*** Bare wind area parallel to the point load azimuth of the point load multiplied by the appropriate force coefficient or drag factor depending on the shape of the appurtenance (m^2 or ft^2).
- ***Lateral Wind Area (Iced):*** Iced wind area parallel to the point load azimuth of the point load multiplied by the appropriate force coefficient or drag factor depending on the shape of the appurtenance and the ice accretion (m^2 or ft^2).
- ***Weight (Bare):*** Bare weight of the load (kN or kips)
- ***Weight (Iced):*** Weight of the load including ice (kN or kips)
- ***TX Line Type:*** Type of transmission lines associated with that point load and can be chosen from a database available.
- ***No. of TX Lines:*** Quantity of TX lines associated with that point load.
- ***Comments:*** A comments field that does not get displayed on the profile.
- ***Gh:*** Gust factor for the point load. Default is equal to gust factor for the tower but can be overwritten by User.
- ***Antenna?:*** Tick box to indicate if the point load is an antenna.
- ***Leg Azimuth from North (Deg.):*** Angle between true geographical north and tower north measured clockwise. The tower north is the line going through tower centre and tower apex (for triangular towers) or through tower centre perpendicular to face (for square towers). This azimuth allows Users to specify point loads azimuths and orientation referring to true north.

Following functions are available:

- Select: Marks point load for copying or deleting
- Copy: Allows to copy selected point load
- Delete: Allows to delete selected point load
- Sort: Sorts point loads by elevation
- Delete All: Allows to delete all point loads

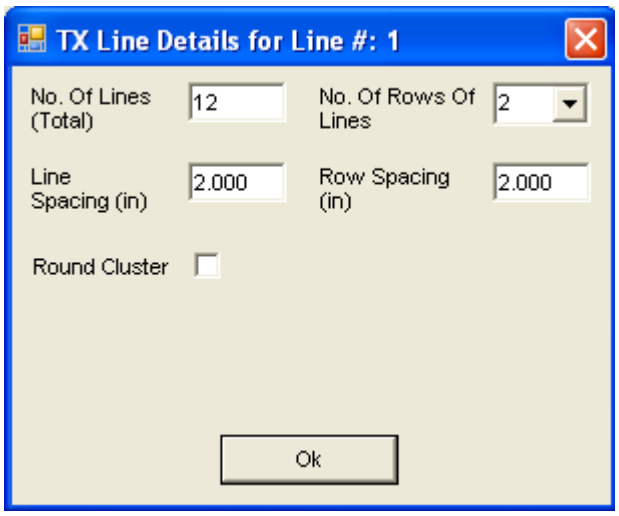
TRANSMISSION LINES DEFINITION

From the main menu choose **Appurtenance** and then sub-menu **General Appurtenances** and then select **TXLines,** the following window is displayed.



In this window the tower elevation is shown along with the tower cross-section at the bottom of the Tx-line. To add a new line or group of lines select **New** and a blank line with a line type (none) is shown. The user inputs the following data as defined below:

- ***Bot. Elev.:*** Elevation of the bottom of the lines (m or ft.)
- ***Top Elev.:*** Elevation of the top of the lines (m or ft.)
- ***Type:*** Type of lines and can be chosen from the tx-lines database available. To choose line type click on this field and the required type and size can be specified as explained in the sequel.
- ***No of lines:*** Number of lines having the same properties shown on that line. If this box is clicked another form will be shown (see below). User will enter total number of lines, number of rows of lines, line spacing and row spacing. If Round Cluster check box is ticked then this group of lines will be considered as round cluster and User should enter the round cluster diameter.



- ***Azimuth:*** Group of lines azimuth (specified in degrees).
- ***Radius:*** Radius is measured from the pole center to the center of the lines group (m or ft.).
- ***Orient.:*** Angle between the pole's north and the lines group radius measured in the clockwise direction.
- ***Vertical:*** User may select "Yes", which will force the lines to run vertically from bottom of line to top of line. If "No" is selected the line(s) will follow the slope of the tower.
- ***Locate on Face:*** Following selections are available: "No", "Yes-Out" and "Yes-In". This feature helps in positioning the line on face of the tower (either on outer side of the face or on inner side of the face). Note: For design/analysis using CSA S37-01 or EIA-222-G the program presents following choices: "No", "Yes-Out", "Yes-In" and "Yes-NS". If "Yes" is selected the line(s) will be considered as part of face and their respective EPAs and weights will be added to structure face. "Yes-NS" means no shielding of structure by lines or lines by structure is applied. These options are available for both bare and iced cases (User may choose the line to be part of face for bare cases only, for bare and iced cases or for iced cases only.
- ***Antenna:*** Antenna description for cases where the TX lines were defined for a specific antenna or point load.
- ***User Def. Ka:*** An indicator ("Yes" or "No") showing if User defined Ka (shielding factor) was applied..
- ***User Ka:*** Users may overwrite the default Ka (shielding factor). If this is done the indicator above is changed to "Yes".

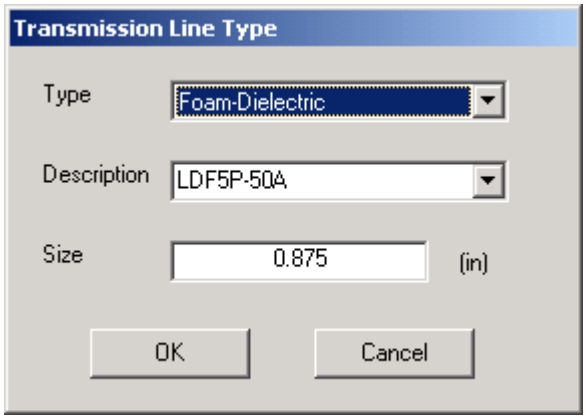
Following functions are available:

- Zoom In: Allows to zoom in the sketch of lines

- Zoom Out: Allows to zoom out the sketch of the lines
- Zoom All:
- Cross-Section View: Opens another form with enlarged cross-sectional view at selected elevation. This form is printable.
- Show all TX Lines*:* Tick box allowing to show only selected line or all lines.
- Select: Marks a line for copying or deleting
- Copy: Allows to copy selected line
- Delete: Allows to delete selected line
- Sort: Sorts lines by elevation
- Delete All: Allows to delete all lines

Transmission Line Type:

Click on type in the tx-lines table and the following window is displayed and from which the tx-lines type and size is specified.



The following Transmission Line types are available:

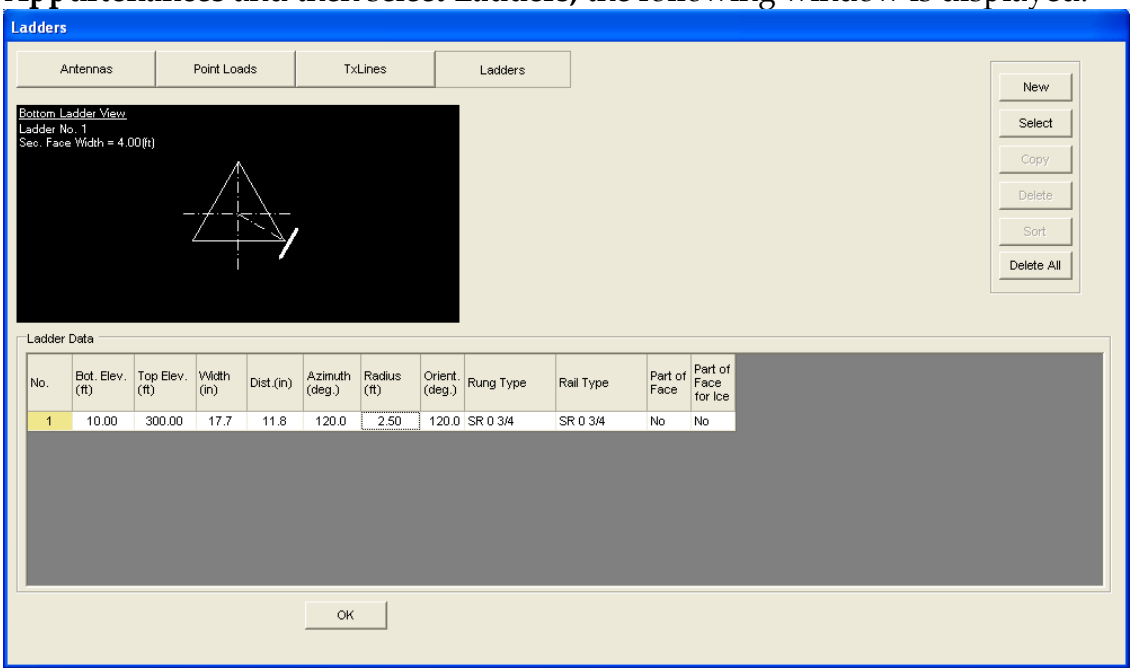
Air-Dielectric
Foam-Dielectric
Elliptical Waveguide
Circular Waveguide
Rectangular Waveguide

Different sizes can be chosen from the description field and the actual size for the chosen lines is displayed in the size field (mm or in.).

Users can create their own Transmission Line types through "Database Management"/"Antennas DB".

LADDER DEFINITION

From the main menu choose **Appurtenances** and then sub-menu **General Appurtenances** and then select **Ladders,** the following window is displayed.



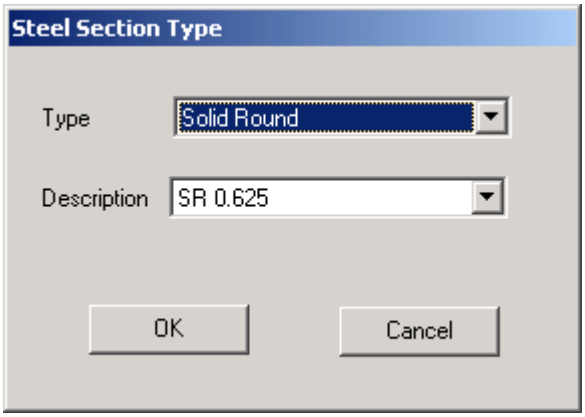
In this window the pole elevation is shown along with the plan cross-section at the bottom of the ladder. To add a new ladder select **New** and a blank line with a rail type, and rung type (none) is shown. The user inputs the following data as defined below:

- ***Bottom Elevation:*** Elevation of the bottom of the ladder (m or ft.)
- ***Top Elevation:*** Elevation of the top of the ladder (m or ft.), please note that the length of ladder, located above the tower top, will be ignored by the program for loading
- ***Width:*** Width of ladder rungs (mm or in.)
- ***Dist.:*** Distance (spacing) between rungs (mm or in.)
- ***Azimuth:*** ladder azimuth (specified in degrees).
- ***Radius:*** Radius is measured from the pole center to the center of the ladder (m or ft.).
- ***Orient.:*** angle between north and the ladder's center measured in the clockwise direction.
- ***Rung Type:*** Rung size chosen from available steel sections database for angles and solid rounds.
- ***Rail Type:*** Rail size chosen from available steel sections database for angles and solid rounds.

- ***Part of Face:*** Default is "No" but for cases of all-welded tower, where the ladder components are welded and from part of the face the ladder may be considered as part of face.
- ***Part of Face for Ice:*** Default is "No" but for cases of all-welded tower, where the ladder components are welded and from part of the face the ladder may be considered as part of face.

Steel Section Type:

Click on rail type or rung type in the ladder data table and the following window is displayed, from which the steel section type and size is specified.



The following steel sections for ladders are available:

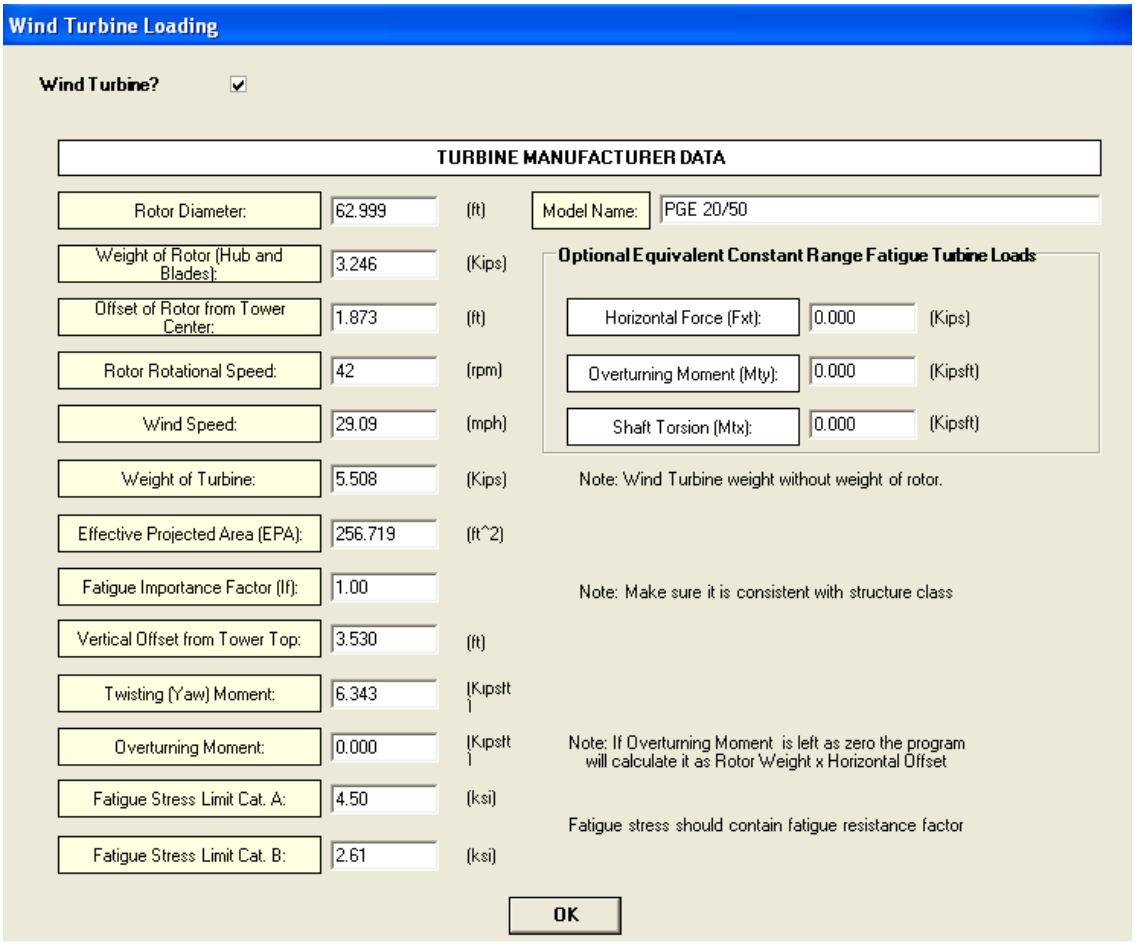
Solid Round (SR13, SR16, SR19, SR25, SR50)
(SR.5, SR.625, SR.75, SR1.0, SR2.0)

Angle (L51x51x6, L51x51x13)
(2x2x1/4, l2x2x1/2)

WIND TURBINE DEFINITION

The loading for wind turbine definitions are based on ANSI/TIA-222-G-DS1 "Design Supplement for Small Wind Turbine Support Structures". Small Wind Turbines (SWT's) defined as wind turbines with rotor swept areas less than 2,200 sq. ft. [200 sq. m].

From the main menu choose **Appurtenances** then sub-menu **Wind Turbine,** the following window is displayed.



The user inputs the following data as defined below:

- ***Wind Turbine?*** Tick-box indicating if the Wind Turbine should be considered in analysis.
- ***Rotor Diameter:*** Rotor diameter (m or ft.)
- ***Weight of Rotor (Hub and Blades):*** Combined weight of hub and blades (kN or kips)
- ***Offset of Rotor from Tower Center:*** Horizontal offset of turbine weight from vertical centerline of turbine base (m or ft.)
- ***Rotor Rotational Speed:*** Rotational rotor speed at AWEA electrical power rating of turbine (RPM)
- ***Wind Speed:*** Wind speed at hub height associated with the specified maximum turbine horizontal thrust (m/s or mph)
- ***Weight of Turbine:*** Weight of turbine (kN or kips)
- ***Effective Projected Area (EPA):*** The effective projected area (m^2 or ft^2) of a turbine shall be calculated in accordance with

ANSI/TIA-222-G-DS1 unless the effective projected area is specified by the turbine manufacturer. The effective projected area of a turbine shall be considered to be constant for all wind directions with a wake interference factor, K_a, equal to 1.0.

- ***Fatigue Importance Factor:*** Importance factor for fatigue from Table 11-1 of ANSI/TIA-222-G-DS1
- ***Vertical Offset from Tower Top:*** Hub height above turbine base connection to supporting structure (m or ft.)
- ***Twisting (Yaw) Moment:*** A specified twisting (yaw) moment considered to act about the vertical centerline of the turbine base in a counterclockwise direction in the plan view (kNm or kipft)
- ***Overturning Moment:*** A specified overturning moment shall be considered to occur in the same direction as the wind. Rotor Weight times Horizontal Offset (kNm or kipft). If this value is left as zero the program will calculate it automatically.
- ***Fatigue Stress Limit Cat. A:*** Equivalent fatigue damage stress ranges for Category A components and shall not exceed the design stress ranges specified in Sections 11.4.1 of ANSI/TIA-222-G-DS1 (MPa or ksi)
- ***Fatigue Stress Limit Cat. B:*** Equivalent fatigue damage stress ranges for Category B components and shall not exceed the design stress ranges specified in Sections 11.4.2 of ANSI/TIA-222-G-DS1 (MPa or ksi)
- ***Model Name:*** Name of the Wind Turbine Model
- ***Horizontal Force (Fxt):*** Equivalent constant range turbine horizontal force, (kN or kips). This entry is optional and if left as zero the program will calculate it.
- ***Overturning Moment (Mty):*** Equivalent constant range turbine overturning moment, (kNm or kipft). This entry is optional and if left as zero the program will calculate it.
- ***Shaft Torsion (Mtx):*** Equivalent constant range turbine rotor shaft torsion (kNm or kipft). This entry is optional and if left as zero the program will calculate it.

Chapter 3 ANALYSIS

This Chapter explains the input of the code-related data and material data. Also, the assumptions and the underlying theory of the analysis are explained in the sequel.

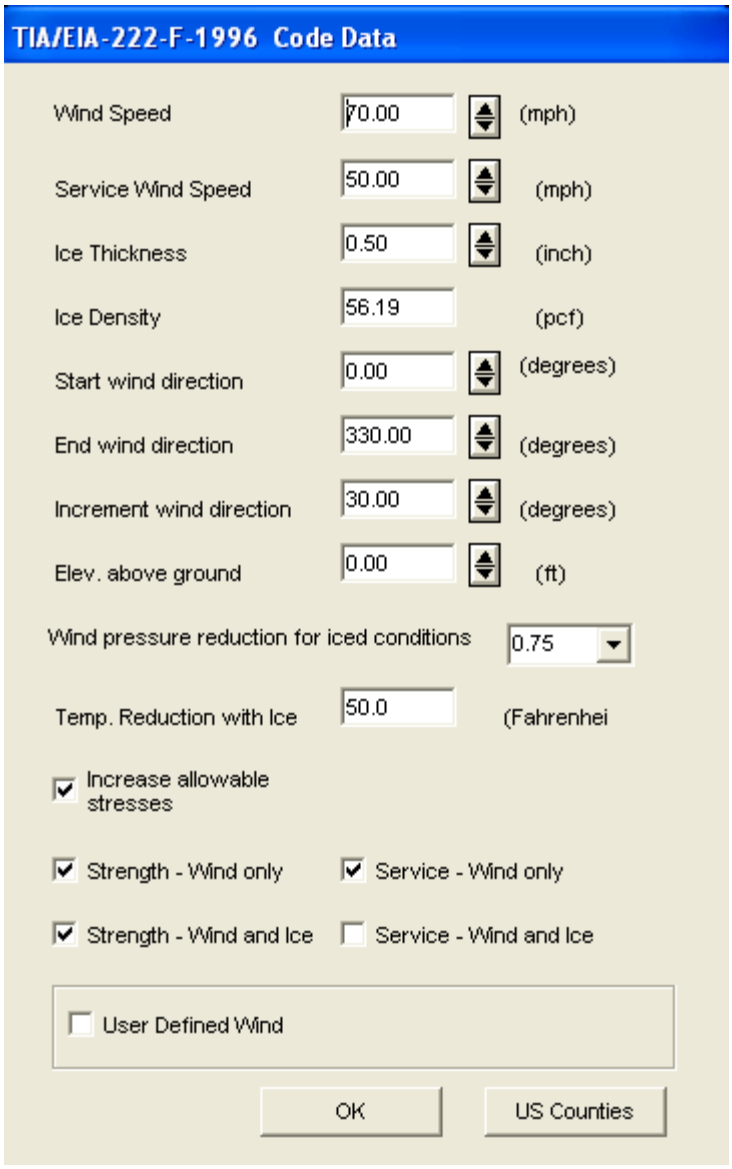
CODE DATA

Wind Loads

From the main menu choose **Load Cases,** and then he following window will appear depending on the design standard specified:

a) TIA/EIA 222-F

Having the EIA 222-F as the design code, the wind Loads window will be as shown:



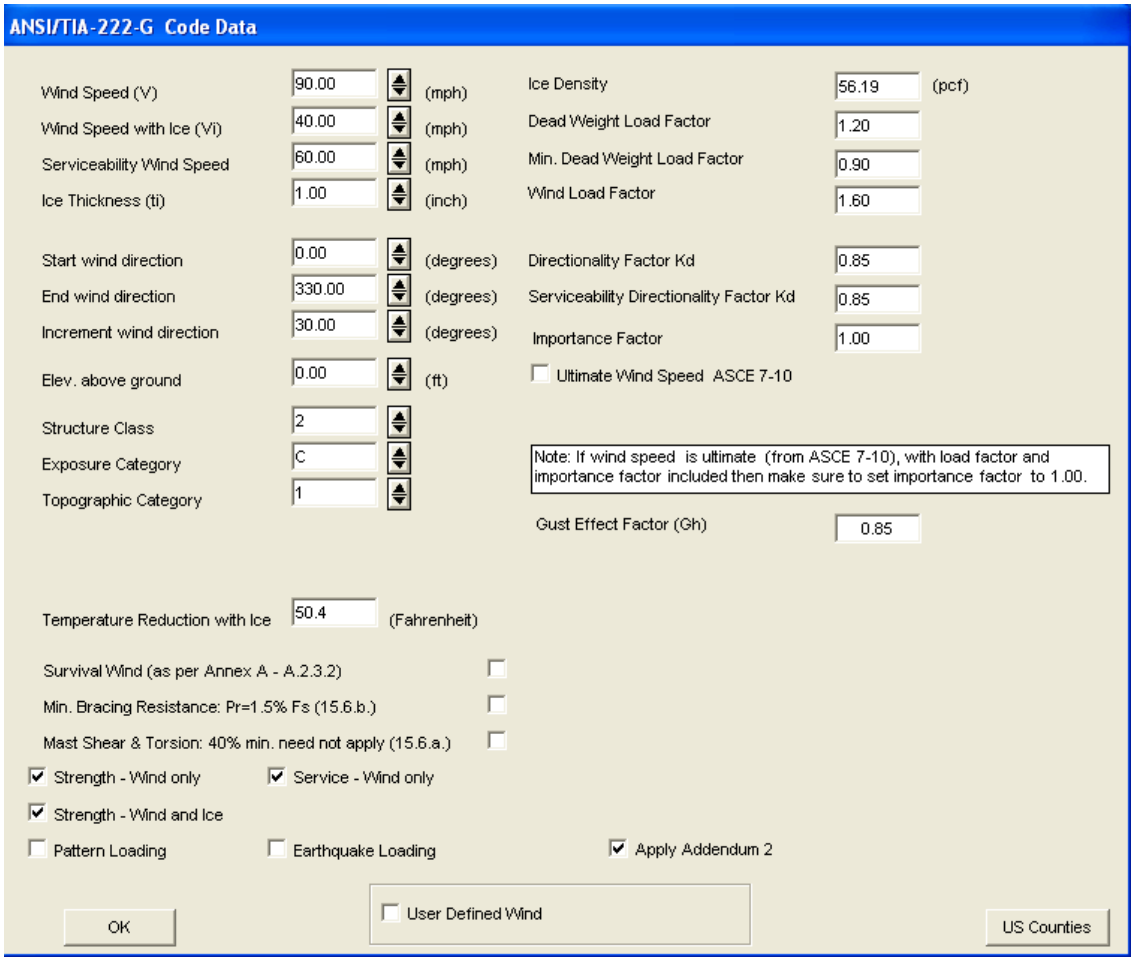
The input data is explained as follows:

- Wind Speed: Design wind speed for the structure (m/sec or mph).
- Service Wind Speed: for the calculation of deflections (tilt/twist) under service load conditions (m/sec or mph).
- Ice Thickness: specified ice thickness for the design of the structure depending on the location (mm or inches).
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered

- Increment Wind direction: Wind increment in degrees at which wind loads are calculated. For example, in the above shown example wind will be considered at the following directions (0, 30, 60, 330)
- Elevation above ground: The elevation of the structure base above ground this is used for wind loads calculations. The height factor is increased accordingly.
- Allowable stress factor: For the structural assessment of the tower, the allowable stress is increased to 1.33 as per EIA 222-F when applicable. However, the user has the option to limit the allowable stress to 1.0
- Wind pressure reduction for the case of iced conditions (default is 0.75 as per code).
- Options to mark the analysis loading cases for wind only or for combined case of wind and ice under strength conditions and service conditions.

b) ANSI/TIA-222-G-2005

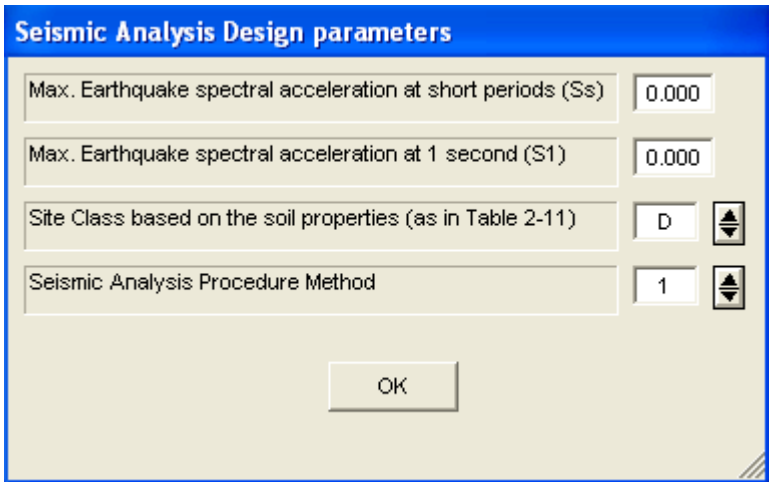
Having the TIA–222-G as the design code, the Wind Loads window will be displayed as shown:



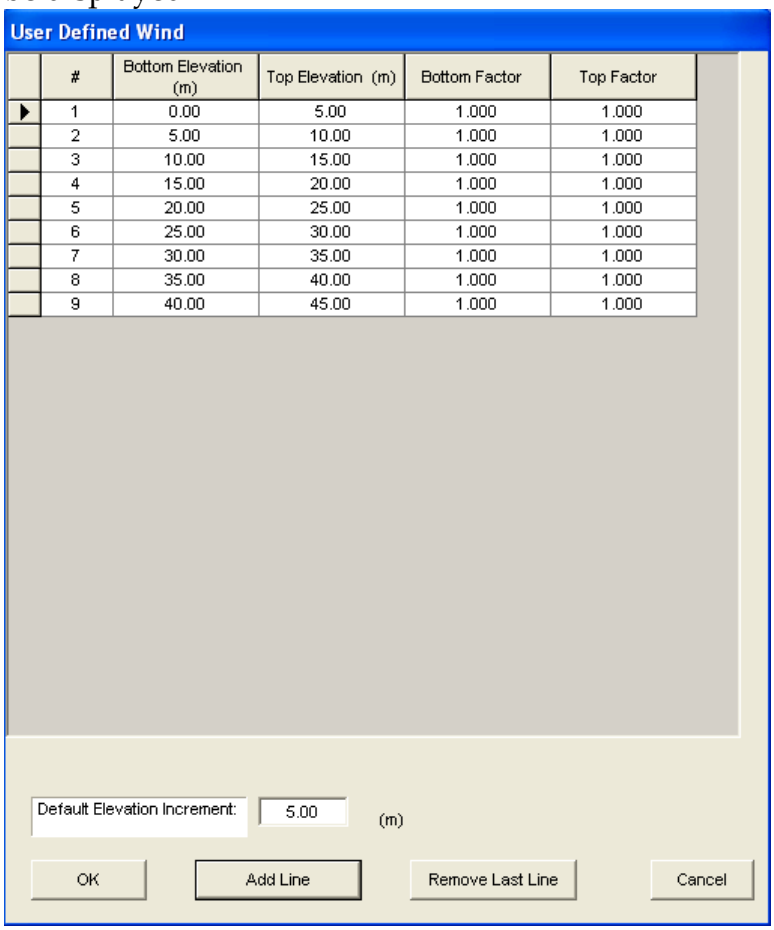
The input data is explained as follows:

- Basic Wind Speed (V): 3-second gust speed for the structure (m/sec or mph).
- Wind Speed with Ice (Vi): 3-second gust speed concurrent with the design ice at 33 ft above the ground (m/sec or mph).
- Serviceability Wind Speed: for the calculation of deflections (tilt/twist) under service load conditions (m/sec or mph).
- Ice Thickness (ti): specified ice thickness for the design of the structure depending on the location (mm or inches).
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered
- Increment Wind direction: Wind increment in degrees at which wind loads are calculated. For example, in the above shown example wind will be considered at the following directions (0, 30, 60, 90, 120, 150, 180, 210, 240, 270, 300 and 330)

- Elevation above ground: The elevation of the structure base above ground this is used for wind loads calculations. The height factor is increased accordingly.
- Structure Class as defined in Table 2-1 of ANSI/TIA-222-G. Values are 1 through 3 with class 2 as default.
- Exposure Category as defined in clause 2.6.5.1 of ANSI/TIA-222-G. Options are Exposure B, C or D with Exposure B as default.
- Topographic Category as defined in clause 2.6.6.2 of ANSI/TIA 222-G ranging from Category 1 through 5 with Category 1 as default.
- Ice Density: Density of ice (mm or inches)
- Dead Weight Factor: Default 1.20, can be overwritten by user
- Min. Dead Weight Factor: Default 0.90, can be overwritten by user
- Wind Load Factor: Default 1.60, can be overwritten by user
- Directionality Factor (k_d): Default 0.85, can be overwritten
- Serviceability Directionality Factor (k_d): Default 0.85, can be overwritten
- Importance Factor: Default 1.00, can be overwritten
- Ultimate Wind Speed: Tick box, if selected then the wind speed is understood as Ultimate Wind Speed defined by ASCE 7-10
- Gust Effect Factor (G_h): Default 1.10, can be overwritten
- Option to use survival wind speed as defined in Annex A of the standards. This option assumes that the wind speed used is a survival wind speed with Load factors, gust factors and height factors of 1.0 as per clause A2.3.2.
- Options to mark the analysis loading cases for wind only or for combined case of wind and ice under strength conditions and service conditions.
- Earthquake Loading: Tick box, if selected then following data entries will be showing and relevant data should be entered by User

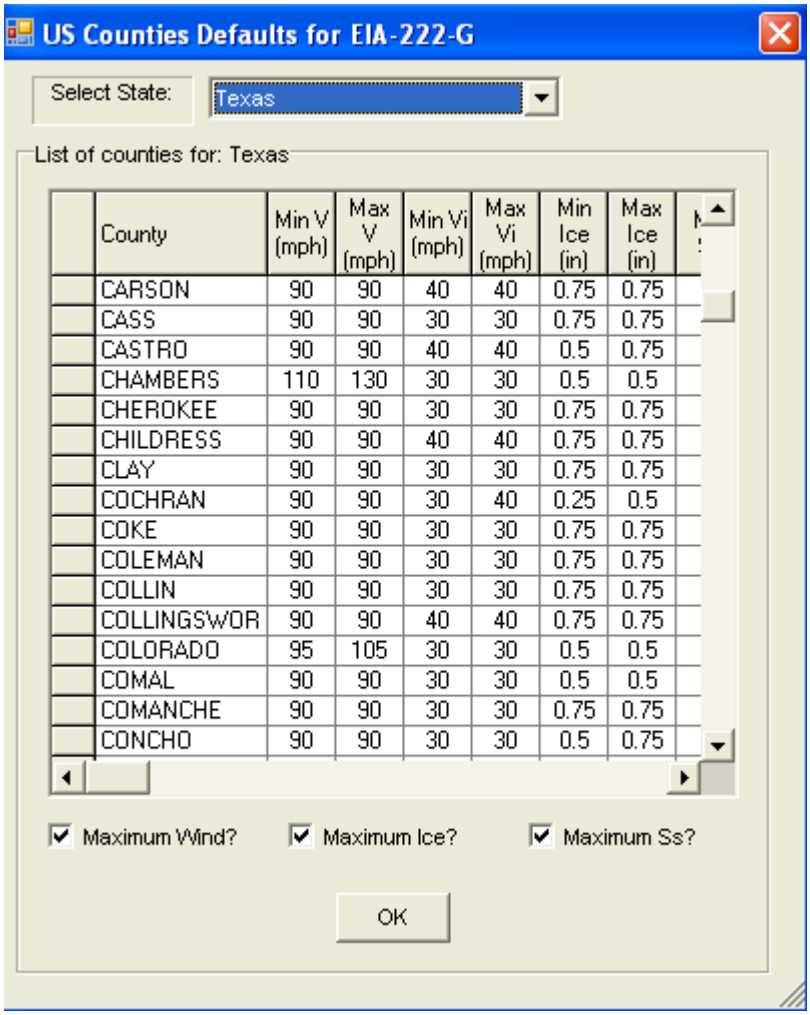


- Apply Addendum 2: Tick box, by default it is selected and in such case the Addendum 2 of the ANSI/TIA-222-G will be applied
- User Defined Wind: If this option is selected following form will be displayed



User should declare elevation increments and the using "Add Line" button create lines to cover entire structure height. Then the bottom height factor and top height factor should be entered. If this option is applied the program will apply height factors at each tower elevation as entered by User.

- US Counties: If this button is clicked following form appears, allowing to select State, County and to automatically apply wind speed, ice thickness and Ss matching the selection.

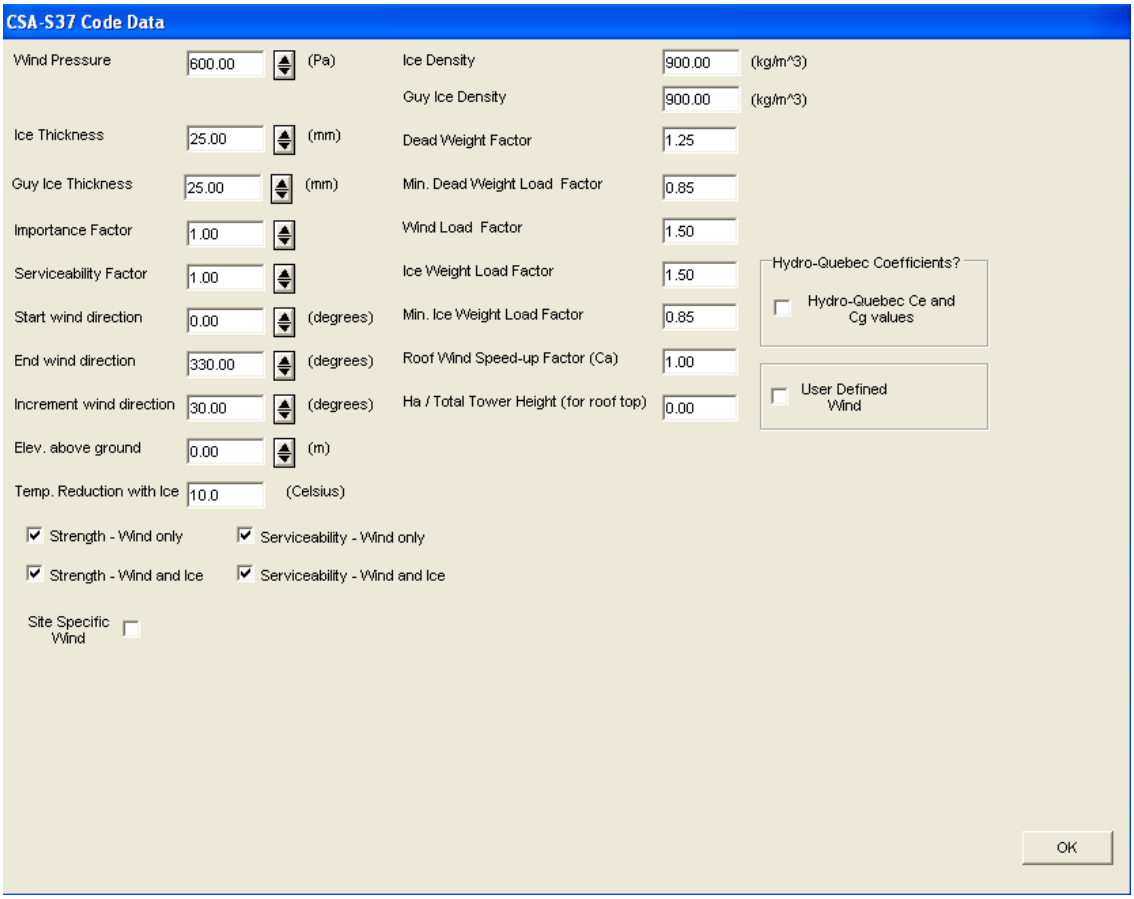


County	Min V (mph)	Max V (mph)	Min Vi (mph)	Max Vi (mph)	Min Ice (in)	Max Ice (in)
CARSON	90	90	40	40	0.75	0.75
CASS	90	90	30	30	0.75	0.75
CASTRO	90	90	40	40	0.5	0.75
CHAMBERS	110	130	30	30	0.5	0.5
CHEROKEE	90	90	30	30	0.75	0.75
CHILDRESS	90	90	40	40	0.75	0.75
CLAY	90	90	30	30	0.75	0.75
COCHRAN	90	90	30	40	0.25	0.5
COKE	90	90	30	30	0.75	0.75
COLEMAN	90	90	30	30	0.75	0.75
COLLIN	90	90	30	30	0.75	0.75
COLLINGSWOR	90	90	40	40	0.75	0.75
COLORADO	95	105	30	30	0.5	0.5
COMAL	90	90	30	30	0.5	0.5
COMANCHE	90	90	30	30	0.75	0.75
CONCHO	90	90	30	30	0.5	0.75

Please note that in addition to wind, wind and ice, and serviceability loading cases the program may optionally run the pattern loading as defined in Figure 3-3. The Earthquake Loading case may be analyzed as well with usage of Seismic Analysis Procedure Method 1 as defined by ANSI/TIA-222-G.

c) CSA S37-94

Having the CSA S37-94 as the design code, the wind Loads window will be displayed as shown:

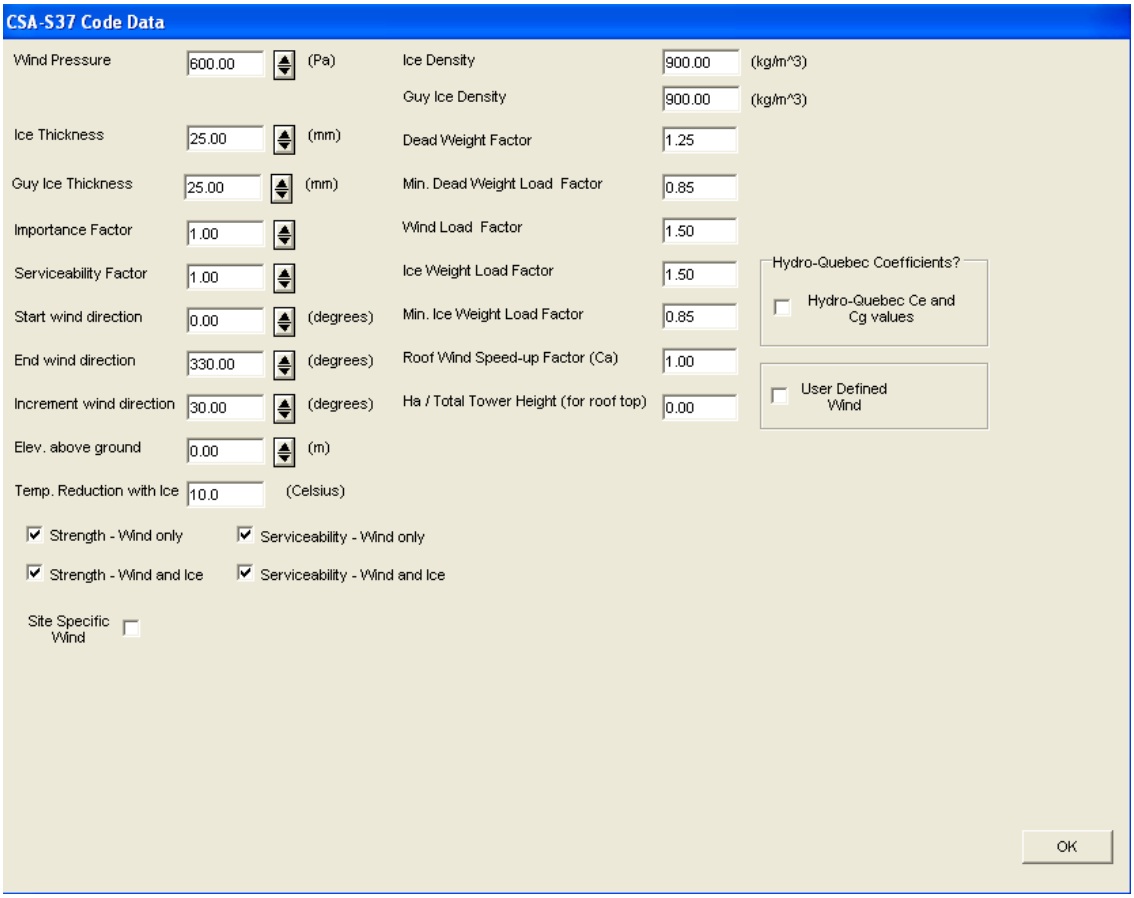


The data for which is explained as follows:

- Wind Pressure: Reference wind pressure (q) as per CSA S37-94
- Ice thickness: Radial ice thickness for the design
- Importance factor: Importance factor as per S37-94
- Serviceability factor: Serviceability factor for service load conditions as per S37-94
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered
- Increment Wind direction: Wind increment in degrees at which wind loads are calculated. For example, in the above shown window, wind will be considered only from 0 degrees.
- Elevation above ground: The elevation of the structure base above ground. This is used for wind loads calculations.
- Options to mark the analysis loading cases for wind only or for combined case of wind and ice under strength conditions and service conditions.
- Option to use site specific wind data

d) CSA S37-01

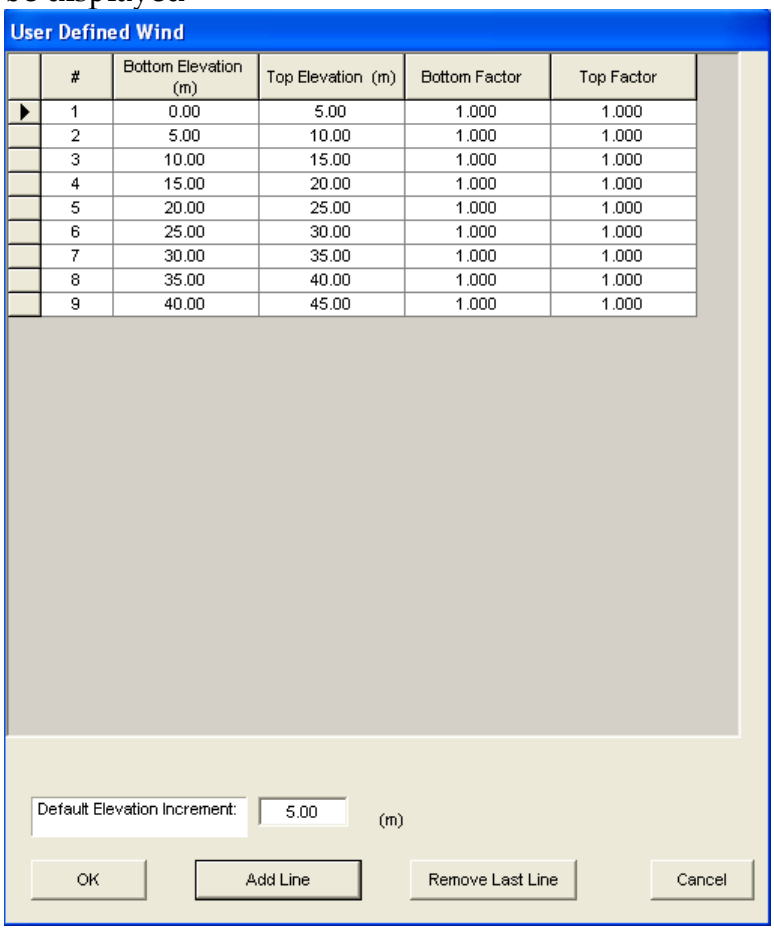
Having the CSA S37-01 as the design code, the wind Loads window will be displayed as shown:



The data for which is explained as follows:

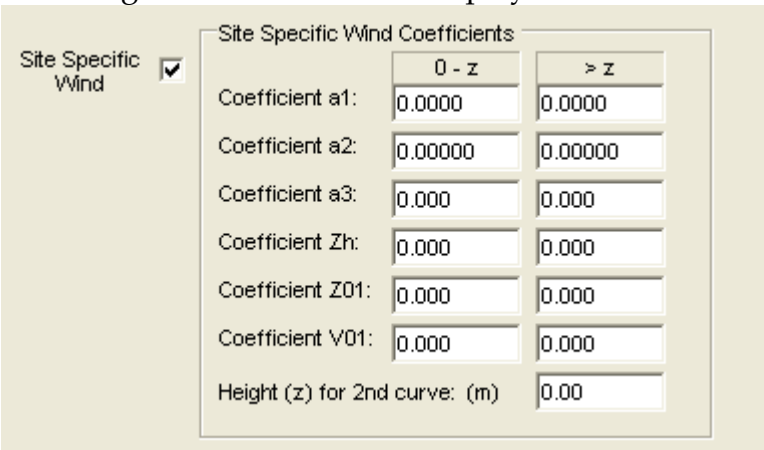
- Wind Pressure: Reference wind pressure (q) as per CSA S37-01
- Ice thickness: Radial ice thickness for the design
- Importance factor: Importance factor as per S37-01
- Serviceability factor: Serviceability factor for service load conditions as per S37-01
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered
- Increment Wind direction: Wind increment in degrees at which wind loads are calculated.
- Elevation above ground: The elevation of the structure base above ground. This is used for wind loads calculations.
- Ice Density: Density of ice (mm or inches)
- Dead Weight Factor: Default 1.25, can be overwritten by user

- Min. Dead Weight Factor: Default 0.85, can be overwritten by user
- Wind Load Factor: Default 1.50, can be overwritten by user
- Ice Weight Load Factor: Default 1.50, can be overwritten by user
- Min. Ice Weight Load Factor: Default 0.85, can be overwritten by user
- Roof Wind Speed-up Factor (C_a): Applicable for roof top structure
- Ha / Total Tower Height (for roof top): Fraction of tower height, for which the C_a as entered will be applied and above which the C_a = 1.0 will be applied
- Hydro-Quebec C_e and C_g values: if this option is selected the height factor and gust factor will follow the specifications of Hydro-Quebec
- User Defined Wind: If this option is selected following form will be displayed



User should declare elevation increments and the using "Add Line" button create lines to cover entire structure height. Then the bottom height factor and top height factor should be entered. If this option is applied the program will apply height factors at each tower elevation as entered by User.

- Site Specific Wind: If Site Specific Wind option is selected following data entries will be displayed:

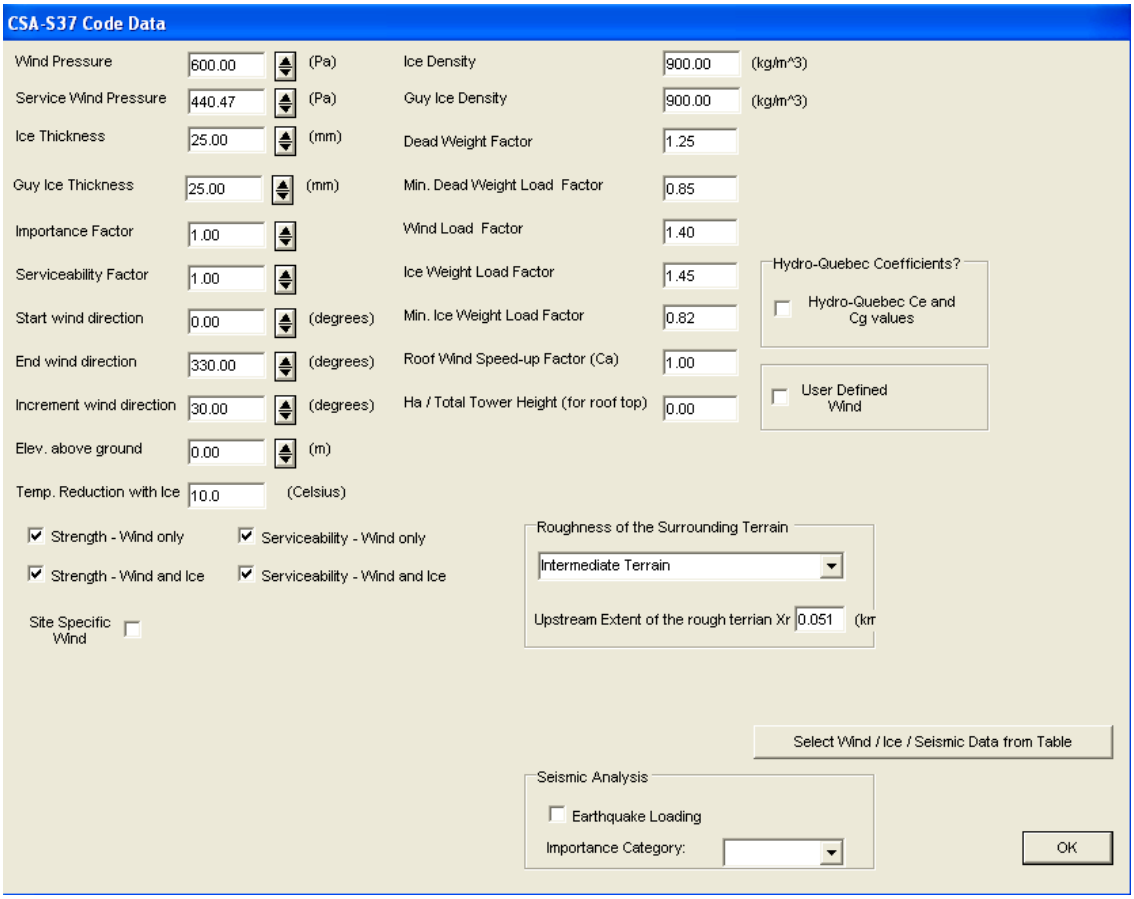


User should enter the site specific wind coefficients as per data supplied by Environment Canada.

Notes: Selections of User Defined Wind, Hydro-Quebec Wind and Site Specific Wind are mutually exclusive. Only one selection will be applied for analysis.

e) CSA S37-13

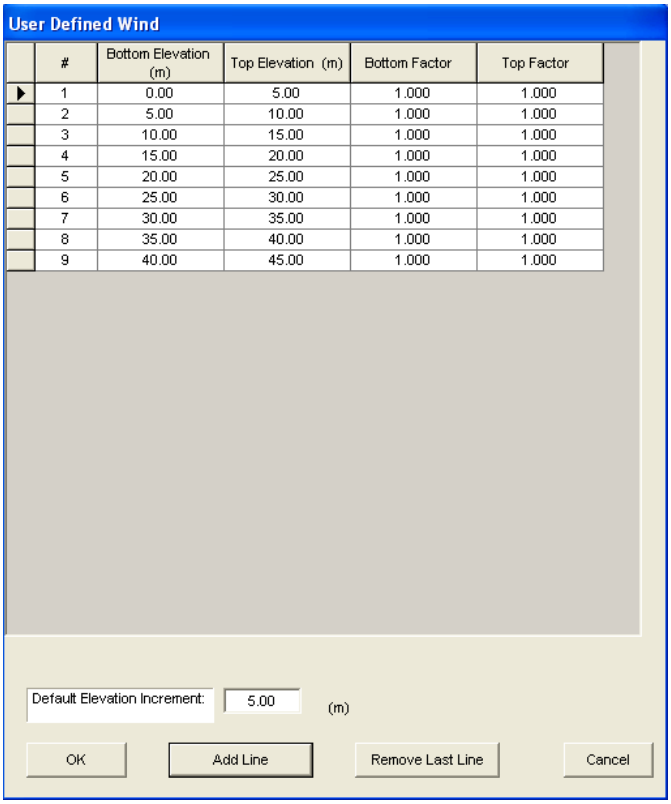
Having the CSA S37-13 as the design code, the wind Loads window will be displayed as shown:



The data for which is explained as follows:

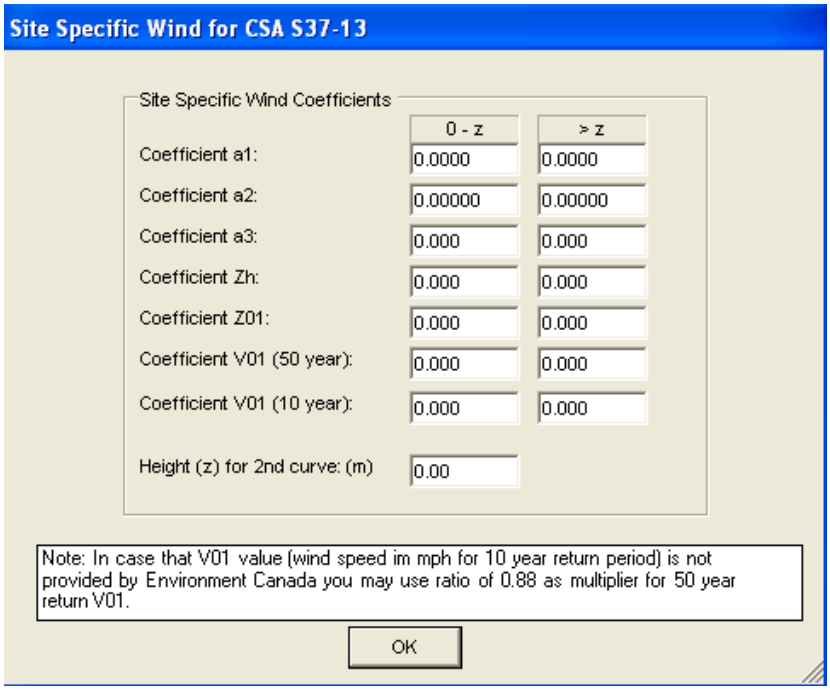
- Wind Pressure: Reference velocity pressure (q) as per CSA S37-13 (50-year return period mean hourly wind pressure at 10 m above ground level), (Pa or psf)
- Service Wind Pressure: Service velocity pressure as per CSA S37-13 (10-year return period mean hourly wind pressure at 10 m above ground level), (Pa or psf)
- Ice thickness: Reference radial ice thickness (t_i) on structure, (mm or inches)
- Guy Ice thickness: Reference radial ice thickness (t_i) on guys, (mm or inches)
- Importance factor: Importance factor as per S37-13
- Serviceability factor: Serviceability factor for service load conditions as per S37-13
- Start Wind Direction: Starting wind directions from which the wind loads are considered.
- End wind directions: Last wind direction for which the wind loads are considered

- Increment Wind direction: Wind increment in degrees at which wind loads are calculated. For example, in the above shown window, wind will be considered only from 0 degrees.
- Elevation above ground: The elevation of the structure base above ground. This is used for wind loads calculations (m or ft).
- Ice Density: Density of ice (mm or inches)
- Dead Weight Factor: Default 1.25, can be overwritten by user
- Min. Dead Weight Factor: Default 0.85, can be overwritten by user
- Wind Load Factor: Default 1.40, can be overwritten by user
- Ice Weight Load Factor: Default 1.45, can be overwritten by user
- Min. Ice Weight Load Factor: Default 0.82, can be overwritten by user
- Roof Wind Speed-up Factor (C_a): Applicable for roof top structure
- Ha / Total Tower Height (for roof top): Fraction of tower height, for which the C_a as entered will be applied and above which the C_a = 1.0 will be applied
- Options to mark the analysis loading cases for wind only or for combined case of wind and ice under strength conditions and service conditions.
- Roughness of the surrounding terrain, following options are available:
  - Open terrain (default)
  - Rough terrain
  - Intermediate terrain (if this option is selected the user should enter the upstream extent of the rough terrain, x_r (km))
- Hydro-Quebec C_e and C_g values: if this option is selected the height factor and gust factor will follow the specifications of Hydro-Quebec
- User Defined Wind: If this option is selected following form will be displayed



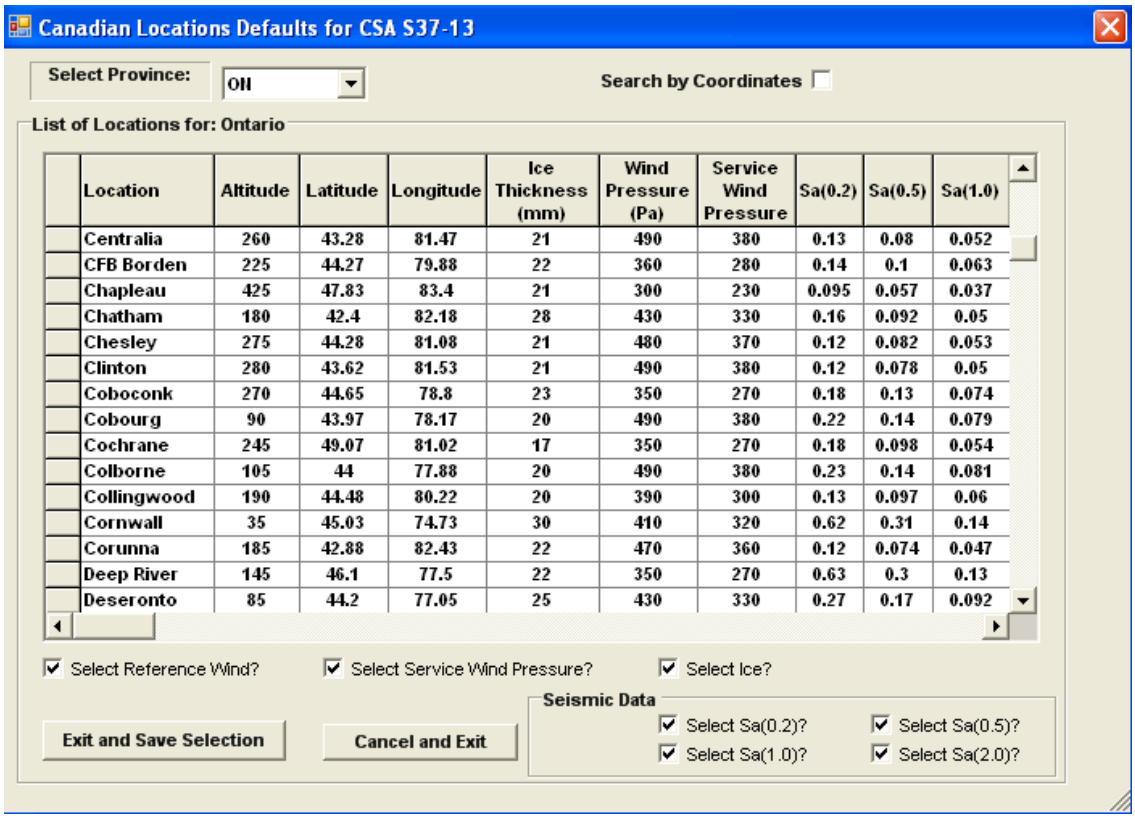
User should declare elevation increments and the using "Add Line" button create lines to cover entire structure height. Then the bottom height factor and top height factor should be entered. If this option is applied the program will apply height factors at each tower elevation as entered by User.

- Site Specific Wind: If Site Specific Wind option is selected following form will be displayed:

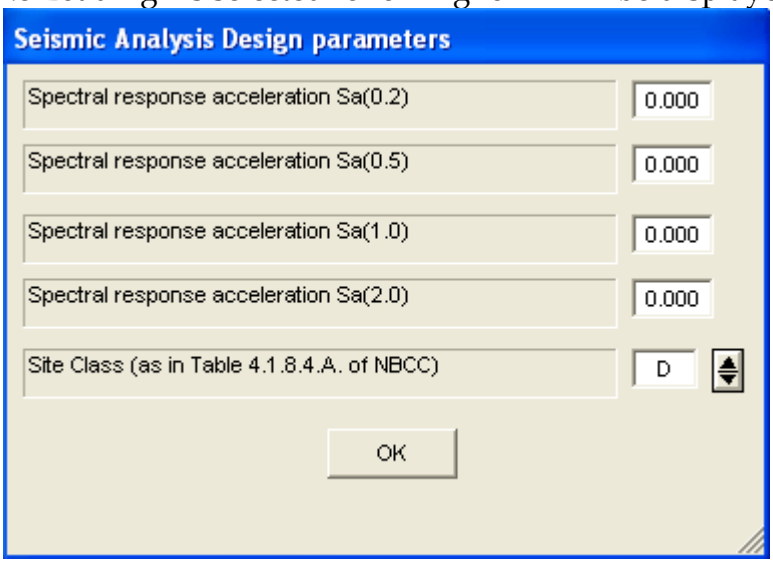


User should enter the site specific wind coefficients as per data supplied by Environment Canada.

Select Wind/Ice/Seismic Data from Table: This option will open following form



Seismic Analysis
If "Earthquake Loading" is selected following form will be displayed:



User should enter Spectral response acceleration factors Sa(0.2) through Sa(2.0) and Site Class.
The Importance Category shall be defined. Options are: Low, Normal, High and Post-Disaster.

Notes: Selections of User Defined Wind, Hydro-Quebec Wind, Site Specific Wind and Wind Data from Table are mutually exclusive. Only one selection will be applied for analysis

ANALYSIS THEORY

The analysis is based on an elastic three-dimensional beam-column where the mast is modeled as equivalent three-dimensional beam-column members supported by cables represented as cable elements. Wind load calculations and analysis are based on the following assumptions:

Wind Loads Calculations

1. Wind is assumed horizontal and is blowing from a certain direction throughout the whole mast height.

2. Force coefficients (drag factors) are calculated as per applicable code.

3. Tx-lines declared as part of the face are considered in the force coefficient calculations of the structure. Tx-Lines as part of the face and outside shield the structural members and vice versa for lines inside the tower (that is not applied for ANSI/TIA-222-G and CSA S37-13 standards).

4. Ice built up is considered uniform on the structure and appurtenances except for ANSI/TIA-222-G and CSA S37-13 standards where the ice effect is changing in function of the height.

5. Wind loads considered on iced structures are reduced as per applicable code.

6. For wind load calculations, wind load is calculated for each section.

7. Loads that are offset from the tower center are applied at the tower center with the corresponding moments (torsional and bending).

8. Loads that extend beyond the height of the structure are applied at the top of the structure with the corresponding additional moments (torsional and bending).

Analysis and Capacities

1. Each member is modeled as two-nodded three-dimensional truss elements with three degrees of freedom at each node.

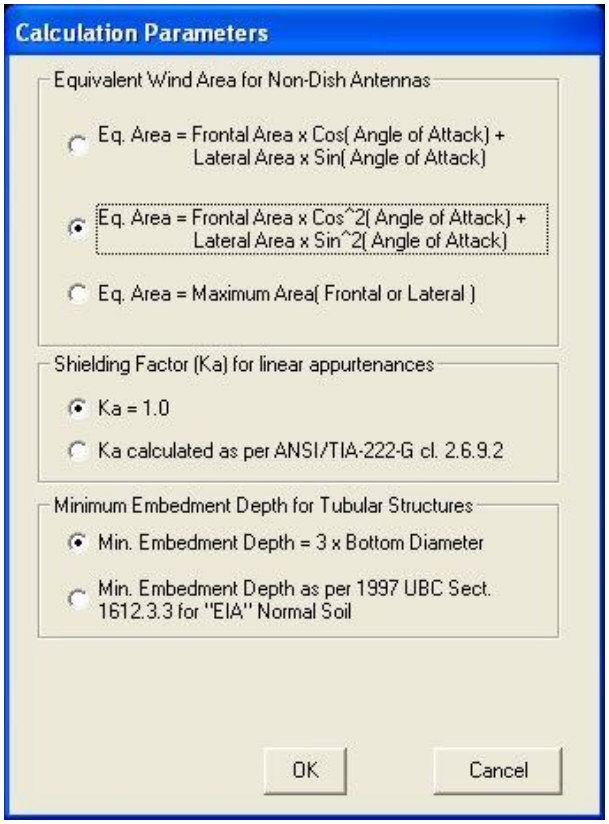
2. Element properties are assumed constant for the full length of the member.

3. Uniform loads applied to the tower are distributed to each level of the section at the three or four leg points.

4. Capacities are calculated based on applicable codes and the structure is assessed for each member.

Calculation Parameters

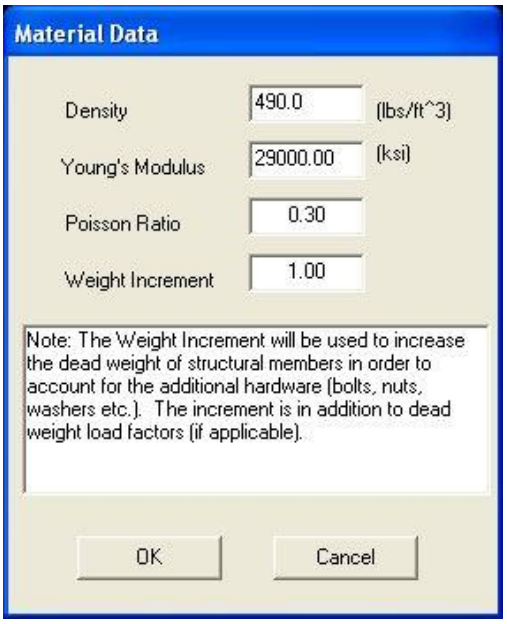
From the main menu, choose "setup" and select "Calculation Parameters". In this menu, the user may change the equivalent wind area calculations for non-dish antennas and setup the shielding factor Ka for linear appurtenances (for CSA-S37 and for EIA-222-F standards).



For non-dish antennas, the user can select the calculation method for the equivalent wind area of non-symmetric antennas.

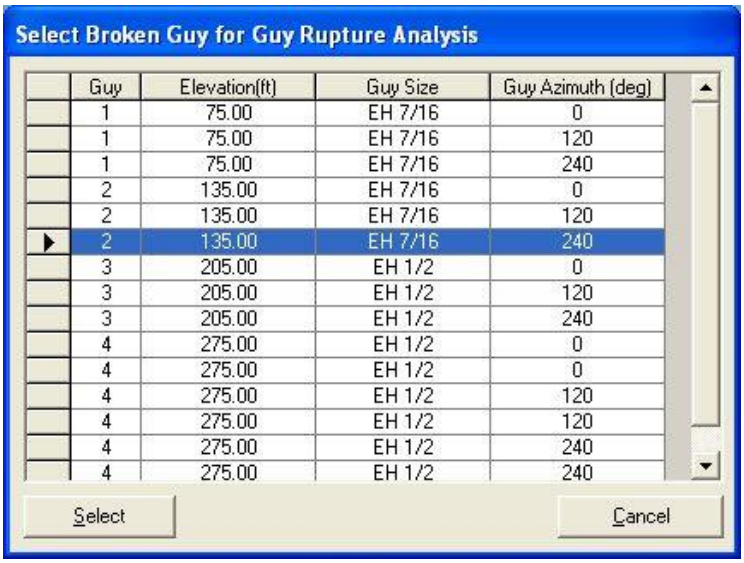
Material Data

Clicking on **Setup** from the main menu the material data can be defined. The user inputs the material density (lbs/ft³ or kg/m³), Young's modulus (ksi or MPa) and Poisson's ratio.



Guy Rupture Analysis

Please note, that for ANSI/TIA-222-G standard the program can perform the guy rupture analysis as per Annex E of the standard.
When this option is selected from the Run menu the program shows following form:



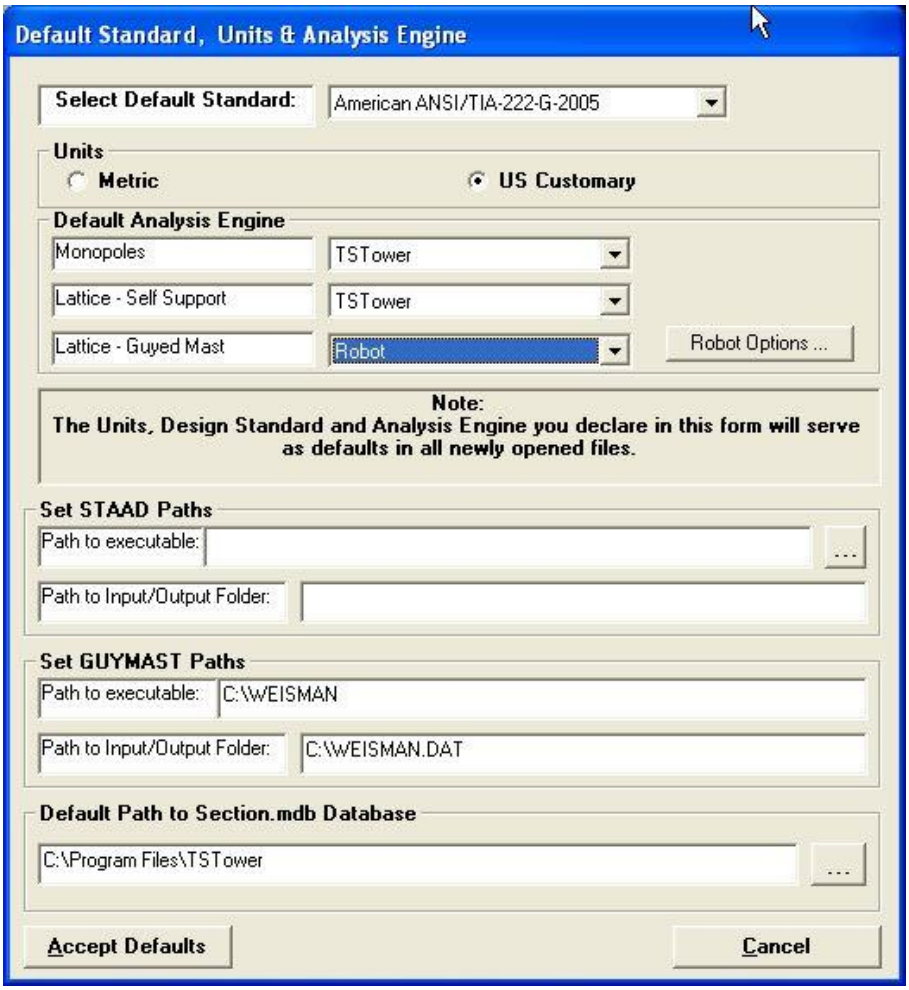
The User may select any guy elevation and any guy number at this elevation to perform this analysis.

Analytical Engine

In addition to using the TSTower finite element analytical engine the users may select two additional programs (the User shall have the license to use these programs in order to utilize these options):

- GUYMAST by Weisman Consultants Inc.
- Robot Millenium by RoboBat

In order to run the analysis with one of these engines the User should change the analytical engine from Menu Setup/ Default Standard, Units & Analysis Engine as shown in the screen shot below:



Analysis Options

Following types of analysis can be performed:

1. Full analysis for ultimate limit states and serviceability limit states (as selected by User) for selected wind directions.
2. Guy rupture analysis (applicable for ANSI/TIA-222-G standard as defined in Annex E of the standard).
3. Modal analysis.
4. Fatigue strength analysis, provided that User selects Wind Turbine loading.

Notes:

- With selection of EIA-222-F the analysis is performed for working loads (not factored).
- In case of applying Wind Turbine loading both full analysis and fatigue analysis should be performed.

Chapter 4 RESULTS

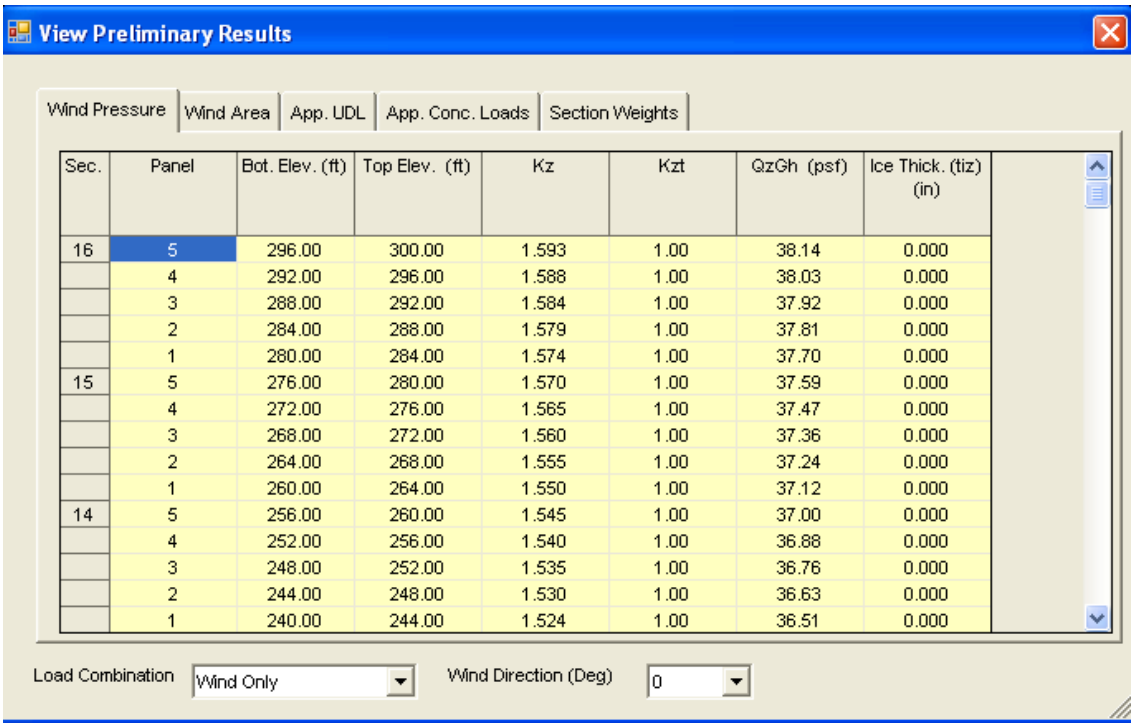
In this Chapter the printed and viewable output options for the program are explained. After the input data phase is completed, the user chooses **Run** from the main menu. The program performs the wind load calculations and executes the structural analysis. From the main menu the user may choose **Results** and the user chooses one from the following menu options:

- Preliminary Results
- Final Results
- Print
- .DXF Profile

Preliminary Results

Choosing Preliminary results the following screen is displayed which allows the user to see the different calculated wind pressure and effective projected areas for each of the different load combinations and wind directions.

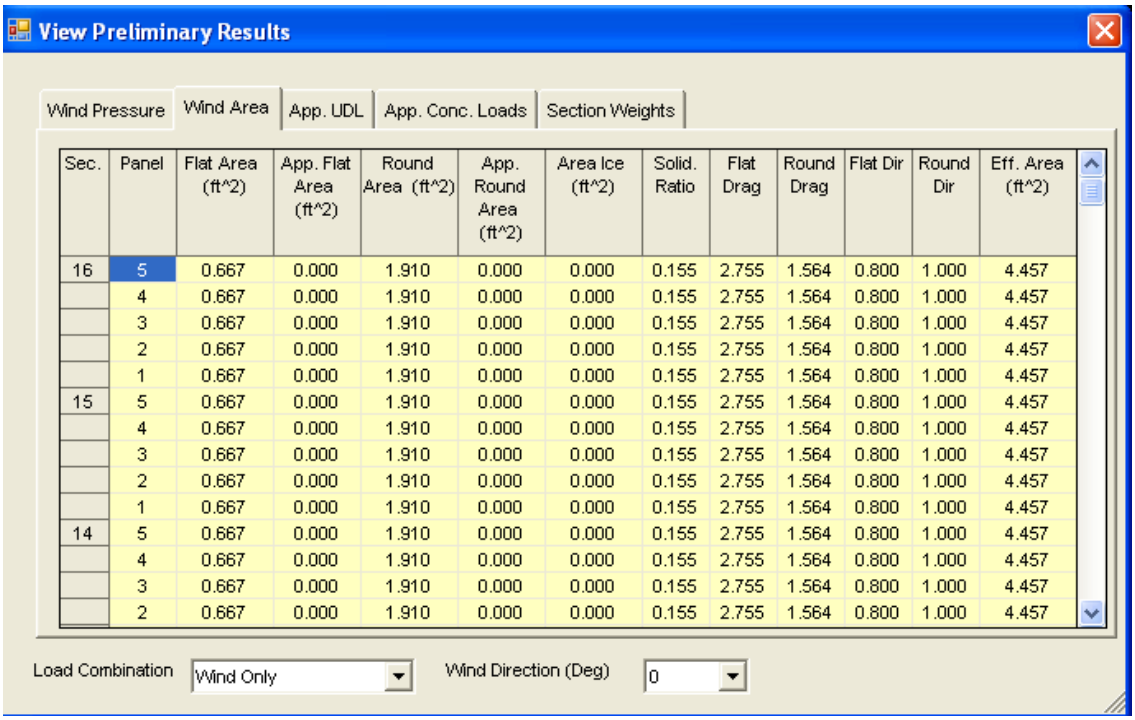
Wind Pressure



Sec.	Panel	Bot. Elev. (ft)	Top Elev. (ft)	Kz	Kzt	QzGh (psf)	Ice Thick. (tiz) (in)
16	5	296.00	300.00	1.593	1.00	38.14	0.000
	4	292.00	296.00	1.588	1.00	38.03	0.000
	3	288.00	292.00	1.584	1.00	37.92	0.000
	2	284.00	288.00	1.579	1.00	37.81	0.000
	1	280.00	284.00	1.574	1.00	37.70	0.000
15	5	276.00	280.00	1.570	1.00	37.59	0.000
	4	272.00	276.00	1.565	1.00	37.47	0.000
	3	268.00	272.00	1.560	1.00	37.36	0.000
	2	264.00	268.00	1.555	1.00	37.24	0.000
	1	260.00	264.00	1.550	1.00	37.12	0.000
14	5	256.00	260.00	1.545	1.00	37.00	0.000
	4	252.00	256.00	1.540	1.00	36.88	0.000
	3	248.00	252.00	1.535	1.00	36.76	0.000
	2	244.00	248.00	1.530	1.00	36.63	0.000
	1	240.00	244.00	1.524	1.00	36.51	0.000

This screen shows bottom and top elevation of each section the applied wind pressure and the relevant height factors and ice thickness for each of the load combination and wind directions.

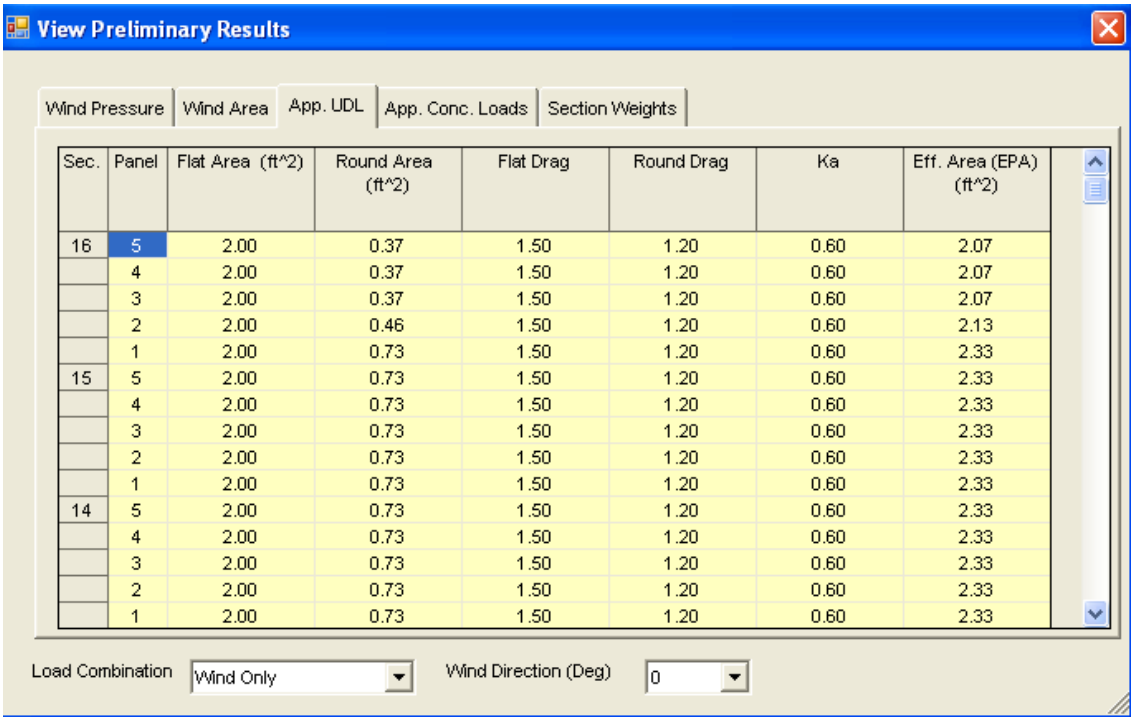
Wind Area



Sec.	Panel	Flat Area (ft^2)	App. Flat Area (ft^2)	Round Area (ft^2)	App. Round Area (ft^2)	Area Ice (ft^2)	Solid. Ratio	Flat Drag	Round Drag	Flat Dir	Round Dir	Eff. Area (ft^2)
16	5	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	4	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	3	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	2	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	1	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
15	5	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	4	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	3	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	2	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	1	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
14	5	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	4	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	3	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457
	2	0.667	0.000	1.910	0.000	0.000	0.155	2.755	1.564	0.800	1.000	4.457

This screen shows flat and round areas of the structure and that of the appurtenances included as part of the structure in the calculations. Solidity ratios are also show for each panel and the calculated force coefficients (drag coefficients) for both flat and round members. Also the directionality factors and the total effective wind areas are shown for each of the load combinations and wind directions.

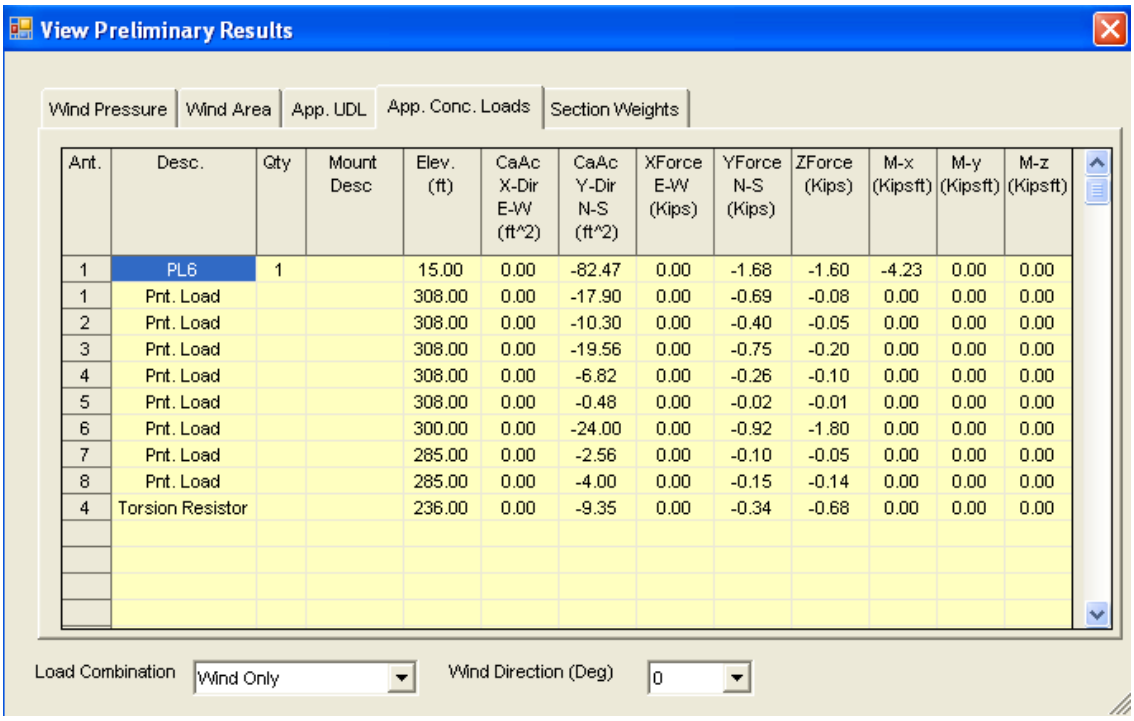
Appurtenances Uniformly Distributed Loads (UDL)



Sec.	Panel	Flat Area (ft^2)	Round Area (ft^2)	Flat Drag	Round Drag	Ka	Eff. Area (EPA) (ft^2)
16	5	2.00	0.37	1.50	1.20	0.60	2.07
	4	2.00	0.37	1.50	1.20	0.60	2.07
	3	2.00	0.37	1.50	1.20	0.60	2.07
	2	2.00	0.46	1.50	1.20	0.60	2.13
	1	2.00	0.73	1.50	1.20	0.60	2.33
15	5	2.00	0.73	1.50	1.20	0.60	2.33
	4	2.00	0.73	1.50	1.20	0.60	2.33
	3	2.00	0.73	1.50	1.20	0.60	2.33
	2	2.00	0.73	1.50	1.20	0.60	2.33
	1	2.00	0.73	1.50	1.20	0.60	2.33
14	5	2.00	0.73	1.50	1.20	0.60	2.33
	4	2.00	0.73	1.50	1.20	0.60	2.33
	3	2.00	0.73	1.50	1.20	0.60	2.33
	2	2.00	0.73	1.50	1.20	0.60	2.33
	1	2.00	0.73	1.50	1.20	0.60	2.33

The appurtenances uniformly distributed loads not included as part of the face are shown on this screen. The window shows the calculated flat and round areas as well as the corresponding force coefficients (drag factors) for each of the load combinations and wind directions.

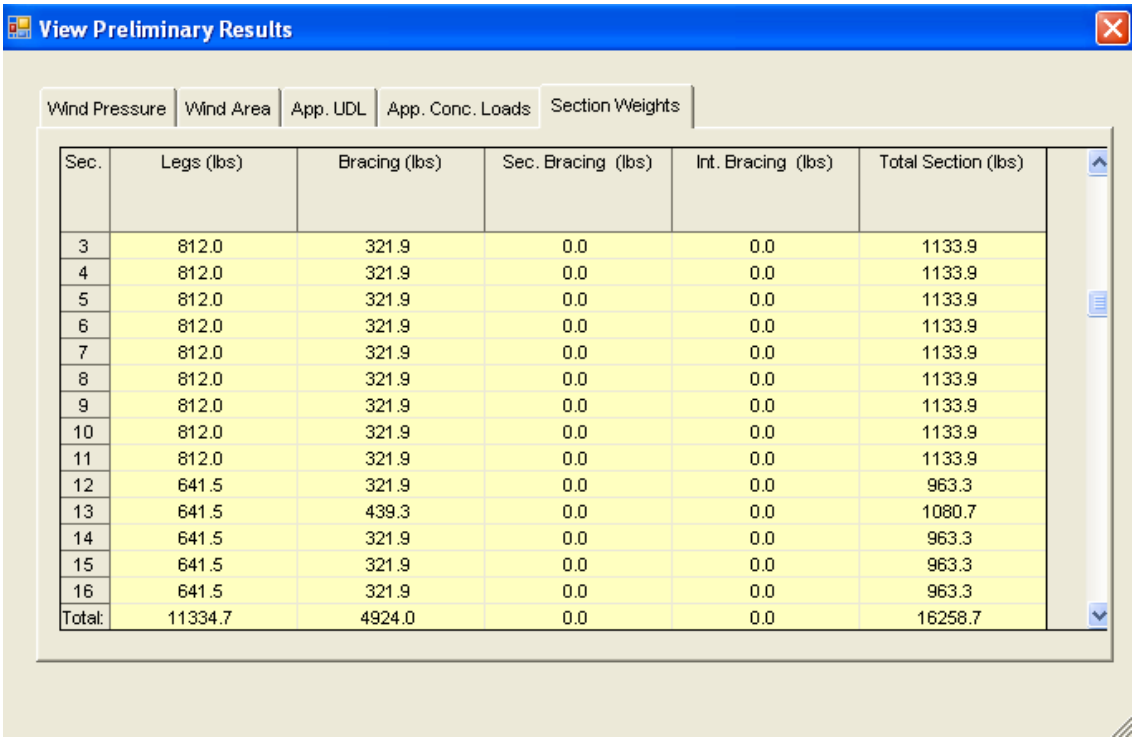
Appurtenances Concentrated Loads (UDL)



Ant.	Desc.	Qty	Mount Desc	Elev. (ft)	CaAc X-Dir E-W (ft^2)	CaAc Y-Dir N-S (ft^2)	XForce E-W (Kips)	YForce N-S (Kips)	ZForce (Kips)	M-x (Kipsft)	M-y (Kipsft)	M-z (Kipsft)
1	PL6	1		15.00	0.00	-82.47	0.00	-1.68	-1.60	-4.23	0.00	0.00
1	Pnt. Load			308.00	0.00	-17.90	0.00	-0.69	-0.08	0.00	0.00	0.00
2	Pnt. Load			308.00	0.00	-10.30	0.00	-0.40	-0.05	0.00	0.00	0.00
3	Pnt. Load			308.00	0.00	-19.56	0.00	-0.75	-0.20	0.00	0.00	0.00
4	Pnt. Load			308.00	0.00	-6.82	0.00	-0.26	-0.10	0.00	0.00	0.00
5	Pnt. Load			308.00	0.00	-0.48	0.00	-0.02	-0.01	0.00	0.00	0.00
6	Pnt. Load			300.00	0.00	-24.00	0.00	-0.92	-1.80	0.00	0.00	0.00
7	Pnt. Load			285.00	0.00	-2.56	0.00	-0.10	-0.05	0.00	0.00	0.00
8	Pnt. Load			285.00	0.00	-4.00	0.00	-0.15	-0.14	0.00	0.00	0.00
4	Torsion Resistor			236.00	0.00	-9.35	0.00	-0.34	-0.68	0.00	0.00	0.00

The appurtenances calculated concentrated loads are shown on this screen. The window shows the calculated flat and round areas in the direction of each axis and the corresponding forces and moments in the three axes. This value can be selected for each of the load combinations and wind directions.

Section Weights



Sec.	Legs (lbs)	Bracing (lbs)	Sec. Bracing (lbs)	Int. Bracing (lbs)	Total Section (lbs)
3	812.0	321.9	0.0	0.0	1133.9
4	812.0	321.9	0.0	0.0	1133.9
5	812.0	321.9	0.0	0.0	1133.9
6	812.0	321.9	0.0	0.0	1133.9
7	812.0	321.9	0.0	0.0	1133.9
8	812.0	321.9	0.0	0.0	1133.9
9	812.0	321.9	0.0	0.0	1133.9
10	812.0	321.9	0.0	0.0	1133.9
11	812.0	321.9	0.0	0.0	1133.9
12	641.5	321.9	0.0	0.0	963.3
13	641.5	439.3	0.0	0.0	1080.7
14	641.5	321.9	0.0	0.0	963.3
15	641.5	321.9	0.0	0.0	963.3
16	641.5	321.9	0.0	0.0	963.3
Total:	11334.7	4924.0	0.0	0.0	16258.7

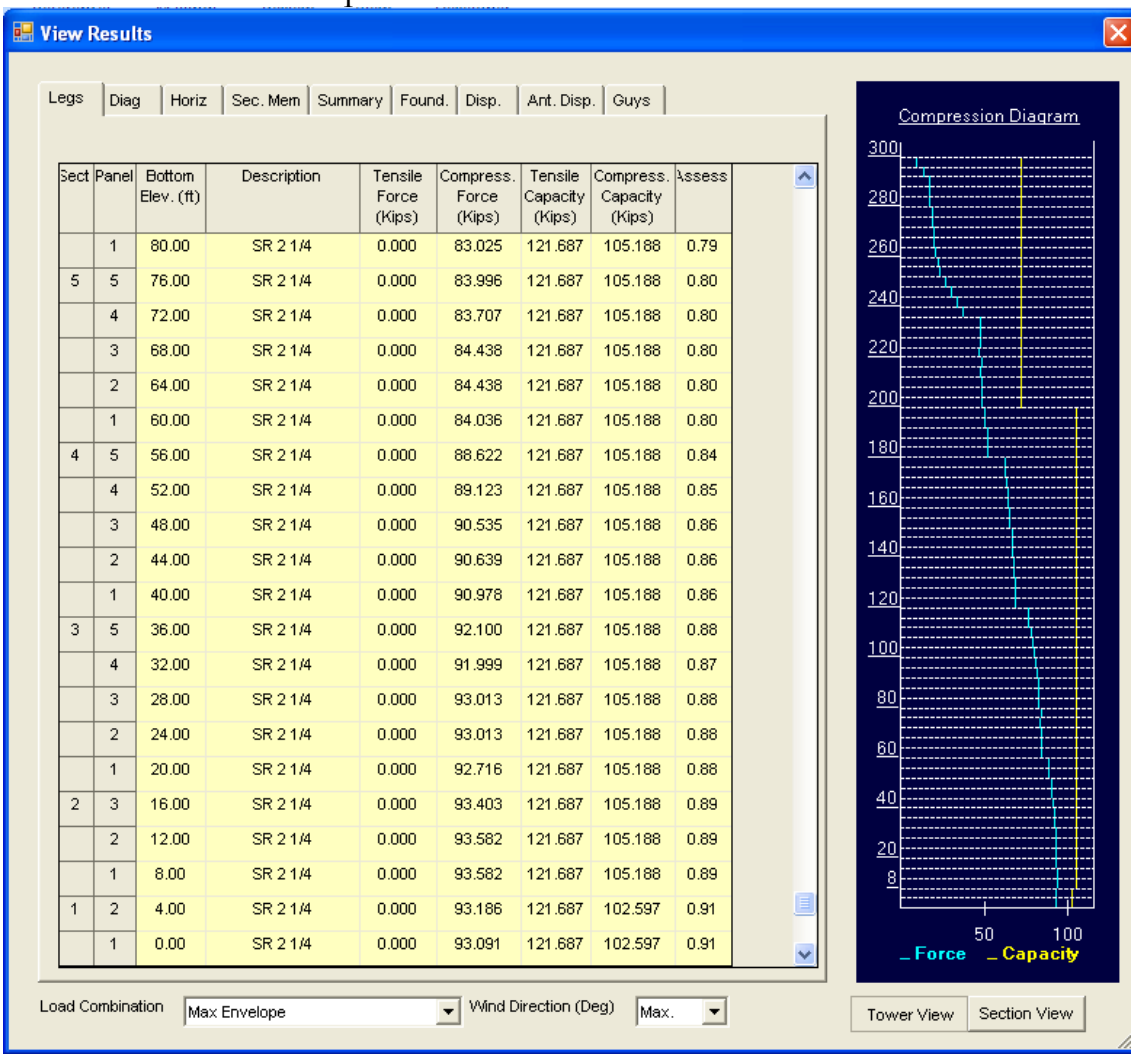
The section weights with divisions for Legs, Bracing, Secondary Bracing, Internal Bracing and Totals are shown in this grid.

Final Results

Choosing final results the following screen is displayed which allows the user to view the results of the analysis for either the maximum envelope or any of the different load combinations and wind directions.

Legs

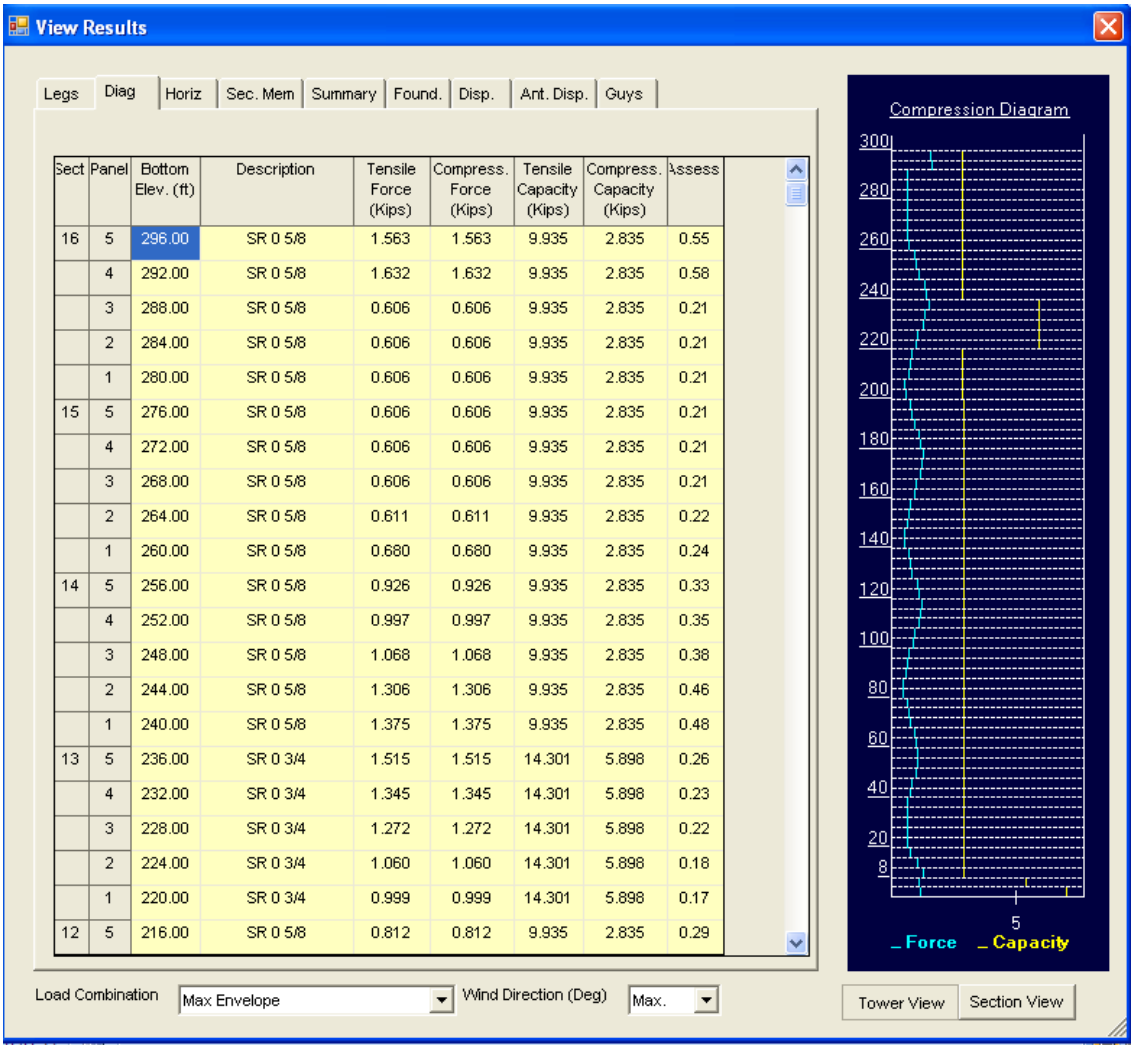
The assessment of the tower legs is shown in tabular form and graphical format by plotting the tensile or compression forces versus the corresponding capacities of the member. The diagram is refreshed based on the selection of either the tensile or compression columns.



Sect	Panel	Bottom Elev. (ft)	Description	Tensile Force (Kips)	Compress. Force (Kips)	Tensile Capacity (Kips)	Compress. Capacity (Kips)	Assess
	1	80.00	SR 2 1/4	0.000	83.025	121.687	105.188	0.79
5	5	76.00	SR 2 1/4	0.000	83.996	121.687	105.188	0.80
	4	72.00	SR 2 1/4	0.000	83.707	121.687	105.188	0.80
	3	68.00	SR 2 1/4	0.000	84.438	121.687	105.188	0.80
	2	64.00	SR 2 1/4	0.000	84.438	121.687	105.188	0.80
	1	60.00	SR 2 1/4	0.000	84.036	121.687	105.188	0.80
4	5	56.00	SR 2 1/4	0.000	88.622	121.687	105.188	0.84
	4	52.00	SR 2 1/4	0.000	89.123	121.687	105.188	0.85
	3	48.00	SR 2 1/4	0.000	90.535	121.687	105.188	0.86
	2	44.00	SR 2 1/4	0.000	90.639	121.687	105.188	0.86
	1	40.00	SR 2 1/4	0.000	90.978	121.687	105.188	0.86
3	5	36.00	SR 2 1/4	0.000	92.100	121.687	105.188	0.88
	4	32.00	SR 2 1/4	0.000	91.999	121.687	105.188	0.87
	3	28.00	SR 2 1/4	0.000	93.013	121.687	105.188	0.88
	2	24.00	SR 2 1/4	0.000	93.013	121.687	105.188	0.88
	1	20.00	SR 2 1/4	0.000	92.716	121.687	105.188	0.88
2	3	16.00	SR 2 1/4	0.000	93.403	121.687	105.188	0.89
	2	12.00	SR 2 1/4	0.000	93.582	121.687	105.188	0.89
	1	8.00	SR 2 1/4	0.000	93.582	121.687	105.188	0.89
1	2	4.00	SR 2 1/4	0.000	93.186	121.687	102.597	0.91
	1	0.00	SR 2 1/4	0.000	93.091	121.687	102.597	0.91

Diagonals

The assessment of the tower diagonals is shown in tabular form and graphical format by plotting the tensile or compression forces versus the corresponding capacities of the member. The diagram is refreshed based on the selection of either the tensile or compression columns. Similar results can be viewed for tower horizontals from the next tab.

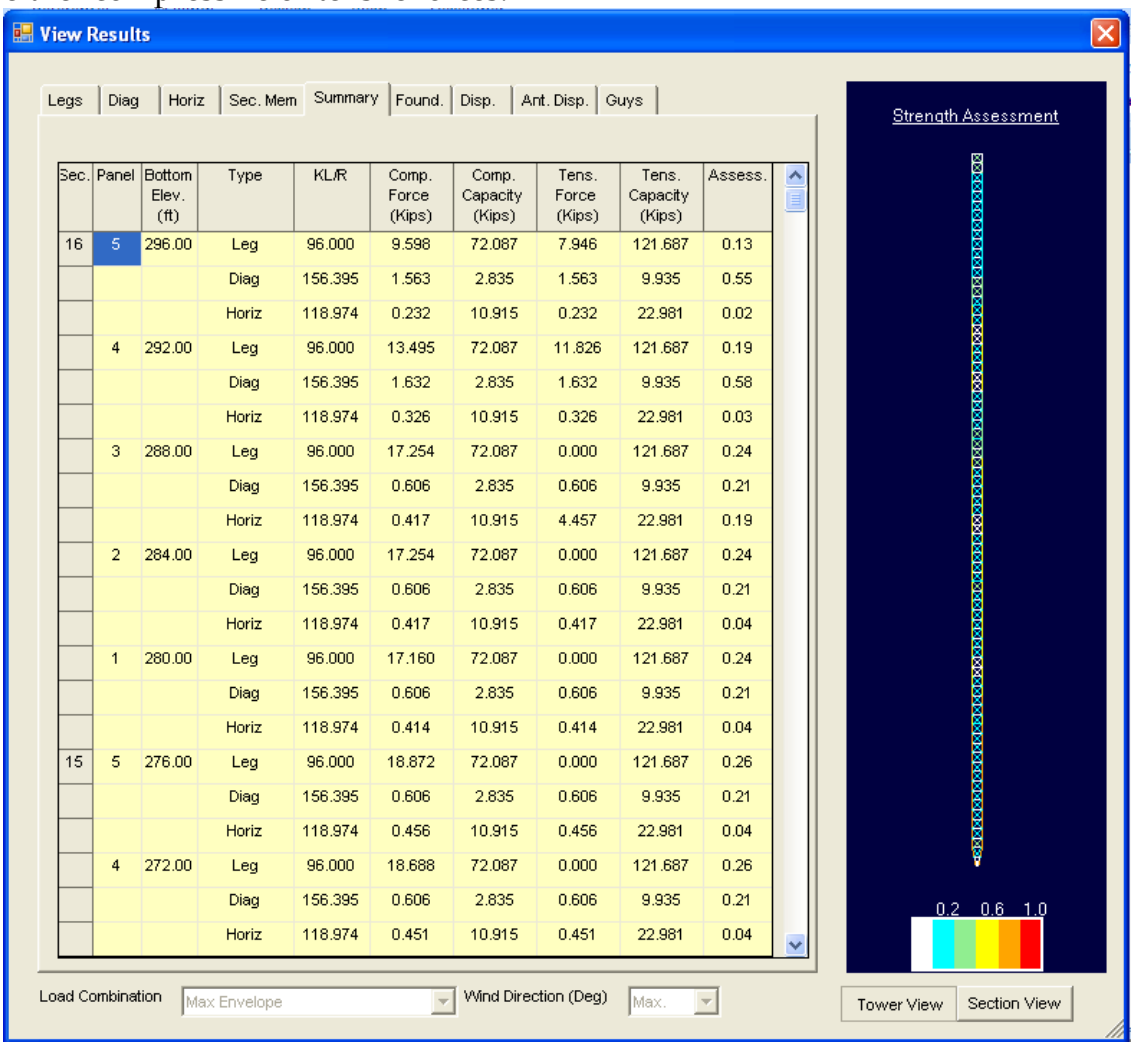


Sect	Panel	Bottom Elev. (ft)	Description	Tensile Force (Kips)	Compress. Force (Kips)	Tensile Capacity (Kips)	Compress. Capacity (Kips)	Assess
16	5	296.00	SR 0 5/8	1.563	1.563	9.935	2.835	0.55
	4	292.00	SR 0 5/8	1.632	1.632	9.935	2.835	0.58
	3	288.00	SR 0 5/8	0.606	0.606	9.935	2.835	0.21
	2	284.00	SR 0 5/8	0.606	0.606	9.935	2.835	0.21
	1	280.00	SR 0 5/8	0.606	0.606	9.935	2.835	0.21
15	5	276.00	SR 0 5/8	0.606	0.606	9.935	2.835	0.21
	4	272.00	SR 0 5/8	0.606	0.606	9.935	2.835	0.21
	3	268.00	SR 0 5/8	0.606	0.606	9.935	2.835	0.21
	2	264.00	SR 0 5/8	0.611	0.611	9.935	2.835	0.22
	1	260.00	SR 0 5/8	0.680	0.680	9.935	2.835	0.24
14	5	256.00	SR 0 5/8	0.926	0.926	9.935	2.835	0.33
	4	252.00	SR 0 5/8	0.997	0.997	9.935	2.835	0.35
	3	248.00	SR 0 5/8	1.068	1.068	9.935	2.835	0.38
	2	244.00	SR 0 5/8	1.306	1.306	9.935	2.835	0.46
	1	240.00	SR 0 5/8	1.375	1.375	9.935	2.835	0.48
13	5	236.00	SR 0 3/4	1.515	1.515	14.301	5.898	0.26
	4	232.00	SR 0 3/4	1.345	1.345	14.301	5.898	0.23
	3	228.00	SR 0 3/4	1.272	1.272	14.301	5.898	0.22
	2	224.00	SR 0 3/4	1.060	1.060	14.301	5.898	0.18
	1	220.00	SR 0 3/4	0.999	0.999	14.301	5.898	0.17
12	5	216.00	SR 0 5/8	0.812	0.812	9.935	2.835	0.29

Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. Specific wind direction may be chosen or simply the envelope of the maximum values.

Results Summary

The user may view the summary of the assessment based on colored contours of the tower. The contours summarize the results depending on the governing either compressive or tensile forces.

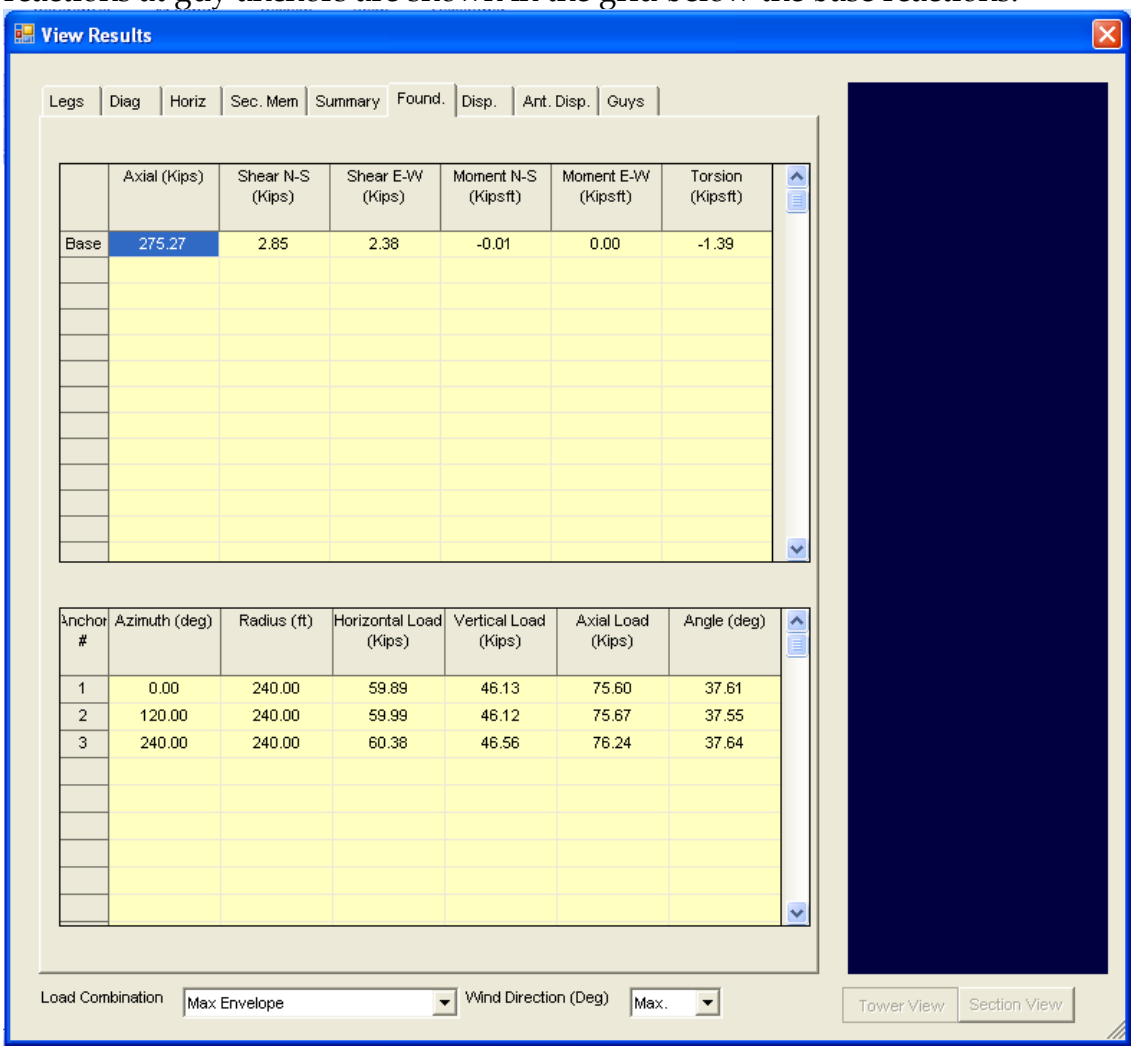


Sec.	Panel	Bottom Elev. (ft)	Type	KL/R	Comp. Force (Kips)	Comp. Capacity (Kips)	Tens. Force (Kips)	Tens. Capacity (Kips)	Assess.
16	5	296.00	Leg	96.000	9.598	72.087	7.946	121.687	0.13
			Diag	156.395	1.563	2.835	1.563	9.935	0.55
			Horiz	118.974	0.232	10.915	0.232	22.981	0.02
	4	292.00	Leg	96.000	13.495	72.087	11.826	121.687	0.19
			Diag	156.395	1.632	2.835	1.632	9.935	0.58
			Horiz	118.974	0.326	10.915	0.326	22.981	0.03
	3	288.00	Leg	96.000	17.254	72.087	0.000	121.687	0.24
			Diag	156.395	0.606	2.835	0.606	9.935	0.21
			Horiz	118.974	0.417	10.915	4.457	22.981	0.19
	2	284.00	Leg	96.000	17.254	72.087	0.000	121.687	0.24
			Diag	156.395	0.606	2.835	0.606	9.935	0.21
			Horiz	118.974	0.417	10.915	0.417	22.981	0.04
	1	280.00	Leg	96.000	17.160	72.087	0.000	121.687	0.24
			Diag	156.395	0.606	2.835	0.606	9.935	0.21
			Horiz	118.974	0.414	10.915	0.414	22.981	0.04
15	5	276.00	Leg	96.000	18.872	72.087	0.000	121.687	0.26
			Diag	156.395	0.606	2.835	0.606	9.935	0.21
			Horiz	118.974	0.456	10.915	0.456	22.981	0.04
	4	272.00	Leg	96.000	18.688	72.087	0.000	121.687	0.26
			Diag	156.395	0.606	2.835	0.606	9.935	0.21
			Horiz	118.974	0.451	10.915	0.451	22.981	0.04

Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. Specific wind direction may be chosen or simply the envelope of the maximum values.

Foundations

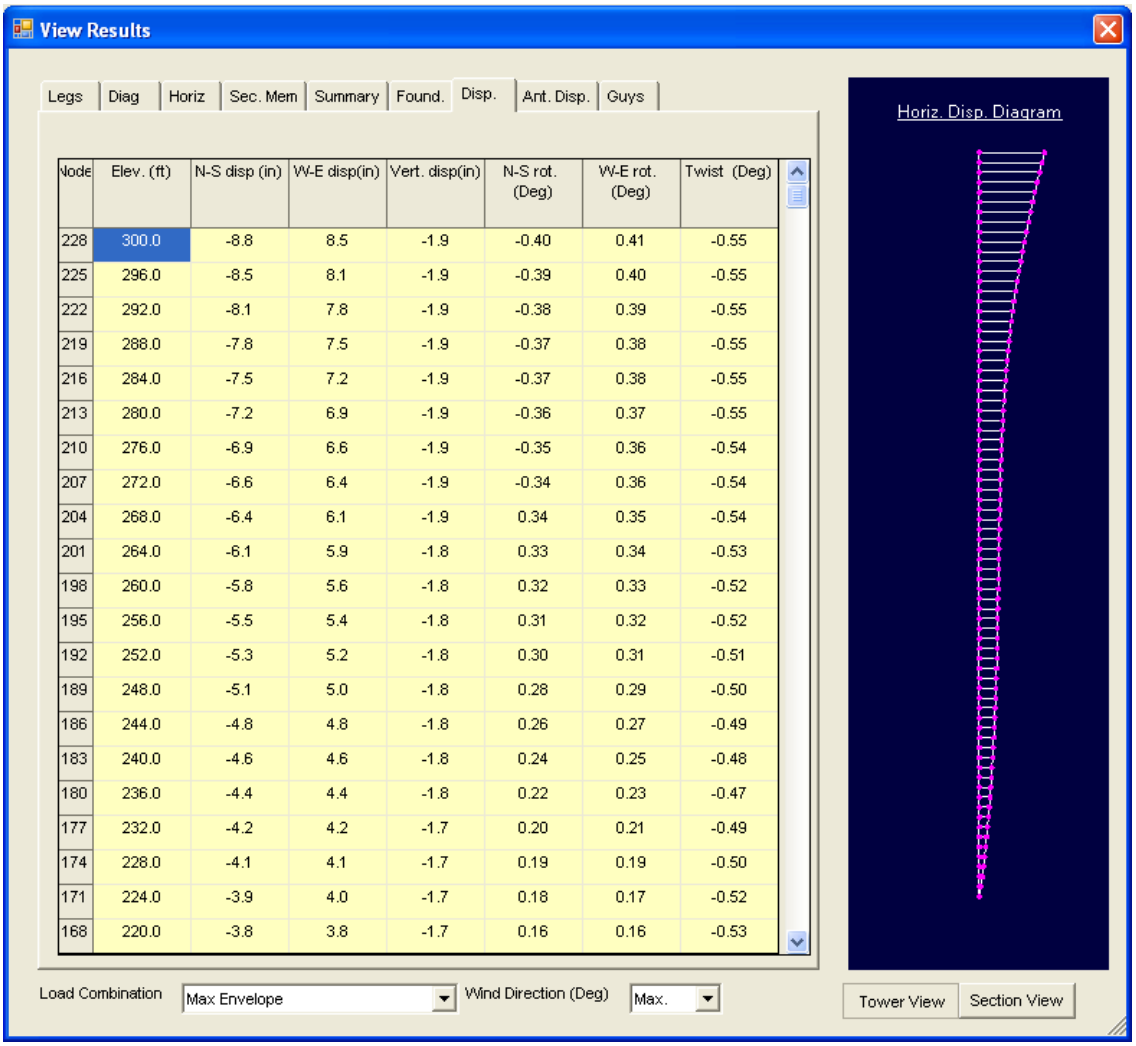
The program calculates the foundations reactions for the mast's base. The reactions at guy anchors are shown in the grid below the base reactions.



Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. Specific wind direction may be chosen or simply the envelope of the maximum values.

Tower Displacements

The user may view the tower displacement and rotations in the three directions from the **Disp.** Tab.

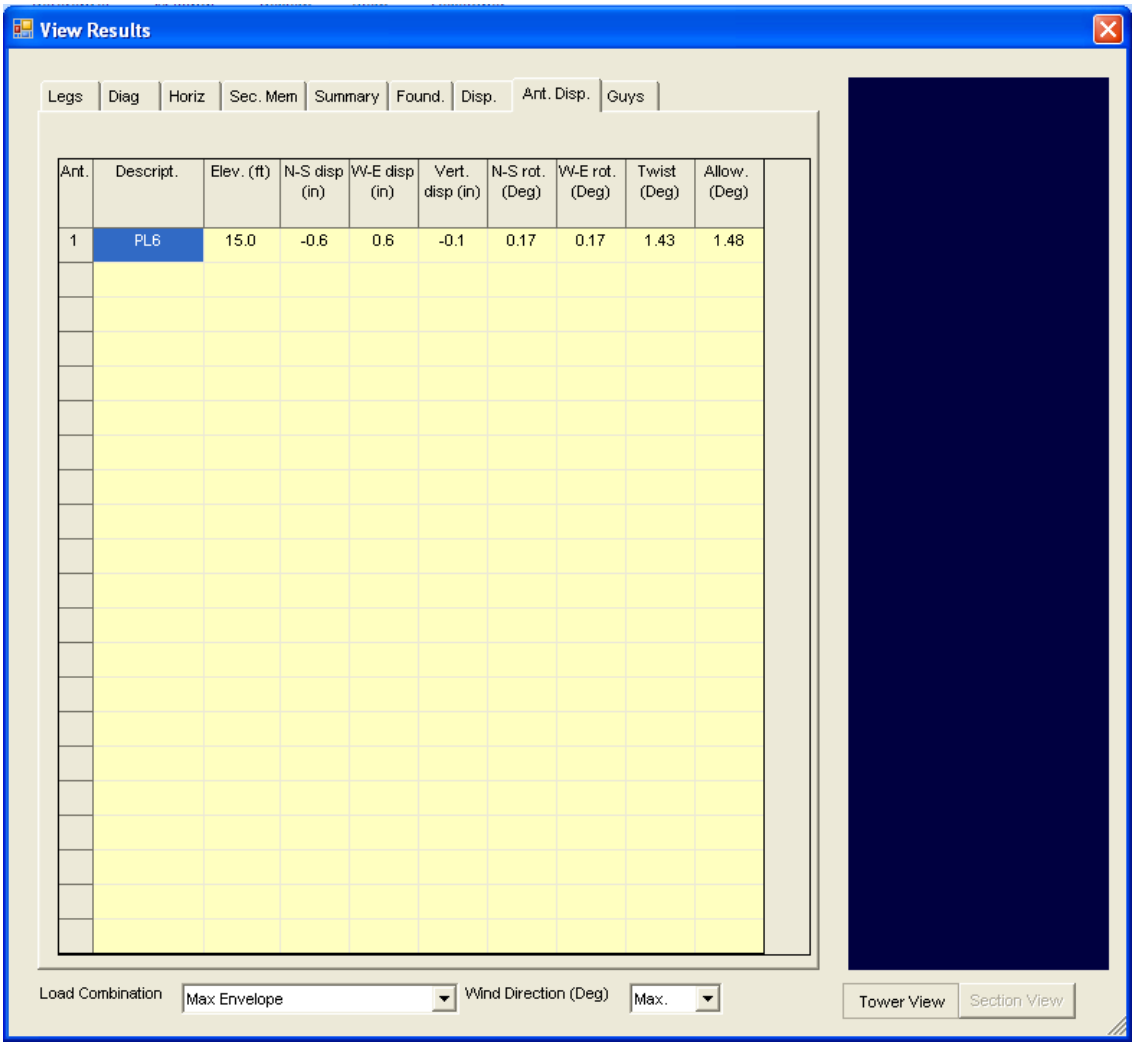


Node	Elev. (ft)	N-S disp (in)	W-E disp(in)	Vert. disp(in)	N-S rot. (Deg)	W-E rot. (Deg)	Twist (Deg)
228	300.0	-8.8	8.5	-1.9	-0.40	0.41	-0.55
225	296.0	-8.5	8.1	-1.9	-0.39	0.40	-0.55
222	292.0	-8.1	7.8	-1.9	-0.38	0.39	-0.55
219	288.0	-7.8	7.5	-1.9	-0.37	0.38	-0.55
216	284.0	-7.5	7.2	-1.9	-0.37	0.38	-0.55
213	280.0	-7.2	6.9	-1.9	-0.36	0.37	-0.55
210	276.0	-6.9	6.6	-1.9	-0.35	0.36	-0.54
207	272.0	-6.6	6.4	-1.9	-0.34	0.36	-0.54
204	268.0	-6.4	6.1	-1.9	0.34	0.35	-0.54
201	264.0	-6.1	5.9	-1.8	0.33	0.34	-0.53
198	260.0	-5.8	5.6	-1.8	0.32	0.33	-0.52
195	256.0	-5.5	5.4	-1.8	0.31	0.32	-0.52
192	252.0	-5.3	5.2	-1.8	0.30	0.31	-0.51
189	248.0	-5.1	5.0	-1.8	0.28	0.29	-0.50
186	244.0	-4.8	4.8	-1.8	0.26	0.27	-0.49
183	240.0	-4.6	4.6	-1.8	0.24	0.25	-0.48
180	236.0	-4.4	4.4	-1.8	0.22	0.23	-0.47
177	232.0	-4.2	4.2	-1.7	0.20	0.21	-0.49
174	228.0	-4.1	4.1	-1.7	0.19	0.19	-0.50
171	224.0	-3.9	4.0	-1.7	0.18	0.17	-0.52
168	220.0	-3.8	3.8	-1.7	0.16	0.16	-0.53

The displacements can also be viewed for the maximum envelope or any of the load combinations or wind directions.

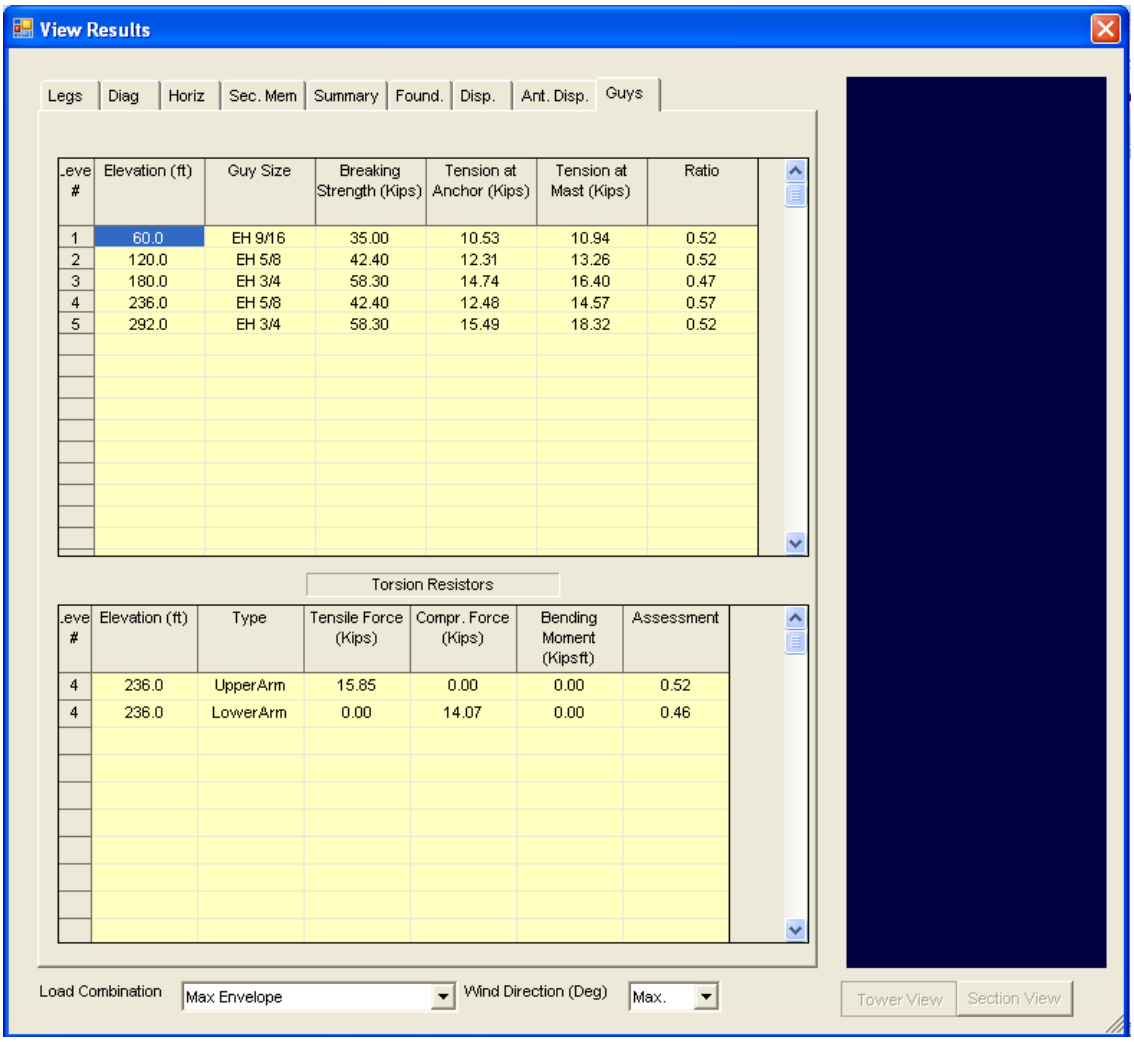
Antenna Displacements

On this screen, the user can see the antenna number (for this example, the tower has 1 antenna at 100 ft. elevation), and the elevation at each antenna, displacements in the orthogonal three directions, rotations and twist.



Guys Assessment

This screen shows the assessment of the guys for each guy level as well as the assessment of the Torsion Resistors (if applicable).



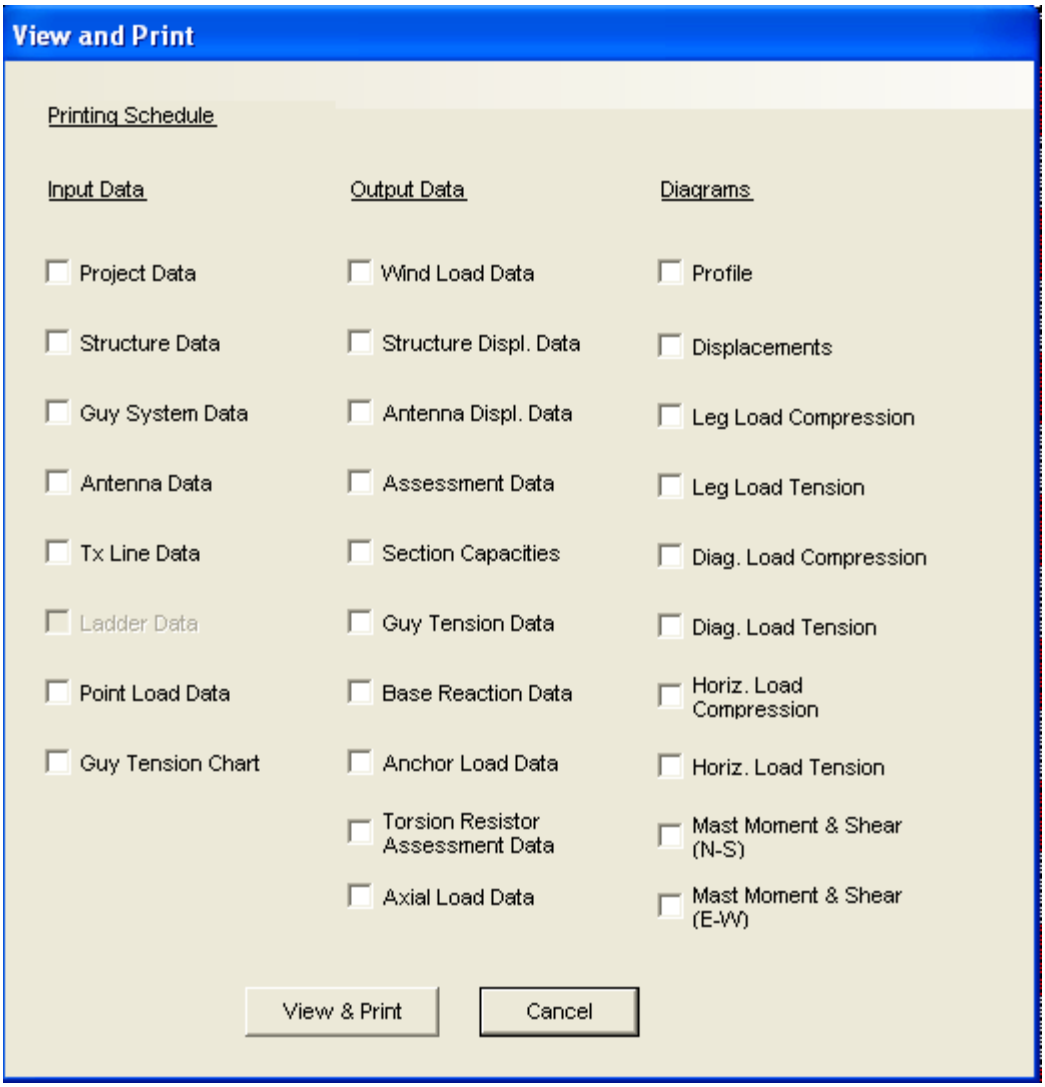
Leve #	Elevation (ft)	Guy Size	Breaking Strength (Kips)	Tension at Anchor (Kips)	Tension at Mast (Kips)	Ratio
1	60.0	EH 9/16	35.00	10.53	10.94	0.52
2	120.0	EH 5/8	42.40	12.31	13.26	0.52
3	180.0	EH 3/4	58.30	14.74	16.40	0.47
4	236.0	EH 5/8	42.40	12.48	14.57	0.57
5	292.0	EH 3/4	58.30	15.49	18.32	0.52

Leve #	Elevation (ft)	Type	Tensile Force (Kips)	Compr. Force (Kips)	Bending Moment (Kipsft)	Assessment
4	236.0	UpperArm	15.85	0.00	0.00	0.52
4	236.0	LowerArm	0.00	14.07	0.00	0.46

Also, on the bottom of the screen, the user may select the load combination for which results are to be displayed. Load combinations selected from the analysis menu are shown in the load combination list. Specific wind direction may be chosen or simply the envelope of the maximum values.

Printout

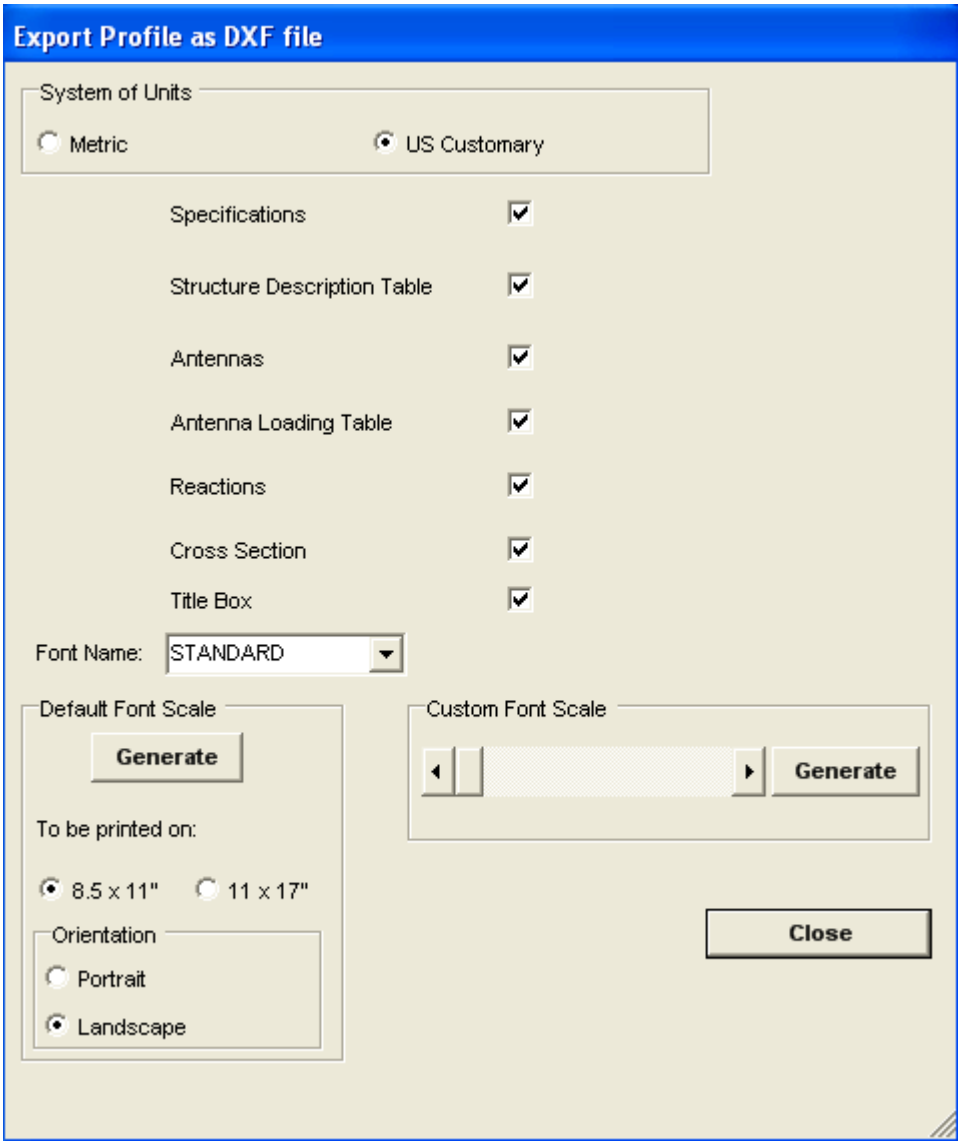
From the main menu, click on **Results**, and the following screen will be displayed. From this screen the user may choose the required printout sections by clicking on the corresponding check box.



Note that for analysis including several load combinations and different wind directions as well the user has the option to select the required sections for printing and also the desired wind directions. This may result in a relatively large number of pages in the printout. By choosing the profile, the program prints a simple profile on which the project data, design specification, base reactions and tower geometry are printed in a graphical format.

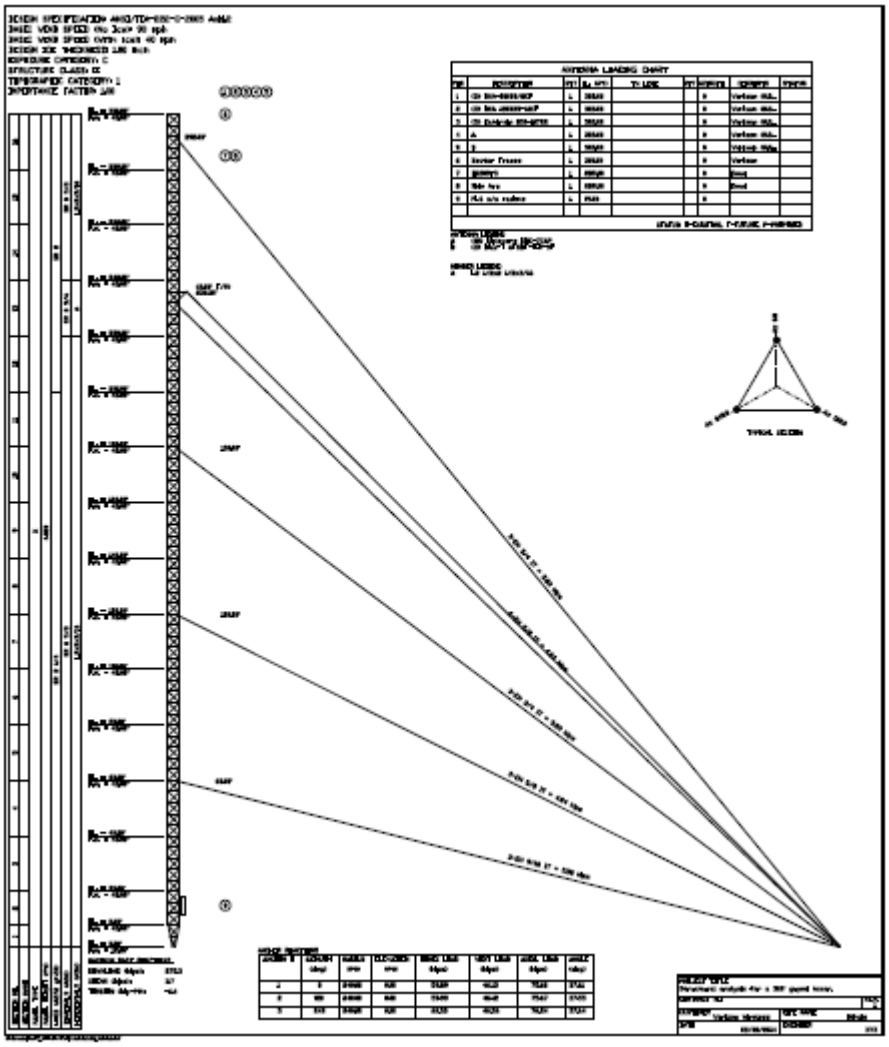
Export Profile

Select sub-menu "**Export Profile (*.dxf)**" from main menu "**Results**". Following screen will be displayed.



Click on "**Generate**" button.

If you have selected DXF viewer you see the profile as shown below.



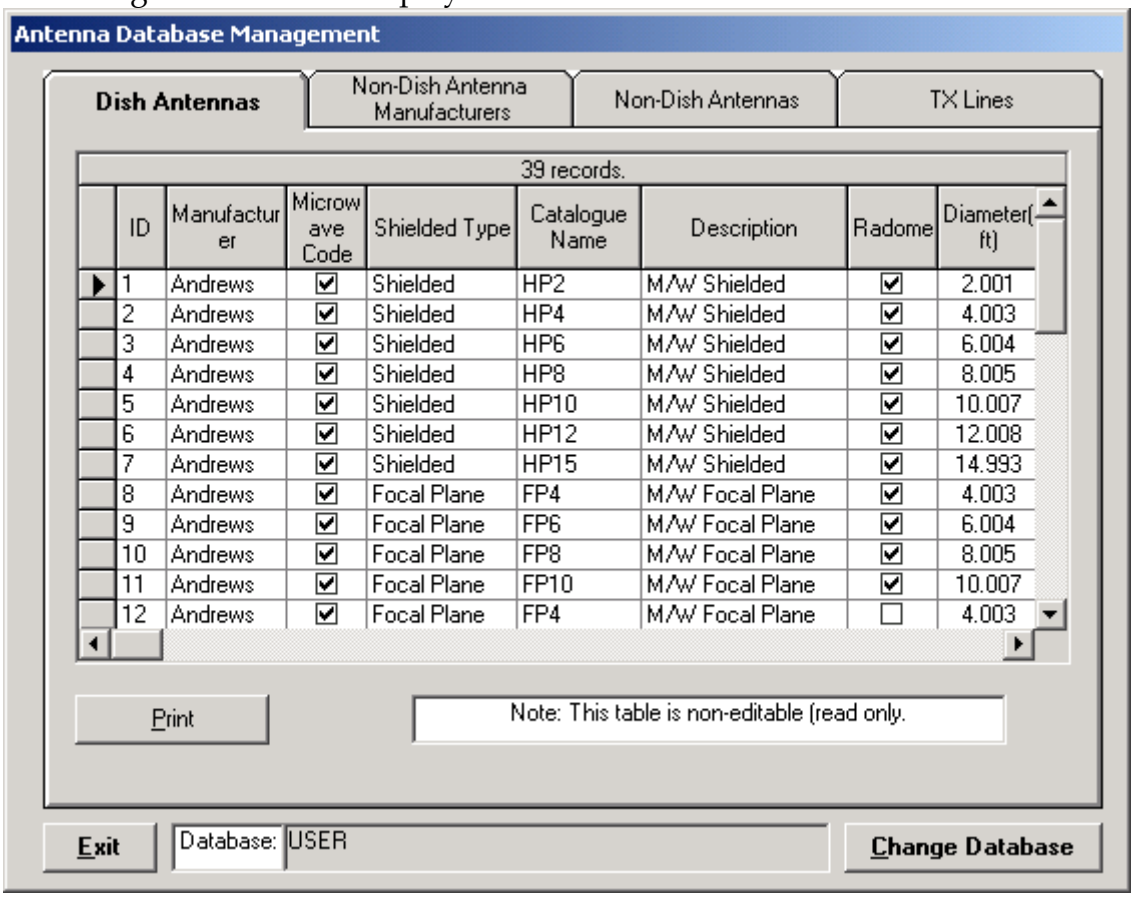
Attached is sample printout of the .dxf profile that can be further edited using AutoCad.

Chapter 5 DATABASE MANAGEMENT

The program uses "USER" Antenna database for selection of the antennas, mounts and TX Lines.
Apart from the "USER" database there is "MASTER" database (not to be modified) and "REMOTE" database or databases, which can be placed in commonly accessed server and thus shared by different users typically within the same organization.
The "USER" and "REMOTE" databases can be modified by the users.

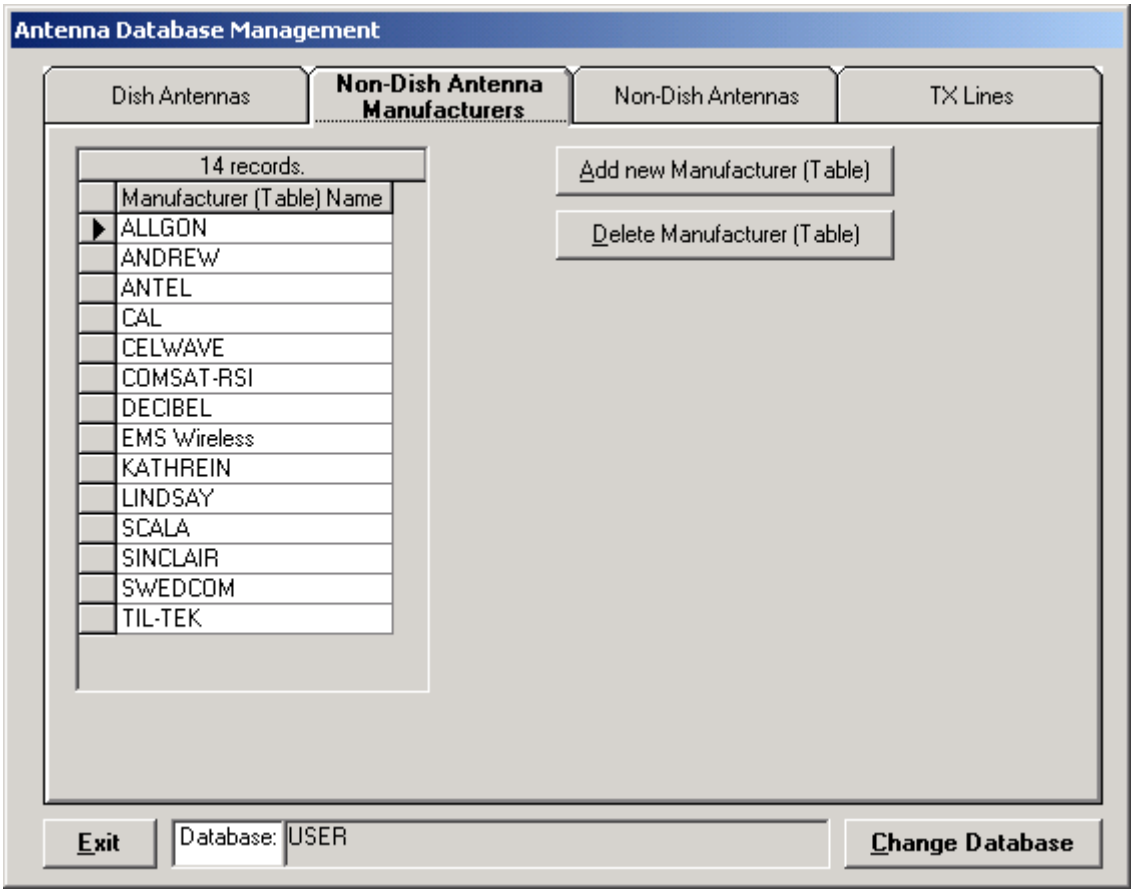
Antennas Database

From the main menu, click on **Database Management** and **Antennas** submenu. Following screen will be displayed.



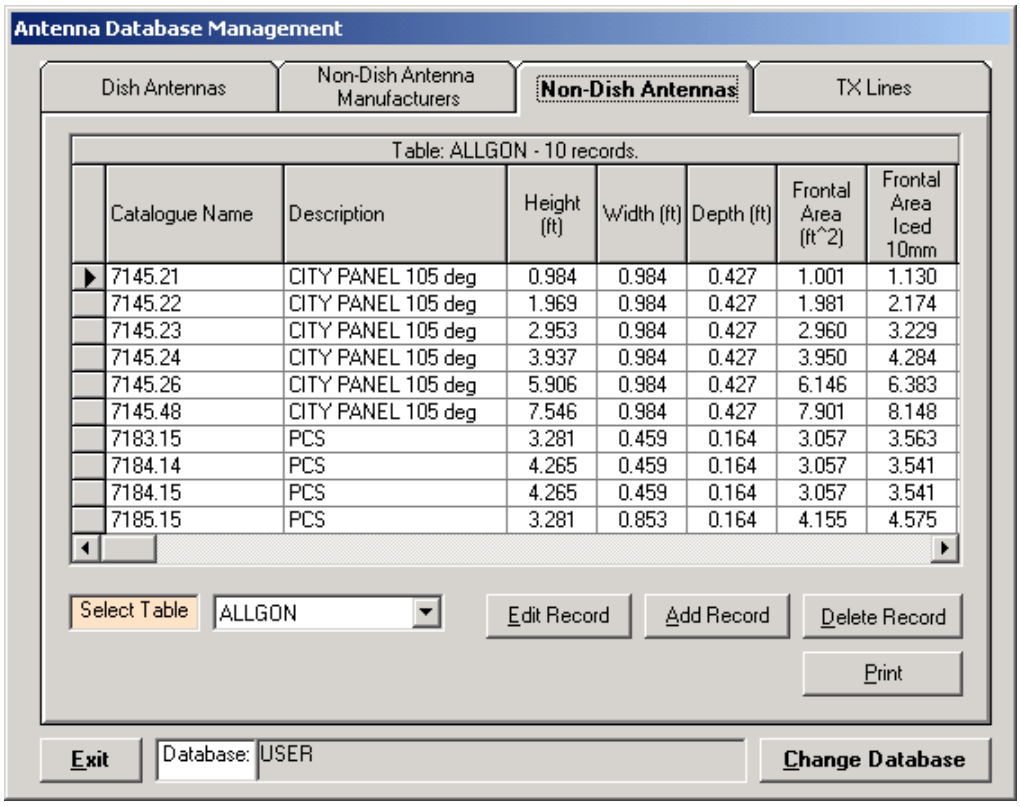
The first group "Dish Antennas" is non-editable (read only), as it covers practically all types and sizes of Microwave Dish Antennas.

Non-Dish Antenna Manufacturers

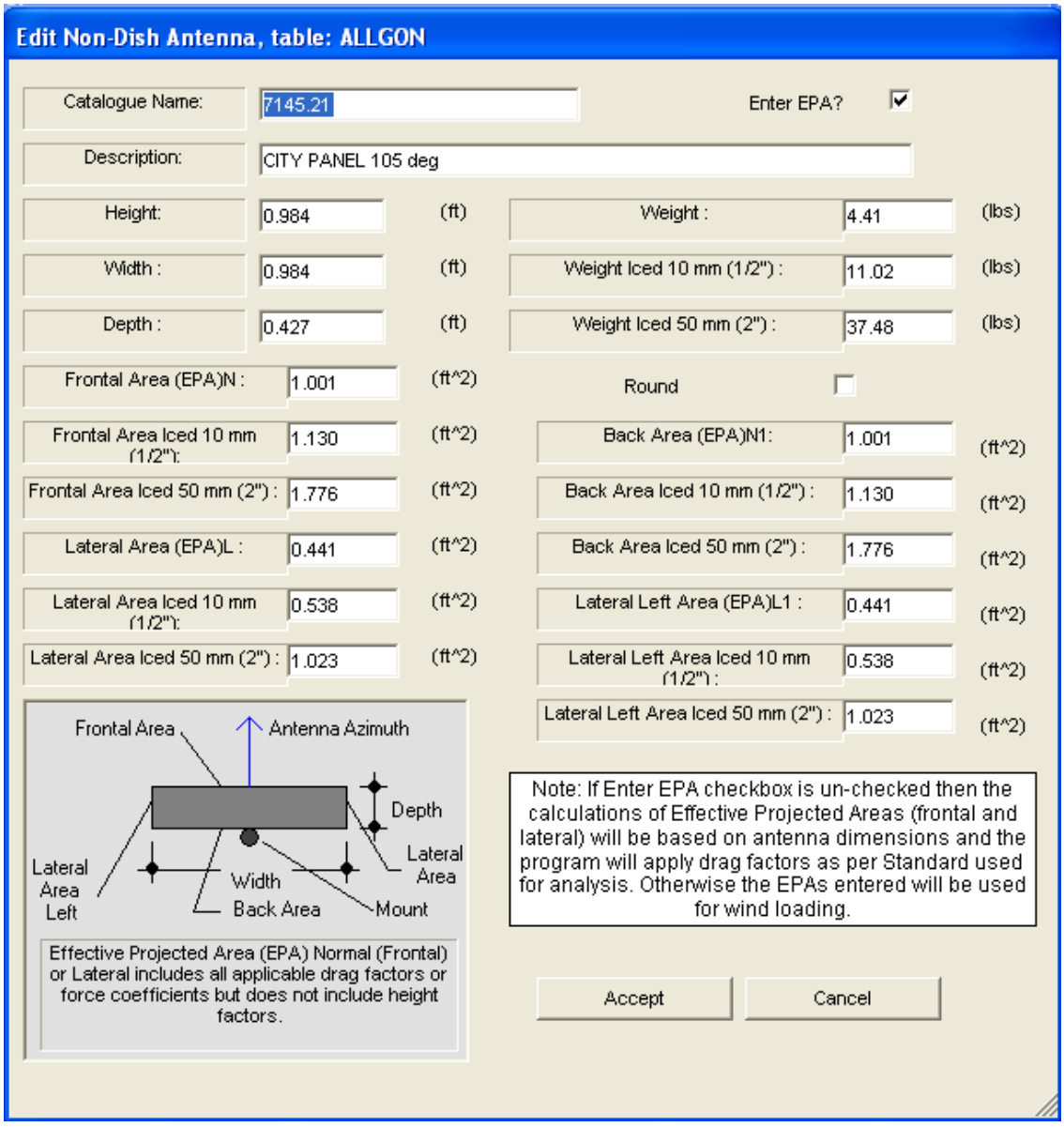


User can add new manufacturers. Such action will create an empty table and the user will then add records to such table. Deletion of manufacturer will cause removal of the manufacturer name from the list as well as deletion of a table of non-dish antennas associated with the manufacturer.
Default database is "USER" but any other "REMOTE" database can be selected (if present) and then the modifications are performed in the selected database. Once such "Remote" database is created and located remotely (on a server) other users can synchronize their local "USER" database with the "REMOTE".

Non-Dish Antennas

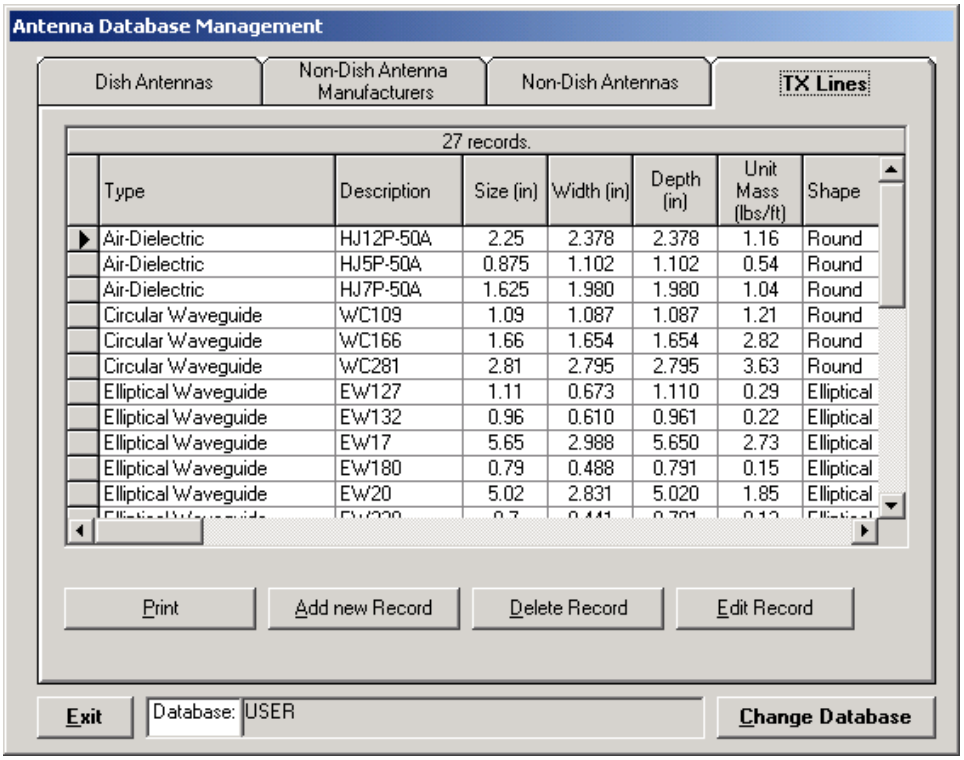


To edit or add record for selected table click on "Edit Record" or Add Record" buttons. Following screen will be displayed.

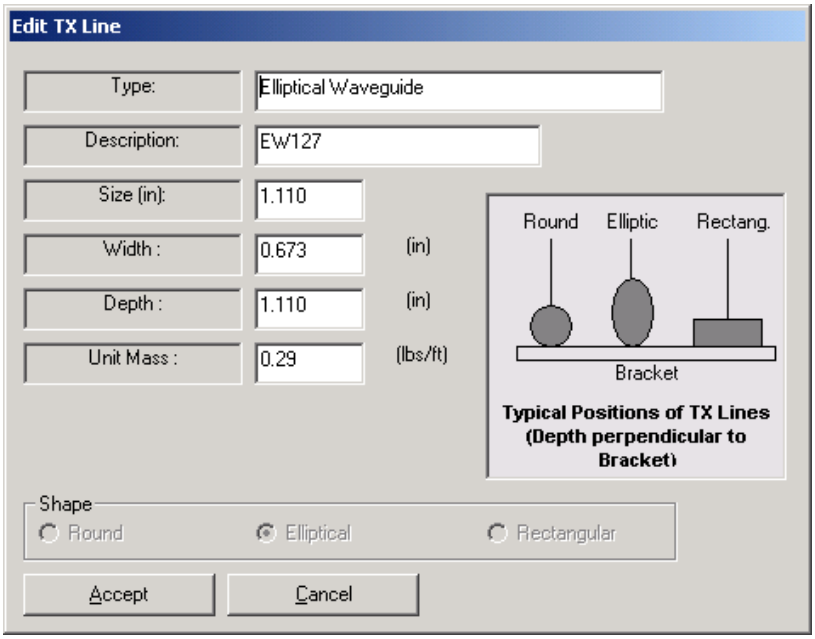


Changes of entries will become effective after "Accept" button is clicked.

TX Lines



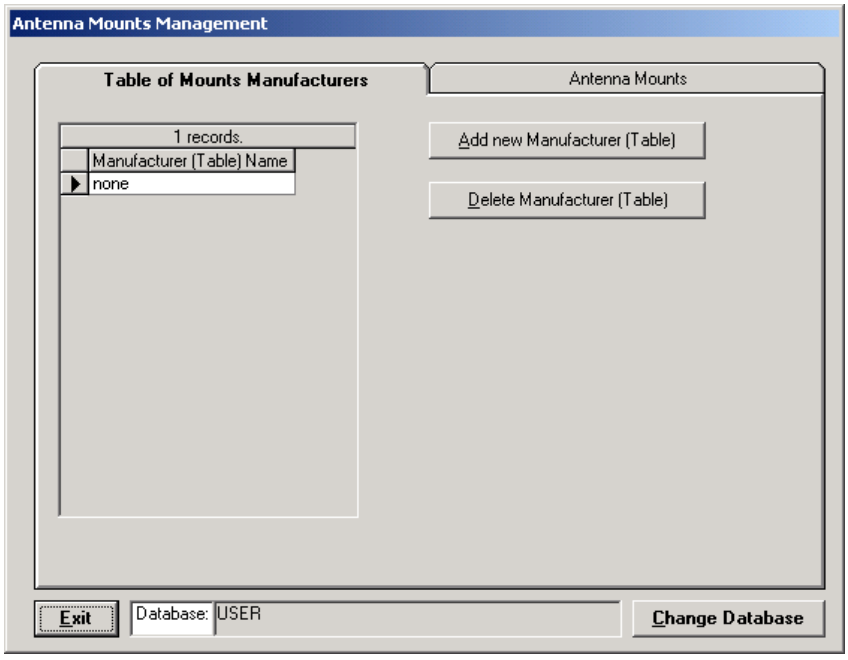
To add or edit a record click on "Add new Record" or "Edit Record" buttons respectively. Following screen will be displayed.



Changes of entries will become effective after "Accept" button is clicked.

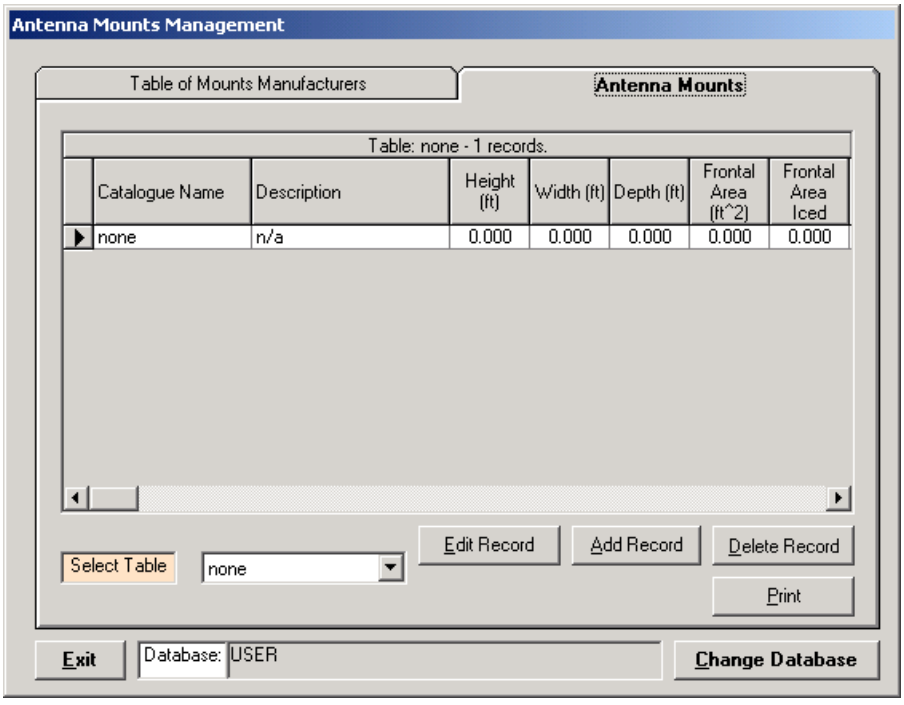
Mounts

From the main menu, click on **Database Management** and **Mounts** submenu. Following screen will be displayed.

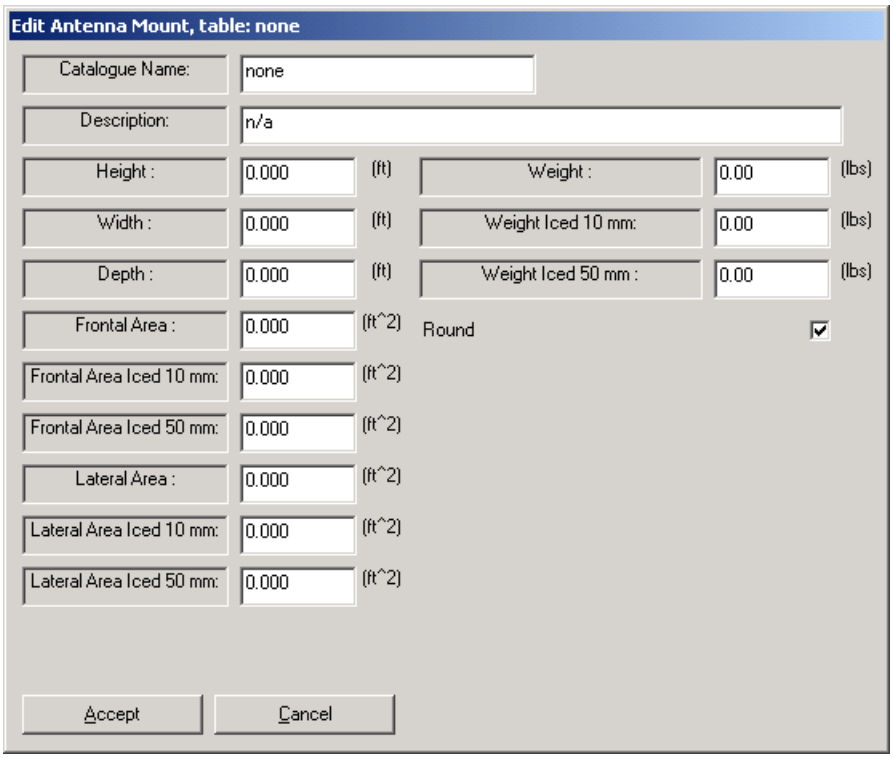


User may add new manufacturers (as is the case with Non-Dish Antennas) or delete them.

Antenna Mounts



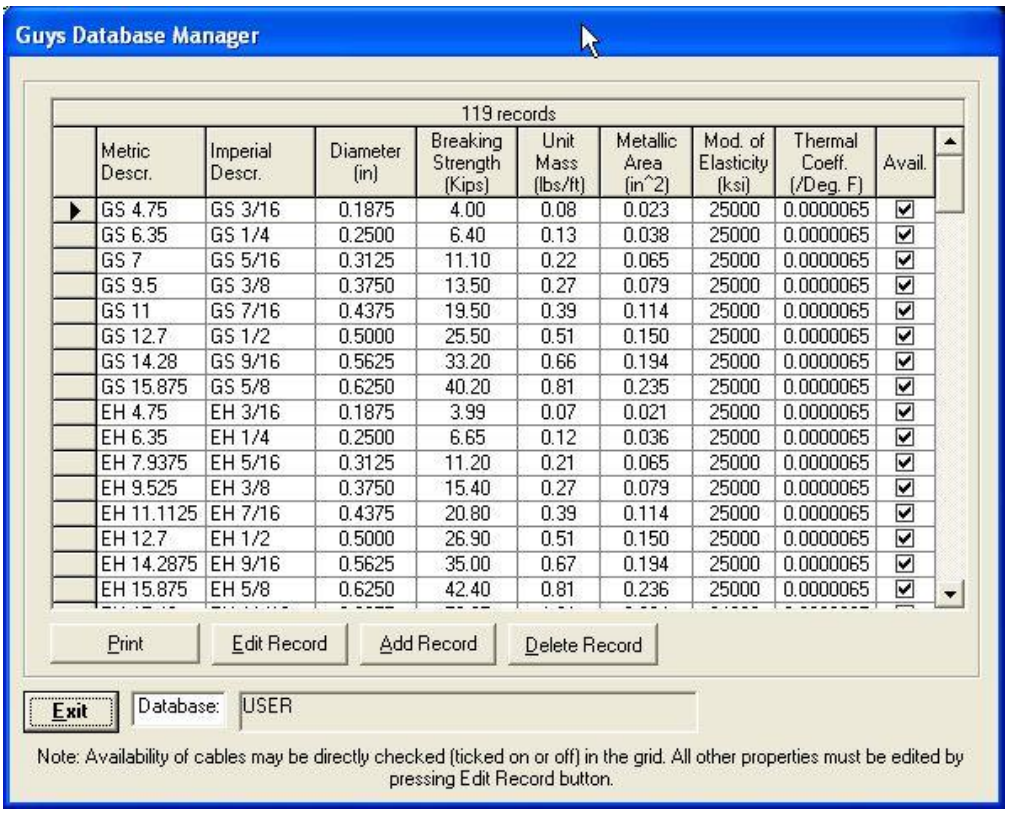
To add or edit a record click on "Add new Record" or "Edit Record" buttons respectively. Following screen will be displayed.



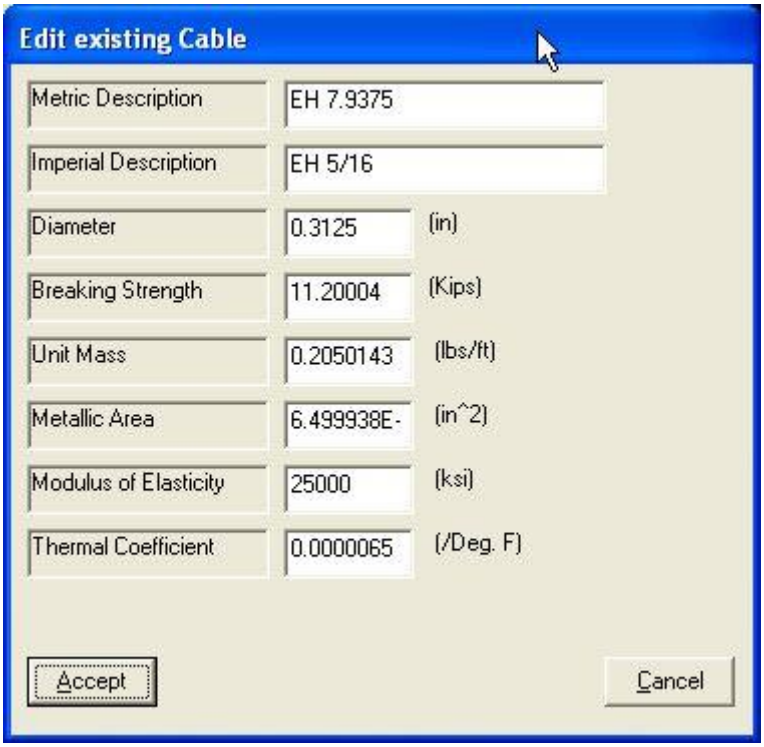
Perform the editing and press "Accept" button to update or "Cancel" otherwise.

Guys Database

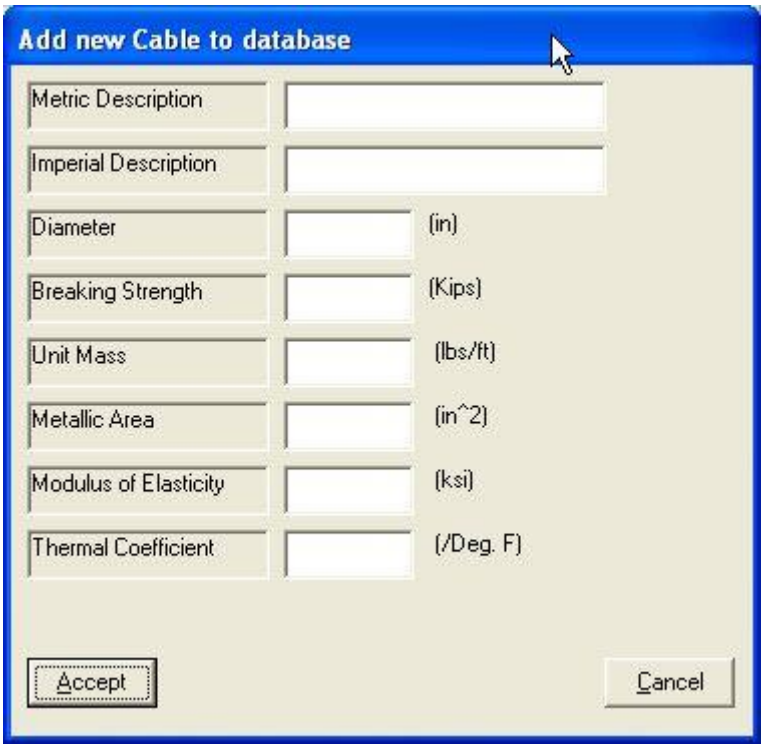
From the main menu, click on **Database Management** and **Mounts** submenu. Following screen will be displayed.



When "Edit Record" button is clicked following form appears:



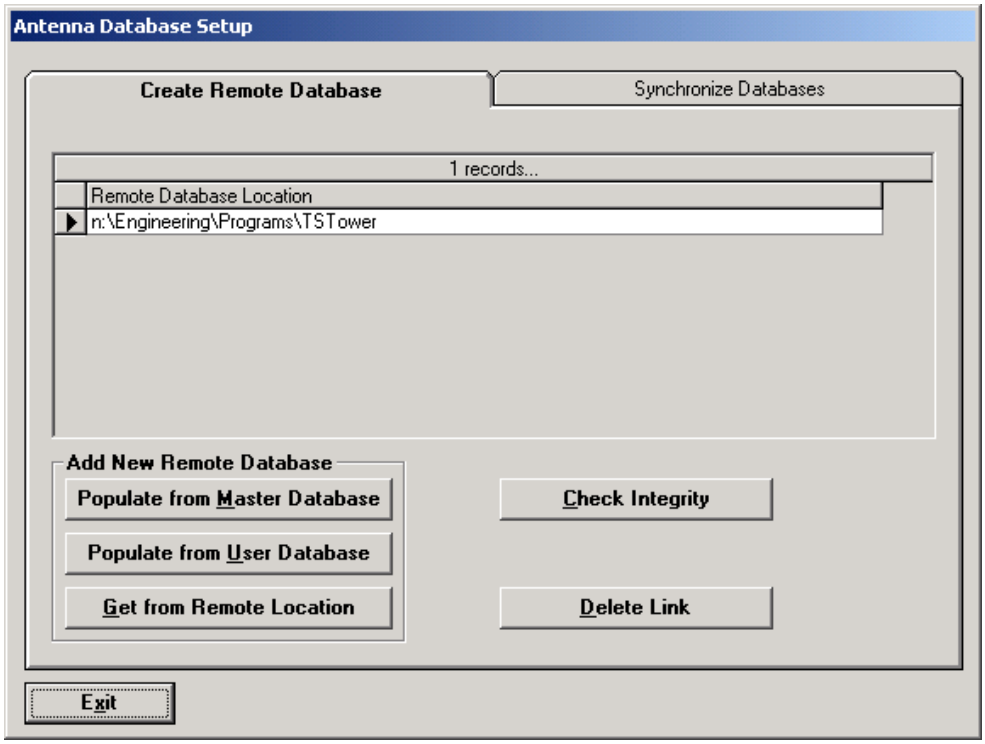
When "Add Record" button is clicked following form shows:



After the data is entered for the new addition clicking on "Accept" button will add the new guy to the table. "Cancel" button should be used to ignore the entries (new Guy will not be added).

Database Setup

From the main menu, click on **Database Management** and **Database Setup** submenu.
Following screen will be displayed.



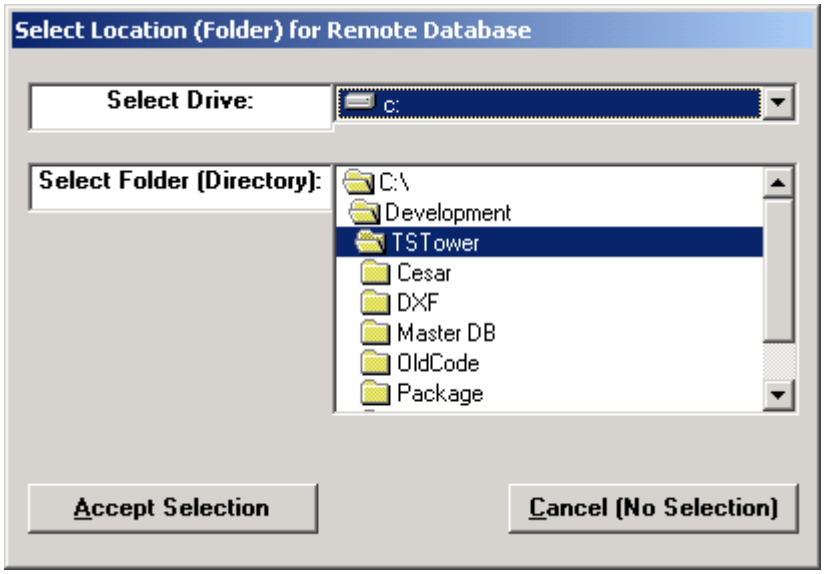
New remote database(s) can be created in three different ways:

"Populate from Master Database" – new "remote" database will be replicated from "Master Database" and then user will indicate the location of the new database – screen showing selection of "drives" and "folders" will be displayed.

Populate from Master Database

"Populate from User database" – new "remote" database will be replicated from "User Database" and then user will indicate the location of the new database – screen showing selection of "drives" and "folders" will be displayed.

"Get from Remote Location" this action will prompt you to select existing "remote" database not linked to you computer so far.



Once you selected the "remote" database it will be displayed in the "Remote Database Location" grid and you will have access to it via Database Management for Antennas or Mounts.

"Check Integrity" will perform the comparison of the records between selected remote database and "Master Database" and display differences.

"Delete Link" will remove a link between selected "remote" database and the program (such database will not be accessible to you).

Synchronize Databases – instructions as shown on attached screenshot.